Gear Design Software KISSsoft

KISSsoft
Release 2024

User Manual

Sharing Knowledge

Table of Contents

I

General

Chapter 1 - 12

1 Installing KISSsoft

1.1 Basic installation

After you have inserted the KISSsoft CD in the appropriate disk drive, the setup program starts automatically. If it does not, you can run the setup.exe file directly in the CD root directory by double-clicking on it.

The setup program guides you through the installation process step by step. All you need to do is select an installation folder and the required language for the installation. If you change the default installation folder, it is advisable to include the version descriptor as part of the directory name of the new installation folder (e.g. C:/Programs/KISSsoft xx-20xx).

At the end of installation, we recommend that you install the latest Service Pack (patch). Download the latest patch from our website. You can choose between an installation program (.exe) and zipped files (.zip). The installation program automatically copies the necessary files after you specify which installation folder it is to use. However, not all companies permit .exe files to be downloaded. If not, you must unpack the ZIP file and manually copy the files it contains into your installation folder. Any files that are already present must be overwritten by the ones contained in the patch.

Once you have installed KISSsoft, you will need a license for it (see chapter 1.3, Licensing). If KISSsoft is not licensed, it will only run as a demo version.

► **Note:**

If you are installing KISSsoft on a server, we recommend that you perform the installation from a client (workstation computer). This means that all the necessary directory entries will automatically be added to the KISS.ini file (see chapter 2.6.1, Definitions in [PATH]) correctly. Otherwise, you will have to change these directory entries from the local drive name (e.g. C:/...) to the appropriate share name in the network, later, manually, using an editor.

1.2 Downloading a license file

1. Go to the KISSsoft website at www.KISSsoft.com.
2. Register in the **MyKISSsoft** area there, and inform your commercial contact representative or send an email to info@KISSsoft.com.
3. After you have been assigned the required rights by KISSsoft AG, select the menu at the top of the **Customer area** and then download your license file.

► **Note:**

You will see license files for different KISSsoft versions in your personal download area. Ensure that you download the appropriate license file for the version you have just installed.

1.3 Licensing

After you have completed the KISSsoft installation (see chapter 1.1, Basic installation), you must license the software either by downloading a license file or activating the program's license. Please read the relevant section for your license type.

1.3.1 Test version

1. If you run KISSsoft from the client (workstation computer), the user account for the test version will become active.
2. Select **License tool** in the **Extras** menu, and then click on the Activate license tab.
3. **Activating the license online:** If your computer has Internet access, and you have received an online code from us, enter this code under the **Activate Test or Student version** option and then click on the **Activate license** tab.
4. **Activating the license directly:** Under the **Activate test version by phone** option, you see a question code. Call the telephone number you see there and tell us this code. We will then give you the appropriate answer code. Input this in the appropriate field, and then click on the **Activate license** tab.

1.3.2 Student version

1. Copy your license file (which you will usually be given by the place you are studying for your qualification) to your License directory (see chapter 2.6.2, Definitions in [SETUP]).
2. Select **License tool** in the **Extras** menu, and then click on the Activate license tab.
3. To input your online code (which you will usually be given by the place you are studying for your qualification), select the **Activate test or student version (online code required)** option and then click on **Activate license**.

1.3.3 Single user version with dongle (protection key)

1. Copy your license file (see chapter 1.2, Downloading a license file) to your License directory (see chapter 2.6.2, Definitions in [SETUP]))
2. Now, simply plug in the dongle supplied with the system.

► **Note**

The single user version of KISSsoft can also be installed on a central server. Local clients (workstation computers) can then run the software directly from this server. Please note that, in this case, the dongle must always be plugged into the particular client on which you want to use KISSsoft.

1.3.4 Single user version with license code

1. Start KISSsoft from the client (workstation computer) for which the software is to be licensed.
2. Select **License tool** in the **Extras** menu, and then click on the Activate license tab.
3. Enter your contact data under the **Request license file** option and click on **Send** to send your computer-specific access data directly to us. Alternatively, you can first save this access data in a file and then send us this file by email.
4. You will receive an email as soon as we have created your license file.
5. Download your License file (see chapter 1.2, Downloading a license file) and copy it to your License directory (see chapter 2.6.2, Definitions in [SETUP]).

1.3.5 Network version with dongle (protection key)

For the network version with dongle a server program has to be installed in addition to the licensing of the KISSsoft installation.

1.3.5.1 Installation on the server

1. Copy the KISSsoft dongle/MxNet installation directory onto a server.
2. Start MxNet32 on the server. You will see a dongle icon in the task bar.
3. Double-click this icon to start the user interface.
4. Now enter **Application**: KISSsoft and any file with the file extension .mx as the **server file**. The clients must have both read and write access to this file. Now click **New Entry** to add this entry.
5. Then click the **Active Users** button to check who is using KISSsoft. You can also reactivate a license that has already been used.

1.3.5.2 Licensing the KISSsoft system

1. Copy your license file (see chapter 1.2, Downloading a license file) to your License directory (see chapter 2.6.2, Definitions in [SETUP]).

2. Complete the necessary details in the "ServerFile: serverfilepath" line after the checksum line in the license file. The "serverfilepath" is the path to the server file that is defined in the server program.

► **Note**

The KISSsoft installation will also run if the client is not connected to the network and if the dongle is inserted in the client instead of in the server. You can also "check out" the license if you remove the dongle.

1.3.6 Network version with a license code

1. Start KISSsoft from a client (workstation computer).
2. Select **License tool** in the **Extras** menu and go to the General tab.
3. Select an access directory on a server. Please note: If you change this, you will need a new license.
4. Open the Activate license tab.
5. Enter your contact data under the **Request license file** option and click on **Send** to send your computer-specific access data directly to us. Alternatively, you can first save this access data in a file and then send us this file by email.
6. You will receive an email as soon as we have created your license file.
7. Download your License file (see chapter 1.2, Downloading a license file) and copy it to your License directory (see chapter 2.6.1, Definitions in [PATH]).

2 Setting Up KISSsoft

2.1 Directory structure

If there are several users, it is advisable to store shared data (**databases**, user-defined **report templates** and **standard files**) on a server. This ensures that, if there are changes and upgrades, all users will be able to work with one uniform set of data. To set this up, put the UDB, EXT and TEMPLATE directories on a server that can be accessed by all users, and then set the corresponding variables, UDBDIR, EXTDIR and TEMPLATEDIR, in the KISS.ini (see chapter 2.6.1, Definitions in [PATH]) file.

In contrast, if there are several users, the temporary directories should be defined locally on their workstations. Otherwise, the interim results generated for individual users might overwrite each other. For each installation, KISSsoft uses the temporary user directory set in the operating system. The CADDIR and TEMPDIR variables can, however, be tailored in the KISS.ini (see chapter 2.6.1, Definitions in [PATH]) file.

If you want to open or save a calculation file or report, KISSsoft displays your own personal **user directory** as the first choice storage location. This saves you frequent searches in the directories on your system. You can define this user directory via the USERDIR variable in the KISS.ini (see chapter 2.6.1, Definitions in [PATH]) file. The user directory will be ignored if you have selected an Active working project (see chapter 6.3, The active working project). In this case, KISSsoft offers you the project directory as the first choice storage location.

2.2 Language settings

KISSsoft is available in nine languages: German, English, French, Italian, Spanish, Russian, Portuguese, Chinese and Japanese. When you select a language, the program differentiates between the language used for the user interface and the language used for the reports. This makes it possible to operate KISSsoft in one language and simultaneously display reports in a different language. Messages will be displayed either in the same language as the user interface or as the reports.

In the program, select **Extras > Language** to change between the languages available in your license. To make global language settings, you need to edit the KISS.ini (see chapter 2.6.2, Definitions in [SETUP]) file. The user can change the language used for reports by selecting **Report > Settings**.

2.3 Systems of units

KISSsoft recognizes two systems of units: the metric system and the US Customary Units system. For global settings, you need to edit the KISS.ini (see chapter 2.6.2, Definitions in [SETUP]) file. You can also quickly toggle between systems of units in the program by selecting **Extras > System of units**. In addition to changing the system of units, it is possible to switch the unit used for a particular value input field (see chapter 5.2.1, Value input fields).

2.4 Defining your own template files

Anyone who frequently carries out the same, or at least similar, calculations has to repeatedly select or enter the same values in selection lists and value input fields. Thanks to template files, KISSsoft makes it much easier to do this. For each calculation module, there is an internal default setting for all values. If, however, you have defined your own template file, this template file will be used when you open a calculation module or load a new file.

To define a template file, you open a new file in the appropriate calculation module and enter your default settings. Click on **File > Save as template** to transfer your values to the template file. All template files will be saved in the directory that has been defined as TEMPLATEDIR (see chapter 2.6.1, Definitions in [PATH]).

Project-specific template files can also be created. To define special standards for a project (see chapter 6, Project Management), select this project in the Project Tree (see chapter 4.2.2, The project tree) and open its properties by selecting **Project > Properties**. There, select **Use own templates for this project** and specify a directory for the template files. To define the template files, first select this project as the active working project (see chapter 6.3, The active working project).

2.5 Rights

You can restrict the rights for selected areas of KISSsoft for some users.

Right	**Implementation**
Change general settings	Write-protect the KISS.ini: (see chapter 2.6, Global settings - KISS.ini) file
Change or add data to databases	Write-protect databases (files of the type .udb), and the DAT and EXT/DAT directories (write rights for UDBDIR (see chapter 2.6.1, Definitions in [PATH]) should be retained).
Change report templates	Write protect RPT, EXT/RPT and EXT/RPU directories
Change template files	Write protect the TEMPLATE directory

2.6 Global settings - KISS.ini

Global settings for KISSsoft are defined in the KISS.ini file, which is located directly in the installation folder. Most of these settings can also be defined directly in the software and are then saved to the KISS.ini file.

2.6.1 Definitions in [PATH]

Variable name	Description	Note
KISSDIR=<INIDIR>	The KISSsoft installation folder path is generally defined with the **INIDIR** variable.	
HELPDIR	Directory for user manual and help figures	
DATADIR	Directory for .dat files	**Attention:** You should not carry out any upgrades or make any changes in this directory. Save your own files in the DAT subdirectory in the EXTDIR directory.
RPTDIR	Directory for report templates (*.rpt)	**Attention:** You should not carry out any upgrades or make any changes in this directory. Save your own files in the RPT subdirectory in the EXTDIR directory.
USERDIR	Default directory for opening and saving	
CADDIR	Default directory for CAD export	Should be located locally on a workstation. %TEMP% sets the temporary directory to the operating system default.
TMPDIR	Directory for temporary files	Should be located locally on a workstation. %TEMP% sets the temporary directory to the operating system default.
UDBDIR	Directory for user-defined databases (*.udb)	If several users are using the system, we recommend you store the databases on a server to ensure data uniformity if there are changes and upgrades.
KDBDIR	Directory for KISSsoft's databases (*.kdb)	KISSsoft datasets containing data that cannot be modified.

EXTDIR	Directory for user-defined report templates and additional DAT files	If there are several users, it is advisable to store this directory on a server.
TEMPLATEDIR	Directory for template files (STANDARD.*).	If there are several users, it is advisable to store this directory on a server.
LICDIR	Directory for the license files	You can install this directory on a server so that all users can access the new license files.

Table 2.1: Table of variables used in the PATH environment

► **Note**

You should have write permission for the directories set in these variables: **TMPDIR**, **CADDIR**, **USERDIR** and **UDBDIR**.

Depending on the configuration, you might not have write permission in these directories: **C:\ Program Files\ KISSsoft directory name** or **C:\ Programs\ KISSsoft directory name**. Any files you create are then diverted to the operating system's internal directories. In this case, select directories with write permission.

The **UDBDIR**, **TMPDIR**, **CADDIR**, **USERDIR** and **EXTDIR** directories can also be defined in the 'Directories' tab in the 'Program settings' dialog (**Extras**>**Settings**).

2.6.2 Definitions in [SETUP]

Variable name	Description	Values
USCUSTOMARYUNITS	Sets the system of units	**0**: metric, **1**: US customary units
MATERIALSSTANDARD	Specifies the standard according to which the materials are defined (configuration tool)	**0**: DIN, **1**: BS, **2**: AISI, **3**: UNI, **4**: AFNOR, **5**: JIS, **6**: CN
REPORTLANGUAGE	Sets the language in which reports are displayed	**0**: German, **1**: English, **2**: French, **3**: Italian, **4**: Spanish, **5**: Russian, **6**: Portuguese, **7**: Chinese, **8**: Japanese, **11**: English with US Customary Units
SHOWCALCTIME	Displays the amount of time taken to perform a calculation	**0**: No, **1**: Yes
SHOWPROGRESSBAR	Shows the progress bar for time-intensive calculations	**0**: No, **1**: Yes

DISPLAYLANGUAGE	Sets the language in which the user interface is displayed	**0**: German, **1**: English, **2**: French, **3**: Italian, **4**: Spanish, **5**: Russian, **6**: Portuguese, **7**: Chinese, **8**: Japanese
DISPLAYFONTSIZE	Sets the font size in KISSsoft (FONT)	**0**: System size or else the direct font size
MESSAGESINREPORTLANGUAGE	Sets the language in which messages are displayed	**0**: as interface, **1**: as reports
MESSAGESSHOWSTATE	Defines which messages are to be displayed in a message box.	**0**: all, **1**: Information only in message window, **2**: Information and warnings only in message window
EDITOR	Path to the external editor	
USEEXTERNALEDITOR	Defines whether the external editor is to be used.	**0**: No, **1**: Yes
DATEFORMAT	Date format, e.g. DD.MM.YYYY	
TIMEFORMAT	Time format, e.g. hh.mm.ss	
ENABLENETWORKING	Defines whether the network/internet can be accessed (for example, to display product news).	**0**: No, **1**: Yes
CHECKFORUPDATES	Defines whether the system is to search for updates when the program starts.	**0**: No, **1**: Yes
USETEMPORARYDATABASE	Defines whether the databases are to be copied to a temporary directory when the program starts	**0**: No, **1**: Yes
RECENTFILESCOUNT	Set number of most recently used files in the File menu displayed.	
FORCEEXCLUSIVEOPEN	Defines whether files can only be opened exclusively.	**0**: No, **1**: Yes
CALCONOPEN	Defines whether calculations are to be performed on a file immediately, when it is loaded.	**0**: No, **1**: Yes, **2**: no if KISSsoft is started from KISSsys, otherwise yes
CALCINTERFACEOUT	Defines whether temporary reports for manufacturing data	**0**: No, **1**: Yes

	are to be written during a calculation.	
ENABLEUSERSETTINGS	Defines whether the settings in kiss.ini can be overwritten by local settings.	**0**: No, **1**: Yes
USEFILEEXPLORER	Defines whether the Explorer is to be displayed in the "View" menu list. This process will slow down KISSsoft considerably.	**0**: No, **1**: Yes
USEHIGHDPIICONS	Use scaled icons for high-resolution screens.	**0**: No, **1**: Yes

Table 2.2: Table of variables used in the SETUP environment

2.6.3 Definitions in [REPORT]

Variable name	**Description**
SIZE	A number, **0-9**, that specifies the scope of the report.
INCLUDEWARNINGS	**0/1**: Warnings are contained in the report
FONTSIZE	Number that sets the font size in the report.
PAPERFORMAT	Paper format: A3, A4, A5, Letter, Legal
PAPERORIENTATION	**0/1**: Portrait/Landscape
PAPERMARGINLEFT	Distance from the left-hand page margin [mm].
PAPERMARGINRIGHT	Distance from the right-hand page margin [mm].
PAPERMARGINTOP	Distance from the top page margin [mm].
PAPERMARGINBOTTOM	Distance from the bottom page margin [mm].
COMPARE	**0/1**: Adds date/time to the report in comparison mode.
LOGO	Sets the picture file displayed in the header and footer.
HEADER	Contains the header usage definition.
USEHEADERFORALLPAGES	**0/1**: Header only on first page/on all pages
FOOTER	Contains the footer usage definition.
USEFOOTERFORALLPAGES	**0/1**: Footer only on first page/on all pages
READONLY	**0**: Reports can be edited and saved as editable text documents

	1: Reports cannot be edited and can only be exported as PDF documents

Table 2.3: Table of variables used in the REPORT environment

2.6.4 Definitions in [REPORTEDITOR]

Variable name	**Description**
SAVEFORMAT	**0-5**:DOCX, PDF, ODT, TXT, HTM, MD
SYNCHSCROLL	**0**: Synchronous scrolling is disabled. **1**: Synchronous scrolling is enabled.
KEEPZOOMFACTOR	**0**: The zoom factor is reset to 100% when you open a new report. **1**: The zoom factor is retained when you open a new report.
KEEPSCROLLPOSITION	**0**: The scroll position is reset when you open a new report. **1**: The scroll position is retained when you open a new report.
SPLITONOPEN	**0**: When you open a file with KISSedit, it is displayed in full screen mode. **1**: When you open a file with KISSedit, it is displayed in split view.
TABBED	**0**: is not in a tab **1**: is in a tab
LOCKED	**0**: Reports are unlocked for editing. **1**: Reports are locked for editing.
PLAINTEXTFONT	Font (e.g. Consolas)
PLAINTEXTFONTSIZE	Font size (by default: 12)
PLAINTEXTENCODEUTF8	**0**: Text in ASCII format **1**: Text in UTF8 format
TITLECONFLICTRESOLUTION	Adds a number or date to the file name when opening multiple reports with the same name. **0**: Adds a number **1**: Adds a date
DRAWFRAME	**0**: Report frame is not displayed. **1**: Report frame is displayed.

Table 2.4: Table of variables used in the REPORTEDITOR environment

2.6.5 Definitions in [GRAPHICS]

Variable name	**Description**
BACKGROUND	**0**: black, **15**: white

CLIPDIAGRAMCURVES	**0**: Display complete curve progression in diagrams, **1**: Limit curves in diagrams to current axis range
PERSPECTIVEPROJECTION	**0**: 3D graphics in parallel projection, **1**: 3D graphics in perspective projection
USESETTINGS	**0**: Don't save graphics settings, **1**: Save graphics settings for specific users

Table 2.5: Table of variables used in the GRAPHICS environment

Other variables are described in (see chapter 23.9, Settings).

2.6.6 Definitions in [LICENSE]

Variable name	Description
LOGGING	Number used to configure the logging of license usage. 0: no log file 1: Log in, Log out, No license, Used and Missing rights 2: Log in, Log out, No license 3: Log in, Log out, No license, Missing rights In network versions, the user's uptime is also displayed , in seconds.
LICENSELOGFILE	**.log** file for generating reports of license usage.
TIMEOUT	Time [min] until an unused floating license is activated on the network again.
LICENSEMANAGEMENTSTATE	If this parameter is set to 1, the License dialog is displayed immediately, when KISSsoft is started, and the "General" tab is active. This enables the user to select the license with which KISSsoft is to be used.

Table 2.6: Table of variables used in the LICENSE environment

2.6.7 Definitions in [CADEXPORT]

Variable name	Description
USEDXFHEADER	**0**/**1**: Use DXF header for DXF export.
DXFVERSION	**0**/**1**: Version 12/15
INPUTLAYER	Name of the layer for import.

OUTPUTLAYER	Name of the layer for export.
DXFPOLYLINE	**0**/**1**/**2**: Use polygonal course, lines or points for the export.

2.7 table: Table of variables used in the CADEXPORT environment

2.6.8 Definitions in [INTERFACES]

Variable name	Description
DEFAULT	Name of the CAD system:
	Solid Edge
	SolidWorks
	Inventor
	CATIA
	Creo
	HiCAD
GEAREXPORT3D	Displays the CAD system name in lists (see **DEFAULT**).
SYMMETRIC	**0**/**1**: Full tooth space/half tooth space mirrored (symmetrical) (default = 0)
SAVEFILENAME	**0**/**1**: Saves the entire file contents/Saves only the file name and the path. (Default = 1)
MESSAGECADVERSION	**0**/**1**: You see a message/no message if the CAD version is no longer supported by the interface. (Default = 1)

2.8 table: Table of variables used in the INTERFACES environment

2.6.9 Definitions in [SOLIDEDGE]

Variable name	Description
LIBRARY	Interface dll (**kSoftSolidEdge.dll**) directory
SIMPLIFIEDPRESENTATION	**0**/**1**: Set the variable to 1 to also generate a simplified gear
SMARTPATTERN	**0**/**1**: Fastpattern/Smartpattern
APPROXIMATION	**1**/**2**/**3**/**4**: Polygonal course (supported)/Arcs of circle (supported)/Quadratic splines (supported)/Cubic splines (default)

USERPARTTEMPLATE	Template file directory (e.g. C:\Template\metric.par) or just the template file name (e.g. metric.par): the path is then taken from the settings in Solid Edge
USERDRAFTTEMPLATE	Template file directory (e.g. C:\Template\metric.dft) or just the template file name (e.g. metric.dft), in which case the path is then taken from the settings in Solid Edge.

Table 2.9: Table of variables used in the SOLIDEDGE environment

2.6.10 Definitions in [SOLIDWORKS]

Variable name	Description
LIBRARY	Interface dll (**kSoftSolidWorks.dll**) directory
SIMPLIFIEDPRESENTATIONNAME	Setting this variable generates a simplified gear with this name
APPROXIMATION	**1**/**2**/**3**/**4**: Polygonal course (supported)/Arcs of circle (supported)/Quadratic splines (supported)/Cubic splines (default)

2.10 table: Table of variables used in the SOLIDWORKS environment

2.6.11 Definitions in [INVENTOR]

Variable name	Description
LIBRARY	Interface dll (**kSoftInventor.dll**) directory
APPROXIMATION	**1**/**2**/**3**/**4**: Polygonal course (supported)/Arcs of circle (supported)/Quadratic splines (supported)/Cubic splines (default)

2.11 table: Table of variables used in the INVENTOR environment

2.6.12 Definitions in [CATIA]

Variable name	Description
LIBRARY	Interface dll (**kSoftCatia.dll**) directory
LIBRARYSWMS	Directory containing the interface manufacturer's **.dll** file
LANGUAGEFILE	Directory containing the interface manufacturer's **.ini** file
DEBUG	Interface manufacturer's variable
DEBUGPATH	Interface manufacturer's variable
HELPFILE	Interface manufacturer's variable

LASTSETTING_CONSTRUCTION	Interface manufacturer's variable
LASTSETTING_GEARNAME	Interface manufacturer's variable
LASTSETTING_PRODUCTIONINFO	Interface manufacturer's variable
LASTSETTING_CALCINFO	Interface manufacturer's variable
LASTSETTING_FLAGINFO	Interface manufacturer's variable
APPROXIMATION	**1**/**2**/**3**/**4**: Polygonal course (not supported)/Arcs of circle (not supported)/Quadratic splines (default)/Cubic splines (not supported)

2.12 table: Table of variables used in the CATIA environment

2.6.13 Definitions in [PROENGINEER]

The ProEngineer interface has an individual subsection/menu for each version (for example, Wildfire 5, 32bit). However, the definitions in "kiss.ini" are the same in every 3D interface to Creo Parametric (ProEngineer) chapters.

Variable name	Description
LIBRARY	Interface dll directory (**kSoftProEngineer.dll**)
INTERFACECOMMAND	Directory containing the interface manufacturer's **.exe** file
USCUSTOMARYUNITS	**0**/**1**: System of units used in the metric or US Customary Units model
APPROXIMATION	**1**/**2**/**3**/**4**: Polygonal course (not supported)/Arcs of circle (default)/Quadratic splines (not supported)/Cubic splines (not supported)

2.13 table: Table of variables used in the PROENGINEER environment

2.6.14 Definitions in [HICAD]

Variable name	Description
LIBRARY	Interface dll directory (**kSoftHiCAD.dll**)
APPROXIMATION	**1**/**2**/**3**/**4**: Polygonal course (not supported)/Arcs of circle (default)/Quadratic splines (not supported)/Cubic splines (not supported)

2.14 table: Table of variables used in the HICAD environment

2.6.15 Definitions in [VIDEOENCODING]

Variable name	Description	Values
CODEC	Specifies the video codec which is to be used to encode videos. Not all operating systems support all possible values.	0: H.264, 1: H.265 (default = 0)
HARDWAREENCODING	Specifies whether hardware video encoding is to be used, if available.	0: No, 1: Yes (default = 1)
MODE	Specifies the encoding mode. In some circumstances, a different mode will be used if the hardware, operating system or selected codec do not support the initially selected mode.	0: CBR (fixed bitrate) 1: Unconstrained VBR (variable bitrate without maximum) 2: Constrained VBR (variable bitrate with maximum) 3: Quality level (target quality, without specifying the bitrate) (Default = 3)
WIDTH	Specifies the video width. The width can be either defined automatically or fixed. However, very small or very large values may cause the video recording to fail.	0: Use the current width of the graphic in the Graphics window 1-32767: Use this width (in pixels) (Default = 0)
HEIGHT	Specifies the video height. The height can be either defined automatically or fixed. However, very small or very large values may cause the video recording to fail.	0: Use the current height of the graphic in the Graphics window 1-32767: Use this height (in pixels) (Default = 0)
QUALITY #	Specifies the quality level to be used if MODE=3 is set.	0-51: The quality level to be used (default = 24)

Variable name	Description	Values
	Not all codecs or operating systems support all possible values.	
AVGBITRATE	The video's target bitrate in bit/s. If MODE=0, this specifies the video's constant bitrate. If MODE=1/2, this specifies the video's average bitrate. However, very small or very large values may cause the video recording to fail.	0: The bitrate is calculated automatically using the Kush gauge Other values: The bitrate to be used (Default = 0)
MAXBITRATE	The video's maximum bitrate in bit/s, if MODE=2. This value should be greater than AVGBITRATE. However, very small or very large values may cause the video recording to fail.	0: The bitrate is calculated automatically using the Kush gauge Other values: The bitrate to be used
FPS	Specifies the number of images per second in the recorded video. However, very small or very large values may cause the video recording to fail.	Recommended values: 30 or 60 (default = 30)

2.7 User-defined settings

User-defined settings can be reset via **Extras > Configuration** tool.

2.7.1 Configuration tool

In the **General** tab, you can select the older version's "kdb" database directory (prior to 03-2017. After that, it is called "udb".) by selecting the **Update database** option. Click "Run" to transfer the datasets you defined yourself in the older version to the current version, to ensure these datasets are available in the current version.

Click **Update external data** to select the older version's "ext" directory. This then automatically copies the "dat", "rpt" and "rpu" subdirectories to the current release.

Click **Update settings** to transfer your personal settings from the previous version to the current release.

Select **Connect file extensions** to link all the KISSsoft files with the current version so that you can double-click on any file to open it in the current release.

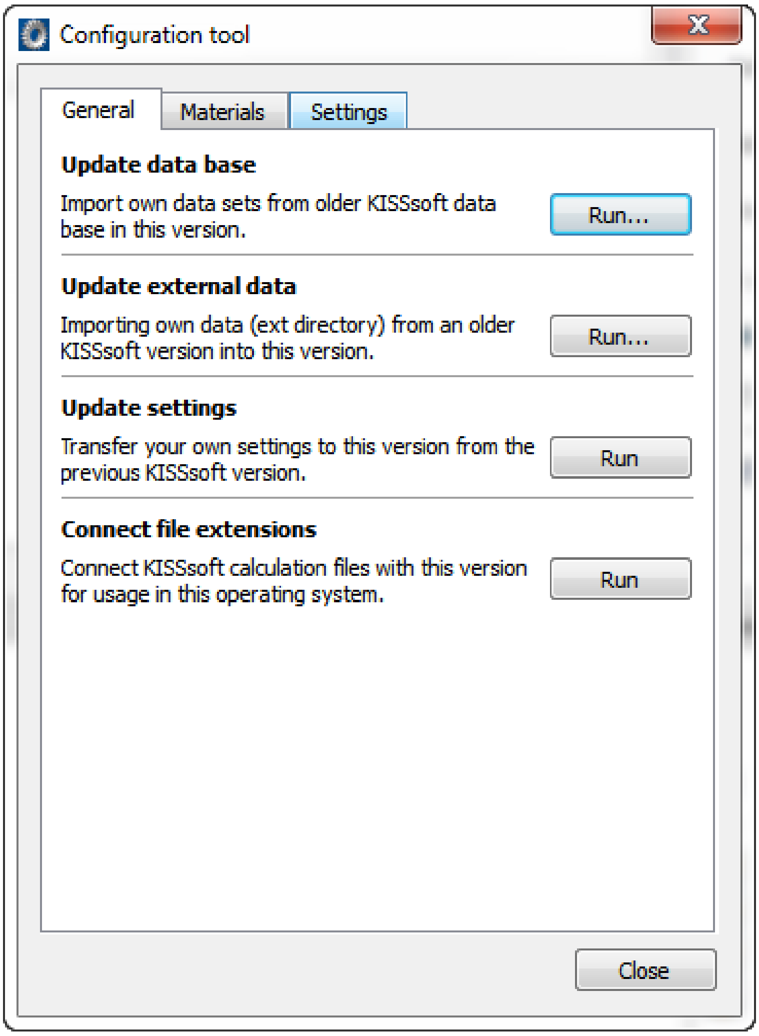


Figure 2.1: General tab in the Configuration tool window

In the **Materials** tab, you can specify the standard with which the material descriptions in the database are to comply.

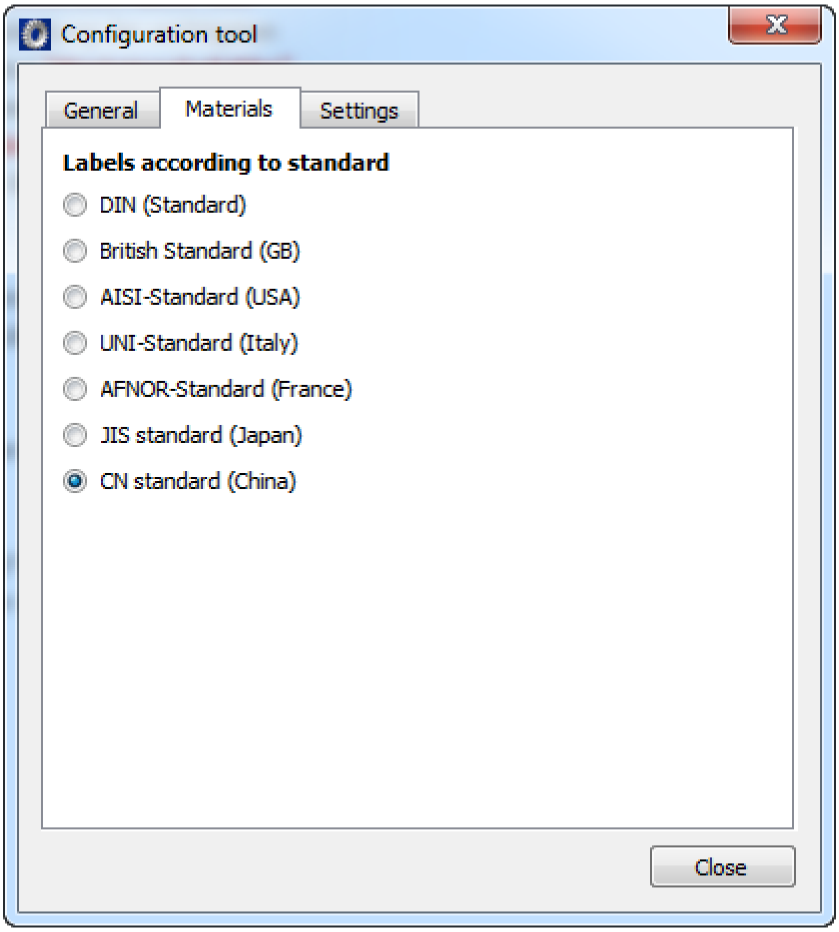


Figure 2.2: Materials tab in the Configuration tool window

In the **Settings** tab, you can delete the user-defined settings (divided into groups). This reloads the default values.

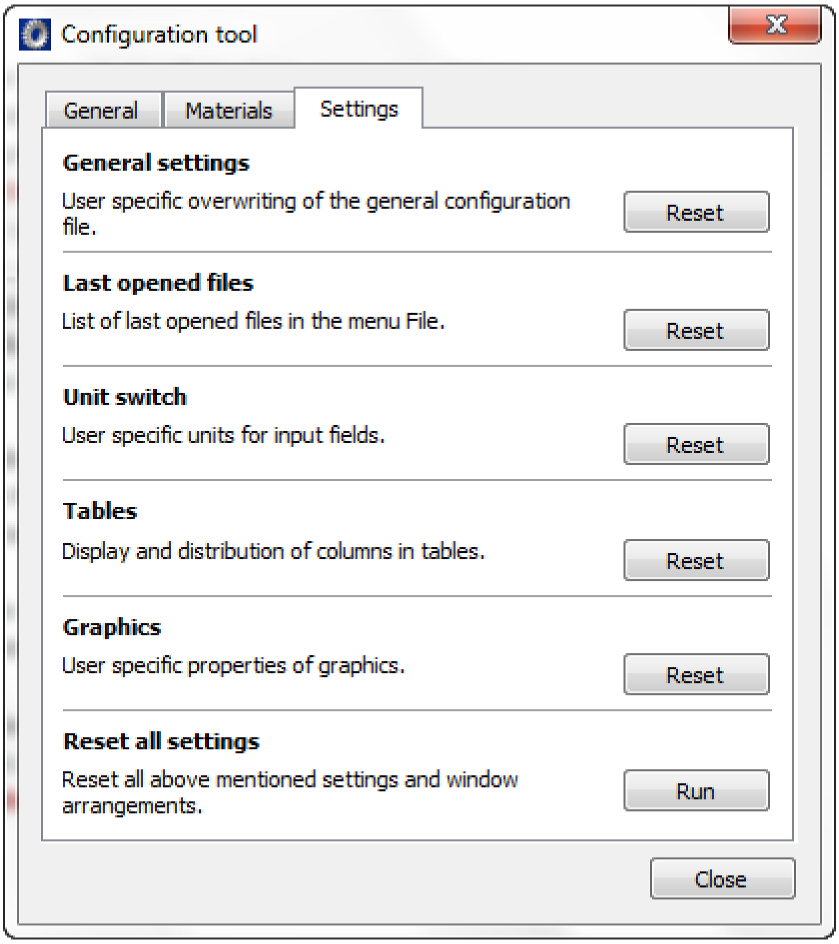


Figure 2.3: Settings tab in the Configuration tool window

2.8 Rules

Rules are used to ensure that in-house guidelines for the ranges of validity of parameters are applied and complied with. This typically concerns the maximum and minimum limits of input values, calculated values and the relationships between these values i.e. length/width ratios, length/diameter relationships or even the relationship between the module and the center distance.

These rules are defined by being stored in a **module**.rls file, where **module** stands for the calculation module's in-house label, e.g. Z012 for cylindrical gear pairs.

These rules are subdivided into those that must be fulfilled before the calculation is performed and those that must be checked afterwards. If a rule is infringed, the appropriate messages can be displayed. In the case of rules that must be checked before the calculation, variables can also be set to constant or calculated values.

The following statements are possible:

precalc: marks the beginning of the rules that must be checked before a calculation is performed.

postcalc: marks the beginning of the rules that must be checked after a calculation.

assert(Condition): The **condition** is ensured. In this case, the **condition** usually represents a comparison in which both the right-hand and left-hand side of the comparison can also be calculated.

action msg Message: If the **condition** of the previous assert has not been fulfilled, the **message** is output. Here the **message** can include variables, in the same way as report templates.

action set Assignment: If the **condition** of the previous assert has not been fulfilled, the assignment is performed. The assigned value can be a constant, or can be calculated from variables, in the same way as for the report templates.

Defining an assignment is only really useful in the precalc section because changing the contents of variables after the calculation merely leads to inconsistent results and has no other effects.

Here is an example file for a helical gear calculation:

```
precalc

assert (ZR[0].x.nul < 1)

action msg "Profilverschiebung Rad 1 zu gross, Ist {ZR[0].x.nul}, Maximum
1. Wird auf 1 gesetzt."

action set ZR[0].x.nul = 1

assert (ZR[1].x.nul < 1)

action msg "Profilverschiebung Rad 2 zu gross, Ist {ZR[1].x.nul}, Maximum
1. Wird auf 1 gesetzt."

action set ZR[1].x.nul = 1

postcalc

assert ((ZP[0].a/ZS.Geo.mn) < 200)
```

action msg "Center distance too big for module (a={ZP[0].a}, mn={ZS.Geo.mn}, a/mn={ZP[0].a/ZS.Geo.mn})."

Explanations:

The "precalc" statement opens the section of the rules that must be executed before the calculation.

The first "assert" statement checks whether the nominal profile shift of gear 1 is less than 1.0.

If this "assert" is not fulfilled, the "action msg" statement outputs the message that the profile shift is too big, displays the current value and tells you that the profile shift has been set to 1.0.

The "action set" then sets the profile shift to 1.0.

The second "assert" statement checks the same values for gear 2.

The "postcalc" statement signifies the end of the set of rules to be executed before the calculation and opens the section containing the rules that are to be checked after the calculation. The example shows a definition of an "assert" statement. This checks the ratio between the center distance to the module. If the rule is infringed, the "action msg" statement triggers a message to the user. However, there is no point in changing one of these two values after the calculation, and this is why the "action set" statement is not present here.

Permitted operators and functions in the formulae (see chapter 8.5.3.3, Calculation variables).

The file containing the rules is stored in the template directory (TEMPLATEDIR, usually the "template" (see chapter 2.1, Directory structure) subdirectory). As the template directory can also be project-specific, you can also define project-specific rules.

2.9 FEM related settings

By default, KISSsoft uses CM2 FEM®/CM2 MeshTools® (referred to below as the CM2 Library). This CM2 Library is integrated in KISSsoft, so no further action is required by the user.

3 Running KISSsoft

3.1 Start parameters

You can run KISSsoft from the input prompt with the following start parameters:

Parameter	Description
INI=**directory**	The KISS.ini (see chapter 2.6, Global settings - KISS.ini) file will be loaded from the specified location. You can transfer a file name, with its directory path, or only a directory name.
START=**module**	The specified calculation module will be started. The module descriptor is, for example, **M040** for bolt calculation or **Z012** for cylindrical gear pair calculation.
LOAD=**file name**	The calculation module belonging to the file is started and the file is loaded. If the supplied file name does not include a path, the system looks for the file in the User directory (see chapter 2.6.1, Definitions in [PATH]).
LANGUAGE=**number**	KISSsoft starts with the language specified for the interface and reports. (0: German, 1: English, 2: French, 3: Italian, 4: Spanish, 5: Russian, 6: Portuguese, 7: Chinese, 11: English with US Customary Units)
DEBUG=**file name**	A log file containing debug information will be written. It can be very helpful for error-tracking. It is advisable to define the file name with a complete path, so that you can find the log file easily later.
LIC=**license file**	If a license file is specified here, KISSsoft starts with that license file instead of the specified default license file.
File name	The calculation module belonging to the file is started and the file is loaded. This also provides a way to associate KISSsoft with the appropriate file name extensions in Windows.

3.2 Disconnect license from the network

If KISSsoft has not been properly shut down, users may remain registered if a network version is in use. This may lead to licenses being blocked even though some users are no longer working with KISSsoft. You can disconnect a license from the network by selecting the required license (the user and start time are also displayed). To do so, select the **Extras > License tool** option in the **Network** tab. The system then also deletes the appropriate cookie file and activates the blocked license on the network again.

Unused licenses will be activated after a certain time, as soon as the next user logs on. This time-span can be predefined via the TIMEOUT (see chapter 2.6.6, Definitions in [LICENSE]) variable in the KISS.ini (see chapter 2.6, Global settings - KISS.ini) file.

► **Note**

A user who has been disconnected from KISSsoft can no longer carry out calculations in the current session. The user must restart KISSsoft. However, they can still save data.

4 Elements in the KISSsoft user interface

KISSsoft is a Windows-compliant software application. Regular Windows users will recognize user interface elements, such as the menus and context menus, docking window, dialogs, tooltips and status bar, from other applications. Because the internationally valid Windows Style Guides are applied during development, Windows users will quickly become familiar with how to use KISSsoft.

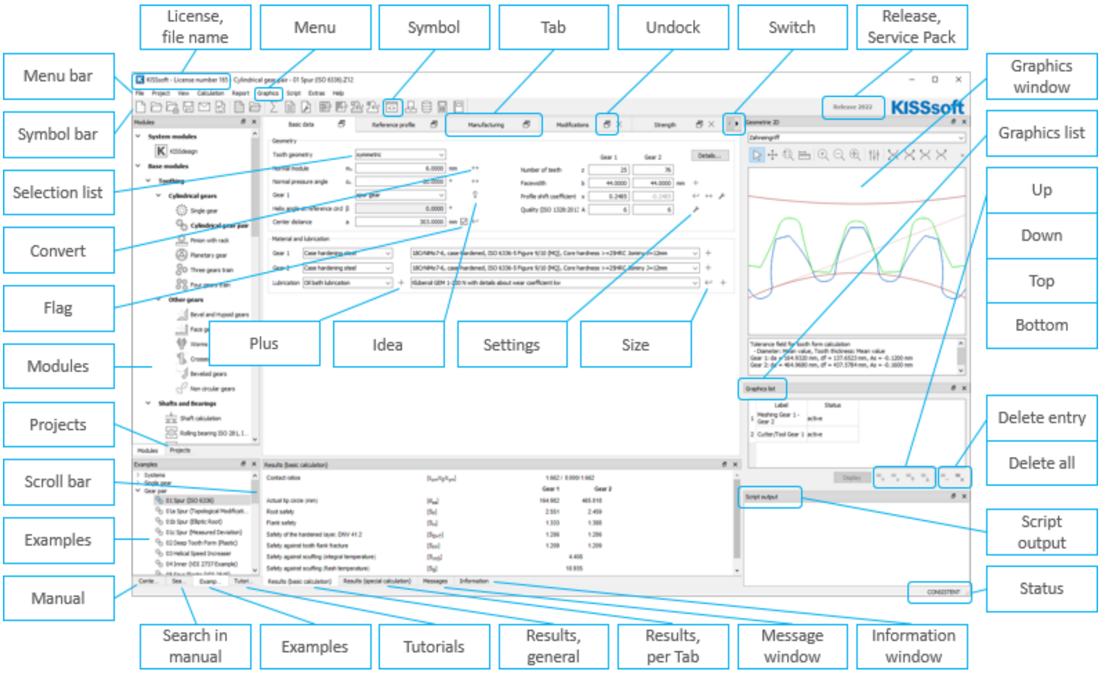


Figure 4.1: Elements in the KISSsoft user interface

4.1 Menus, context menus and the tool bar

In the **File** main menu, you can open and save calculation files, and send them as e-mail attachments, restore previous calculation stages, view file properties and exit KISSsoft. Click **File > Save as template** to retain user-defined default values (see chapter 2.4, Defining your own template files).

You can use the KISSsoft Project Management (see chapter 6, Project Management) functionality from both the **Project** main menu and the Project Tree (see chapter 4.2.2, The project tree). You can open, close and activate projects, insert files into a project, or delete them, and also view project properties.

Each individual docking window (see chapter 4.2, Docking window) in the user interface can be hidden or displayed in the **View** main menu. If you are in the report or helptext viewer, select **View > Input window** to return to the calculation module input dialog.

In the **Calculation** main menu, you can run the current calculation (see chapter 5, KISSsoft Calculation Modules), add more calculations to the calculation module as default tabs or special tabs and call subcalculations as dialogs. Select **Calculation > Settings** to change the Module specific settings.

In the **Report** main menu you will find actions for generating and opening a report. The system always generates a report for the current calculation. Click **Report > Drawing data** to display drawing data (see chapter 8.3, Drawing data) for the element currently selected in the Report Viewer (see chapter 4.4.1, Report Viewer). Click **Report > Settings** to change the report's font size, page margins and scope. The options for saving, sending and printing are only active if a report is open.

You can open and close the Graphics window (see chapter 4.3, Graphics window) of a calculation module in the **Graphics** main menu. Select **3D export** to access KISSsoft's CAD interfaces. Select **Graphics > Settings** to choose the CAD system into which you want to export the selected element.

In the **Extras** menu, you will find the license tool, the configuration tool and the database tool. In this main menu, you can start the Windows calculator and change the language (see chapter 2.2, Language settings) and system of units (see chapter 2.3, Systems of units). In **Extras > Settings**, you can change general program settings such as the formats for time and date values.

In accordance with Windows conventions, at the end of the menu bar you will find the **Help** icon, which you can use to navigate in the KISSsoft manual. In **Help > Info** you will find information on the program version and on the support provided by KISSsoft.

In addition to the main menu, KISSsoft uses **context menus** in many locations. Context menus give you access to actions for a particular area or element of the software. Normally, you click the right-hand mouse button to display context menus.

The **tool bar** gives you faster access to actions from the menus that are used particularly frequently. You should also read the tool tips: they display information about the actions in the tool bar and also the more detailed explanations in the status bar (see chapter 4.5, Tooltips and status bar).

► **Note**

The **Calculation**, **Report** and **Graphics** main menus are only active if a calculation module is open. The actions available in these menus may vary depending on the current calculation module.

4.2 Docking window

Beside the menu bar, tool bar and status bar, the docking windows are important elements in the KISSsoft user interface. Docking windows are windows that can either be moved freely on the desktop, like a dialog, or docked on the program pages in any arrangement that suits you. Several docking windows can be placed on top of each other and be displayed as tabs.

You can release a docking window by double-clicking in its title bar. To move a docking window, click the left-hand mouse button in the title bar and move the mouse while holding down the mouse button. If you move the mouse close to the edge of the main window, a new position for the docking window will be displayed. Docking windows can be arranged in several lines and columns around the main window. Release the mouse button to place the window.If you press and then hold down the Ctrl button while moving a docking window, the docking window then becomes free-floating.

Docking windows can be displayed and hidden via the View menu (see chapter 4.1, Menus, context menus and the tool bar) .

The individual arrangement of docking windows in each calculation module is saved in the user-defined settings. As a result, every calculation module always starts in the same place as when it was last used.

4.2.1 The module tree

The module tree shows all KISSsoft calculation modules in an easy to understand and logically structured list. Any calculation modules for which you have not purchased a license are grayed out. To open a module, double-click on it with the left mouse button. The current calculation module will be shown in bold.

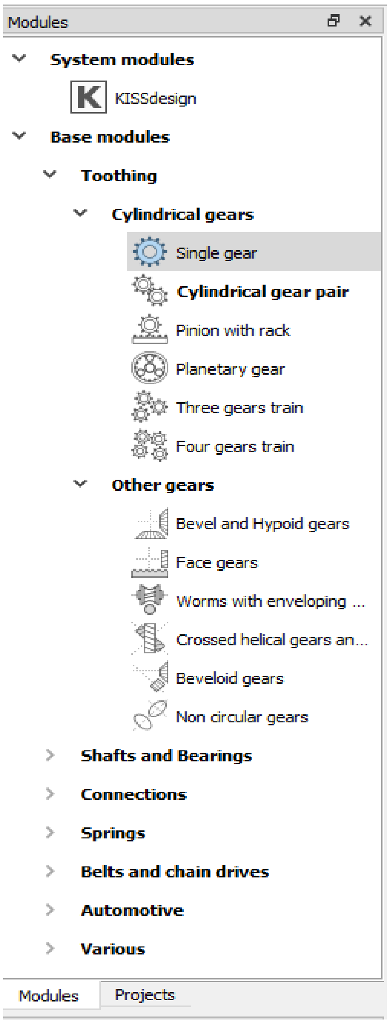


Figure 4.2: KISSsoft calculation modules

4.2.2 The project tree

The Project Tree gives you an overview of the open projects, and the files belonging to these projects, and highlights the active working project (see chapter 6.3, The active working project) in bold. You use project management functions (see chapter 6, Project Management) via the **Project** menu or from a context menu (see chapter 4.1, Menus, context menus and the tool bar).

4.2.3 The Results window

The KISSsoft results window displays the results of the last calculation. For each calculation module, the basic calculation results and the results for the current special calculation are displayed in two results windows. The results can be saved and printed via the context menu.

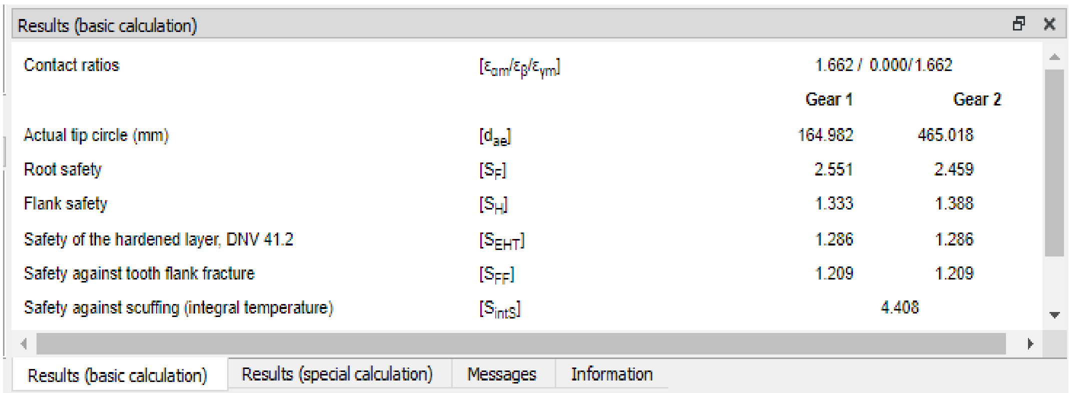


Figure 4.3: The KISSsoft results window

4.2.4 The Messages window

The messages window displays all information messages, warnings and errors. Generally, all additional messages are not only displayed, but also appear in a message box. You can change the way information and warnings are displayed in a message box by selecting **Extras > Settings**.

4.2.5 The info window

The Info window displays information that is displayed when you click on an Info button (see chapter 5.2.1, Value input fields) in the calculation module. You zoom and print the information via a context menu (see chapter 4.1, Menus, context menus and the tool bar).

4.2.6 Manual and Search

The manual's Table of Contents and search function are also available as docking windows. When you select an entry, by double-clicking on it, the Helptext viewer (see chapter 4.4.2, Helptext Viewer) opens and the relevant section in the manual is displayed.

4.3 Graphics window

In KISSsoft, you can open as many graphics windows as you need at the same time and arrange them in the same way as the other docking windows (see chapter 4.2, Docking window). This means you can see all the graphics and diagrams you require for your calculations at a glance. To make working with graphics more effective, you can use the tool bar (see chapter 4.3.1, Tool bar and context menu), the Comment field, the context menu (see chapter 4.3.3, Context menu) and the Properties (see chapter 4.3.4, Properties).

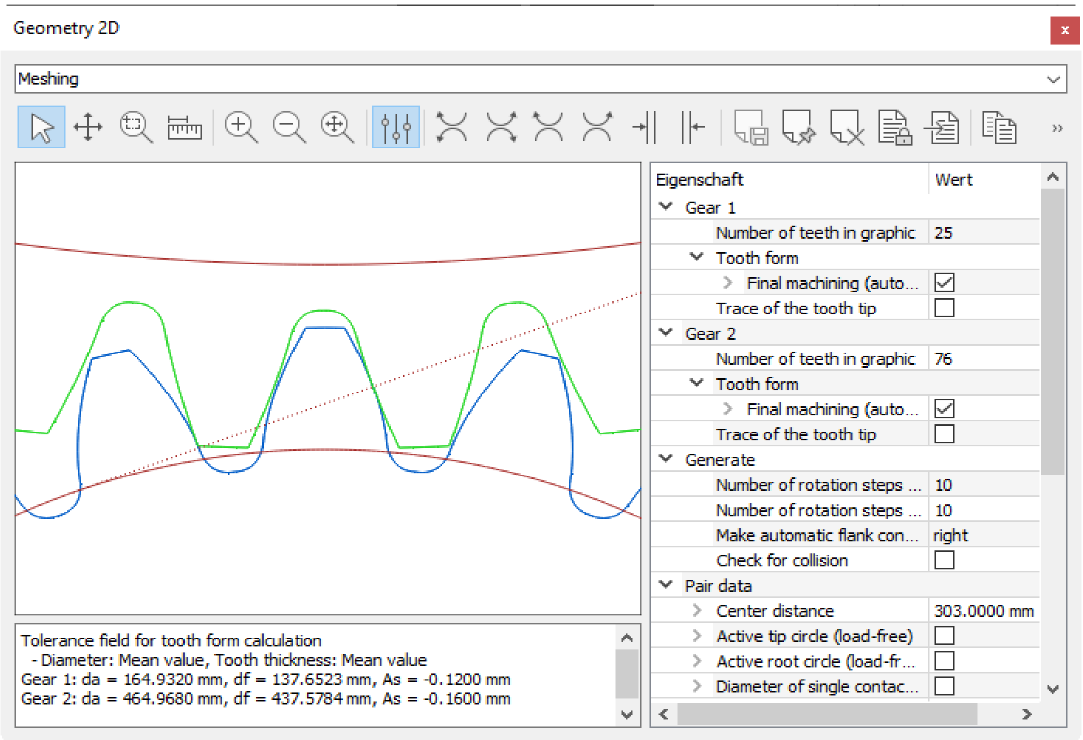


Figure 4.4: Components of the graphics window

4.3.1 Tool bar and context menu

Use the selection list in the tool bar to switch from one graphic to another in a group. You will also see various icons for saving, printing and locking a graphic, and also functions for displaying or graying out its properties.

Save graphic as

This saves the graphic as a DXF, IGES or other image or text format, with the name you enter here.

Saving diagrams in a DXF file usually creates a conflict between the diagram axis units and the unit used in the DXF file. For this reason, when you save a diagram, the program opens a dialog in which you can specify the drawing area to which the diagram is to be projected, in the file.

Print graphic

Prints the current section of the graphic. The information underneath the graphic is defined by graph*.rpt report templates (see chapter 8.5).

Lock

This is useful for comparing two calculation results. In this way, you can, for example, generate a **Specific sliding** graphic for a toothing scenario, lock this graphic and then, after having changed the gear parameters, open a new graphics window that shows the new calculation results. The locked window will no longer be updated.

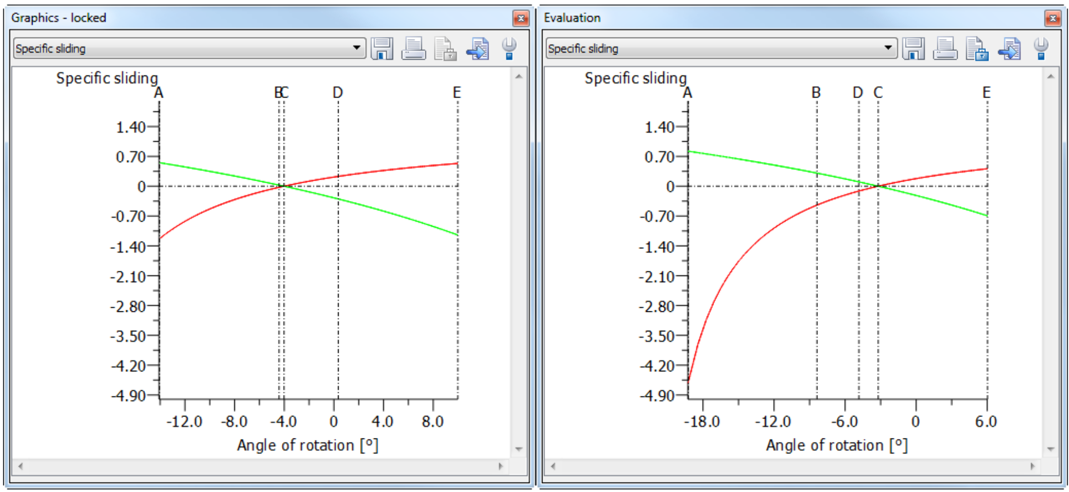


Figure 4.5: Locking graphics windows (a) Locked window and (b) Window with new calculation results

When you lock a graphics window, a dialog will open, in which you can enter a title for the window. This makes it easier for you to compare different graphics.

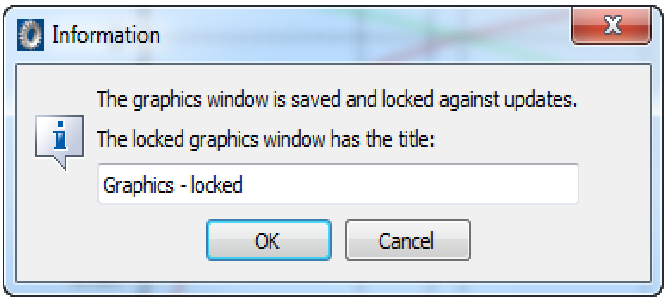
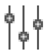


Figure 4.6: Dialog window for inputting the window title



Properties

This opens a list with the Properties (see chapter 4.3.4, Properties) of the current graphic in the same window.

Video recording

Starts recording a video of the 3D graphic. All of the graphic's animations and movements are recorded when video recording is enabled. Click again to stop recording. You can then either save or delete the video file.

The size of the graphic cannot be changed while video recording is active.

4.3.2 Comment field

In the Comment information is displayed about the graphic. You can change the Comment to suit your needs and it is included in the print output.

4.3.3 Context menu

Here, use the left-hand mouse button to select, move, zoom and measure elements in a graphic. You can permanently select which action is to be performed in the context menu. You can access this more quickly by using these combinations: Move: Shift, Zoom: Ctrl and Measure: You can select multiple single items by pressing the Alt key while holding down the left-hand mouse button.

Other actions in the context menu are: Zoom In (plus), Zoom out (minus) and Fit window (Pos1 or Home). Use the direction keys to move the current section of the graphic.

4.3.4 Properties

In Properties, you can display or hide elements in a graphic and change its colors and line styles. You can make different modifications, depending on the graphic: for diagrams and such like, you can modify the value ranges and units to match the axes, or, for a meshing, you can change the center distance.

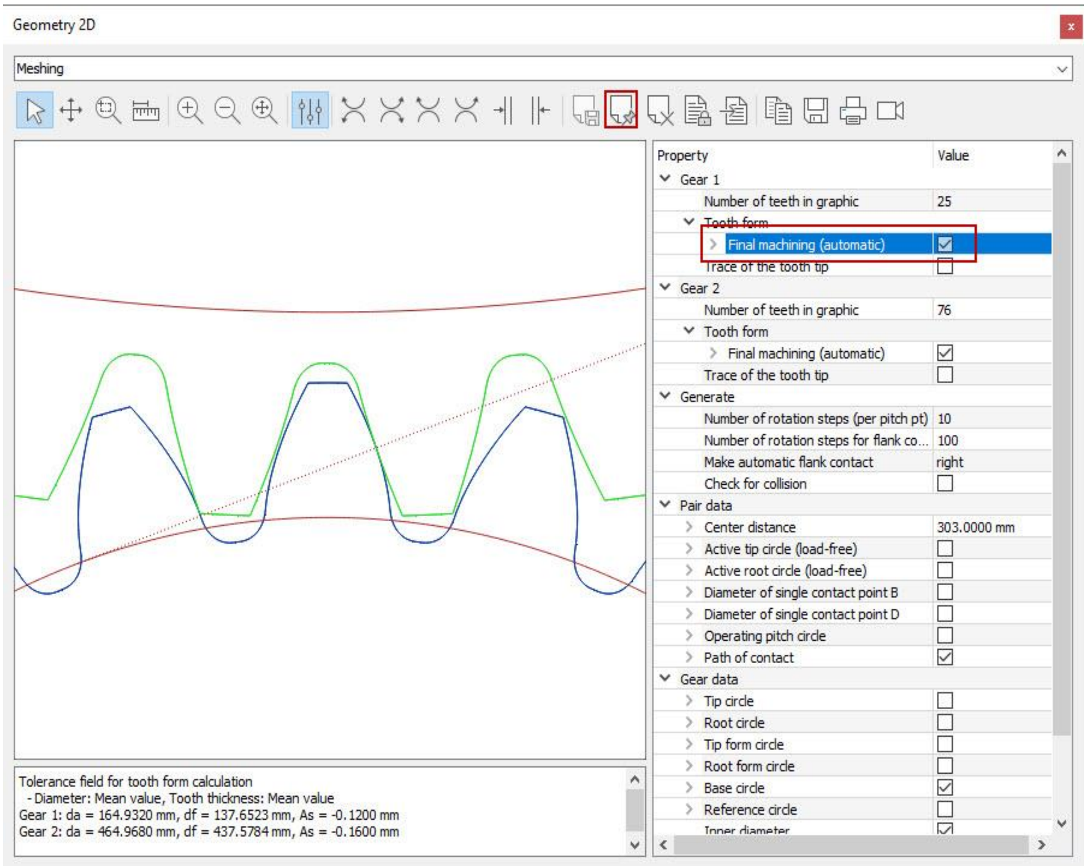
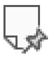


Figure 4.7: Graphic properties

If the properties are displayed, you will see three other icons in the tool bar. You use them to store curves in a graphic as text, or in the graphic itself.

Save curve as text

Stores the coordinates of the curve selected in Properties in a text file. This makes it easy to transfer curves to, for example, an Excel file.



Save curve

Stores the curve selected in Properties in the graphic. This function is ideal for comparing the graphical outputs of a calculation while you change its parameters.

Delete memory

Deletes the curve from the memory.

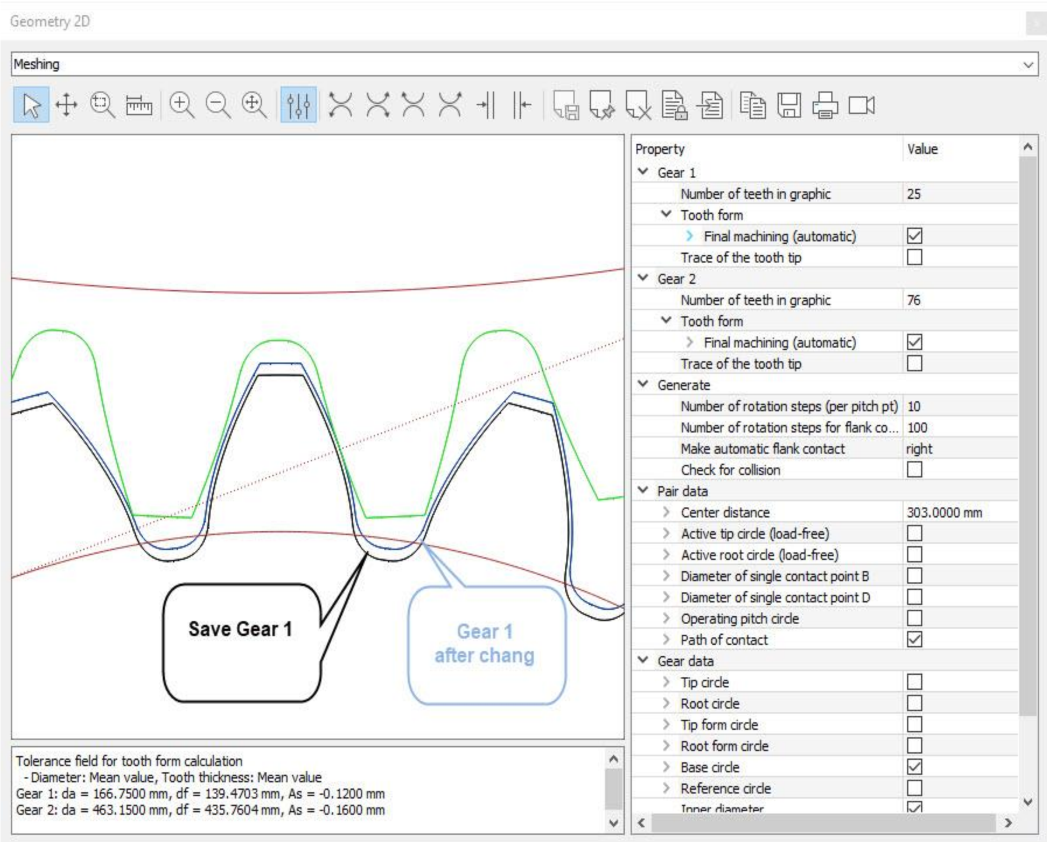


Figure 4.8: Graphics with saved and different curves

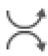
4.3.5 Toothing

If you select Toothing, additional icons are displayed for generating the gear pair and creating the flanks when you open the **Geometry** graphics window.

Rotate to the left

Generates the gear pair to the left.

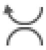
Key combination: Ctrl + left direction key



Rotate to the right

Generates the gear pair to the right.

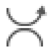
Key combination: Ctrl + right direction key



Rotate one gear independently to the left

One gear remains static while the other is rotated to the left. The profiles overlap.

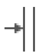
Key combination: Alt + left direction key



Rotate one gear independently to the right

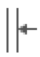
One gear remains static while the other is rotated to the right. The profiles overlap.

Key combination: Alt + right direction key



Make flank contact left

The gears are rotated until the flanks of both gears touch on the left.



Make flank contact right

The gears are rotated until the flanks of both gears touch on the right.

▶ **Note:**

If you press and hold down a button, to rotate it, the gears rotate continuously (movie).

▶ **Note:**

Click Properties (see chapter 4.3.4, Properties) to specify the number of rotation steps for the rotation. The number of rotation steps here refers to the pitch.

4.4 Main input area

The main input area shows a calculation module's input window. In addition, it is used to display the internal report viewer or the internal help viewer.

4.4.1 Report Viewer

When you generate a report in KISSsoft, the report viewer opens in the main input area, the entries in the **Report** menu are activated and the report viewer tool bar is displayed. The report viewer is a text editor that supports the usual functions for saving and printing a text file. In KISSsoft, you can save reports in portable document format (.pdf), in Microsoft Word format (.doc) or as ANSII text (.txt).

The report viewer's other functions are Undo/Redo, Copy, Cut and Paste, and Search, with the usual shortcuts. You can zoom in on the view and edit the report later on by changing the font size, bold, italics and underlining style. To change the general appearance of the report, select **Report** > **Settings**.

4.4.2 Helptext Viewer

The KISSsoft manual is displayed in the Helptext viewer in HTML format. To open the manual, select something in the Table of Contents or the Search function. If you press function key **F1**, the system displays more information about where the cursor is currently located in KISSsoft.

4.5 Tooltips and status bar

Whenever it is useful, tool tips are provided in KISSsoft, to give you additional information about program elements. Tooltips appear automatically if you slowly move the mouse over a program element.

If you position the mouse over a particular menu option, the system will display detailed information on all actions available in that menu, in the left-hand area of the Status bar. If the mouse is positioned over a selection list, the currently selected list entry will be displayed in the status bar. This is especially helpful if the display is restricted by the width of the selection list.

In the right-hand area of the status bar, the system will display the current status of the calculation. The flag is set to CONSISTENT if the results are current. INCONSISTENT shows that a new calculation needs to be carried out.

5 KISSsoft Calculation Modules

5.1 Standard and special tabs

The input window for most calculation modules is subdivided into different tabs. This ensures that inputs are separated logically. The system does not automatically display all the existing tabs for more complex calculations, such as for a cylindrical gear pair. When you open a new calculation, you only see the tabs that contain the absolutely essential inputs. (For example, for a cylindrical gear pair, this would be the **Basic data, Reference profile, Manufacturing** and **Tolerances** tabs.) In the **Calculation** menu, you can add more tabs if needed (e.g. for a cylindrical gear pair, you would need to add the **Modifications** tab if you wanted to modify the gears).

KISSsoft calculation modules use two types of tabs: Standard tabs and special tabs (see Figure 5.1).

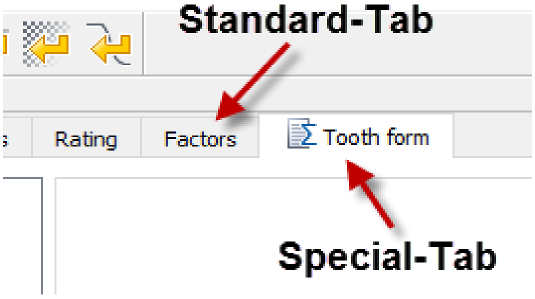


Figure 5.1: Standard and special tabs

If a standard tab (e.g. **Basic data**) is active when the calculation is run, then the basic calculation will be performed and the results for this basic calculation will be displayed in the (basic calculation) Results window (see chapter 4.2.3, The Results window). The basic calculation report is created when a report for this data is generated.

Special tabs are marked with the icon. If this type of special tab is active when the calculation is run, then a special calculation will be performed in addition to the basic calculation. (For example, for a cylindrical gear pair, the path of contact under load is calculated.) The (special calculation) Results window then displays the results for the additional calculation along with the results for the basic calculation,. When a report is generated, the results for the additional calculation are also output.

5.2 Input elements

All KISSsoft calculation modules use the same input elements for input. These input elements are described in more detail in the sections that follow.

5.2.1 Value input fields

In general, a value input field always includes the variable label, a formula character, the edit field and a unit. If the edit field is grayed out, this variable cannot be predefined. Instead it will be determined during the calculation. One or more of the following buttons can follow a value input field:

☐

You can retain a value by selecting the Check button.

○

You can set a radio button to specify which values in a group should be calculated and which should be retained.

↵

Click the Sizing button to calculate the value using calculation methods.

↔

Click the Convert button to calculate the value using conversion formulae.

+

Click the Plus button to display additional data for a value.

Click the Settings button to display additional setting options for a value.

Click the Info button to display information in the Info window (see chapter 4.2.5, The info window).

5.2.2 Formula entry and angle input

In some cases it is advisable to use a small auxiliary calculation to determine a value. Right-click in a value input field's edit field (see chapter 5.2.1, Value input fields) to open a formula editor. In it, you can enter a formula, which must be one of the four basic calculation types: +, -, * and /. Additionally, you can use all the functions that are supported by the report generator (see Table 8.2). Confirm the formula by pressing Enter. The system will evaluate the formula. The formula itself will be lost: if you return to the formula entry dialog, the calculated value will be shown there instead of the formula.

In value input fields (see chapter 5.2.1, Value input fields) that display an angle, a dialog in which you can input degrees, minutes and seconds will be displayed instead of the formula editor.

5.2.3 Unit switch

In KISSsoft, you can switch all the units in the value input fields (see chapter 5.2.1, Value input fields) and in the tables (see chapter 5.2.1, Value input fields). To do this, right-click on a unit. This opens a context menu in which you see all the possible units for the value. If you select a different unit from the one that is currently in use, KISSsoft converts the current value in the value input field to the new unit.

To switch between metric and US customary units globally, select **Extras > Systems of units**.

5.2.4 Tables

In some modules, data is displayed or entered in a table. You select a row by double-clicking, just like when you select a field for input. For tables, additional information is often displayed in a tooltip (see chapter 4.5, Tooltips and status bar). In general, the following buttons come after tables so that you can input data:

+

Click the Add button to add a row to the table.

—

Click the Remove button to delete the selected row from the table

×

Click the Clear button to delete all entries in the table

5.3 Calculating and generating a report

Click **Calculation >Run** to perform the current calculation. In addition, the tool bar and the **F5** function key give you quick, convenient access to this action. Here, please note that a calculation module can contain other special calculations, in addition to the basic calculation. These special calculations are only performed if the appropriate special tab (see chapter 5.1, Standard and special tabs) is active.

Select **Report > Generate** to generate a report about the current calculation. Also note that the default report and the reports about the special calculations are displayed separately in the special tabs (see chapter 5.1, Standard and special tabs).

The status of a calculation is consistent if it was possible to perform it without error. As soon as you change data in the input window, the calculation becomes inconsistent, which means the results of the calculation in the Results window and the graphics no longer match the data in the interface. The current status of the calculation is displayed in the status bar (see chapter 4.5, Tooltips and status bar).

5.4 Messages

A calculation sends different types of messages to the input window: information, warnings and errors. Information and warnings should always be taken note of to ensure accurate results. If an error has occurred, the calculation is interrupted.

Normally, all the messages are displayed in a message box and in the Messages window (see chapter 4.2.4, The Messages window). You can change the way information and warnings are displayed in a message box by selecting **Extras** > **Settings**, and clicking on the Messages tab.

6 Project Management

KISSsoft has its own project management system, which you can use to organize your calculation files and external files. The most important area in the project management system is the KISSsoft Project Tree (see chapter 4.2.2, The project tree). In it, you can see which projects are currently open or active, and you can see all the information about the files that belong to the individual projects.

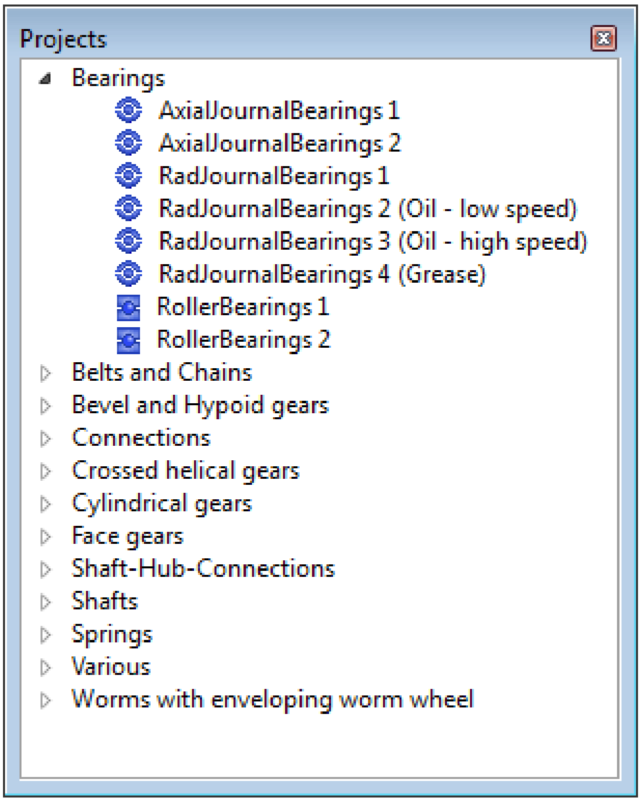


Figure 6.1: The KISSsoft project tree

6.1 Generating, opening and closing projects

Select **Project > New ...** to create a new project. A dialog opens in which you enter the name of the project, the project directory, descriptions and comments, and also the directory for the template files (see chapter 2.4, Defining your own template files) that are to be used. The newly created project is inserted into the Project Tree and defined as the Active working project (see chapter 5.2.3, Unit switch).

If you open an existing project (**Project > Open...**) this will also be inserted into the project tree and defined as the Active working project (see chapter 6.3, The active working project).

You close a project by selecting it and then selecting **Project > Close**. You will also find this action in the context menu (see chapter 4.1, Menus, context menus and the tool bar) in the Project Tree. The project will still be retained, and you can open it again at any time.

6.2 Adding and deleting files

Files can be added and deleted either via the Project properties (see chapter 6.5, Project properties) or the context menu (see chapter 4.1, Menus, context menus and the tool bar). As well as calculation files from KISSsoft, you can insert any external files in a project.

6.3 The active working project

The project tree shows all the open projects. It is not absolutely necessary to define an active working project. If you have defined an active working project, it is highlighted in bold. You can also set a project as an active working project by selecting **Project > Set as working project** or by activating it via the context menu. If you select **Project > Work without project**, this deactivates the active working project.

The current calculation file does not have to belong to the active working project.

6.4 Storage locations

Files that belong to a particular project do not have to be stored in that project's directory. This means files can belong to several projects at the same time. However, if you have defined an active working project (see chapter 6.3, The active working project), KISSsoft will prompt you with its project directory as the first choice storage location whenever you want to open or save a calculation file or a report. If you are working without a project, the system will display your personal user directory (see chapter 2.6.1, Definitions in [PATH]) as a default storage location.

6.5 Project properties

To display the project properties for the selected project, either select **Project > Properties**, or use the Project Tree's context menu (see chapter 4.1, Menus, context menus and the tool bar).

7 Dynamic User Interface

The KISSsoft interface is defined by its editable text files (descriptive data). The elements it contains are fixed components of the software. However, any user can decide how these elements are divided up and arranged. Frequently used entries can be given priority in the tabs and dialog and less commonly used entries can be either hidden or write-protected. KISSsoft can therefore easily be adapted to suit the requirements of individual users.

7.1 Modified tabs and dialogs supplied with the system

The description files for the tabs and dialogs supplied with the system are stored in the kui (kisssoft user interface) directory. These files should never, under any circumstances, be modified by the user. This is because interface upgrades, which are supplied with a patch, always overwrite any user modifications. To modify the interface to suit your own requirements, copy the appropriate description file to the ext/kui directory and change it there. KISSsoft evaluates the files in this directory first. The description files are assigned to the appropriate calculation module by their file name and the file extension .kui. This is why the file name must not be changed.

7.2 Adding additional tabs and dialogs

The description files for additional tabs and dialogs are stored in the ext/dui (dynamic user interface) directory. KISSsoft evaluates the files in this directory every time a module is started. You can give these files any name you want, although the file extension must always be *.dui.

The `<module>` tag tells KISSsoft which calculation module the description file was defined for. This entry is mandatory for tabs. The titles of the tabs or dialogs are defined by the `<title>` tab. The tag can contain either an actual text or the ID (number) of a text from the KISSsoft Glossary (wpoolUi_.txt).

Use the `<before>` tag to define the position of the additional tabs. If you do not see the `<before>` tag, the additional tab is placed after the standard system tabs. An additional tab can also be used to replace a standard tab. To exclude a tab, set the `<exclude>` tag.

Example of an additional tab:

```
<KISSsoft filetype="userinterface">

<module>Z012</module>

<title>My own title</title>
```

```
<before>Z012_Tolerances</before>

<exclude>Z012_BasicData</exclude>

<element>a</element>

<element>Q</element>

</KISSsoft>
```

An additional tab is always displayed. Set the <permanent> tag to define that the tab can be enabled via the **Calculations** menu.

```
<permanent>false</permanent>
```

Additional tabs always work in the same way as the standard tabs supplied with the system. Insert the `<calculation>`, `<report>` and `<results>` tags to represent the behavior of a special tab. The `<calculation>` tag executes a COM function. All the functions that are available via the COM interface are also available here. The name of the corresponding template is set for the report and the results (see chapter 8, Results and Reports).

Use the `<setup>` tag to assign a COM function to additional dialogs. This function is executed when those dialogs run.

Examples of additional description files can be requested from KISSsoft AG.

7.3 Formatting

7.3.1 Elements

Set the <element>element name</element> to add an element. The elements in the description file appear in the same sequence as they appear in the interface.

The following element types are available:

Value input fields	For entering whole number values or floating values
Selection lists (drop-down lists)	For selecting list entries, database entries, materials, lubricants or load spectra
Checkboxes	For selecting/deselecting calculation options
Titles and texts	For structuring the interface

Click on kui in the Help directory to display lists of available elements.

7.3.2 Columns

Set the <column> tag to add a column. The columns in the description file appear in the same sequence as they appear in the interface. You will not usually need more than two columns.

Example of a two-column layout:

```
<column>

<element>Element1</element>

<element>Element2</element>

</column>

<column>

<element>Element3</element>

<element>Element4</element>

</column>
```

7.3.3 Groups

Set the <group> tag to add a group. The groups in the description file appear in the same sequence as they appear in the interface. Groups can also contain columns. Groups cannot be nested.

Set the <title> tag to define a group's title. The tag can contain either an actual text or the ID (number) of a text from the KISSsoft Glossary (wpoolUi_.txt).

Example of a group:

```
<group>

<title>145</title>

<element>Element1</element>
```

```
<element>Element2</element>

</group>
```

7.3.4 Tabs

Dialogs can also have tabs. Set the `<tab>` tag to add a tab. The tabs in the description file appear in the same sequence as they appear in the dialog. Each tab includes elements that are arranged in groups or columns. Sub-tabs are not supported in the tabs in a calculation module.

Set the `<title>` tag to define the title of a tab. The tag can contain either an actual text or the ID (number) of a text from the KISSsoft Glossary (wpoolUi_.txt).

7.3.5 Attributes

The following attributes can be set for an element:

Attribute	Value	Description	Usage
de, en, fr, it, es, pt, ru (all: obsolete!)	Actual text or the ID (number) of a text in the KISSsoft Glossary (wpoolUi_.txt)	Overwrites the element's label for a language. Use this option to create company-specific or regional glossaries.	element, title
prompt	ID (number) of a text in the KISSsoft Glossary (wpoolUi_.txt)	Overwrites the element's label. Use this option to create company-specific or regional glossaries.	element
showPrompt	false	The label is not displayed.	element (type selection list)
ignoreTitle	true	The group is displayed without a label.	group
dynamic	true	The label is determined using a function.	title
readOnly	true	Set this attribute to write-protect the associated element. Use this option to predefine values (see "Defining your own default files" in the manual) and prevent other users from changing them.	element
decimals	2	The set unit of decimal places is then used as the default in the interface.	element (value input type)

unit	DEGREE, MILLIMETER, INCH, etc.	The set unit is then used as the default in the interface.	element
index	1, 2, 3, etc.	Elements with multiple entries are reduced to a fixed index.	element
count	1, 2, 3, etc. Function, for example, GetGearCount	Sets the number to be used for elements with multiple entries.	element
visibleCondition	Function, for example, IsOwnInput	If this attribute is set, the associated element is only displayed if this function returns "true".	element, group, tab
shrink	true	The element can be narrower than its contents.	element (type selection list)
layout	table	The table fills the entire range (including label, formula symbol, unit).	element (type table)
joinLayout	off	The group or tab is not linked to the automatic layout.	group, tab
alignment	left, right, center	The input elements are left-justified, right-justified, or centered.	element, text, button
hSpacer	skip	The automatic horizontal placeholder is not set.	group
vSpacer	skip	The automatic vertical placeholder is not set.	dlg, tab, column
geometry	1000x450, etc.	The dialog is displayed in the predefined size.	dlg
editButton	no Function e.g. IsOwnInput dummy	The element is not provided with an EditButton, even if one is present (a Checkbox or RadioButton is not displayed). The element is not provided with an EditButton if the function is not fulfilled, even if this button is present. The element is given a placeholder.	element

button	KUI file	The element is provided with a button that displays a dialog as defined in the KUI file.	element

Table 7.1: <element attribute1="wert" attribute2="wert">Name</element>

7.3.6 Comments

Comments in a description file are a useful way of explaining how the file is structured. Comments start with //.

`<title>32</title>` // Basic data

7.3.7 Special elements

7.3.7.1 Separator

A (horizontal) separator can be added like this

`<line></line>`

7.3.7.2 Text

To insert a (horizontal) text:

`<text>975</text>` // ID (number) of a text in the KISSsoft glossary (wpoolUi_.txt)

`<text>My own text</text>` // actual text

7.3.7.3 Button

A button can be added in this way:

<button prompt="1799">MyDialog.kui</button> // Name of a KUI file for opening a dialog.

<button prompt="1799">MyFunction</button> // Name of a function that is to be executed by clicking the button.<accept>MyFunction</accept> // Name of a function that is to be executed by clicking the OK button in a dialog.

<action prompt="2053">ExportToothForm</action> // Name of a function that is to be executed by clicking an additional action button in the button area in a dialog.

<report>MyReportFile.rpt</report> // Name of a report template that is to be executed by clicking the report button in the button area in a dialog.

8 Results and Reports

8.1 Results of a calculation

KISSsoft displays the results of a calculation in the Results window (see chapter 4.2.3, The Results window). If no results are displayed, an error has occurred during the calculation. If this happens, a system message appears in a message box to alert you to the error. An indicator in the status bar (see chapter 4.5, Tooltips and status bar) shows whether the results are consistent, i.e. whether the results match up with the data in the user interface.

8.1.1 Add your own texts in the results window

To enable this, define a new file in the KISSsoft installation folder in "…\ext\.rpt\". This file must then be named using this convention: "Modulname + result.RPT" (e.g. for a cylindrical gear pair Z012result.RPT).

Then define the new parameters or values that are to be added. These values then also appear at the end of the "Results" window.

The syntax corresponds exactly to the entries for the report templates.

8.2 Calculation reports

Select **Report > Generate** to generate reports about your calculations. In addition, the tool bar and the **F6** function key give you quick, convenient access to this action. The report contents depend on which tab is currently active (see chapter 5.1, Standard and special tabs). The Length (see chapter 8.5.2, Scope of a report) and Appearance (see chapter 8.5.3, Formatting) of standard reports can be influenced by user-defined report templates (see chapter 8.5, Report templates).

A calculation module can contain further reports which you can access via the **Report** menu.

Reports are usually displayed in the KISSsoft Report Viewer (see chapter 4.4.1, Report Viewer). **Important**: The report is not saved when you return from the report viewer to the input window. To make it permanently available, you must save it with a new name!

► **Note**

In general, a report should only be created if the calculation is consistent (see chapter 5.3, Calculating and generating a report). If this is not the case, you can still generate the report, but the status of the calculation will then be noted in the report.

► **Note**

When you generate a standard report, the system generates a report file with the module's label as its file name. The file is saved in the directory defined as the TEMPDIR (see chapter 2.6, Global settings - KISS.ini) in the KISS.ini file (see chapter 2.6.1, Definitions in [PATH]).

8.3 Drawing data

Depending on the calculation module, you can select **Report > Drawing data** to generate a report which can be used to output drawings.

8.4 Report settings

Select the **Report >Settings** menu option to tailor the automatic generation of reports. All the settings can also be defined globally in the KISS.ini (see chapter 2.6.3, Definitions in [REPORT]) file.

8.4.1 General

Here you define the scope of the report (see chapter 8.5.2, Scope of a report) and whether warnings from the calculation are to be included in it. You can also set the font size, language and the standard format used to save reports.

The report can be viewed in two different modes: "overwrite" or "compare".

If a report is generated, and a previous report is still open, the data will be updated. The cursor in the editor will remain in the same line it was in before this. This feature will help you analyze specific values using different inputs.

In the report settings, change the report mode to "compare" if you need to compare two or more reports at a time. This mode can only by set if you are using KISSedit as the editor. You can also synchronize the reports and scroll through them all at the same time.

You can also set these report settings directly in the KISS.ini file.

8.4.2 Page layout

Here you can define the paper size and the page margins used to generate reports automatically.

8.4.3 Header and footer

In KISSsoft, reports are usually generated with headers and footers. You can define your own header and footer lines. There are a number of placeholders available for this.

Placeholder	Description
%logo	Picture file
%date	Dated
%time	Time
%pn	Number of pages
%pc	Number of pages
%t	Tab

The %logo placeholder uses the selected graphics file to integrate a user-defined logo (company label). The date and time are output in accordance with the details specified under **Extras > Settings**.

8.4.4 Start and end block

Reports in KISSsoft are usually generated with a start block and an end block. You can define these start and end blocks yourself. The start and end blocks are defined in template files which are stored in the **rpt** directory in the installation folder.

Language	Start block file	End block file
German	kissd.rpt	kissfd.rpt
English	kisse.rpt	kissfe.rpt
French	kissf.rpt	kissff.rpt
Italian	kissi.rpt	kissfi.rpt
Spanish	kisss.rpt	kissfs.rpt
Russian	kissr.rpt	kissfr.rpt
Portuguese	kissp.rpt	kissfp.rpt
Chinese	kissc.rpt	kissfc.rpt

Commands that can be used in these templates and what they mean:

Command	Description
DATE	Date (select "Extras > Settings" and then set your preferred output format.)
TIME	Time (select "Extras > Settings" and then set your preferred output format.)
PROJECT	Project name
PROJECTDESCRIPTION	Description of the project
FILENAME/DESCRIPTION	File name
FILENAME.EXT	File name with extension (e.g. "Example1.Z12")
FILEPATH	Path with file name (e.g. "C:\Temp\GearPair.Z12")
DESCRIPTION	Description of the file
COMMENT	Comment for the file
CUSTOMER	Customer name as defined in the project
USER	User name (Windows user name)
RELEASE	Version number (e.g. "04-2010")
COMPANY	Company name (as defined in the license file)
NLINES	Number of lines in the report
IMPERIALUNITS	Whether US customary units are specified for IF statements
METRICUNITS	Whether metric units are specified for IF statements
PROJECTUSED	Whether projects are used for IF statements

8.5 Report templates

For each calculation module, KISSsoft provides report templates to define the form and content of the reports. You can use these supplied templates as the basis for generating user-defined templates for producing reports that meet your requirements. However, you must ensure the Formatting (see chapter 8.5.3, Formatting) and Storage locations (see chapter 8.5.1, Storage locations and descriptions) remain the same.

8.5.1 Storage locations and descriptions

The report templates supplied by KISSsoft are stored in the directory that has been set as RPTDIR (see chapter 2.6, Global settings - KISS.ini) in the KISS.ini (see chapter 2.6.1, Definitions in [PATH]) file. If RPTDIR (see chapter 2.6.1, Definitions in [PATH]) was not defined in KISS.ini (see chapter 2.6, Global settings - KISS.ini), you will find the templates in the installation folder under rpt. It is essential that user-defined report templates are stored in the RPT subdirectory, in the EXTDIR (see chapter 2.6.1, Definitions in [PATH]) directory. This is the only way to prevent your templates from being overwritten if a patch is installed. When the system generates a report, it uses the user-defined template from the EXTDIR directory, if present. Otherwise it uses the template from the RPTDIR to create the report.

The report template labels have this structure: *MMMMlsz.rpt*It is made up of:

MMMM	Module descriptor	e.g. *M040*
l	For historical reasons,	always = *l*
s	Language of the report	*s = d, e, f, i, s or a*
z	For historical reasons,	always = *0*
.rpt	File type	

► **Examples**

Bolt calculation:	
M040LD0.RPT	Bolt calculation, German printout
M040USER.RPT	Default printout via the interface, results in the M040USER.OUT file
Cylindrical gear calculation:	
Z012LD0.RPT	Cylindrical gear pair, German printout
Z012USER.RPT	Default printout via the interface, results in the Z012USER.OUT file
Z10GEAR1.RPT	Output via interface, contains only data
	for gear 1, results in file Z10GEAR1.OUT
Z10GEAR2.RPT	Output via interface, contains only data
	for gear 2, results in file Z10GEAR2.OUT
Z011LD0.RPT	Single gear, German printout
Z013LD0.RPT	Rack, German printout
Z014LD0.RPT	Planetary gear, German printout
Z015LD0.RPT	3 gears, German printout
Z016LD0.RPT	4 gears, German printout
Spring calculation:	

F10SPRING.RPT	Default printout for drawing data results in the F10SPRING.OUT file
English printout:	
M040LE0.RPT	Bolt calculation, English printout
American printout:	
M040LA0.RPT	Bolt calculation, American printout

8.5.2 Scope of a report

To preset the scope or length of a report, on a scale of 1 to 9, select the **Report > Settings** menu option. 9 will produce a complete report, and 1 will produce a short report. In the report template, you see a number between 1 and 9 at the beginning of every row. This number works together with the setting described above to determine whether or not the row is to be read.

Example: If you entered 5 (medium) as the report length, all the lines in the report template that start with 1, 2, 3, 4 or 5 are read. Rows with 6, 7, 8 and 9 will be not read.

8.5.3 Formatting

Both the report template and the report generated from this are text files that are created with the Microsoft Windows font. You should always edit text in MS Windows, otherwise accented characters such as ä, ö, ü, as well as some special characters, may be represented incorrectly.

The following statements and key words are defined in the report format:

- Texts that are to be output
- Comments that are not to be output
- Descriptions and formatting of calculation variables
- Limited branchings (*IF ELSE END*)
- Iterations (*FOR*-loops)

8.5.3.1 Text formatting features

You can use these text formatting features in RPT:

Description	Start	End
Underline	<UL>	</UL>
Cross out	<STRIKE>	</STRIKE>

Grease	<BF>	</BF>
Italic	<IT>	</IT>
Superscript	<SUPER>	</SUPER>
Subscript	_	
Font size	<FONTSIZE=xx>	
Enlarge font size	<INCFONTSIZE>	</INCFONTSIZE>
Reduce font size	<DECFONTSIZE>	</DECFONTSIZE>
Page break	<NEWPAGE>	
Line break	 	
Text color red	<RED>	<BLACK>
Text color green	<GREEN>	<BLACK>
Text color blue	<BLUE>	<BLACK>
Http link or e-mail address	<LINK=destination>	</LINK>
Blank space	<SPACE>	
Insert figure	<IMAGE=name,WIDTH=xx,HEIGHT=yy>	
Insert image	<INCLUDEGRAPHIC=name,WIDTH=xx,HEIGHT=yy>	
Adding a report template	<EXECUTE=name.rpt>	

8.5.3.2 Comments

Comment lines begin with //. Comments are ignored when a report is created.

► **Example**

```
// Hier habe ich am 13.12.95 die Protokollvorlage geändert, hm

Aussendurchmesser mm : %10.2f {sheave[0].da}
```

In this case, only the second line will be output.

8.5.3.3 Calculation variables

You cannot define your own variables (apart from the number variables used for FOR loops (see chapter 8.5.3.5, FOR loop), which you (as the user) specify, and which can output a value.

Placeholder

Use placeholders to specify the file type and formatting for a variable:

- *%i* stands for a whole number
- *%f* stands for a floating point number
- *%n*1.*v*2*f* stands for a formatted floating point number with *v*1 places in total (including sign and decimal point) and *v*2 decimal places
- *%s* stands for a left-justified character string (text)
- *%ns* stands for a right-justified character string in a field with length *n* characters (*n* is a whole number).

The data types must match the definition in the program. The value is returned in exactly the place where the placeholder is positioned. The syntax of the formatting corresponds to the C/C++ standard.

► **Examples**

- *%10.2f* returns a floating point number in a field that is 10 characters long and has 2 decimal places. In this case, the decimal point is in the position of the "%" character.
- *%i* returns a whole number that is right-justified to the left of the percentage sign. This number is therefore positioned directly underneath the whole number part of a floating point number and in the same position.
- *%30s* stands for a right-justified character string in a field with length 30 characters. If the number 30 is omitted, the numbers are left-justified when they are output.

► **Counter-examples**

- *%8.2i* is an invalid formatting because a whole number has no decimal places.
- *%10f2* outputs a right-justified 10-digit floating point number. However, the 2 decimal places are ignored and output as text 2. The default setting is to output floating point numbers to 6 decimal places.

Variables

The variable to be displayed must stand after the placeholder in the same row. The variable is identified by being enclosed in curly brackets. If these brackets are left out, the variable name will be displayed as normal text.

Important: It is essential that the number of placeholders exactly matches the number of pairs of brackets {}.

► **Example**

%f {sheave[0].d} returns the value of the variable *sheave[0].d* in the location *%f* as a floating point number with 6 decimal places.

Basic calculation types - output of changed variables

You can output changed variables in the report. They can be multiplied or divided with a coefficient. You can also add or subtract a number. This functionality is also available in the arguments used in the *IF* or *FOR* statements (see below).

Value of the variable multiplied	*%3.2f*	*{Var*2.0}*
Value of the variable divided	*%3.2f*	*{Var/2.0}*
Value of the variable added	*%3.2f*	*{Var+1.0}*
Value of the variable subtracted	*%3.2f*	*{Var-2}*

The two Degree and Gear functions are also available for converting variables to degrees or radians:

Angle %3.2f {degrees(angle)}

Variables can also be linked with each other directly, e.g. in the form *{sheave[0].d- sheave[1].d}*. More than two numbers can be linked. Numbers that have signs must be enclosed in brackets, for example *{ZR[0].NL*(1e-6)}*.

The available functions are listed in Table 8.2.

Function	**Meaning**
sin(angle)	sine of angle in the radian measure
cos(angle)	cosine of angle in the radian measure
tan(angle)	tangent of angle in the radian measure
asin(val)	arcsine of val, returns radian measure
acos(val)	arccosine of val, returns radian measure
atan(val)	arctangent of val, returns radian measure
abs(val)	|val|
exp(val)	eval
log(val)	Return value x in ex = val
log10(val)	Return value x in 10x = val
sqr(val)	Return value val2
sqrt(val)	Return value $\sqrt{val}$
int(val)	Whole number of val
pow(x;y)	Return value xy

sgn(val)	Return value $\begin{matrix} 1 & wenn\ val > 0 \\ 0 & wenn\ val = 0 \\ -1 & wenn\ val < 0 \end{matrix}$
sgn2(val)	Return value $\begin{matrix} 1 & wenn\ val \geq 0 \\ 0 & wenn\ val < 0 \end{matrix}$
grad(angle)	Converting from the radian measure to degrees
rad(angle)	Converting from degrees to radian measure
DegMinSec(angle)	Return angle as string (10°5'55")
mm_in(val)	Return value val/25.4
celsius_f(val)	Return value $\frac{9}{5}$ val + 32
min(v_1; ...; v_5)	The return value is the minimum of $v_1,\ldots,v_5$
max(v_1; ...; v_5)	The return value is the maximum of $v_1,\ldots,v_5$
and(v_1; v_2)	binary and function
or(v_1; v_2)	binary or function
xor(v_1; v_2)	binary exclusive or function
AND(v_1; ...; v_5)	logical and function
OR(v_1; ...,v_5)	logical or function
NOT(val)	Return value $\begin{matrix} 0 & wenn\ val \neq 0 \\ 1 & wenn\ val = 0 \end{matrix}$
LESS(v_1; v_2)	Return value $\begin{matrix} 1 & wenn\ v_1 < v_2 \\ 0 & wenn\ v_1 \geq v_2 \end{matrix}$
EQUAL(v_1; v_2)	Return value $\begin{matrix} 1 & wenn\ v_1 = v_2 \\ 0 & wenn\ v_1 \neq v_2 \end{matrix}$
GREATER(v_1; v_2)	Return value $\begin{matrix} 1 & wenn\ v_1 > v_2 \\ 0 & wenn\ v_1 \leq v_2 \end{matrix}$
ROUND(x;n)	Rounds off x to n places
strlen(str)	Length of character string
strcmp(str1;str2)	Compare character string Return value: 1 if str1 = str2 0 otherwise

Table 8.1: Functions available for calculations in the report

8.5.3.4 Condition query IF ELSE END

The condition query or branching enables you to output certain values and texts only if a particular condition has been fulfilled. The following conditions are supported:

Combination of characters	**Meaning**
==	equal to
>=	greater than or equal to
<=	less than or equal to
!=	not equal to
<	less than
>	greater than

Table 8.2: Boolean functions available in the report

This condition is entered as follows:

IF (condition) {Var}
Case 1
ELSE
Case 2
END;

▶ **Example**

IF (%i=0) {Zst.kXmnFlag}
Addendum modified no
ELSE
Addendum modified yes
END;

If the *Zst.kXmnFlag* variable is 0, the first text is output. If not, the second text is output. There can be any number of rows between *IF, ELSE* and *END*. For each branching opened with *IF* you must use *END*; to close it again (do not forget the semicolon after *END*). The key word *ELSE* is optional. It reverses the condition. Branchings can be nested within each other up to a depth of 9.

▶ **Example of a simple branching**

IF (%i=1) {ZP[0].Fuss.ZFFmeth}
Calculation of tooth form coefficient according to method: B
END;

If the *ZP[0].Fuss.ZFFmeth* variable is 1, the text is output. If not, no text is output.

► **Example of encapsulated branchings**

IF (%f≤ 2.7) {z092k.vp}				
	Regular manual lubrication			(Text1)
ELSE				
	IF (%f<12) {z092k.vp}			
		Lubrication with drop dispenser (2 to 6 drops per minute)		(Text 2)
	ELSE			
		IF (%f<34) {z092k.vp}		
			Lubrication with oil bath lubrication	(Text 3)
		ELSE		
			Lubrication with circulation system lubrication	(Text4)
		END;		
	END;			
END;				

If the *z092k.vp* variable is less or equal to 2.7, text 1 is output. Otherwise you are prompted to confirm that *z092k.vp* is less than 12. If yes, text 2 is output. Otherwise you are prompted to confirm that *z092k.vp* is less than 34. If yes, text 3 is output, otherwise text 4 is output.

8.5.3.5 FOR loop

In KISSsoft you can also use *FOR*loops in the report generator. A numerical variable will be incremented (or decremented) within a FOR loop. You can use constructs that are nested down to 10 levels.

This loop is specified as follows:

FOR varname=%i TO %i BY %i DO {initial value}{final value} {step}
// access to variable with #varname or $varname
...
END FOR;

- Instead of *%i* or *%f* you can also use fixed figures (static *FOR*loop):

  FOR varname=0 TO 10 BY 1 DO
  ...
  END FOR;

- or a mixture:

  FOR varname=5 TO %i BY -1 DO {final value}
  ...
  END FOR;
- Each *FOR*loop must end with the statement *END FOR;* (including semicolon). Each defined numerical variable (*varname*) within the loop can be addressed with the statement *#varname* .
- The increment can also be selected as a negative value (for example -1). However, it must never be 0. You must always specify the intervals.
- The *#varname*statement can be used to define a variable. For example:
- Number of teeth: *%3.2f {ZR[#varname].z}*
- The *$varname*statement can be used to output the variable value as a letter. The value 0 corresponds to A, 1 corresponds to B etc. For example:

  FOR quer=0 TO 3 BY 1 DO
  Cross section *$quer-$quer: %8.2f {Qu[#quer].sStatisch}*
  E*ND FOR;*

► EXAMPLE OF A SIMPLE LOOP

FOR i=0 TO 10 BY 1 DO
Run number #i $i
END FOR;

Results in the following output:

Run number 0 A
Run number 1 B
Run number 2 C
Run number 3 D
Run number 4 E
Run number 5 F
Run number 6 G
Run number 7 H
Run number 8 I
Run number 9 J
Run number 10 K

The numerical variable can be used anywhere within the loop, even for arrays.

9 Database Tool and External Tables

In addition to the unique data you input for calculations, you may also encounter recurring data such as a material's characteristics. KISSsoft will store these characteristics in databases. You view and change them with the **database tool**, whose use will be explained in the following sections. Tables form the elements of the databases and are contained in your program package as editable ASCII files. The External tables (see chapter 9.4, External tables) section deals with the setting up and handling of external tables (also called "look-up tables).

In KISSsoft there are four databases:

KMAT	- Materials
M000	- Shaft-Hub Connection and Bolts
W000	- Shafts and Bearings
Z000	- Gears

This table (see Table 9.1) uses the **M000** database as an example to show how data is organized in KISSsoft. As shown there, the **F040NORM** and **M090MAT** tables belong to the group of shaft-hub connections.

KMAT
M000
W000
Z000
(a) Databases

9.1 table: Table: How data is organized in KISSsoft (1 of 2)

KMAT	
M000	**F040NORM**
	...
	M090MAT
W000	

Z000
(b) tables

9.2 table: Table: How data is organized in KISSsoft (2 of 2)

Up to now, the following tables have been created in the databases: Center distance tolerances, Reference profiles, Bore standard, Thread type bolt, Production process for hypoid bevel gears, Manufacturing process for bevel gears, V-belt standard, Spline standard, Chain type DIN 8154, Chain type DIN 8187, Chain type DIN 8188, Glue materials, Load spectra, Soldering materials, Key standard, Polygon standard, Woodruff key standard, Lubricants, Bolt type, Washer standard, Multi-spline standard, Rolling bearing, Materials for glued and soldered joint, Material, Tooth thickness tolerances, Toothed belt standard.

9.1 Viewing database entries

To open the database, select the **Extras** > **Database tool** menu option, as shown in (Figure 9.5, ①). A dialog window appears with the question whether you want to open the database with write authorization (②). If you click on **Yes**, you can edit the database entries, otherwise they are write protected. If you select **No**, the actual database tool window (③) opens, but in read-only mode. There, you can select a table from a list that is assigned to a particular database. The row of a table contains the values that set the parameters for the database entry. The columns contain the parameters for the database entries, i.e. values for the yield point of different materials. This section describes how to edit database entries. You can also display table entries by selecting a row in the database tool window and then confirming this by clicking **Display** (④). The **Display entry** window opens with a structured display of the value amount from a table row (⑤).

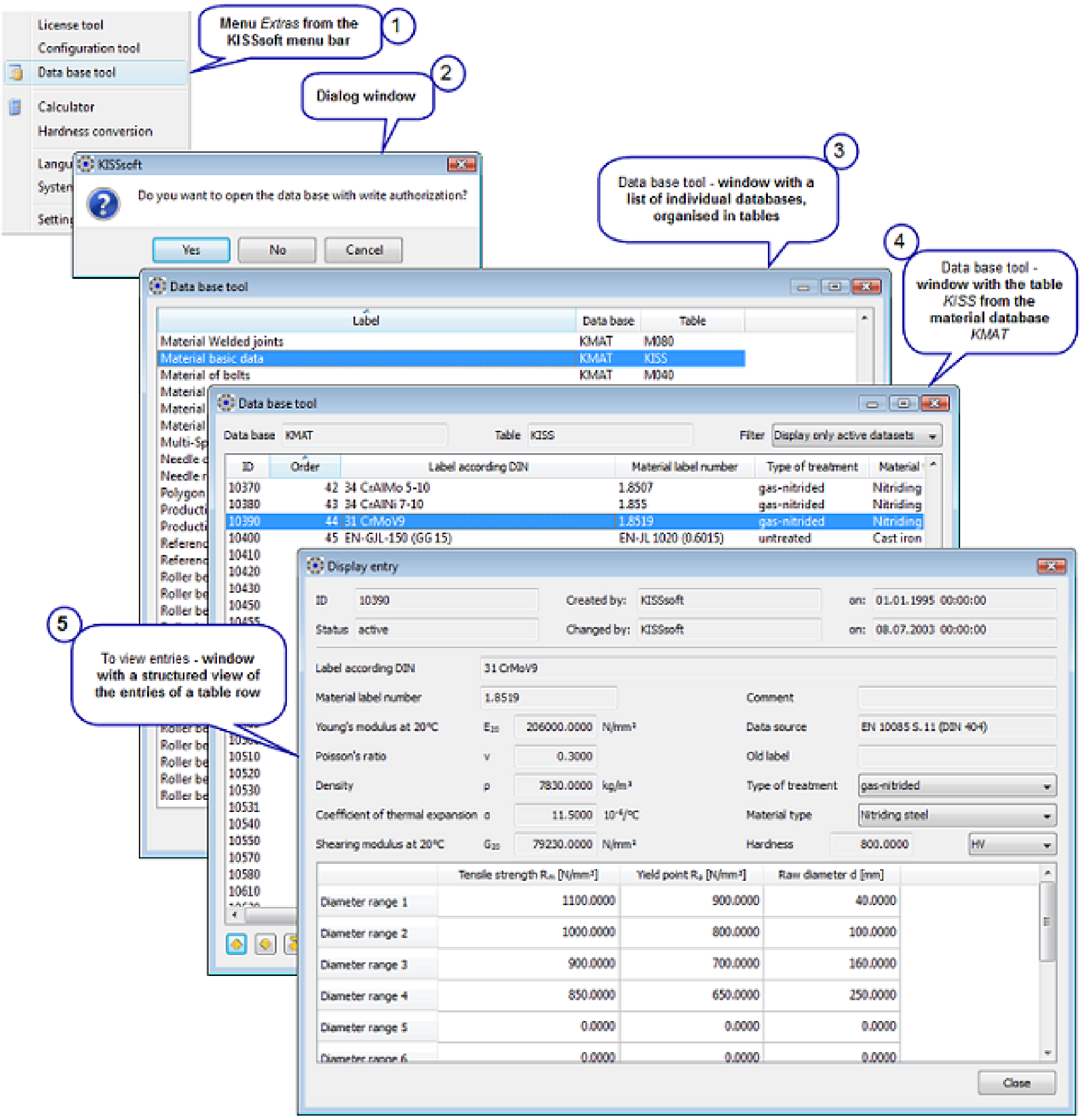


Figure 9.1: Accessing database entries

▶ **Note:**

With the KISSsoft database tool you can change the databases and expand them with your own entries. The data stored in the databases are in a sense "sensitive", so that incorrectly entered values can have consequences that are initially imperceptible, yet eventually far-reaching and serious. For this reason, when you open the database you are asked whether the access should have write authorization. If you answer this question with "No"", you can view the data in the tables but not change it.

If you want to make absolutely sure that the databases remain unchanged, you can write protect their corresponding files (*.**udb**). Any attempt to open a table with write authorization results in an error message and the table will normally be opened in write protected mode. To change a file's write protection attribute, right-click on the file in Windows® Explorer, and then click on **Properties**. Click in the **Properties** dialog field, on the **General** tab, and then click the **Write-protected** checkbox. If you want to make changes to a write protected file, first either deselect the **Write-protected** checkbox or save the file with a different name.

9.2 Managing database entries

If you want to change one of your own entries in a table in the database, you must work in write authorization mode. To do so, click **Yes** in the 5ba0bcdba6a56 dialog window. In the list that you see next, (③) select the required table by double-clicking on the appropriate row or single-clicking on the **Edit**button at the bottom right of the window, after you have selected the row. The database tool window now shows a list of the table entries (④) and a row of new buttons is displayed on the bottom left in the window:

≡↑	Moves the selected item up one row
≡↓	Moves the selected item down one row
≡⤒	Moves the selected item to the start of the list
≡⤓	Moves the selected item to the end of the list
≡+	Adds a new item to the list
≡−	Moves the selected item into the list of hidden datasets.

Select the **Filter** drop-down menu option on the top right of the window to choose between displaying active datasets, hidden datasets, or both. Active datasets can be used within the calculation modules, hidden ones cannot.

9.2.1 Creating a database entry

If you click on the ≡+button without having selected a row, the **Display entry** window (⑤) opens and the input fields in it are empty. Only the **Name** field contains the entry **_NEW**, which normally identifies the new table entry. After you have transferred the necessary data, confirm your entries by clicking on **OK** and then **Save**in the database tool window. The new entry is assigned an identification number (**ID**) ≥ 20000 and is then transferred into the list of active datasets. Click the **Edit** button to change entries with an ID of ≥ 20000.

Click the ≡+button after having selected a row, the **Display entry** window opens and contains predefined values in the input fields according to the table entry. The suffix **_NEW** will automatically be attached to the name, in order to differentiate it from the original dataset. In all remaining steps, you then proceed as described above.

Example: Creating a database entry
Let's assume you want to add a new spring material to the **KMAT.F000** table. As described in the procedure, you would select the **F000** table from the **KMAT** database. Click on the ≡+button in it, to

add a new entry/new row to the table, and then transfer the new data into the input fields in the **Display entry** window. However, as only a few parameters can be freely selected there, the next question is, where can the other values such as the yield point and Young's modulus be changed? The reply is: in the base material input fields, i.e. in table **KMAT.KISS**. You must always specify a base material before you can introduce a new spring material. If this is not present, you must first define it in the **KMAT.KISS** table and then make the missing entries in **KMAT.F000**.

► **Note**

All material-specific tables such as **KMAT.F000** or **KMAT.Z080** - with the exception of **KMAT.KLUB** - have a Check button beside the Base material drop-down menu. If you have marked the checkbox, you have the option of selecting an alternative base material in the associated drop-down list-menu. If the checkbox is empty, access to the menu of the base materials is blocked. This option helps prevent unwanted changes when the base material is being assigned.

9.2.2 Deleting a database entry

Datasets in KISSsoft will never be deleted. It is only possible to move entries with an ID ≥ 20000 into the table in which hidden datasets are stored. Select the corresponding entry by single clicking on it in the window ④and then click on the ≡−button. The selected row will be copied into the range that contains the hidden datasets and is removed from the list of active datasets. To access the table of inactive datasets, select the **Display only hidden datasets** option in the **Filter** drop-down list menu in the top right of the database tool window.

9.2.3 Restoring a database entry

In the hidden datasets table, select the appropriate row with a single mouse click and then click on the ≡+button. The entry will be copied into the table of active datasets and then removed from the inactive datasets range.

9.3 Import and export data from the database tool

The datasets from every table in the database tool can be exported to a file or imported from a file. These datasets can be imported or exported as a single dataset or imported from a list of datasets.

To import a list of datasets, first save it in a file, preferably an Excel spreadsheet with the extension ".csv". The entries in the spreadsheet columns should match the database table columns.

The software can also interpret lists that are saved as text files with the ".txt" extension. List entries should be separated by a "comma" or a "semicolon". Which separator is used depends on your operating system configuration.

- Use the **Dataset** button to import datasets from a file with the extension ".kds", or export them from one.

- Use the **List** button to import datasets from a file with the extension ".csv", or export them from one.

► **Important notes:**

1. Only "user-defined datasets" (ID >= 20000) can be exported or imported.
2. An existing "user-defined dataset" can be overwritten if you are processing individual datasets.
3. The names of the columns in the ".kds" files is case-sensitive and must exactly match the names in the database tool. You could export a dataset to verify the column names.
4. A new ID will automatically be assigned to every dataset if an entire list is imported or exported.

9.4 External tables

KISSsoft uses external tables, also called look-up tables, to handle larger data volumes. One or more output values from these external tables are assigned to one or more input values (see Figure 9.2).

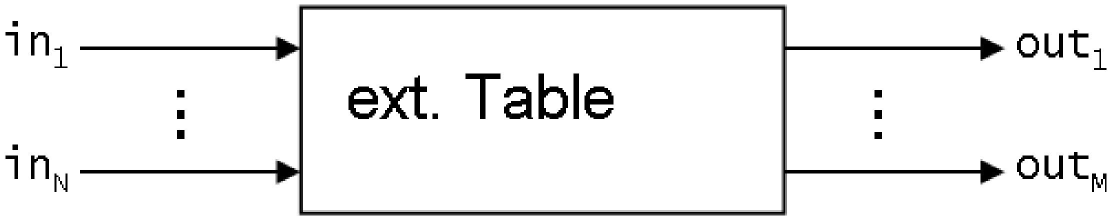


Figure 9.2: Principle of functionality of external tables

The output data that is assigned to the input data are contained in the table.

The external tables are stored in the /**KISSsoft installation directory/dat**. If a new table name is entered in a database, a file with the same name and the file extension **.dat** must also be created manually.

Because tables are located externally, KISSsoft can only determine how many of them there are during program execution. The user directly benefits from the fact that they can generate their own files with data tables, in a similar way to the files supplied by KISSsoft. The tables are readable ASCII files and can therefore be edited and expanded by the user. It would, for example, be possible to use an internal standard as an alternative to the ISO base tolerances.

Figure (see Figure 9.3) shows the three table types used by KISSsoft in one diagram:

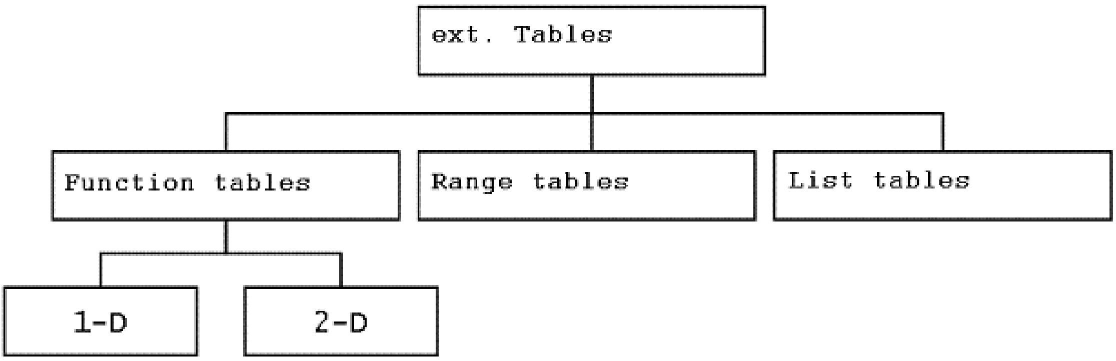


Figure 9.3: Types of external tables

A table always has the following structure, no matter what type it is:

:TABLE type variable or label
table header
DATA
data
END

Use the **:TABLE** command to mark the external table as an external table. You must use one of the following designations for the **Type** argument:

FUNCTION	Functions tables
RANGE	Range tables
LIST	List tables

► **Note**

You can mark blanks in tables with *****, **-** or blank spaces. Note here that no space characters can be used if they are followed by more values. KISSsoft interprets blank space as value separators.

The structure of the table header and the body data, which is dependent on the type, is described with example applications in the following sections.

9.4.1 Functions tables

Functions tables are tables that expect one or two input values (1D or 2D table) and which return exactly one corresponding value.

► **Example 1D table**

The angle coefficient (**factor**) is determined on the basis of a specified (**angle**). For example: if the input value **angle = 45** supplies an output value of **factor = 0.35**.

-- table type: Functions table; output variable: **factor**					
:**TABLE FUNCTION factor**					
-- INPUT X angle defines the input parameter angle;					
-- interim values will be interpolated linearly					
	INPUT X angle TREAT LINEAR				
-- Data content: 1st line: input values, 2nd line: output values					
DATA					
	0	30	60	90	. . .
	0.1	0.25	.45	.078	. . .
END					

INPUT is a key word, i.e. a word that is reserved by the Table Interpreter, and is followed by an argument **X**, which assigns a dimension to the **angle** input parameter. The key word **TREAT**, with associated **LINEAR** argument, specifies that interim values are to be interpolated linearly. The output value **factor** will determined using the value of the **angle** variable. The first row of data content in the 1D table (between **DATA** and **END**) corresponds to the input value **angle**, and the second row corresponds to the output value. The data content in a 1D table is therefore always a (2 × n) matrix, i.e. both rows must contain the same number of values.

► **Example of a 2D table**

The nominal power is defined on the basis of the speed and the sheave diameter. For example: if the input values **diameter = 60** and **speed = 60** supply an output value **power = 8.6**.

-- table type: Functions table; output variable: **power**						
:**TABLE FUNCTION power**						
-- INPUT X diameter defines the input parameter **diameter**;						
-- INPUT Y speed defines the input parameter **speed**;						
-- interim values will be interpolated linearly in both dimensions						
	INPUT X angle TREAT LINEAR					
	INPUT Y Speed TREAT LINEAR					
-- Data content: (see chapter 10.3, Example: Interference fit calculation)						
DATA						
		50	100	200	300	. . .
	50	4	7	12	25	. . .

	75	12	25	30	35	. . .
	. . .	. . .	. . .	. . .	. . .	. . .
END						

Here, the variable **power** is defined with the input variables **INPUT X** and **INPUT Y**. Interim values running down the columns (Y) should be interpolated linearly. The same applies across the rows (X). The first row in the table corresponds to the values of the **INPUT X** entry variables. The first column corresponds to the values of the **INPUT Y** entry variables. The values placed at the points where the entry values intersect are values which correspond to the output variables (see Figure 9.4).

	X_1	X_2	X_3
Y_1			
Y_2	$f(X_1, Y_2)$		
Y_3			

Figure 9.4: Data schema of 2D tables

If would be possible to use this method to define an inverse table. Assuming that, in your **XY** belt catalog, the table displaying power output shows the speed in the first row, and the diameter in the first column, then there is no need for you to turn your table upside down. Instead, simply change the assignment in the table header (i.e. replace **X** with **Y**).

9.4.2 Range tables

Range tables check whether a given value is moving within a defined range.

► **Example**

-- table type: range table; Name of the table: 'A'	
:**TABLE RANGE 'A'r**	
-- INPUT X drehzahl defines the **drehzahl** (speed) input parameter.	
-- interim values will be interpolated logarithmically.	
-- INPUT Y leistung defines the **leistung** (power) input parameter.	
	INPUT X drehzahl I TREAT LOG
	INPUT Y leistung

-- Data content: 1st line: INPUT X, 2nd line: INPUT Y upper limit						
-- 3rd line: INPUT Y lower limit						
DATA						
		200	300	500	1000	4000
	LOWER	1.5	2.0	3.0	10	20
	UPPER	10	15	20	15	40
END						

The two input variables are **drehzahl** (speed) and **leistung** (power). The output value represents the decision about whether the power in dependency with the speed is moving within a defined range and does not have to be declared. Interim values of the speed will be interpolated logarithmically. The first row of the body data corresponds to values of the **drehzahl** (speed) variable. The other rows correspond to values of the **leistung** (power) variable with **LOWER** as the lower, and **UPPER** as the upper, limit. The input value of **leistung** (power) is compared with these limits and a report sent to the program stating whether the **leistung** is located below, within, or above, the given range **A**.

9.4.3 List tables

Several output values are defined in list tables that contain at least one input value. The sequence of the input values is important if more than one input value is entered. The reading direction goes from left to right and the first input value defines the range of the next input value, which in turn defines that of the next one, etc. up to the last. All input values apart from the last one must match the entries in the body data (**TREAT DIRECT**, (see chapter 9.4.4, List of key words used)).

► **Example 1**

If the following three input values are assumed:
g.d = 2.0; g.P = 0.8; s.l = 6
The output values would be in accordance with the code given below:
s.l = 7; s.k = 3; s.k = 4.5.

-- Table type: list table. Output variable: **s.norm**	
:TABLE LIST s.norm	
-- INPUT g.d defines the input parameter **g.d**;	
-- INPUT g.P defines the input parameter **g.P**;	
	INPUT g.d
	INPUT g.P
-- IN_OUT s.l defines s.l as phase variable	

-- TREAT NEXT_BIGGER specified how interim values are handled					
	IN_OUT s.l			TREAT NEXT_BIGGER	
-- OUTPUT s.k, s.dk declares s.k and s.dk as output variables					
	OUTPUT s.k,s.dk				
-- Data content: A (N × Nin) matrix					
DATA					
	2.0	0.4	0		
	2.0	0.8	5	3	4.5
	2.0	0.8	7	3	4.5 - relevant data row
	2.0	0.8	10	3	4.8
END					

In contrast to functions tables, **s.norm** in the first row of the code specifies the name of the external table, and not the output variable. **IN_OUT s.l** declares a variable **s.l**, which is used both as an input and output variable (phase variable). **TREAT** functions again as a key word for processing the interim values: **NEXT_BIGGER** shows that input values are to be evaluated it they are not present in the appropriate column in the body data. In the example, the input value **s.l = 6** lies between the values 5 and 7 and, in accordance with **NEXT_BIGGER**, will be promoted to the next bigger value. **OUTPUT s.k, s.dk** declares not only **s.l.** but also the output values **s.k** and **s.dk**. The number of the columns in the body data must at least correspond to the number of input variables and, at most, correspond to the number of input variables + output variables, in this case: $3 < N_{in} > 5$.

► **Example 2**

Two input values are used to determine the different measurements for a bolt: the bolt type, here represented by the **typ** variable, and the bolt length, specified by **l**.

:TABLE LIST schraube.geometrie (meaning "bolt.geometry")								
	INPUT typ							
	INPUT l				**TREAT NEXT_SMALLER**			
	OUTPUT M, dw, (s), e, bez, vorrat							
DATA								
	. . .							
	12x2.5	20	12	14.57	23.78	5.75	ID 1	1
	12x2.5	25	12	15.78	24.88	5.75	ID 2	1
	. . .							

END

This table is called **schrauben.geometrie** (meaning "bolts.geometry"). The sequence in the table header defines the sequence within the columns. The first column therefore corresponds to the **typ** variable, the second to the **l** variable, etc. The **typ** and **l** variables are used as inputs, where the value for the **typ** variable must be present in the list. If an interim value is given for the **l** variable, the row with the next smaller value will be interpreted as the result. Blanks are not permitted, i.e. in this type table values must always be present. It may happen that individual variables are shown in brackets in the output definition. This has the effect that the appropriate column is ignored, i.e. this variable will not be specified.

► **Note**

Commented-out output definitions cannot be changed by the user.

9.4.4 List of key words used

--	The Interpreter ignores everything in a row that follows this comment character.
DATA	The data matrix is below this.
END	Ends the input area of the external table.
INPUT [<*dim*>**]** <*var*>	Input variable, with definition of the dimension if required.
IN_OUT <*var 1*>**[,** <*var 2*>**, ...]**	List tables: Phase variables
LOWER	Range tables: Lower limiting value.
OUTPUT <*var 1*>**[,** <*var 2*>**, ...]**	Output value(s)
:TABLE <*type*>	Defines the type of the external table.
TREAT DIRECT	Interim values: none permitted. The values input in the appropriate column/row must match those of the body data.
TREAT NEXT_SMALLER	Interim values: The next smallest value is assigned.
TREAT NEXT_BIGGER	Interim values: The next highest value is assigned.
TREAT LINEAR	Interim values: Linear interpolation.
TREAT LOG	Interim values: Logarithmic interpolation.
UPPER	Range tables: Upper limiting value.

9.5 Description of database tables

The individual database tables have very different structures. The next section describes these database tables and their specific fields.

The **Label** field is displayed in every table, and is only described here. You must enter a unique name for the dataset in this field. This name is then used to select the datasets in the program.

Note: Fields in which file names are to be entered have an auto-fill function. To perform this, the software searches in the ..\dat and ..\ext\dat folders, and also in the current project directory.

9.5.1 Center distance tolerances

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). The tables used for center distance tolerances begin with K10-???.dat. The center distance tolerances specified in ISO 286 are imported directly from the program code and not from a file.

9.5.2 Machining allowance for cylindrical gear

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). The tables for the cylindrical gear machining allowance begin with ZADDT-???.dat.

9.5.3 Reference profiles

You enter reference profile data directly in the database. However, each individual value depends on the other.

- **Description in accordance with ISO, the standard on which this is based**
- **Comment: Text field for your own use**
- **Data source: Text field for your own use**
- **Definable reference profile data: Dedendum coefficient h^*_{fP}, root radius coefficient ϱ^*_{fP}, addendum coefficient h^*_{aP}, tip radius coefficient ϱ^*_{aP}, topping, protuberance height coefficient h^*_{prP}, protuberance angle α_{prP}, tip form height coefficient h^*_{FaP}, ramp angle α_{KP}**

9.5.4 Compression springs standard

You can store data from geometry standards for compression springs.

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). The tables for compression spring standards begin with f010-??.dat.
- **Tolerance:** tolerance data for the geometry standard

9.5.5 Hobbing cutter selection

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). The table for cutter data according to DIN 3972 is called Z000-BP.dat.

9.5.6 Basic material Glued and Soldered joints

- **Tensile strength R_m:** [N/mm2] Data about the material's tensile strength is required to calculate glued and soldered joints.

9.5.7 Manufacturing process for bevel and hypoid gears

These values are only necessary for calculations using the Klingelnberg method. They correspond to tables for machine types that use the Klingelnberg in-house standard.

- Values that must be defined: **machine type, cutter tip cutter radius r_0**[mm]**, No. of blade groups cutter z_0, maximum machining distance MD_{max}**[mm]**, minimum normal module $m_{n,min}$**[mm]**, maximum normal module $m_{n,max}$**[mm]

9.5.8 V-belt standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for the V-belt standard begin with Z090-???.dat.
- **Calculation method:**
    - 1) Narrow V-belts (Fenner)
    - 2) Narrow V-belts/force belts
    - 3) Conti belts
- More definitions: **Maximum belt speed v_{max}**[m/s]**, elasticity E:** [N]**, weight per length q:** [kg/m]**, coefficient of friction μ_r**

9.5.9 Spline Standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for spline standard norms begin with M02C-???.dat.
- **Calculation method:** the appropriate calculation method is selected for each spline.

9.5.10 Chain profiles ISO 606

- Values to be defined for this table: **type, pitch p:** [mm]**, number of strands n_s, maximum roller diameter d_1**[mm]**, maximum bolt diameter d_2**[mm]**, minimum width between inner plates b_1**[mm]**, maximum width over inner link b_2**[mm]**, total width b_{tot}**[mm]**, maximum inner plates depth h_2**[mm]**, ratio t_H/t_S**

9.5.11 Adhesives

- **Comment:** Text field for your own use.
- Definable values: **Minimum** and **Maximum shear strength** $\tau_{B,min}$**,** $\tau_{B,max}$ **[N/mm**2].

9.5.12 Modifications

The different modifications applied to gears are defined as database classes. If a dataset is hidden in the database, it will no longer appear in the modifications selection list.

Although you can add new datasets to the database, these will not be visible in the calculation module.

9.5.13 Load spectra

All entries (frequency, power, speed) can be defined either as coefficients or as values. If coefficients are used to input these values, the power and speeds are entered as coefficients of the nominal power. In the calculations, the factor for torque (load factor/speed factor) is used for forces and torques. You cannot change the reference gear, torque, speed or power if the input is set to values. These input fields are inactive.

You can either import load spectra from a file or enter them directly. If you input this data directly, the number of load cases is defined by the number of lines you enter.

- **Input:** Specify whether the factors are for power or torque. This also applies if the load spectrum is imported from a file.
- **Link with file:** This option is displayed if the load spectrum is set to "Own input". If the option is displayed, load spectrum values can be imported from a selected file. The import can be performed in two ways: If the flag is not set, the imported load spectrum values can be modified. If the flag is set, the load spectrum values are automatically overwritten by the values in the selected file, and cannot be modified.
- **Own input of load spectra:** You can input the load spectrum directly, or import it from a file.
- **File name:** Click the button to select a file via the directories. The file containing the load spectrum must be a text file (.dat). You will find a sample load spectrum file called

"Example_DutyCycle.dat" in the "dat" directory. You should store load spectra you define yourself in the "EXT/dat" directory, to ensure they are always available even after a version upgrade.

```
0.1  1    1
0.1  0.9  1
0.1  0.8  1
0.1  0.7  1
0.1  0.6  1
0.1  0.5  1
0.1  0.4  1
0.1  0.3  1
0.1  0.2  1
0.1  0.1  1
```

Example of a file used to input a load spectrum

- **Frequency:** H_0 ... H_{19}, the sum of these frequencies must be 1.
- **Load factor (torque factor):** P_0 ... P_{19} $0<P_n<,\infty$.
- **Speed factor:** N_0 ... N_{19}, $0<N_n<\infty$.

Load bins with negative torque are interpreted as a change from driving to driven.

9.5.14 Solders

- Definable values: **Minimum** and **Maximum shearing strength** $\tau_{B,min}$, $\tau_{B,max}$ [N/mm^2].

9.5.15 Surface roughness of shafts and shaft-hub connections

You can also input your own data for shaft-hub connections in the "Own Input" field. The Rz values for the shafts are taken from the list.

- **Comment:** Text field for your own use.
- Definable values: **Mean peak-to-valley roughness** R_z [μm] and **arithmetic mean roughness value** R_a[μm].

9.5.16 Key standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for key standards begin with M02A-???.dat.

9.5.17 Polygon standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for polygon standards begin with M02D-???.dat.

9.5.18 Woodruff key standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for Woodruff key standards begin with M02E-???.dat.

9.5.19 Bolts/pins

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for bolt/pin standards begin with M03A-???.dat.

9.5.20 Lubricants

- **Comment, description as specified in ISO, data source:** Text fields for your own use
- **Additive for rolling bearings:**
    - **Without additives:** Lubricants without additives, or with those additives whose effectiveness in rolling bearings has not been tested.
    - **With additives:** Lubricants whose effectiveness has been tested in rolling bearings
- **Oil/Grease:** specify whether the lubricant is an oil or a grease.
- **Kinematic viscosity at 40°C and at 100°C** ν_{40}**,**ν_{100}**:** [mm2/s]
- **Lubricant base:** Selection options:
    - Mineral oil
    - Polyglycol-based synthetic oil
    - Polyether-based synthetic oil
    - Polyalphaolefin-based synthetic oil
    - Ester-based synthetic oil

Polyalphaolefin: similar to mineral oil, easily mixable with mineral oil, some approved for use with foodstuffs.

Esters: some approved for use with foodstuffs, some biodegradable.

- **Test procedure scuffing:** Selection options:
    - No information about scuffing
    - FZG Test A/8.3/90; ISO 14635-1 (normal)

        - FZG Test A/16.6/90
        - FZG Test A/16.6/120
        - FZG Test A/16.6/140
        - FZG Test A10/16.6R/120; ISO 14635-2
        - Input of scuffing temperature
        - FZG test A10/16.6R/90
        - FZG test S-A10/16.6R/90
- **Failure load stage scuffing FZG test:** Input load stage scuffing as specified in the FZG test. These values are required for gear calculations.

1= weakest stage; 12=best stage

Good gear lubricants all have a load stage scuffing value of 12.

- **Scuffing temperature** θ_S**:** You can also enter the scuffing temperature for the scuffing test procedure.
- **Test procedure micropitting:** Selection options
    - No information about micropitting available
    - C-GF/8.3/90/ with Ra=0.50 θ=90° (FZG)
    - C-GF/8.3/60/ with Ra=0.50 θ=60° (FZG)
    - C-GF at current oil temperature
- **Load stage micropitting test:** The best achievable load stage is 10.
- **Density** ϱ**:** [kg/dm^3]
- **Cone penetration at 25°C (grease) P_e:** [0.1mm] This value is only required to calculate grease-lubricated plain bearings.
- **Soap proportion (grease) c_s:** [Vol%] This value is only required to calculate grease-lubricated plain bearings.
- **k coefficient, s coefficient (compression viscosity) k, s:** Coefficient used to calculate compression viscosity (AGMA 925):

$$\alpha = k \cdot \eta_M^S$$

$$\eta_P = \eta_{atm} \cdot e^{\alpha_p}$$

If you do not know these values, you can input 0. The values are then taken from the standard (AGMA 925-A03, Table 2).

- **Lower/upper service temperature limit** θ_{min}**,** θ_{max} **:** [°C]

9.5.21 Bolts: Tightening factor

- **Tightening technique:** Select "not yield point- or rotation-angle-controlled tightening" and "yield point- or rotation-angle-controlled tightening"
- **Minimum tightening factor α_{Amin}:** Minimum value, can be defined per tightening technique.
- **Medium tightening factorα_{Amid}:** Medium value, can be defined per tightening technique.
- **Maximum tightening factor α_{Amax}:** Maximum value, can be defined per tightening technique.

9.5.22 Bolts: Bore

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for bores begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be read in mm or inches.

9.5.23 Bolts: Strength grade

- **Comment:** Text field for your own use.
- Definable values: **Yield point R_p** [N/mm2], **tensile strength R_m** [N/mm2], **shearing strength ratio I_b/R_m[-]**

► **Note:**

The yield point and tensile strength for the lower diameter limit for strength classes 8.8 and SAE J429 Grades 2 and 5 are always displayed in the database. If the diameter is greater than the diameter limit, this is corrected in the program.

The shearing strength ratios have been taken from Table 5.5/2 in VDI 2230 (2015) according to the strength classes. Undefined SAE classes are set to the ratio 0.6.

9.5.24 Bolts: Nut strength grade

- **Comment:** Text field for your own use.
- Definable values: **Vickers hardness for standard thread for diameter D <= 16 mm** [HV], **Vickers hardness for standard thread for diameter D > 16 mm** [HV],**Vickers hardness for fine thread for diameter D <= 16 mm** [HV], **Vickers hardness for fine thread for diameter D > 16 mm** [HV]

► **Note:**

The minimum value specified in the DIN EN ISO 898-2:2012 standard is assumed in the strength classes. However, as this value depends on the diameter, in some strength classes, both values are defined in the database. In the program, you can then specify which diameter the value is to be used for. The same applies to differentiating between fine and standard threads. The program handles the value that is being used differently, according to the selection in the interface.

9.5.25 Bolts: Coefficients of friction classes

- **Comment:** Text field for your own use.
- Definable values: **Minimum coefficient of friction** μ_{min}[-] and **Maximum coefficient of friction** μ_{max} [-]

▶

▶ **Note:**

The minimum and maximum coefficients of friction are always displayed in the database for the coefficient of friction classes. The values are used to size the moments of friction for bolts.

The predefined values are taken from Table A5 in VDI 2230, Sheet 1 [1].

9.5.26 Bolts: Thread type

- **Name:** Text field for your own use.
- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for threads begin with M04-???.dat.
- **Coefficient used to calculate the flank diameter/core diameter**
- **Flank angle** α**:** [°]

9.5.27 Bolts: Nuts

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for nuts begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be read in mm or inches.

9.5.28 Bolts: Type

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for bolt types begin with M04-???.dat.
- **Name:** Text field for your own use.
- **Screw thread type:** selection list to show which screw thread type this bolt has.
- **Unit in use**: select whether the values in the file are to be read in mm or inches.

9.5.29 Bolts: Washer

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for washers begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be read in mm or inches.

9.5.30 Selection of pinion type cutters

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for pinion type cutters begin with Z000-Cutter-?.dat.

9.5.31 Disc spring standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for disc springs begin with F040-?.dat.

9.5.32 Tolerances standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for tolerances begin with K10-???.dat.

The tolerances according to DIN EN ISO 286 have been programmed directly into KISSsoft. For fit (tolerance) classes H, h, JS and js, the tolerance has been extended up to the nominal length 10,000 mm (according to the standard, up to 3,150 mm). The values were determined by extrapolation.

9.5.33 Beam profiles

- **Drawing file:** image displayed on screen when a shaft is calculated.
- Values for profiles: **height h** [mm], **width b** [mm], **cross section A** [cm2], **moments of gyration of a surface relating to x-/ z-axis I_x/ I_z** [cm4], **moment of inertia in torsion I_t** [cm4], **moments of resistance relative to the x- or z-axis W_x/ W_z**[cm3], **moment of resistance in torsion W_t** [cm3]

9.5.34 Multi-Spline standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for multi-spline profiles begin with M02b-???.dat.

9.5.35 Materials

The materials consist of a database table **Basic data Materials** and the particular table for the modules. The Basic data table lists the general material data. As the materials can then be transferred to the individual module tables, you therefore only need to define the basis data once. Module-specific data is then defined in the module tables.

In module-specific tables, you must always select one **Base material**.

9.5.35.1 Basic data: Materials

- **Label according to DIN, BS, AISI, UNI, AFNOR, JIS, CN, Old label, Material number, Origin of data, Comment:** Text fields for your own use
- **Young's modulus at 20°C E_{20}:** [N/mm²]
- **Poisson's ratio ν:** [-]
- **Density ϱ:** [kg/dm³]
- **Coefficient of thermal expansion α:** [10-6/K]
- **Shearing modulus at 20°C G_{20}:** [N/mm²]
- **Type of treatment:** Select the type of treatment in this list.
- **Material type:** Select the material type in this list.
- **Hardness value:** This value is purely for information purposes and has a negligible effect on the calculation.
- **Unit of hardness:** You can select this in this list.
- **Core hardness value:** This value is used, for example, for the strength calculation according to AGMA 6101-F19/AGMA 6001-F19. However, for other calculations, the value is purely for information purposes and has only a minor influence on the calculation. For case-hardened case hardening steels, the value for the tensile strength Rm was taken from the DIN EN 10084 standard for a diameter between 16 and 40 mm, and was then calculated with the hardness conversion of the Brinell hardness HBW.
- **Core hardness unit:** You can select this from a list. The hardness values according to Brinell, Rockwell C and Vickers are possible as input values. Depending on the input, the required hardness values are converted into the hardness unit used.
- **Tensile strength R_m:** [N/mm2] A maximum of 10 different diameter ranges can be defined.
- **Yield point R_p:** [N/mm2] A maximum of 10 different diameter ranges can be defined.
- **Raw diameter d:** [mm] A maximum of 10 different diameter ranges can be defined.

9.5.35.2 Material Spring calculation

This table applies to Compression (F010), Tension (F020) and Leg springs (F030):

- **Admissible shear stress:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for springs begin with F01-???.dat. In this file, you can view or define the Admissible shear stress, the values for the Goodman diagram and the values for the relaxation diagram. If the curves of the relaxation diagram are only defined with 2 points, you must set the values for tau3 and rel3 to 0 so KISSsoft can recognize them.
- **Comment:** Text field for your own use.
- **Minimum** and **maximum wire diameter** d_{min}, d_{max} [mm]
- **Shearing modulus depending on temperature** α_G**:** [1/K]
- **Use:** selection list with the cold and thermo-formed variants

9.5.35.3 Material of plain bearings

- **Comment:** Text field for your own use.

9.5.35.4 Material of enveloping worm wheels

The table applies to worm wheels (Z080):

- **Comment:** Text field for your own use.
- **Material characteristics:** Selection list (such as CuSn-bronze/CuAl bronze/GJS40/GC25/PA-12)
- **Mineral oil coefficient** W_{MLOel}**:** Material/lubricant factor for mineral oil
- **Polyglycol coefficient (DIN)/ (ISO)** W_{MLGDIN}**/** W_{MLGISO}**:** Material/lubricant factor for polyglycol
- **Polyalphaolefin coefficient** W_{MLA}**:** Material/lubricant factor for polyalphaolefin
- **Material factor** Y_W**:** (see DIN 3996, Table 5)
- **Pitting strength** σ_{HlimT}**:** [N/mm2] (We recommend you use reduced values as specified in ISO 14521)
- **Shear fatigue strength** τ_{FlimT}**:** [N/mm2]
- **Reduced shear fatigue strength** $\tau_{FlimTred}$**:** [N/mm2] (If no slight deformation is permitted, you must include reduced strength values in the calculation.)

9.5.35.5 Material Interference fit

- **Comment:** Text field for your own use.

9.5.35.6 Material of bolts

The table applies to the bolt module (M040):

- **Comment:** Text field for your own use.
- **Permissible pressure p_G:** [N/mm2] (Data should be entered as specified in VDI 2230)
- **Shearing strength τ_B:** [N/mm2]

9.5.35.7 Material for disc spring calculation

The table applies to disc springs (F040):

- **Comment:** Text field for your own use.
- **Goodman diagram:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for the Goodman diagram begin with F04-???.dat.
- **Young's modulus depending on temperature α_E:** [1/K]

9.5.35.8 Material of shaft-hub-connection

- **Comment:** Text field for your own use.

9.5.35.9 Housing material

- **Comment:** Text field for your own use.

9.5.35.10 Material Shaft calculation

The table applies to shafts (w010):

- Values for calculating strength according to Hänchen:
  - **Bending fatigue limit σ_{bW}:** [N/mm2]
- Values for strength calculation according to DIN 743:
  - **Reference diameter d_b** [mm], **Tensile strength R_m** [N/mm2], **Yield point R_p** [N/mm2], **Bending fatigue limit** σ_{bW} [N/mm2], **Tension/compression fatigue limit** σ_{zdW} [N/mm2], **Torsion fatigue limit** τ_{tW} [N/mm2]
  - **Experimental data:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for the experimental Haigh diagram begin with W01-???.dat.
  - **CrNiMo case hardening steel:** When determining the size factor K1 for case hardening steel, you need to specify whether the steel is CrNiMo

case hardening steel. The materials 20NiCrMo13-4, 17NiCrMo6-4, 17NiCrMoS6-4, 18NiCr5-4, 17CrNi6-6, 18CrNiMo7-6, 14NiCrMo13-4, 22CrMoS3-5 and 15NiCr13 belong to this group. According to DIN 10084, there is no significant drop in the tensile strength of these materials. These materials also belong to this group according to Prof. Linke.

- Values for strength calculation according to FKM:
  - **Tensile strength for reference diameter** $R_{m,N}$ [N/mm2], **Yield point for reference diameter** $R_{e,N}$ [N/mm2], **Effective reference diameter for** $R_{p,N}$ $d_{eff,N,p}$ [mm], **Effective reference diameter for** $R_{m,N}$ $d_{eff,N,m}$ [mm], **Effective reference diameter for A** $d_{eff,N,a}$ [mm], **Constant for the calculation of** $K_{d,p}$ **(flow)** $a_{d,p}$, **Constant for the calculation of** $K_{d,m}$ **(fracture)** $a_{d,m}$, **Constant for the calculation of** $K_{d,a}$ **(elongation at break)** $a_{d,A}$
  - **Elongation at break A:** [%]
  - **FKM Group:** Selection list showing the material group to which the entry belongs.

9.5.36 Material of gears

- **Comment:** Text field for your own use.
- **File for hardness curve:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for the hardness curve begin with Z22-???.dat. The material's measured hardness value, to be displayed as a graphic in module Z22. Does not influence the calculation. Here is an example of how to create this type of hardness curve in an external table.

```
-- ------------------------------------------------------------------------
-- File   = ~modname: z22-100.dat~
-- Verlauf der Haerte des Werkstoffs/Curve of the material hardness
-- Werkstoff/Material: GGG50 180HB
--
-- (c) KISSsoft AG, CH-8608 Bubikon
--
-- 1.0     14.10.1995  em        Inital Version
--
-- INPUT  nummer      Punktnummer/ number of the point
-- OUTPUT Tiefe, HaerteHV Tiefe(mm), HV-Wert/Depth(mm), HV-Value
-- ------------------------------------------------------------------------

:TABLE LIST haerteHV
  INPUT nummer
  OUTPUT Tiefe, HaerteHV
DATA
1    0.0   890
2    0.1   523
3    0.2   470
4    0.3   295
5    0.4   290
6    0.5   207
7    0.6   207
8    0.7   207
9    1.0   207
10   1.5   207
END
```

Figure 9.5: Example of a hardness curve definition (Z22-100.dat)

- **Woehler line file:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for S-N curves (Woehler lines) begin with .dat.
  You **must input a file name for plastics** here. The file contains the material data (Woehler lines, Young's modulus, etc.) used in the calculation.
  For metallic materials you can also input a file name here. The file contains the S-N curves (Woehler lines) for bending strength and for Hertzian pressure that are used in the calculation, if the **Calculate with own Woehler line** flag is set.

```
-- Data for significant no. of cycles (edge points for interpolation of Woehler line)
-- Important: This list must contain ALL cycle values of FlankSigH and FootSigF in numerical order!
:TABLE FUNCTION EdgeCycle
      INPUT X number TREAT NEXT_BIGGER
DATA
      1      2      3      4      5      6      7
    1E3    1E5    2E6    3E6    5E7   1E10   1E99
END

-- Data for Hertzian pressure sigH
:TABLE FUNCTION FlankSigH
      INPUT X Cycles TREAT LOG
DATA
      0    1E3    1E5   3E6    5E7   1E10   1E99
   2700   2700   1920   1485   1200   1104   1104
END


-- Data for fatigue strength tooth root sigF
:TABLE FUNCTION FootSigF
      INPUT X Cycles TREAT LOG
DATA
      0    1E3    1E5    2E6    1E10   1E99
   1080   1080    423    230    195    195
END
```

Figure 9.6: Example of a file with S-N curves (Woehler lines) for a metallic material

- **Endurance limit root (ISO, DIN/ AGMA 2101)** σ_{Flim}**/**s_{at}**, Endurance limit flank (ISO, DIN AGMA 2101)** σ_{Hlim}**/**s_{ac}**:** [N/mm2] Endurance limit values specified in DIN 3990 or ISO 6336 Part 5.
- **Endurance limit root (AGMA 2001)** s_{at}**, Endurance limit flank** s_{ac} **(AGMA 2001):** [lbf/in2] Strength calculation based on AGMA 2001.
- **Average roughness height root/flank** R_{zF}**/** R_{zH}**:** [µm]
- **Thermal contact coefficient** B_M**:** [N/mm/s0.5/K] This coefficient is needed to calculate the flash factor. You will find more information about this in DIN 3990, Part 4, Equations 3.11, 4.17, 4.18 and 4.19. For the most commonly used materials, it is 13.795.

9.5.37 Rolling bearings

Rolling bearing tables are sub-divided into three different tabs:

- Basic data tab
- Additional data tab
- Internal geometry tab

9.5.37.1 Rolling bearing basic data

- **Bearing label:** The codes for the bearing series are as specified in DIN 623 Part 1.
- **Comment:** more detailed description of the bearing (e.g. SKF-Explorer, X-Life, etc.)

- Main dimensions of the bearing: **Inside diameter d** [mm], **External diameter D** [mm], **Bearing width b** [mm], **Corner radius** r_{smin} [mm]
- **Basic dynamic load rating C:** [kN], Dynamic radial load rating C_r: [kN] (if radial and axial values are predefined, e.g. cross roller bearing)
- **Basic static Load rating C_0:** [kN], Static radial load rating C_{0r}: [kN] (if radial and axial values are predefined, e.g. cross roller bearing)
- Dynamic axial load rating C_a: [kN] (if present, e.g. cross roller bearing)
- Static axial load rating C_{0a}: [kN] (if present, e.g. cross roller bearing)
- **Hybrid bearing:** [Yes/No] Yes: if a hybrid bearing (ceramic rolling body); No: if not a hybrid bearing
- Coefficients **X1, Y1, X2, Y2, e, e0, X01, Y01, X02, Y02**

	Defining individual coefficients:
X1,Y1,e:	Coefficients in formula *P = X1*Fr + Y1*Fa* for *Fa/Fr <= e*
X2,Y2:	Coefficients in formula *P = X2*Fr + Y2*Fa* for *Fa/Fr > e*
X01,Y01:	According to ISO: Coefficients in formula *P0 = max(Fr; X0*Fr + Y0*Fa):* X0 is used for X01,Y0 and Y01. According to Schaeffler*:* Coefficients in formula *P0 = X01*Fr + Y01*Fa* for *Fa/Fr <=e0*
X02,Y02, e0:	According to ISO: e0 and X02 are set to 0 (not present). According to Schaeffler: Coefficients in formula *P0 = X02*Fr + Y02*Fa* for *Fa/Fr > e0*
X1,Y1,X2,Y2,e:	For some bearings, these values are not imported from the database. Instead they are imported from the files, depending on the axial force.

Determination of e0 according to Schaeffler: *P0 = X0*Fr + Y0*Fa* -> with *P0 = Fr* at limit -> *Fr = X0*Fr + Y0*Fa* -> **e0 = Fa/Fr = (1-X0)/Y0**

Ball bearing: depending on *f0*Fa/C0*

- at bearing clearance C0: Data is imported from file W05-100A.dat (single row bearing) and W05-100B.dat (double row bearing)
- at bearing clearance C3: Data is imported from file W05-101A.dat (single row bearing) and W05-101B.dat (double row bearing)
- at bearing clearance C4: Data is imported from file W05-102A.dat (single row bearing) and W05-102B.dat (double row bearing)
- with no clearance: Data is imported from file W05-104.dat

Angular contact ball bearing: depending on *f0*Fa/C0* and contact angle

- at contact angle 5°: Data is imported from file W05-100A.dat (single row bearing) and W05-100B.dat (double row bearing)
- at contact angle 10°: Data is imported from file W05-101A.dat (single row bearing) and W05-101B.dat (double row bearing)
- at contact angle 15°: Data is imported from file W05-102A.dat (single row bearing) and W05-102B.dat (double row bearing)
- at contact angle 20°< α< 45°: Data is imported from file W05-103A.dat (single row bearing) and W05-103B.dat (double row bearing)

Note:

Bearing clearance class C2 is handled in the same way as C0. These formulae have been taken from the "Die Wälzlagerpraxis" document [2].

- **Speed limit when grease lubrication is n_{Gmax}:** [rpm] Only one speed limit is specified for SKF bearings. There is no distinction between grease and oil lubrication.
- **Speed limit when oil lubrication is n_{Omax}:** [rpm] Only one speed limit is specified for SKF bearings. There is no distinction between grease and oil lubrication.
- **Weight m:** [kg]
- **Contact angle α_0:** [°] Input the contact angle for shaft bearings, tapered bearings, etc.; for four-point contact bearings: If you input 0°, this is set to 35°, for axial spherical roller bearings: If you input 0° this is set to 50°.
- **Permitted axial force F^*_{azul}:** [-] Input the permitted axial force in % of Fr. The permitted axial force is not checked if you input 0.
- **Maximum permitted misalignment α:** [min] If you input 0, the angle adjustability (i.e. a comparison of the permitted angular deviation of the shaft with the effective angular deviation in the bearing) is not checked.
- **Thermal reference speed $n_{\theta r}$**: [rpm]
- Currently not evaluated in KISSsoft: **Availability** (0=in stock; 1=not in stock), **price** [in local currency]
- **Addition A-E:** You can input additional data for specific types in these fields. (see table: Use of additions A-E.)
- **radial** and **axial spring stiffness c_r, c_a**: [N/µm]
- **Spring stiffness for bending c_{rot}:** [Nm/°] Input spring stiffness for inclination.
- **Factor f_0:** Used to define X and Y (e.g., for deep groove ball bearings), because these values depend on the factor *f0*Fa/C0*.
- **Minimum load *P/C*:** The minimum load *P/C* (*P*: dynamic equivalent load: *C*: basic dynamic load rating) is usually:
  - 0.01 for ball bearings with a cage
  - 0.02 for roller bearings with a cage, 0.04 for pure roller bearings with a cage

- If you input 0 in the database, these values are used automatically in the calculation. These entries only apply to radial load. The minimum axial force is calculated directly in the software.

- **Fatigue load limit C_u:** Coefficient for calculating the modified rating life

Type	Addition A	Addition B	Addition C	Addition D	Addition E
Angular contact ball bearing (single row)		Offset a (mm) (*2)			
Angular contact ball bearing (paired)		Offset a (mm) (*2)			
Cylindrical thrust roller bearing		Coefficient A (*1)		Max. axial force (kN)	
Tapered roller bearing (single row)		Width B (mm)	Offset a (mm) (*2)	Dimension C (mm)	
Tapered roller bearing (paired, O)		Dimension T (mm) (* 1)		Dimension C (mm) (*1)	
Tapered roller bearing (paired, X)		Dimension 2B (mm)		Dimension 2T (mm)	
Barrel-shaped and toroidal roller bearings	Calculation coefficient k1 (SKF: CARB, internal geometry)	Calculation coefficient k2 (SKF: CARB, internal geometry)			
Spherical roller thrust bearing	Dimension d1 (mm)	Dimension T2 (mm)	Dimension D1 (mm)	Dimension T1 (mm)	Pivot center (mm)

Table 9.3: Use of additions A-E

Descriptions given in additional data conform with those in the INA/FAG catalog 2017.

(*1) Values are only used for SKF bearings, as specified in the SKF catalog 2013.

(*2) Values for dimension a, for FAG bearings, have been provided up to the center point by the manufacturer. In KISSsoft, half the bearing width has then been added to this value to obtain the database value (This might result in values that vary slightly from those in the bearing catalog). The value at the bearing center was used in this calculation. This corresponds to the data we received from the manufacturer.

9.5.37.2 Rolling bearing Internal geometry

You can define and edit data for the internal geometry of most bearing types in the Internal geometry tab. Many bearing manufacturers do not make this data available directly. However, in KISSsoft, you can estimate this data by using the calculation methods specified in ISO/TS 16281.

You need the details documented below to calculate internal geometry. For some rolling bearings, you can also enter a user-defined roller profile definition file, which is a ".dat" file (see chapter 28.1.2.1, User-defined roller profile).
To handle bearings without an inner or outer ring more effectively, these profiles can be deactivated either individually or together. However, you must define the internal geometry before you can use this function. The corresponding internal and external bearing diameters, d and D, then refer to the (average) operating pitch circle of the corresponding bearing housings and must be modified if necessary. A slightly different reference diameter is therefore used to calculate the bearing clearance. This might result in negligible differences when compared to the identical bearings with rings. If a bearing side is defined as being without a ring, any calculations of fit used in shaft projects are deactivated.

The load-bearing capacity correction factors (fC, fC0) are used to adjust the load-bearing capacity, which is used to approximate the bearing geometry according to:

$C' = C \cdot f\mathrm{C}$

$C'0 = C0 \cdot f\mathrm{C0}$

- Deep groove ball bearing (single row), four-point contact bearing: **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

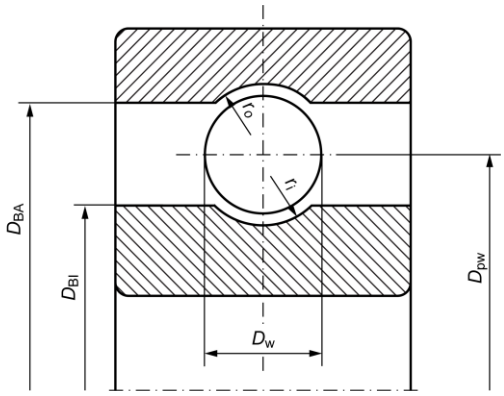


Figure 9.7: Dimensions of the deep groove ball bearing, single row

- Deep groove ball bearing, double row: Same geometry value as for single row deep groove ball bearings. Additional input: Row distance a [mm]

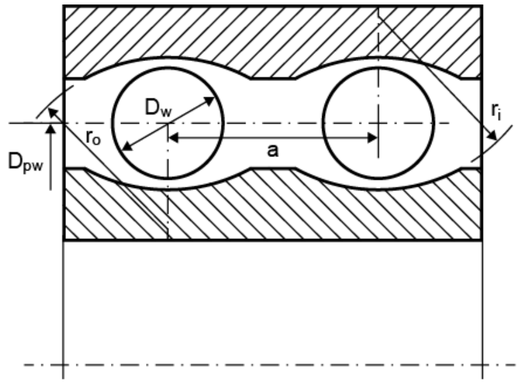


Figure 9.8: Figure: Dimensions of the deep groove ball bearing, double row

- Angular contact ball bearing (single row): **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm], **Minimum initial tension v_{min}** [mm], **Maximum initial tension v_{max}** [**m**m], Minimum pretension force **F_{vmin}** [N], **Maximum pretension force F_{vmax}** [N]

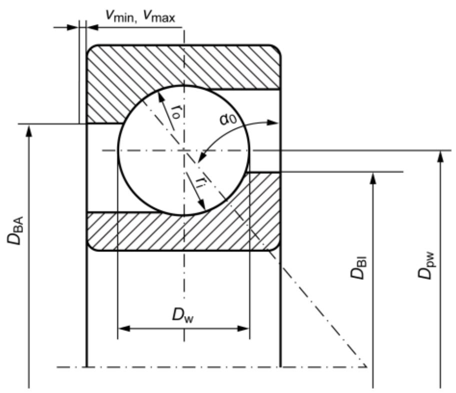


Figure 9.9: Dimensions of the angular contact ball bearing

- Cylindrical roller bearing (single row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Roller length L_{WE}** [mm], **Axial displacement possibility non-locating bearing v_l** [mm], **Axial displacement possibility fixed bearing v_f** [mm]

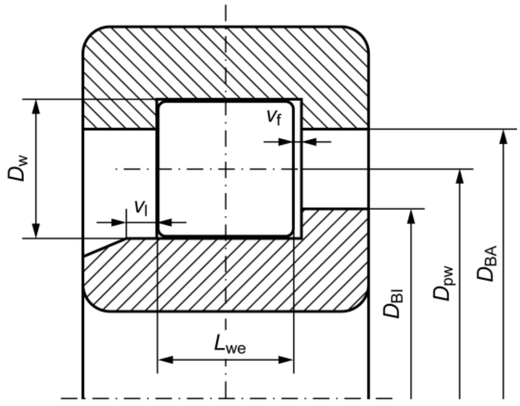


Figure 9.10: Dimensions of the cylindrical roller bearing

- Cylindrical roller bearing (double row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Roller length L_{WE}** [mm], **Row distance a** [mm]

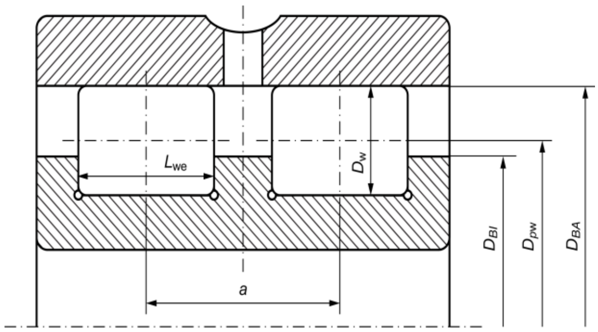


Figure 9.11: Dimensions of the double row cylindrical roller bearing

- Axial angular contact roller bearing: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm]

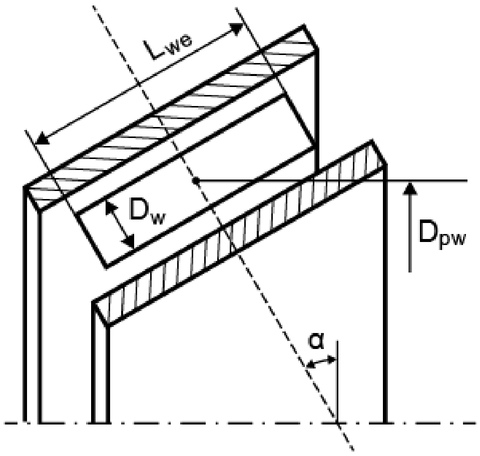


Figure 9.12: Figure: Axial angular contact roller bearing mass

- Tapered roller bearing (single row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm]

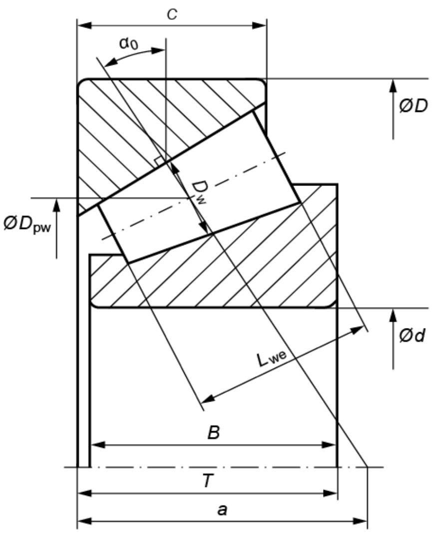


Figure 9.13: Dimensions of the tapered roller bearing

- Double row self-aligning roller bearing: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

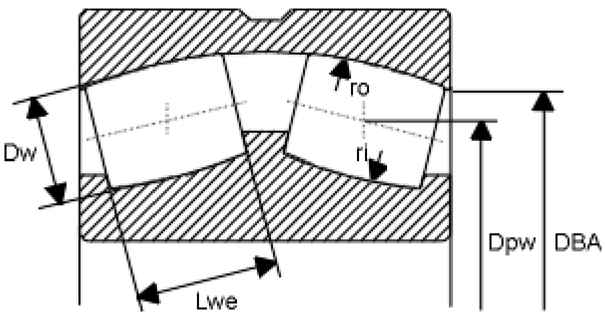


Figure 9.14: Dimensions of the double row self-aligning roller bearing

- Needle roller bearing, needle cage: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm], **Axial displacement possibility non-locating bearing v_l** [mm]

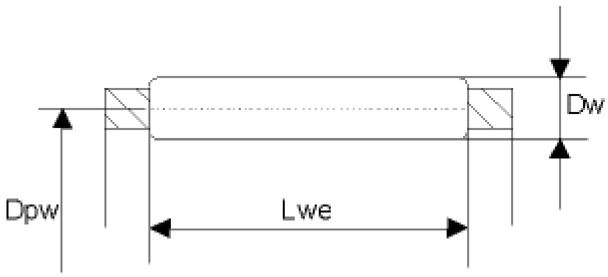


Figure 9.15: Dimensions of the needle roller bearing/needle cage

- Deep groove thrust ball bearing: **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

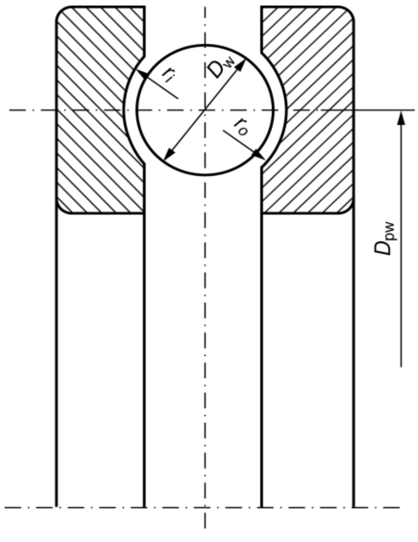


Figure 9.16: Dimensions of the deep groove thrust ball bearing

- Cylindrical roller thrust bearing: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm]

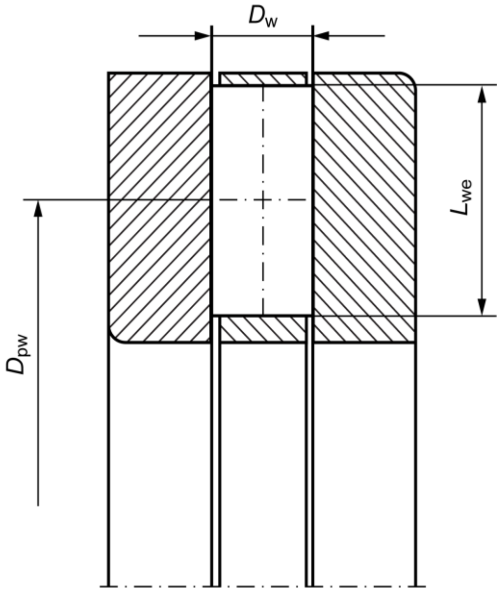


Figure 9.17: Dimensions of the cylindrical roller thrust bearing

- Spherical roller thrust bearing: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm], **Distance L_{WC} of the maximum Roller diameter** [mm], **Radius of curvature, inside r_i** [mm], **Radius of curvature, roller R_p** [mm], **Radius of curvature, outside r_o** [mm]

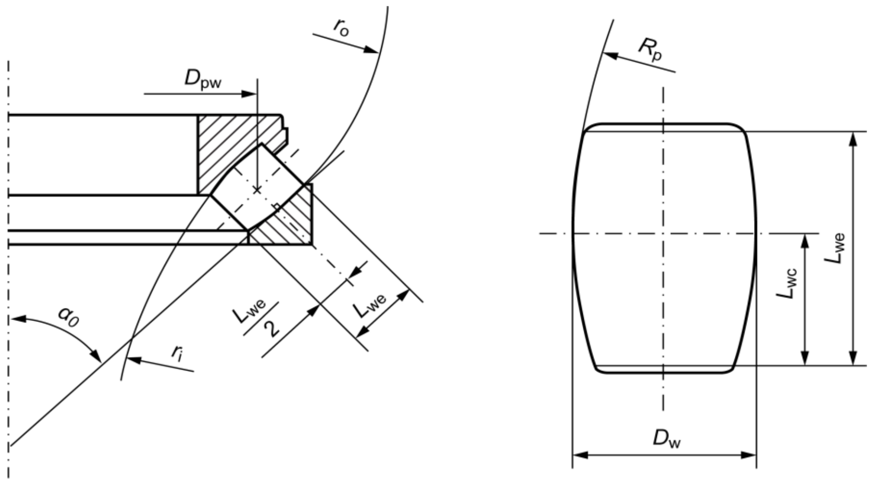


Figure 9.18: Dimensions of the spherical roller thrust bearing

9.5.38 Rolling bearing tolerance

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for rolling bearings begin with W05-??-??.dat.

9.5.39 Rolling bearing fit (tolerance) classes

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for rolling bearing tolerance classes begin with W05-???.dat.

9.5.40 Tooth thickness tolerances

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for tooth thickness tolerances begin with Z01-???.dat or Z9-???.dat.
- **Interpret as:**
    - Tooth thickness allowances: the data is interpreted as tooth thickness allowances.
    - Base tangent length allowances: the data is interpreted as the base tangent length allowances (or normal play).

9.5.41 Toothed belt standard

- **File name:** The database entries refer to external tables (see chapter 9.4, External tables). Tables for the toothed belt standard begin with Z091-???.dat.
- **Calculation method:**
    - 1)"normal" toothed belts (RPP)
    - 2) GT types (PolyChain)
    - 3) AT types (Brecoflex)
    - 4) PG types (PowerGrip)

Differences:

- Special calculation for toothed belts with integrated steel rope (Method 3)
- Calculating the operating factor: the special factor for the speed increasing ratio is added (Method 1, 2 or 4) or multiplied (Method 3)
- Additional performance table for higher performance at greater conversions (Method 2)
- **Calculation method for belt pre-tension:**
    - 1) in % of the circumferential force; slack = 1/50 of the tension length
    - 2) in % of the maximum permitted circumferential force; slack = 1/50 of the tension length

        - 3) in % of (operating factor*power(W)/circumferential speed reference circle (m/s)) (according to DAYCO RPP Panther) Slack = 1/64 of the tension length
        - 4) in % of the circumferential force; Slack = 1/64 of the tension length
    - **Nominal range for power table b:** [mm] belt width, which corresponds to the performance data stored in the file (see file name).
    - **Coefficient for belt pre-tension f:** 0 ... 1.0 (% factor for calculating the belt pre-tension coefficient)
    - **Maximum belt speed v_{max}:** [m/s]
    - **Summand for operation F_S:** No influence
    - **Pitch p:** [mm] pitch of the toothed belt
    - **Elasticity E: [N]** Elasticity = force that doubles the length of a belt (with nominal width). If you do not know this value, enter 0 as the guide value (in this case the elasticity is ignored when the bending test is performed).
    - **Strain ε:** [%] strain along the total length of the belt
    - **Weight per length q:** [kg/m/mm] per meter length and millimeter width

File contents:	
List of suggested standard numbers of teeth for toothed lock washers	*:TABLE LIST z.RadZahne*
List of suggested standard numbers of teeth for belts	*:TABLE LIST z.NormZahne*
Minimum number of teeth, depending on the speed (small disc)	*:TABLE FUNCTION z091k.factorINCR*
Correction factor for powering up, depending on the ratio (this is added to the operating factor)	*:TABLE FUNCTION z091k.factorINCR*
Transmittable power depending on the number of teeth (small disc) and speed (small disc)	*:TABLE FUNCTION z091k.powerNr*
Correction factor for the number of contacting teeth (small disc)	*:TABLE FUNCTION z091k.factorCorrEZ*
Correction factor for belt length	*:TABLE FUNCTION z091k.factorLength*
Correction factor for belt width	*:TABLE FUNCTION belt.bth*
Correction factor for belt width (same values as shown in the table above)	*:TABLE FUNCTION belt.beff*
Disc width depending on belt width	*:TABLE FUNCTION z091k.ScheibenBreite*
Belt type sizing: minimum transmittable power (lower limit) depending on the speed (small disc)	*:TABLE FUNCTION z091k.kWlower*
Belt type sizing: maximum transmittable power (upper limit) depending on the speed (small disc)	*:TABLE FUNCTION z091k.kWupper*

10 Description of the Public Interface

10.1 Interfaces between calculation programs and CAD - Overview

The closest contact point of calculation programs within a CIM concept is the one with the drawing program (CAD). KISSsoft's public data interface can be freely formatted, enabling very powerful communication with third party programs.

All input and output data can be exported in ASCII format. The scope and format of this data is freely definable. For this, each calculation module contains a special, editable report file. The MMMMUSER.RPT files are used as a template for this data transfer. The default setting is that these files are empty. If you want to output data over the interface, you first have to expand the templates. External programs can, in addition, transfer input data (also in ASCII format) to calculation modules. This data is read automatically during start-up, and then displayed on the screen.

*The character string MMMM in a file name is a placeholder for the module to which the file refers. Example: M040USER.RPT

10.1.1 Efficient interfaces

Automated data transfer between the calculation and CAD should only be set up if the benefits are considerably greater than the effort required. For example, an interface between a bolt calculation program and CAD is only of secondary importance since the information to be transferred (for example that, due to the calculation, an M10 bolt has to be selected) is too limited and could be transferred much faster "by hand". If, however, a standard parts library with bolts is available, the bidirectional link between the three components (calculation program, standard parts library and CAD) can prove very efficient.

The following efficient interfaces are available (but this list can be extended):

- General
    - It should be possible to start the calculation programs from the CAD environment (for example by pressing a function key). This enables you to perform a short calculation while you are drawing, transfer the results and then continue drawing.
- Shafts and bearing calculation
    - Output of a contour from the CAD system (i.e. a shaft from detailed or drawing with combined elements) and reading it into the calculation

program. (Problem: in many CAD programs, it is unfortunately rather difficult to define the contour to be exported.)
- Output of a shaft that has been optimized in the calculation program (including rolling bearings etc.), reading it, and importing it into CAD as drawing information.
- Transfer of bending lines and similar data into the CAD system.
- Rolling bearings and plain bearings are calculated, and then the contour is transferred to the CAD system. (Frequently, the CAD system already contains information on rolling bearings, so that only the bearing label is of interest.)

- Gear calculation
    - Calculation of fabrication data in the program and transfer of the required values to the CAD as text. This is a very important function, since the recording of the data is very prone to errors, with potentially serious consequences.
        - Calculation of the exact tooth form in Page view and transfer to the CAD system. (Although this results in very pretty drawings, it usually does not supply any necessary information, except if the data undergoes further processing, i.e. via transfer onto a wire electro-discharge machine.)
- Transfer of the schematic axial section or the Print Preview of the gears to the CAD system (but can also be achieved just as quickly in CAD, manually).
- Machine elements
    - Transfer the contour of calculated machine elements to the CAD such as bolts, V-belt sheaves etc. (Frequently, the CAD station already contains appropriate, preprogrammed information, so that only the parts definition is of interest).
- Shaft-hub connection
    - The sizing or proofing of connections should be implemented directly in a CAD system, so that known data from the CAD can be transferred into the calculation and the results of the calculation can in turn be returned to the CAD system.

10.1.2 Open interfaces concept in KISSsoft

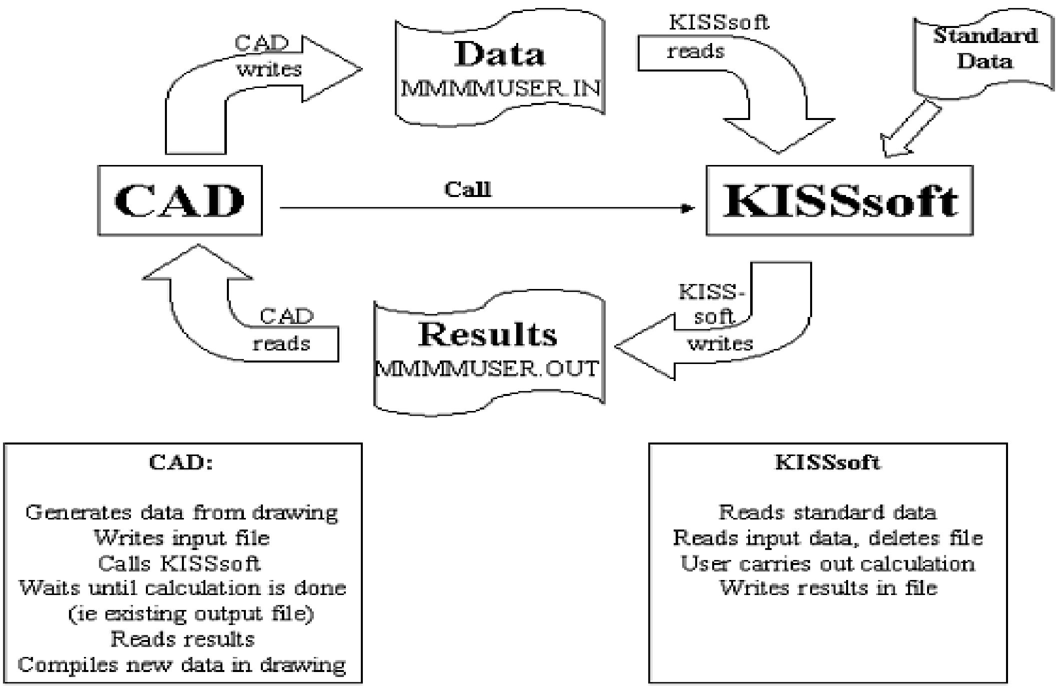
The KISSsoft interfaces concept has a simple, yet very flexible structure.

It should be possible to integrate calculation programs into all kinds of CAD systems as simply as possible, and use them in different environments (operating systems such as MS Windows or UNIX).

The interface mechanism between the CAD system and KISSsoft is based on a text dataset (ASCII file), and an ID is transferred together with the numerical value for all transfer data (see chapter 10.3, Example: Interference fit calculation). This dataset can be of variable length, but only the values that are known in the CAD system will be transferred. This depends on the CAD system and the currently active drawing.

KISSsoft will test the dataset transferred by the third party program, to ensure it is complete and consistent, and, if necessary, you will be prompted to input additional data in the KISSsoft input system. KISSsoft will then run the calculation and write the output data that the CAD system requires to a second text dataset. It then returns this dataset to the CAD system. By using the report generator you can select any format for the output file, i.e. KISSsoft adapts itself to the third party program. The CAD can now read the data required by the situation and process it selectively.

This concept results in simple interface forms, which enables even non-specialists to write applications quickly.



10.2 Defining input and output

10.2.1 Preamble

- In this description the KISSsoft program is always taken as a reference, i.e. an input file for KISSsoft becomes an output file for the third party program and vice versa.

- For automatic data exchange with other programs you will require files with the name *MMMMUSER.RPT* . You can adapt these files to your own requirements. However, if you have purchased KISSsoft interfaces, you should act with caution, since these files are also required for these interfaces.

File name	**Storage location**	**Description**
MMMMUSER.IN	<CADDIR> *)	Input file for KISSsoft (is written by the third party program) User's temporary input file (= will be deleted when imported into KISSsoft)
MMMMUSER.OUT	<CADDIR>	KISSsoft output file (will be written by KISSsoft and read by the third party program). Temporary (= should be deleted by the third party program)
MMMMUSER.RPT	<KISSDIR>	Defines the output format (similar to report), can be permanent/optional (= is usually created once and is retained)
Z10Gear1.RPT *Z10Gear2.RPT* *Z10Gear3.RPT* *Z10Gear4.RPT*	<KISSDIR>	Defines the output format for the manufacturing data in the case of cylindrical gears (see below). Corresponds to MMMMUSER.rpt for this special case.
Z10Gear1.OUT	<KISSDIR>	Output file containing the toothing stamp for cylindrical gears.
Z70Gear1.RPT *Z70Gear2.RPT*	<KISSDIR>	Defines the output format for bevel gears.
Z17Gear1.RPT *Z17Gear2.RPT*	<KISSDIR>	Defines the output format for crossed helical gears.
Z80Gear1.RPT *Z80Gear2.RPT*	<KISSDIR>	Defines the output format for worm wheels.
Z9aGear1.RPT *Z9aGear2.RPT*	<KISSDIR>	Defines the output format for spline connections.
Z??Gear1.OUT *Z??Gear2.OUT*	<CADDIR>	Toothing stamp, similar to definition files.

*) If you specify the entire file name including the directory, it can also be read from any location.

10.2.2 Requirements placed on the third party program

To successfully run and use KISSsoft within a third party program, the following minimum requirements must be met. The third party program must

1. have a query mechanism (i.e. macro language) for providing information, e.g. input data,
2. be able to write and read ASCII files,
3. be able to start a program.

10.2.3 Used files

10.2.3.1 Input file

An input file with the name *MMMMUSER.IN* will be used. It has the same structure and the same function as the saved calculations, except for its temporary status. The values are assigned to the KISSsoft variable names with =. A separate row is used for each variable.

► **Example**

VERSION=2.5;
m02Aw.dWa=30¶
m02Aw.lW=20¶
m02An.lN=25¶

The input file will be read after the default values are predefined (see chapter 2.4, Defining your own template files), i.e. the values of the temporary input file will overwrite the values set by the default.

Note: Temporary input files are used for frequently changing variables such as geometry and/or performance data: data which typically changes from calculation to calculation. It would also be possible to write this data to the template files, since they represent normal input variables. This would, however, mean that the program generating these files had to interpret the data that has already been written, i.e. had to accept permanent constraints, to enable it to completely define the default and to reset to the default data again at the end.

10.2.3.2 Output file

To return the data that is relevant for the KISSsoft calling program, the specified output file *MMMMUSER.OUT* will be generated immediately after a calculation. The scope and the format of the output file will be defined in a report template called *MMMMUSER.RPT* window.

This means that KISSsoft can fully adapt itself to the syntax of a third party program. The range of commands and the report generator's syntax is described in the Reports section (see chapter 8.5, Report templates). Example report files are supplied to help you with this.

10.2.4 Temporary files

The input file *MMMMUSER.IN* input file is generated by the third party program and, after having been read, will be deleted by KISSsoft. The *MMMMUSER.OUT* output file is deleted when KISSsoft starts, and created again after a calculation.

10.2.5 Explicitly reading (importing) and generating data

In addition to the previously described automatic definition you can also explicitly read data by selecting **File >Interface > Read data**, or generate it by selecting **File > Interface > Output data**. You can therefore select any point in time and use it for many varied tasks, i.e. to generate an order form etc.

10.3 Example: Interference fit calculation

The following example of the Interference fit assembly calculation is used to illustrate the way that the KISSsoft interfaces concept works, in more detail.

For the interference fit assembly between the gear rim and the cylindrical gear hub, you need to find the one tolerance pairing that meets the following boundary conditions:

Permanent torque MD = 88000 Nm

The tolerance pairing involves a system of the standard drill hole *(H)*.

Safety	against sliding > 1.4
	against fracture of the hub > 1.5
	against fracture of the gear rim > 1.5
	against the yield point of the hub > 1.1
	against the yield point of the gear rim > 1.1

Procedure:

The necessary information for the geometry is extracted directly from the drawing, with a suitable CAD routine, and converted to the interfaces format defined by KISSsoft:

	m01allg.df=640
	m01n.da=800
	m01w.di=242
	m01allg.l=200.

File contents *M010USER.IN*

Then, start the KISSsoft module. It accepts the geometry data and displays it in the main screen.

In the main screen, enter any parameters that are still missing, the torque, and the materials, and then start the calculation. In KISSsoft, you can also size the tolerance pairing. Here, you are prompted to select suitable tolerance combinations from a list. The system then performs the calculation with your final selection.

After you have exited the calculation, the results file is automatically converted into a format that can be read by the CAD macro. The format of this result file is defined via the templates file *M010USER.RPT*:

[SHAFT]
ntol_max = %f{m01w.tol.max}
ntol_min = %f{m01w.tol.max}
ntol_bez = %s{m01w.tol.bez}
[HUB]
ntol_max = %f{m01n.tol.max}
ntol_min = %f{m01n.tol.max}
ntol_bez = %s{m01n.tol.bez}

File contents *M010USER.RPT*

The result then looks like this:

[SHAFT]
wtol_max = 390.000000
wtol_min = 340.000000
wtol_bez = s6
[HUB]
ntol_max = 50.000000
ntol_min = 0.000000
ntol_bez = H6

File contents *M010USER.OUT*

This data is now attached directly to the appropriate dimension in the CAD system, via the macro.

Summary:

The individual tasks are therefore split up: Each side of the interface will perform only the tasks it is best suited to. The CAD administers the geometry and passes this information on to the calculation program, which knows how to process the data, and which, in turn, will return the result to the CAD.

The CAD system and calculation program can be used efficiently together, with the defined interface.

10.4 Geometry data

KISSsoft has different interfaces for transferring geometry data (contours, drawings):

- DXF format (recommended for communication with most CAD systems)
- IGES format (which exports tooth forms as splines)
- BMP format (Windows bitmap)
- JPG/JPEG format (pixel image)
- PNG (Portable Network Graphic) format

10.5 COM interface

You can control KISSsoft remotely via a COM interface. It can easily be accessed from Visual Basic or Excel.

10.5.1 Registering the server

Now register the KISSsoft COM server on your local computer. There are two different ways of doing this:

- Right-click to display the context menu and then select **As administrator** to display the Windows prompt. Then, go to the "... /bin" subfolder and run the "KISSsoftCOM_Register.bat" file.
- To do this, enter these command lines in a Windows command prompt, in the KISSsoft installation bin directory:

```
KISSsoftCOM.exe /regserver

regsvr32 KISSsoftCOMPS.dll

regsvr32 KISSsoftCOMPS.dll
```

To enable you to use the COM server regardless of release level, there is a version-specific COM interface that has the relevant release number.

To use this variant, run the KISSsoftCOM_RegisterXXXX.bat file or the files listed below. (Here, XXXX represents the version number, e.g. 2021):

```
KISSsoftCOMXXXX.exe /regserver

regsvr32 KISSsoftCOMXXXXPS32.dll

regsvr32 KISSsoftCOMXXXXPS64.dll
```

You will need administrator rights to register each program.

10.5.2 Server functionality

The server has a number of functions that you can use to start a calculation module, read or set values, and perform a calculation.

- **GetModule([in] BSTR module, [in] VARIANT_BOOL interactive)** starts a calculation module from the module descriptor (e.g. Z012 or W010). ‘Interactive’ defines whether the calculation module is to be generated with a graphical user interface.
- **IsModuleValid([out] VARIANT_BOOL isValid)** returns a value that indicates whether the calculation module has been generated with ‘GetModule’ and is present.
- **Calculate()** performs the main calculation for the active module.
- **CalculateRetVal([out, retval] VARIANT_BOOL* isOk)** runs the main calculation for the active module, and returns a value to tell you whether the calculation is OK.
- **SetVar([in] BSTR name, [in] BSTR value)** is a function with which you can set variables to a required value. This data is transferred as text. You will find the variable names in the report templates, but there is no guarantee that all these variables will remain the same in the future.
- **GetVar([in] BSTR name, [out, retval] BSTR* value)** returns a variable from KISSsoft as text.
- **ShowInterface([in] VARIANT_BOOL wait)** displays the graphical user interface. Use the 'wait' parameter to specify whether the function is to wait until the dialog is closed.
- **IsActiveInterface([out, retval] VARIANT_BOOL* isActive)** shows whether a KISSsoft dialog is active.
- **IsActive([out, retval] VARIANT_BOOL* isActive)** shows whether a module has been loaded.
- **ReleaseModule()** releases the loaded module again. You must always release a module again, to reduce the load on the server.
- **LoadFile([in] BSTR file name)** loads the specified file.

- **SaveFile([in] BSTR file name)** saves the calculation in the specified file.
- **CheckLicense ([in] name BSTR, [out, retval] VARIANT_BOOL* isOk)** shows whether the license is valid.
- **GetININame([out, retval] BSTR* name)** supplies the name of the loaded INI file.
- **GetVersionFromFile([in] BSTR filename, [out, retval] BSTR* version)** supplies the version number (e.g. 2.6) of the KISSsoft module in the calculation file (the version number depends on which module is being used).
- **GetModulFromFile([in] BSTR filename, [out, retval] BSTR* name)** supplies the KISSsoft module label (e.g. M040) in the calculation file. You must first fetch a calculation module (GetModule).
- **GetKsoftVersionFromFile([in] BSTR file name, [out, retval] BSTR* kSoftVersion)** supplies the KISSsoft version number (e.g. 03-2011), given in the calculation file.
- **GetKsoftVersion([out, retval] BSTR* kSoftVersion)** supplies the KISSsoft version (e.g. 03-2011) that is registered and was started via the COM interface.
- **GetDBName([in] BSTR db_name, [in] BSTR table, [in] SHORT flag, [in] LONG ID, [in] LONG order, [out,retval] BSTR *name)** Use the 'flag' parameter to specify whether the ID (flag = 0) or the result (flag = 1) is to be used as the input. The output is then either the order and the name of the entry or the ID (in material database BEZ_DIN). No message is displayed if an error occurs. A ‘False’ is returned for the function.
- **GetDBValue([in] BSTR db_name, [in] BSTR table, [in] LONG ID, [in] BSTR fieldname, [out,retval] BSTR *name)** supplies the value present in this database field. No message is displayed if an error occurs. A ‘False’ is returned for the function. Please note that the bearing manufacturers have not approved this function for directly extracting bearing data, which is why this function is disabled.
- **GetKsoftVersionSettings([out, retval] BSTR* kSoftVersionSettings)** supplies the KISSsoft version (e.g. 03-2014) of the temporary settings folder in which the personal settings are stored.
- **SetSilentMode([in] VARIANT_BOOL silent)** defines whether messages are to be hidden or not, so that calculations can be performed without you having to confirm system prompts.
- **Report([in] LONG show)** writes the report. You can specify whether or not this report is to be displayed. The report is created in the Temp directory in the "KISS_?" sub-folder.
- **SetDebugFile([in] BSTR path)** sets the path and file name of a debug file which can be used for error tracking.
- **ReportWithParameters([in] BSTR infile, [in] BSTR outfile, [in] LONG show, [in] LONG type)** creates the report using the specified report template (‘infile’) in the predefined place with the predefined name (‘outfile’) and supplies the file type. You can enter file names either with or without the path. When entering the report template

('infile'), you should also input the file extension (e.g.'Z012ld0.rpt'). If you do not enter a path for this file, the program will search the default directory (see also Reports) for the file. You must also enter the file extension for an output file. If you do not specify the path, the file is saved to the Temp directory with the name 'KISS_?'. The file extension should match the specified type. Use the 'show' parameter to define whether the report is to be displayed, or not. Use the 'art' (meaning 'type') parameter to define the output format. (art=0 → rtf format with *1 ; art=1 → rtf format without *1 ; art=2 → html format with *1 ; art=10 → txt format without *1 ; art=20 → txt format in Unicode without *1) ; art=1000 → pprpt format with *1 ; art=1001 → pprpt format without *1).

- *1 = takes into account the data level
- Examples of possible combinations: With default report templates → RTF format: ReportWithParameters("C:\Program Files (x86)\KISSsoft 2019\rpt\Z070ld0.rpt","C:\Temp\Z070ld0.rtf", 1, 0), HTML format: Call ksoft.ReportWithParameters("Z070ld0.rpt", "C:\Temp\Z070ld0.html", 1, 2), PPRPT format: ReportWithParameters ("C:\Program Files (x86)\KISSsoft 2019\rpt\Z070ld0.rpt","C:\Temp\Z070ld0.pprpt", 1, 1000); with drawing stamp report template → TXT format: ReportWithParameters("Z10GEAR1d.rpt","C:\Temp\Z010GEAR1d.txt", 1, 10)

- **Message([out] VARIANT *strings, [out] VARIANT *types:, [out] LONG *numElem)** returns the messages from the last calculation in the first parameter, as an array containing strings. The second parameter contains the particular message type (error, warning, info). The number of existing messages is shown in numElem.
- **CallFunc([in] BSTR name)** allows you to perform special calculations. A more detailed list of the available calculations is available on request.
- **CallFuncNParam([in] VARIANT paramArray)** allows you to perform special calculations. A more detailed list of the available calculations is available on request.
- **SetLanguage([in] LONG index)** specifies the language used for reports, interfaces and messages (0 = German; 1 = English; 2 = French; 3 = Italian; 4 = Spanish; 5 = Russian; 6 = Portuguese; 7 = Chinese).
- **GetLanguage ([out, retval] LONG* index)** reads the index of the language that is currently set. Indexes are described in the SetLanguage() function description.

10.5.3 Example of a call from Excel

The best way to describe this functionality is to use an example. To use KISSsoft from Excel, you must first select **Extras > References** and then select the KISSsoftCom type library in the Visual Basic Editor.

The first example shows how to use a single gear calculation to define the tip and root circles of a gear:

```
Public Sub ExampleKISSsoftCOM()

Dim ksoft As CKISSsoft

Dim da As String

Dim df As String

' get KISSsoft Instance

set ksoft = New CKISSsoft

' get KISSsoft module for single gear

Call ksoft.GetModule("Z011", False)

' set values

Call ksoft.SetVar("ZR[0].z", "20")

Call ksoft.SetVar("ZS.Geo.mn", "5.0")

Call ksoft.SetVar("ZR[0].x.nul", "0.5")

' Calculate

Call ksoft.Calculate

' get values

da = ksoft.GetVar("ZR[0].da.nul")

df = ksoft.GetVar("ZR[0].df.nul")

' release module

Call ksoft.ReleaseModule

' release server

Set ksoft = Nothing

End Sub
```

The second example shows how to display the KISSsoft user interface:

```
Public Sub ExampleKISSsoftCOM()

Dim ksoft As CKISSsoft
```

```
Dim da As String

Dim df As String

' get KISSsoft Instance

Set ksoft = New CKISSsoft

' get KISSsoft module for single gear

Call ksoft.GetModule("Z011", True)

' show interface

Call ksoft.ShowInterface(True)

' get values

da = ksoft.GetVar("ZR[0].da.nul")

df = ksoft.GetVar("ZR[0].df.nul")

Call ksoft.ReleaseModule

Set ksoft = Nothing

End Sub
```

The same example with "later binding" (the exact property or method is not determined until runtime, which enables you to compile the Visual Basic client without having to know the exact function of the call):

```
Public Sub ExampleKISSsoftCOM()

Dim ksoft As Object

Dim da As String

Dim df As String

' get KISSsoft Object

Set ksoft = CreateObject("KISSsoftCOM.KISSsoft")

' get KISSsoft module for single gear

Call ksoft.GetModule("Z011", True)

' show interface
```

```
Call ksoft.ShowInterface(True)

' get values

da = ksoft.GetVar("ZR[0].da.nul")

df = ksoft.GetVar("ZR[0].df.nul")

Call ksoft.ReleaseModule

Set ksoft = Nothing

End Sub
```

The fourth example shows a contact analysis that was run using the caControll.dat control file (you will find an example file in the dat folder) and they way messages were processed after the calculation:

```
Public Sub ExampleKISSsoftCOM()

On Error GoTo ExitOnErr

Dim ksoft As CKISSsoft

' get KISSsoft Instance

Set ksoft = New CKISSsoft

' get KISSsoft module for gear pair

Call ksoft.GetModule("Z012", True)

' load File – change this to fit to a real file on your machine

Call ksoft.LoadFile("C:\yourPathHere\ExCOM3.z12")

' calculate

Call ksoft.Calculate

Dim ioData(0 To 2) as String

' Which calculation to start

ioData (0) = "CalculatePathOfContactForPairKS"

' controling file

ioData (1) = "C:\ yourPathHere\caControl.dat"
```

```
' Path for results
ioData (2) = "C:\ yourPathHere\prot"
' calculate contact analysis
Call ksoft.CallFuncNParam(ioData)
' Check for messages
Dim mess As Variant
Dim types As Variant
Dim numElem As Long
Dim typesElem As Long
Dim typesElemStr As String
Call ksoft.Message(mess, types, numElem)
If (numElem > 0) Then
Dim msg As String
For i = 0 To numElem - 1
msg = CStr(mess(i))
typesElemStr = CStr(types(i))
typesElem = CLng(types(i))
If (typesElem = 0) Then
Call MsgBox(msg, vbInformation)
ElseIf (typesElem = 1) Then
Call MsgBox(msg, vbExclamation)
Else
Call MsgBox(msg, vbCritical)
End If
Next
```

```
End If

' close ksoft

Call ksoft.ReleaseModule

' no problems, so exit

Exit Sub

ExitOnErr:

MsgBox ("error occured when calling KISSsoft.")

End Sub
```

11 3D Interfaces

11.1 Overview of the available CAD interfaces and their functionality

Feature/CAD	SolidWorks	Autodesk Inventor	Solid Edge: Solution Partner	Siemens NX: Solution Partner	PTC PartnerAdvantage Creo Parametric	Catia	HiCAD	Parasolid / Neutral format Interface (step)
Version	2020 - 2023	2021 - 2024	2021- 2024	NX2007 NX2206, NX2212, NX2306, NX2312	Creo 6-9	V5 R20-32), V5-V6 R2013-2022)	2018 - 2023	
Cylindrical gears, spur/ helical gears	✓	✓	✓	✓	✓	✓	✓	✓
Cylindrical gears, internal/ external teeth	✓	✓	✓	✓	✓	✓	✓	✓
Worm/ cylindrical worm wheel	✓	✓	✓	✓	✓	✓	✓	✓
Worm/ enveloping worm wheel	✗	✗	✗	✗	✗	✗	✗	✓
Rack	✓	✓	✓	✓	✗	✗	✓	✓
Bevel gears, straight	✓	✓	✓	✓	✓	✓	✓	✓
Bevel gears, helical	✗	✗	✗	✗	✗	✗	✗	✓
Bevel gears, spiral	✗	✗	✗	✗	✗	✗	✗	✓
Face gears	✗	✗	✗	✗	✗	✗	✗	✓
Beveloid gears	✗	✗	✗	✗	✗	✗	✗	✓
Splines (Shaft-hub)	✓	✓	✓	✓	✓	✓	✓	✓
Toothing on existing shaft	✓	✓	✓	✓	✓	✓	✗	✗
Shafts	✓	✓	✓	✓	✗	✗	✗	✓
CAD add-in menu	✓	✓	✓	menu-driven only	✗	Selection menu	✗	✗
Manufacturing data	✓	✓	✓	✓	✓	✓	✗	✗

11.2 Generation of 3D gears

You must first perform a gear calculation to ensure that the results are consistent. Click on **Graphics > Settings** to select the CAD system to which you want to export the selected element.

Then, select the **Graphics > 3D Export** menu option and then specify which individual gears you want to generate, and the configuration (only possible as individual gears).

In the case of Siemens NX, generation is only possible if you have started KISSsoft from the NX add-in menu, then run the gear calculation and clicked on the required generation button. In the case of Creo Parametric (ProEngineer) and CATIA, you must run the CAD interface so that you can start the gear generation process from KISSsoft. In the SolidWorks, Solid Edge and Inventor CAD systems, click a generation button to run the CAD process, if it is not already open.

The default setting runs the gear generation process with a permitted tooth form error (tolerance band) of 1 μm. If this tolerance is too large, you can open the **Tooth form** tab to change the tolerance. Once this is changed, you must click **Calculate** again (Tooth form tab active), to transfer the inputs and recalculate the tooth form.
Changing the generation type in the Tooth form tab (polylines, arc of circle approximation, splines) only affects the 2D display. In Siemens NX, SolidWorks and Solid Edge, the part is generated with splines. In Inventor, Creo Parametric (ProEngineer) and CATIA, it is created with arcs of a circle. SolidWorks and Solid Edge also support other generation types, which you can change by entering the additional **APPROXIMATION=1** parameter in the KISS.ini (see chapter 2.6.10, Definitions in [SOLIDWORKS]) file, in the relevant CAD system.
In the case of the gears, the transverse section of the tooth space is usually cut out from a cylinder and then duplicated as a pattern. For worms with a helix angle > 50° and a number of teeth < 4 the tooth space is cut out in the axial section and then duplicated.

Restrictions of the CAD interfaces (except Parasolid):

- Cylindrical Gears: Double helical gearings are not created directly and must be done manually, first one side then the other side.
- Worms: Enveloping worms and worm wheels are not possible (only cylindrical worms with worm wheel).
- Modifications across the face width are not taken into account (only tooth form modifications are considered).

11.3 Generating 3D shafts

Until now, it has only been possible to generate shafts in 3D in the SolidWorks, Solid Edge, Autodesk Inventor and Siemens NX CAD systems.

First, a shaft calculation must be performed to ensure the results are consistent. Click on **Graphics > Settings** to select the CAD system to which you want to export the selected element.

Then, click **Graphics > 3D Export** to select the shaft you require, and configuration (if you want to generate more than one shaft). Each shaft is created individually in the configuration, in sequence, in its own parts.

This enables you to create a 3D shaft in the CAD system at the click of a button, according to the data from a KISSsoft shaft calculation.

11.4 Viewer with neutral format interface

KISSsoft has a 3D viewer for displaying individual gears or a gear system. The viewer is activated from the **Graphics** > **3D Geometry** menu.

In the 3D viewer, you can export the solid model in STEP and Parasolid formats (text and binary). Supported gears (see chapter 11.1, Overview of the available CAD interfaces and their functionality) and details of how to operate the viewer (see chapter 23.3, 3D Geometry). You can change the settings by selecting **Calculation** > **Settings** > **3D Generation**.

11.4.1 Parasolid export of the complete system

The complete 3D assembly can be generated and exported using Parasolid. Select **File > Export... > Complete 3D model** to save the complete system as a STEP, Parasolid text (X_T) or Parasolid binary (X_B) file. The settings for the 3D viewer are used.

11.4.2 Parasolid Export of 3D Shafts

Parasolid can be used to generate the solid model of the shaft. The available data formats for export are STEP, Parasolid text (X_T) and binary (X_B).

Select **File > Export > Shaft > 3D Geometry** to generate the model. If the calculation model contains a number of shafts, you can export these by selecting **File > Export > Geometry 3D System**.

11.4.3 Face gear: 3D geometry

The 3D model of a face gear is generated by simulating the cutting process. In this simulation, there are no limitations involving the helix angle, shaft angle or offset. The reference coordinates of the model are defined according to Roth [3], and the corresponding positions of pinion and gear are defined by equations (1) and (2).

$$L_S = \overline{O_S Q_S} = \frac{r_{t2} + (r_{tS} + x_S \cdot m_n) \cdot \cos\theta}{\sin\theta} \quad (1)$$

$$L_2 = \overline{O_0 Q_2} = \frac{r_{tS} + x_S \cdot m_n + r_{t2}\cos\theta}{\sin\theta} \quad (2)$$

Where r_{tS} is the pinion reference radius and x_S is the pinion profile shift coefficient. r_{tS} in the cutting operation is calculated from the pinion cutter.

To defined the shaft angle and the radial offset (? and a), select **Geometry > Details…**.

The face gear model is generated by simulating the cutting process, and the tooth flank is approximated as a spline surface.

The manufacturing process is based on the Parasolid core, where the quality of the model depends on the settings made in Parasolid modeling (see **Calculation > Settings > Parasolid**).

► **Note:**

The strength calculation is performed with the assumption that the shaft angle is 90° and the radial offset is 0. The shaft angle and radial offset are only used for 3D model generation, so the strength calculation results may not be valid.

11.4.4 Bevel gear: generating a 3D model

The 3D geometry model for straight, helical and spiral bevel gears is defined according to ISO 23509 and the tooth form is calculated for several sections along the facewidth. The tooth form is placed across the planar involutes of the virtual cylindrical gear, at ninety degrees. Then, the tooth flank surface is generated by sweeping the tooth forms of the sections. The tooth forms in the individual sections are transformed by the angle φ_β into the relevant position. The angle of each section φ_β is calculated separately for the generating and face milling processes by using the auxiliary angles φ and η. For this reason, the final tooth form along the facewidth is an extended epicycloid (generating) or circular (milling) form.

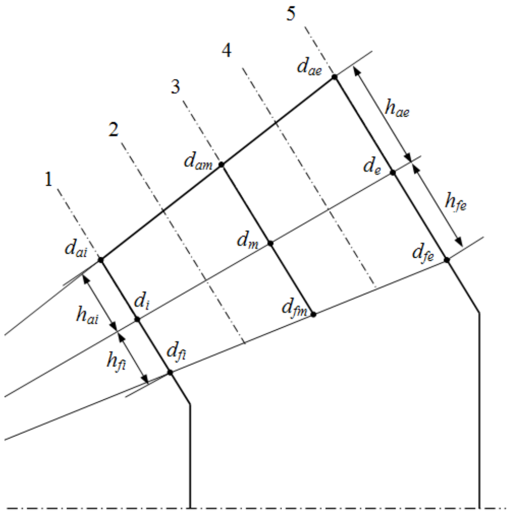


Figure 11.1: Definition of the sections for tooth form calculation

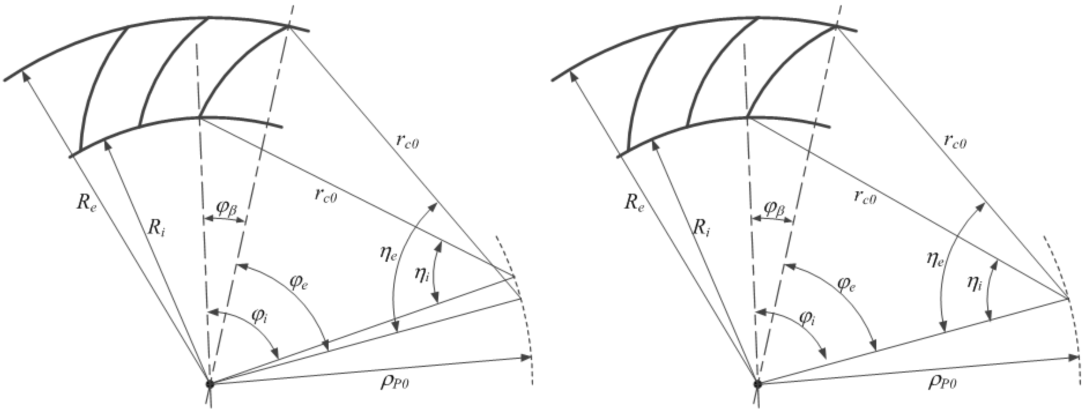


Figure 11.2: Transformation angle of generating (left) and face milling (right) processes

Machine tool manufacturers (such as Klingelnberg and Gleason) also have their own processes for generating tooth forms that differ slightly from the procedures mentioned above. The tooth form is called an octoid, and may differ slightly from our tooth form. However, we have ascertained that the difference between the tooth forms is much less than the tolerance range, and will not cause any problems in practical use.

11.4.5 Worm wheel: generating a 3D model

The 3D model of the enveloping worm wheel is generated by simulating the actual cutting process. The tooth forms at several sections along the facewidth are calculated, and the tooth flank is approximated as a spline surface. The model is generated using the best possible tool to manufacture the worm. Theoretically, the tool generates the worm, with regard to arc of circle, pressure angle, and tooth form. However, if the tool itself was manufactured to these specifications, it would no longer be usable after resharpening, because it would be smaller than the worm. The tools used to manufacture worm wheels are therefore slightly larger than the worm they are to create so that they can be resharpened several times, as required [4]. To generate the model using the larger tool, you can set the oversize factor in the Module specific settings window. You can enter the oversize factor directly in the **Oversize factor for worm wheel cutter (3D)** input window.

In this case, the tool will have a larger tooth thickness, and therefore generate a smaller tooth thickness on the gear. The cutting distance between the hob and the gear will then be changed accordingly, to ensure a consistent result for the root and tip diameters on the gear.

11.4.6 General information about 3D modeling in Parasolid

If the model could not be generated correctly, you can improve it by modifying the Parasolid settings (see **Calculation > Settings > 3D Generation**) or, if gears are involved, by reducing the permitted deviation (**Tooth form > Approximation for export > Permissible deviation** tab).

11.5 3D interface to SolidWorks

Manufacturer: KISSsoft AG

The interface between SolidWorks and KISSsoft is created by direct integration in the 3D CAD system. Use this to run all KISSsoft calculation modules from within SolidWorks. Cylindrical or bevel gears calculated in KISSsoft can be generated directly in SolidWorks as a 3D part (see chapter 11.2, Generation of 3D gears) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see chapter 11.3, Generating 3D shafts) directly in SolidWorks. From within KISSsoft, you can start SolidWorks with one click on a button. The system opens a new part, and generates the appropriate part. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.
You can also add gear teeth to existing shafts (see chapter 11.5.1, Gear teeth if existing shaft data is present). In addition, gear manufacturing data in the 2D range (see chapter 11.5.3.2, Adding manufacturing data) can be automatically inserted on the drawing as a text field, with the interface. The gear manufacturing data is attached to the relevant cutout (tooth space).

11.5.1 Gear teeth if existing shaft data is present

Procedure for manufacturing gear teeth:

1. Select the required area in the CAD system
2. In KISSsoft, select which gear (e.g. **Gear 1**) you want to generate on the cylinder.

Requirements:

- The cylinder diameter must already be the correct external diameter for the toothing before generation starts.
- For internal toothing, a hollow cylinder must already be modeled before the gear teeth can be cut out.

Toothing will be generated for inside and outside cylindrical gears with spur and helical teeth.

11.5.2 Integrating the KISSsoft Add-in (menu options in CAD)

You should register the Add-in when you install it. However, if this doesn't work and the KISSsoft menu is not displayed in SolidWorks, you must register the Add-in.

Go to the KISSsoft installation directory and select the **SolidWorks** sub-folder. In it, double-click on the **SolidWorksRegister64.bat** file to register the interface.

If the KISSsoft Add-in is registered successfully, a message is displayed to confirm that this is the case.

To delete the registration, double-click on the **SolidWorksUnRegister.bat** file in the KISSsoft installation directory. A message is then displayed to confirm that deregistration was successful.

If the Add-in is not displayed directly in SolidWorks, select the **Tools > Add-ins** menu to open a new window. You can then select the **KISSsoftSWAdd-in** in this window.

This integrates the KISSsoft menu options in SolidWorks. The menu remains present, even after a restart, and only needs to be linked once.

The KISSsoft Add-in menu options are available in eight languages (German, English, French, Italian, Spanish, Russian, Portuguese and Chinese). They use the same language as was selected when KISSsoft was being installed. To set the language, open the **kiss.ini** file in the KISSsoft installation directory, click on **DISPLAYLANGUAGE**, and set the language you require (0 = German, 1 = English, 2 = French, 3= Italian, 4= Spanish, 5= Russian; 6= Portuguese, 7= Chinese). This language setting now also applies to your KISSsoft system.

11.5.3 Add-in functions (calls)

11.5.3.1 Calling KISSsoft from the Add-in

Select the **Tools > KISSsoft** menu option to open all the KISSsoft calculation modules directly. The generation of a new/additional gear will then continue as described in the gear generation process (see chapter 11.2, Generation of 3D gears).

11.5.3.2 Adding manufacturing data

The **Add manufacturing data** menu option only works in the Part view. Procedure for adding a gear stamp to a drawing:

1. Open the part and select a tooth's **Cutout**.
2. Select the **Adding manufacturing data** menu option.

This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

11.5.3.3 Opening the calculation file for the created gear

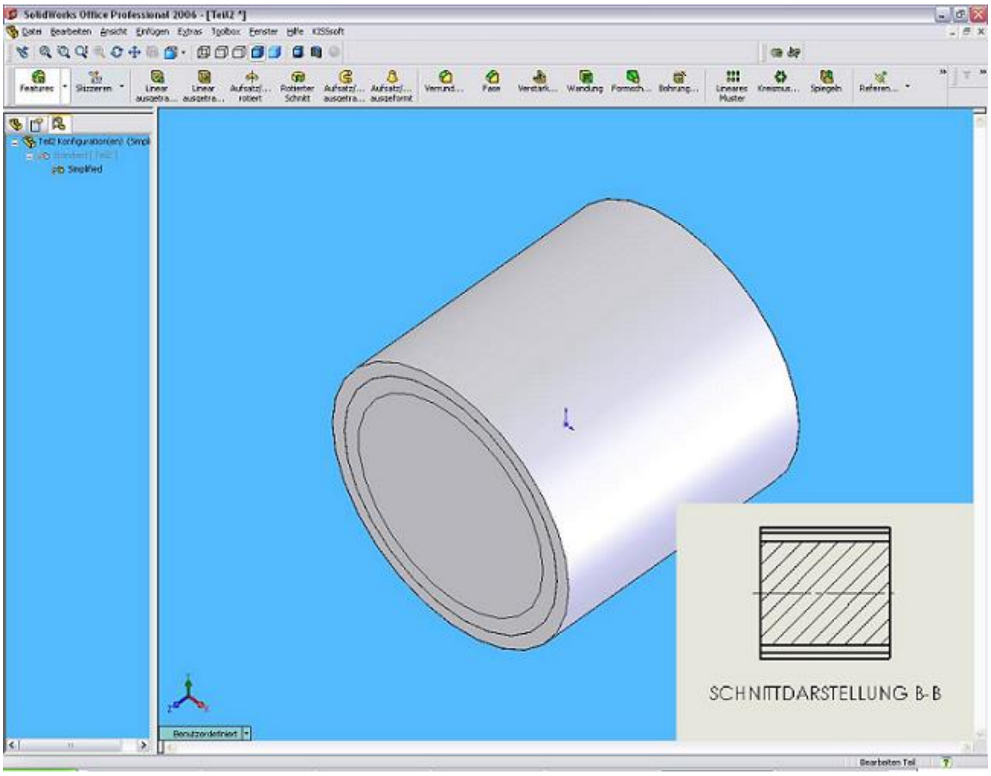
The **Open calculation file** menu option only works in the Part view. Procedure for opening a calculation file:

1. Open the part and select a tooth's **Cutout**.
2. Select the **Open calculation file** menu option.

This starts KISSsoft in each particular calculation module and opens the calculation file.

11.5.3.4 Simplified gear views

You can draw the gear in one of two different views. In the simplified view, you can create a section display view of the gear in the drawing which only contains the gear's edge contours and reference circle. Currently, the simplified view is only available for external teeth. The simplified view option is not the default setting.



To view a simplified display, open the **kiss.ini** file in the KISSsoft installation directory and change this entry:

SIMPLIFIEDPRESENTATIONNAME=**Name**

The name given in the **kiss.ini** file is also the name of the view.

11.6 3D interface to Solid Edge

Manufacturer: KISSsoft AG

This interface creates the direct integration between the Solid Edge 3D CAD system and KISSsoft. Use this to run all KISSsoft calculation modules from within Solid Edge. Cylindrical or bevel gears calculated in KISSsoft can be generated directly in SolidWorks as a 3D part (see chapter 11.2, Generation of 3D gears) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see chapter 11.3, Generating 3D shafts) directly in Solid Edge. You can start Solid Edge from within KISSsoft at the click of a button. The system opens a new part, and generates the appropriate part. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.
You can also add gear teeth to existing shafts (see chapter 11.6.2, Gear teeth if existing shaft data is present). In addition, gear manufacturing data in the 2D range (see chapter 11.6.4.2, Adding

manufacturing data) can automatically be inserted on the drawing as a text field, with the interface. The gear manufacturing data is attached to the relevant cutout (tooth space).

► **Note:**

The default template file (e.g. metric.prt) is used to generate gears. To use a user-specific template file, either define a variable called **USERPARTTEMPLATE** in the **[Solid Edge]** section, in the **kiss.ini** file, or overwrite the default template file and copy it to the user-specific template files folder on the default path. If a user-specific path has been set for templates in Solid Edge, this path is used. Otherwise, the default path for template files is used (e.g. **...\Solid Edge <version>\Template**).

11.6.1 Changing the parameter for generation

In Solid Edge, you can toggle between two settings for copying the tooth space (pattern). The possible modes are: SmartPattern and FastPattern. SmartPattern generates a more accurate tooth form. However, this takes quite some time and creates a very large gear file. FastPattern is a less accurate method, but takes less time and generates a smaller gear file. SmartPattern has always been used to generate gears up to now, since otherwise the gears cannot be created or represented correctly. In the **kiss.ini** (see chapter 2.6.9, Definitions in [SOLIDEDGE]) file in the KISSsoft installation directory, you can set **SMARTPATTERN=0**, to copy the tooth space in FastPattern mode.

11.6.2 Gear teeth if existing shaft data is present

Procedure for manufacturing gear teeth:

1. In Solid Edge, draw a plane on the surface on which you want to cut out the gear teeth.
2. Select this plane
3. In KISSsoft, select which gear (e.g. **Gear 1**) you want to generate on the cylinder.

Requirements:

- The cylinder diameter must already be the correct external diameter for the gear teeth before generation starts.
- For internal toothing, a hollow cylinder must already be modeled before the gear teeth can be cut out.

Toothing will be generated for inside and outside cylindrical gears with spur and helical teeth.

11.6.3 Integrating the KISSsoft Add-in (menu options in CAD)

You should register the Add-in when you install it. However, if this doesn't work and the KISSsoft menu is not displayed in Solid Edge, you must register the Add-in.

Go to the KISSsoft installation directory and select the **SolidEdge** sub-folder. In it, double-click on the **SolidEdgeRegister64.bat** file to register the interface.

If the KISSsoft Add-in is registered successfully, a message is displayed to confirm that this is the case.

To delete the registration, double-click on the SolidEdgeUnRegister.bat file in the KISSsoft installation directory. A message is then displayed to confirm that deregistration was successful.

Select **Tools > Add-Ins** and then **Add-In-Manager**. You can select/deselect the **KISSsoft** Add-in in the Add-In Manager.

The KISSsoft Add-in is displayed in the main menu. This integrates the KISSsoft menu options in Solid Edge. They are retained even after a restart.

The KISSsoft Add-in menu options are available in eight languages (German, English, French, Italian, Spanish, Russian, Portuguese and Chinese). They use the same language as was selected when KISSsoft was being installed. To set the language, open the **kiss.ini** file in the KISSsoft installation directory, click on **DISPLAYLANGUAGE**, and set the language you require (0 = German, 1 = English, 2 = French, 3= Italian, 4= Spanish, 5= Russian; 6= Portuguese, 7= Chinese). This language setting now also applies to your KISSsoft system.

► **Note:**

If the selected language uses Unicode fonts (e.g. Cyrillic for Russian), the Localization must be set to this language (a country with this language) in the operating system.

11.6.4 Add-in functions (calls)

11.6.4.1 Calling KISSsoft from the Add-in

Select the **KISSsoft** menu option to open all the KISSsoft calculation modules directly. The generation of a new/additional gear will then continue as described in the gear generation process (see chapter 11.2, Generation of 3D gears).

11.6.4.2 Adding manufacturing data

The **Add manufacturing data** menu option only works in the Part view. Procedure for adding a gear stamp to a drawing:

1. Open the part and select a tooth's **Cutout**.
2. Select the **Adding manufacturing data** menu option.

This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

11.6.5 Opening the calculation file for the created gear

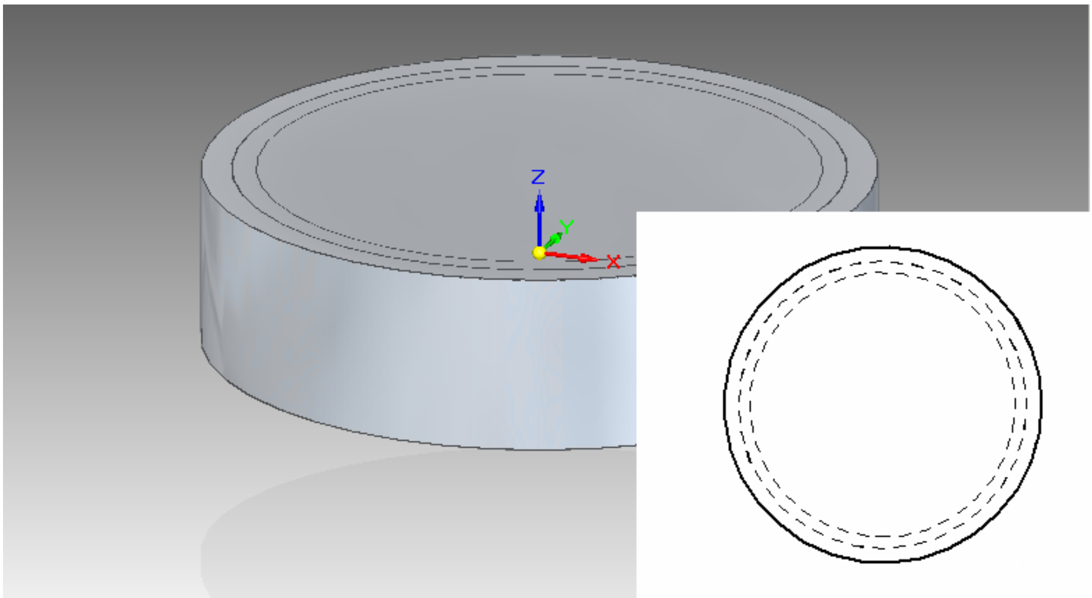
The **Open calculation file** menu option only works in the Part view. Procedure for opening a calculation file:

1. Open the part and select a tooth's **Cutout**.
2. Select the **Open calculation file** menu option.

This starts KISSsoft in each particular calculation module and opens the calculation file.

11.6.6 Simplified gear view

You can draw the gear in one of two different views. In the simplified view, you can create a section display view of the gear in the drawing which only contains the gear's edge contours and reference circle. Currently, the simplified view is only available for external teeth. The simplified view option is not the default setting.



To view a simplified display, open the **kiss.ini** file in the KISSsoft installation directory and change this entry:

SIMPLIFIEDPRESENTATION=**1**

11.7 3D interface to Autodesk Inventor

Manufacturer: KISSsoft AG

The interface between Inventor and KISSsoft is created by direct integration in the 3D CAD system. Use this to run all KISSsoft calculation modules from within Inventor. Cylindrical or bevel gears calculated in KISSsoft can be generated directly in Inventor as a 3D part (see chapter 11.2, Generation of 3D gears) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see chapter 11.3, Generating 3D shafts) directly in Inventor. From within KISSsoft, you can start Inventor with one click on a button. The system opens a new part, and generates the appropriate part. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.
You can also add gear teeth to existing shafts (see chapter 11.7.1, Gear teeth if existing shaft data is present). In addition, gear manufacturing data in the 2D range (see chapter 11.7.3.2, Adding manufacturing data) can automatically be inserted on the drawing as a table, with the interface. The gear manufacturing data is attached to the relevant cutout (tooth space).

11.7.1 Gear teeth if existing shaft data is present

Procedure for manufacturing gear teeth:

1. Select the required area
2. In KISSsoft, select which gear (e.g. **Gear 1**) you want to generate on the cylinder.

Requirements:

- The cylinder diameter must already be the correct external diameter for the gear teeth before generation starts.
- For internal toothing, a hollow cylinder must already be modeled before the gear teeth can be cut out.

Toothing will be generated for inside and outside cylindrical gears with spur and helical teeth.

11.7.2 Integrating the KISSsoft Add-in (menu options in CAD)

For the add-in (in Regfree mode) to be displayed in Autodesk Inventor, the **KISSsoftInventorAddin.dll** and **Autodesk.KISSsoftInventorAddin.Inventor.Addin** files must be copied into one of the following directories, which differ depending on what is to be displayed.

- All users, non-version-specific: Windows 8.1/10/11 - %ALLUSERSPROFILE%\Autodesk\Inventor Addins\

- All users, version-specific: Windows 8.1/10/11 - %ALLUSERSPROFILE%\Autodesk\Inventor 20xx\Addins\
- Per user, version-specific: Windows 8.1/10/11 - %APPDATA%\Autodesk\Inventor 20xx\Addins\
- Per user, non-version-specific: Windows 8.1/10/11 - %APPDATA%\Autodesk\ApplicationPlugins

The **KISSsoftCOM** server should be registered as part of the installation process. However, if this didn't happen, and the KISSsoft interface doesn't work, you must register the Add-in.

Go to the KISSsoft installation directory and select the **Inventor** subfolder. In it, double-click on the **InventorRegister64.bat** file to register the interface.

If the KISSsoft Add-in is registered successfully, a message is displayed to confirm that this is the case.

To delete the registration, double-click on the **InventorUnRegister64.bat** file in the KISSsoft installation directory. A message is then displayed to confirm that deregistration was successful.

The KISSsoft Add-in menu options are available in eight languages (German, English, French, Italian, Spanish, Russian, Portuguese and Chinese). They use the same language as was selected when KISSsoft was being installed. To set the language, open the **kiss.ini** file in the KISSsoft installation directory, click on **DISPLAYLANGUAGE**, and set the language you require (0 = German, 1 = English, 2 = French, 3= Italian, 4= Spanish, 5= Russian; 6= Portuguese, 7= Chinese). This language setting now also applies to your KISSsoft system.

This integrates the KISSsoft menu options in Inventor. The menu remains present, even after a restart, and does not need to be linked.

11.7.3 Add-in functions (calls)

11.7.3.1 Calling KISSsoft from the Add-in

Select the **KISSsoft** menu option to open all the KISSsoft calculation modules directly. The generation of a new/additional gear will then continue as described in the gear generation process (see chapter 11.2, Generation of 3D gears).

11.7.3.2 Adding manufacturing data

The **Add manufacturing data** menu option only works in the Part view. Procedure for adding a gear stamp to a drawing:

1. Open the part and select a tooth's **Cutout**.

2. Select the **Adding manufacturing data** menu option.

This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

11.7.4 Opening the calculation file for the created gear

The **Open calculation file** menu option only works in the Part view. Procedure for opening a calculation file:

1. Open the part and select a tooth's **Cutout**.
2. Select the **Open calculation file** menu option.

This starts KISSsoft in each particular calculation module and opens the calculation file.

11.8 3D interface to Siemens NX:

Manufacturer: KISSsoft AG

The interface between Siemens NX and KISSsoft creates the direct integration in the 3D CAD system. Use this to run all KISSsoft calculation modules directly from within Siemens NX. Cylindrical or bevel gears calculated in KISSsoft can be generated directly in NX as a 3D part (see chapter 11.2, Generation of 3D gears) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see chapter 11.3, Generating 3D shafts) directly in NX. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, worms, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.

If you are generating a new part, the **New** dialog opens first. In it, you can enter the name of the file in which the part should be generated. When you use **Teamcenter**, its dialog is displayed automatically so you can also generate or save the part in the Teamcenter environment.

You also have the option of adding toothing to existing shafts (see chapter 11.8.2.1, Gear teeth if existing shaft data is present). In addition, gear manufacturing data in the 2D range (see chapter 11.7.3.2, Adding manufacturing data) can automatically be inserted on the drawing as a table, with the interface. The gear manufacturing data is attached to the relevant cutout (tooth space).

11.8.1 Integrating the KISSsoft Add-in (menu options in CAD)

First, copy the supplied folder, e.g. **NX1847**, with its **startup** subfolder, to a location that can be accessed by users at any time.

The **kSoftNX_d.men** file contains the definition for the KISSsoft Add-in menu options. This file has different names to reflect which language has been selected. For example, the **_e** in the file name stands for **English**. The other language codes are _d for **German**, _f for **French**, _i: for **Italian**, _s: for **Spanish**, _r: for **Russian**, _p: for **Portuguese**, and _c: for **Chinese**. You can copy the file for the language you require to the **startup** folder. The KISSsoft menu will then be displayed in this language.

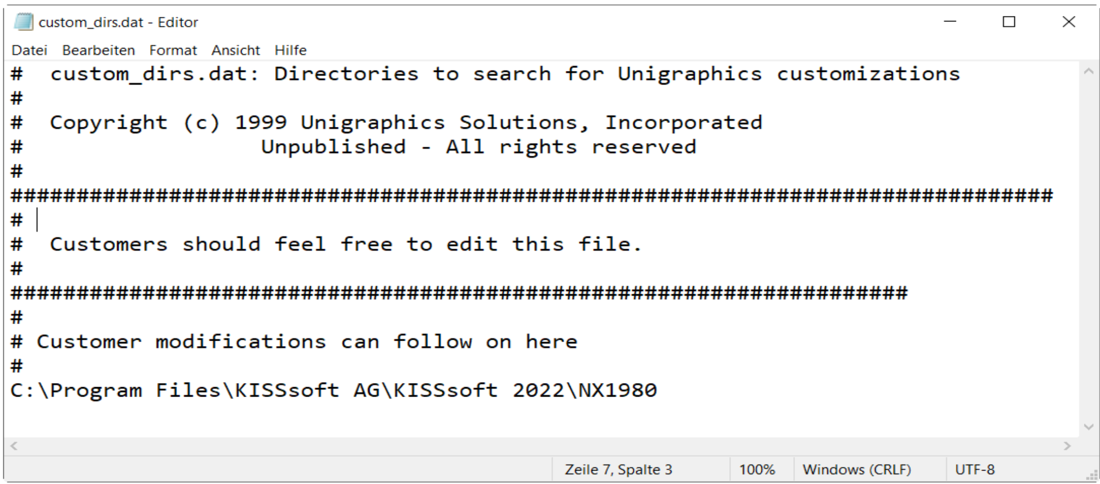
► **Note:**

If the selected language uses Unicode fonts (e.g. Cyrillic for Russian), the Localization must be set to this language (a country with this language) in the operating system.

KISSsoft is also available as a ribbon menu. The English menu with **_e** is embedded as the default setting in the **startup** subfolder. If you want to change the language in which the menu is displayed, delete all the files in the **startup** folder whose name ends with **_e**. The ...**\NX1847** subfolder contains a subfolder for every available language (e.g. **kSoftNXRibbon_e** for English). You can copy the entire contents of the folder that has the language you require to the **startup** subfolder. The menu will then be displayed in this language.

The **kSoftNX1847.dll** file (for example), which contains the links and commands for the menu options, is also stored in this folder.

You must input the path for the previously copied folder, for example, **NX1847**, in the **NX1847\menu\custom_dirs.dat** file, in the NX directory, so that the NX system knows where the files it is to use are stored.



The **KISSsoftCOM** server should be registered as part of the installation process. However, if this didn't happen, and the KISSsoft interface doesn't work, you must register the Add-in.

Go to the KISSsoft installation directory and select the **NX1847** subfolder. In it, double-click on the **NX_Register64.bat** file to register the interface.

If the KISSsoft Add-in is registered successfully, a message is displayed to confirm that this is the case.

To delete the registration, double-click on the **NXUnRegister.bat** file in the KISSsoft installation directory. A message is then displayed to confirm that deregistration was successful.

To ensure the KISSsoft icons are displayed next to the menu options, you must also set a system variable with the path, to tell the program where the KISSsoft icons can be found.

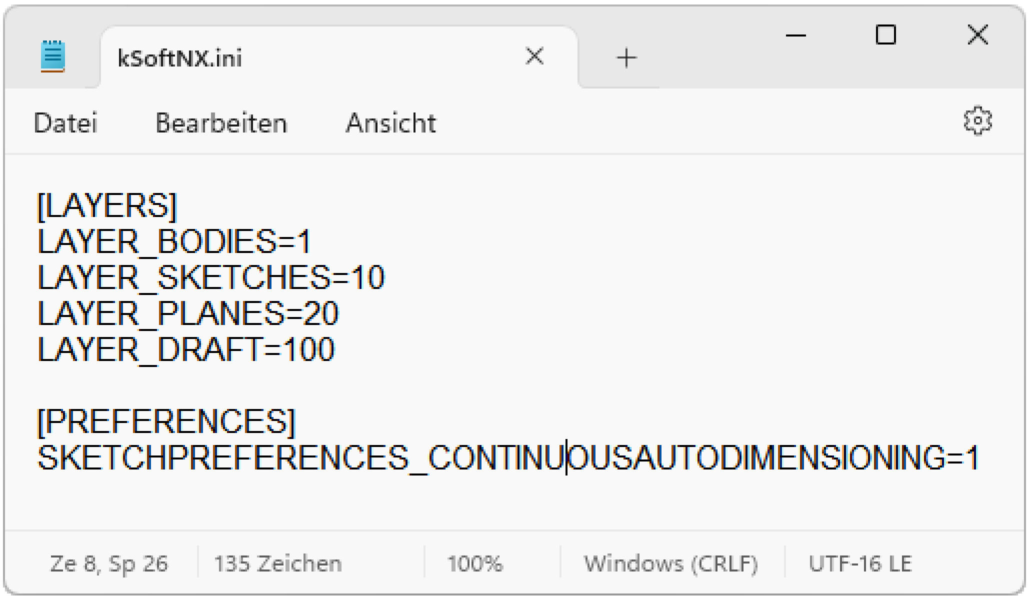
Example: Set a system variable and this value as the path

KSOFT_ICONS

C:\Program Files(x86)\KISSsoft<version>The **startup** folder also contains the **kSoftNX.ini** file, in which you can change the layers of the bodies, sketches, planes and drawings.

You can use the **SKETCHPREFERENCES_CONTINOUSAUTODIMENSIONING** option to define whether dimensions shall be created automatically after modifying a sketch in design applications. If this option is disabled, performance is improved for larger sketches.

In NX, this option can be found under: File>Utilities>Customer Defaults>Sketch>Inferred Constraints and Dimensions (Legacy)>Continuous Auto Dimensioning in Design Applications (Legacy).



11.8.2 Calling KISSsoft from the Add-in

Select the **KISSsoft** menu option to open all the KISSsoft calculation modules directly. By doing this you can perform calculations in KISSsoft quickly and easily during the design process. The **NX1847** (etc.) menu options are inactive while KISSsoft is open. In order to reactivate the CAD program, you must close KISSsoft.

11.8.2.1 Gear teeth if existing shaft data is present

Requirements:

- The cylinder diameter must already be the correct external diameter for the gear teeth before generation starts.
- For internal toothing, a hollow cylinder must already be modeled before the gear teeth can be cut out.

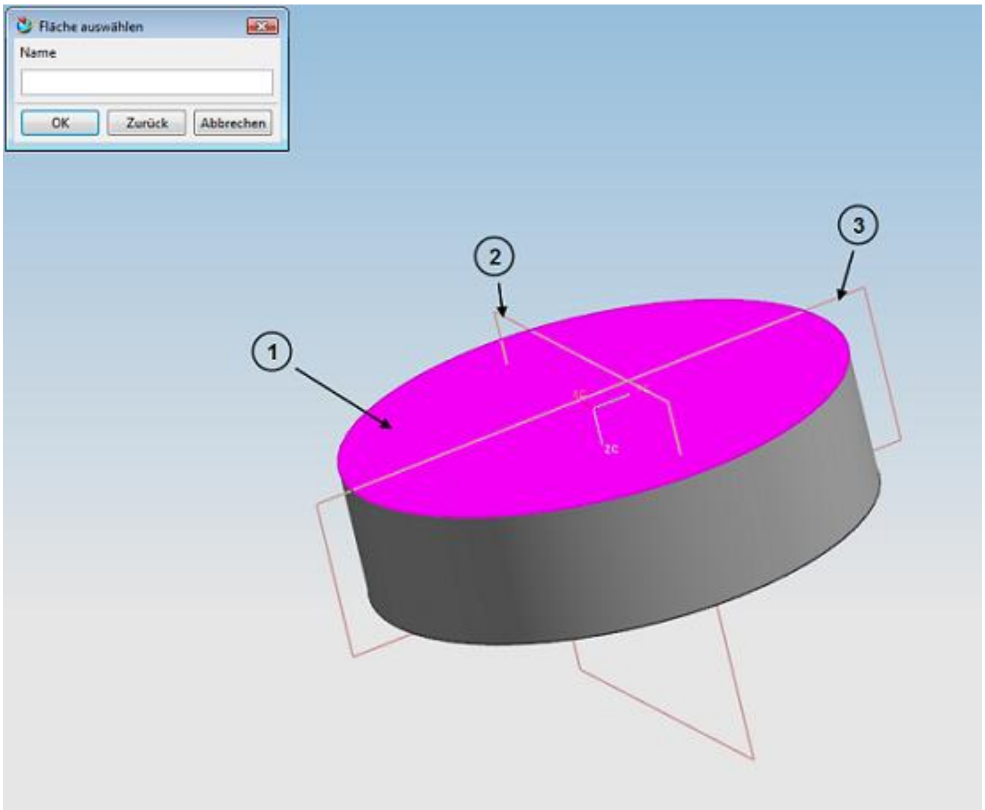
For example, select the cylindrical gear pair calculation in the KISSsoft menu. The procedure for generating the gear (see chapter 11.2, Generation of 3D gears) is identical to the procedure for creating a new one.

If a part is already opened in Siemens NX, a window with 3 selection buttons is displayed:

1. In a new part
2. Available part, absolute positioning
3. Available part, relative positioning

Explanations of the individual selection options and their use:

1. Select **In a new part** to generate the entire gear.
2. Select **Available part, absolute positioning** to select only one side surface on which the gear teeth are to be cut. The generation process now generates fixed levels on which the gear teeth will be positioned.
3. Select **Available part, relative positioning** to select one side surface and two levels (which cut into the side surface), one after the other. The toothing can therefore be positioned at relative levels (DATUM PLANE) and is not dependent on the absolute zero point. This positioning is primarily required for the methodical working method defined in the Teamcenter "Master Model concept".



The generation of toothing on existing cylinders is performed on both inside and outside cylindrical gears with straight or helical toothing.

11.8.2.2 Adding manufacturing data to the drawing

You can select the **Add manufacturing data** menu option to insert a gear stamp of the current gear in a drawing.

- Teamcenter: If you are working in accordance with the **Master Model concept**, the features of the master part are displayed automatically in the non-master drawing when you select **Add manufacturing data**.

After you select this menu option, another window opens, in which you can select the object you require. There, make these selections:

- Straight-toothed cylindrical gears: INSTANCE[0](4)TOOTH(4)
- Helical cylindrical gears/worms/Straight-toothed bevel gears: TOOTH

Click on OK to open a new drawing. The following window opens, and displays the Drawing view. Click with the mouse click to align the upper left corner of the table with the manufacturing data on the drawing. If you want to insert the data into an existing drawing sheet, you must select the tooth space in the Drawing view once the required drawing sheet is opened. You can select the tooth space in the next window that is displayed. You are then prompted to confirm that you want to transfer the manufacturing data to the current drawing sheet.

Click on OK to position the manufacturing data on the drawing (by clicking with the mouse). Click on **Cancel** to display a new drawing sheet into which you can insert the manufacturing data.

11.8.2.3 Opening the calculation file

Select the **Open calculation file** menu option to start KISSsoft. This loads the gear teeth calculation file and the information is saved directly to the gear teeth feature (tooth space). After you select this menu option, a window in which you select the required object is displayed:

- Straight-toothed cylindrical gears: INSTANCE[0](4)TOOTH(4)
- Helical cylindrical gears/worms/Straight-toothed bevel gears: TOOTH

When you click on the OK button, KISSsoft opens in the appropriate module with a loaded gear teeth calculation file.

11.9 3D interface to Creo Parametric (ProEngineer)

Manufacturer: Applisoft Europe (Italy)

Cylindrical or bevel gears calculated in KISSsoft can be generated directly in Creo Parametric as a 3D part (see chapter 11.2, Generation of 3D gears) with a real tooth form. You can create cylindrical gears with straight or helical teeth, which are external or internal, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1.

In addition to the part, the system opens a drawing in which the gear manufacturing data appear in a table. Open the CAD system before you start generating a part with the 3D interface to Creo Parametric.

In the interface to Creo Parametric, you can enter additional variables in the files for the particular gear (e.g. Z10GEAR1CAD.rpt) in the **CAD** directory. These additional variables will later be defined as parameters and saved in Creo Parametric.
The parameters used for the generating process are already defined in Creo Parametric and can no longer be used. Predefined parameters:

- pz, z, b, da, d, df, di, elica, USUnit

If you want to create a model of a part in US customary units (not metric), open the **kiss.ini** file (see chapter 2.6.13, Definitions in [PROENGINEER]) and set the **USCUSTOMARYUNITS** parameter to 1.

You can also change an existing toothing without actually affecting the part (see chapter 11.9.3, Modifying the selected 3D model). You can also cut gear teeth on an existing shaft (see chapter 11.9.2, Cutting gear teeth on an existing shaft).

A new dialog opens as soon as you start the generating process. This dialog has these three options:

1. Generate gear in new file
2. Generate gear on shaft
3. Exit

If you select **Generate gear in new file**, the gear is generated in a new part file.

► **Note:**

If you want to prevent the selection menu or message from appearing, you can specify this in (see chapter 11.9.5, Changing base settings in the interface).

11.9.1 Integrating the KISSsoft Add-in

The **KISSsoftCOM** server should be registered as part of the installation process. However, if this didn't happen, and the KISSsoft interface doesn't work, you must register the Add-in.

Go to the KISSsoft installation directory and select the **ProEngineer** subfolder. In it, double-click on the **ProECreoRegister.bat** file to register the interface.

If the KISSsoftCOM server is registered successfully, a message is displayed to confirm that this is the case.

To delete the registration, double-click on the **ProEUnRegister.bat** file in the KISSsoft installation directory. A message is then displayed to confirm that deregistration was successful.

Use one of the 3 methods described below to ensure the KISSsoft menu is present every time you start Creo Parametric. For Creo 7 and more recent versions, add this line for all 3 variants, in the 'config.pro' file (in Creo):

```
open_protk_unsigned_apps always
```

Variant 1:

Copy the relevant **Protk_EditGear_Creo.dat** file (for the ProE/Creo version involved) to the Creo ...**\Common Files\text** subdirectory (for Creo, copy the file to the**...\text**subdirectory).
Then, rename the file to **Protk.dat**. If this variant is used, the user can change their Creo start directory and the KISSsoft menu always starts along with it.
However, if a different **Protk.dat** file is already present, insert the lines from **Protk_EditGear_Creo.dat** file into the **Protk.dat** file.

For Creo 9 the **Protk_EditGear_Creo9.dat** is to be used.

Variant 2:

Copy the **Protk_EditGear_Creo.dat** file into the initial Creo working directory and rename it **Protk.dat**. For Creo 9 use **Protk_EditGear_Creo9.dat** .
In this case, the **Protk.dat** file is to be copied into the start directory. The path is displayed in the properties of the parametric.exe file.

Variant 3:

The user must add the following lines to their **config.pro** file (in Creo). In other words, the user defines their own path:
protkdat C:\Program Files\KISSsoft<version>
The **Protk_EditGear_….dat** file in the **ProEngineer** installation subdirectory must be renamed to **Protk.dat** to make it work.

Description of the content of the Protk.dat file:

```
NAME EditGear

EXEC_PATH C:\Program Files\KISSsoft AG\KISSsoft
XXXX\ProEngineer\EditGear\bin_nt\Gear...

TEXT_PATH C:\Program Files\KISSsoft AG\KISSsoft
XXXX\ProEngineer\EditGear\text.GB

STARTUP DLL

ALLOW_STOP TRUE

UNICODE_ENCODING FALSE

END
```

EXEC_PATH and **TEXT_PATH** must be the absolute path of the installation.

STARTUP DLL and **UNICODE_ENCODING FALSE** are predefined (do not change them).

Use **ALLOW_STOP TRUE** to stop the Creo Parametric program (**Tools->Auxiliary Application->Stop**).

You can delete this line in the **Protk.dat** file to prevent users from stopping the interface.

NAME EditGear and **END** must be present, although you can change the **EditGear** name if required.

11.9.2 Cutting gear teeth on an existing shaft

If you run KISSsoft from the 3D export, this menu with the following 3 selection options is displayed:

1. Generate gear in new file
2. Generate gear on shaft
3. Exit

To modify an existing model:

1. Select **Generate gear on shaft**
2. In Creo Parametric, open the shaft on which you want to cut the gear teeth.
3. Set a new coordinates system to describe the point at which the gear teeth are to be cut. Select the **GearShaft** menu option in the KISSsoft menu in Creo Parametric.
4. This opens another menu in which you can specify whether the gear teeth are to be cut across the entire width or only across part of the shaft.
5. After you have made your selection, select the coordinates system in which the gear teeth are to be inserted. The coordinates system you select must have a z-axis that is equal to the shaft axis.
6. The gear teeth are then cut on the shaft.

11.9.3 Modifying the selected 3D model

When you export a tooth form from KISSsoft, the model in Creo Parametric is generated in a new part.

To modify an existing model:

1. Import the model you want to modify into Creo Parametric, or use the current part.
2. In the KISSsoft menu, select **Edit** and then **YES**. This imports the current gear teeth.
3. Then, select **Open calculation file**. This menu then imports the appropriate gear teeth data to KISSsoft.

KISSsoft can then regenerate the modified gear teeth. This modifies the current gear teeth.

11.9.4 Modifying the teeth on an existing shaft

Use this procedure to modify the gear teeth generated with the KISSsoft interface on an existing shaft on an existing model:

1. Import the model you want to modify into Creo Parametric, or use the current part.
2. Select **Edit Gear On Shaft**, so you can select which gear teeth data is to be modified. KISSsoft then opens with the data that was stored when the toothing element was generated.
3. Then, modify and recalculate the gear teeth in KISSsoft. You can then restart the 3D export of the relevant gear teeth. Then, click on the cross in the top right-hand corner of the KISSsoft window to close it. You are then prompted to confirm whether to save the temporary change.
4. Click on **Yes** to modify the model. If you click on **No**, the model remains unchanged.

11.9.5 Changing base settings in the interface

There are a number of ways in which you can set up the interface by setting environment variables:

KISS_PROE_INTERFACE_NO_MENU = YES

For users who cannot set up a connection to Creo Parametric (using PRO_COMM_MSG.exe).

Set this environment variable to **YES** to stop the interface trying to use this process to run the connection. This also stops the warning messages, stating that no connection can be created, from appearing.

KISS_PROE_INTERFACE_NO_MENU = NO

If you set this environment variable to **NO**, a warning is displayed if no direct connection to Creo Parametric can be established.

The message describes how to generate the gear despite this.

KISS_PROE_INTERFACE_CLASSIC = YES

The additional dialog in which you can select either **Generate gear in a new file** or **Generate gear on shaft** is then no longer displayed.

KISS_PROE_INTERFACE__CLASSIC = NO

A dialog in which you can select either **Generate gear in a new file** or **Generate gear on shaft** is displayed.

If no environment variables are set, both these values are set to **NO**.

11.9.6 Saving the files to the PTC Windchill working directory

To save the generated interface files to the PTC Windchill working directory in the PLM system, change the **WORK_IN_CURRENT_FOLDER** parameter setting in the **SETUP.txt** file from **NO** to **YES**. The **SETUP.txt** file is located in the **...\ProEngineer\EditGear\SETUP.txt** sub-folder, in the KISSsoft installation.

If you have installed Creo Parametric on the server and started it from the client, temporary files are written to the server, not to the client. To prevent this, set the variables listed below in the **Setup.txt** file with this path.

APSF_WORK_DIR C:\temp

11.10 3D interface to CATIA

Manufacturer: SWMS (DE)

Cylindrical or bevel gears calculated in KISSsoft can be generated directly in CATIA as a 3D part (see chapter 11.2, Generation of 3D gears) with a real tooth form. You can create cylindrical gears with straight or helical teeth, which are external or internal, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1.
You can also add gear teeth to existing shafts.
You will find a more detailed description of the interface in a *.pdf file in the **CATIA** folder in the KISSsoft installation directory.

12 Answers to Frequently Asked Questions

12.1 Changing the output of angles in reports

Is it possible to output angles (in calculations) in the KISSsoft angle report as degree values as well as decimal numbers?

Current form: ##.#### °

Required form: ## ° ## ’ ## ’’

To do this, change the report template (*.rpt) accordingly. Read the notes in the manual about report templates (see chapter 8.5, Report templates) before you do this. The calculation is then performed in the report. A helix angle is used to show this method:

Previous format, as a decimal number:

Helix angle (grd) %11.4f {Grad(ZS.Geo.beta)}=>

Current format, now as a degree value:

Helix angle (grd) %i° %i' %i" {Grad(ZS.Geo.beta)} {(Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60} {((Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60-int((Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60))*60}

12.2 Inputting materials for gear calculations in the database

When comparing the materials used for gear teeth in a particular company, it became evident that not all the required materials were present in the database provided by KISSsoft.

In particular, the following key values, necessary for gear calculation, are missing: include σ_{Flim}/S_{at}, σ_{Hlim}/S_{ac}, R_{zF}, R_{zH} and B_M.

When you redefine materials and their properties, you must compare them with similar materials in our materials database.

First of all, define the basic data for a material in the database. Then, define the gear-specific data for this base material.

Then, calculate the values of $\sigma_{Flim/Sat}$, $\sigma_{Hlim/Sac}$ depending on the hardness values, as described in ISO 6336-5.

To do this, you can either use the relevant material diagram, the conversion function for inputting your own materials (see chapter 15.1.12.1, Materials) or formulae from ISO. The S_{at}, S_{ac} values are converted on the basis of σ_{Flim}, σ_{Hlim}.

A default value is used if no value is input for the thermal contact coefficient BM.

For total heights, specify average values with R_{zF} 10µm and R_{zH} 3µm. You will find more detailed information about this in ISO 6336-2.

ISO 6336, Part 2 provides more information about the influence of the total height on the calculation of flank load capacity when an additional material hardening factor, Z_w, has been introduced.

12.3 How can I test the software?

A demo version of the software is available at (see chapter 1.1, Basic installation). Although the demo version does not have an expiration date, its functionality is limited so that, for example, you cannot change and store material data. The demo version is designed to give you an initial impression of the software. For a detailed trial, request a test version (see chapter 1.3.1, Test version). The test version runs for 30 days, is free of charge and is the same as the full version (without third party programs).

12.4 What licenses are available?

Individual user licenses and network licenses are available for both KISSsoft and KISSsys. A network license enables the software to be used at more than one workplace.

However, network licenses are not available for some of the third party products, for example, some CAD interfaces.

12.5 Add your own texts in the results window

To enable this, define a new file in the KISSsoft installation directory in "…\ext\.rpt\". This file must then be named using this convention: "Modulname + result.RPT" (e.g. for a cylindrical gear pair Z012result.RPT).

Then define the new parameters or values that are to be added. These values are then also displayed at the end of the **Results** window.

12.6 Restoring a previous stage in the calculation

Select File > Restore... (acts like the Undo function) to retrieve an earlier stage of the current calculation file. For this reason, every calculation run stores the current stage as a restore point. The list of restore points is deleted when you open a different file.

II

KISSsoft System Module

13 KISSsoft System Module - the system module in KISSsoft

The system module in KISSsoft is a powerful tool that allows you to model, analyze and evaluate complex gearboxes and transmissions at system level.

13.1 User interface

The main graphical user interface areas are described in the following sub-chapters.

13.1.1 Shaft view

The Shaft view is on the top left-hand side in the standard view. All the shaft calculations (shaft groups) for a modeled gearbox are listed here in the form of a tree structure. In the same way as in the shaft calculation module, every shaft calculation can consist of more than one shaft. Every shaft can have machine elements such as couplings, gears and bearings.

13.1.2 Element View

The Elements view provides more information about the relationships between the individual shaft groups. When filtered, it only displays specific elements and their connections. These selection options are available here:

Gears: The gear calculations that are to be taken into account in the gear units are displayed here. The gear calculations include the associated gears, which you can simply drag and drop into the element from the Shaft view. You can use the mouse to connect the center points of the gears, in the Sketcher, to achieve the same thing.

Shaft-Hub Connections: All connections, such as interference fits, splines, keys, etc. are displayed in this area. One of these calculations can be added, and a gear, coupling or switching element can be dropped in it. In the Sketcher, use the context menu to add shaft-hub connections from any of these types of elements. Data, such as the diameter, forces and materials can then be passed automatically to the corresponding calculation.

Planet carrier: All planet carrier elements are displayed in this window. Click on the associated planet gear shaft and drag it from the Shaft view into this element. The same link can also be created in the Sketcher. You can use the mouse in the same way, in the Sketcher, to connect the planet carrier shaft node, which was previously defined as the carrier, to the planet shaft axis. This gives the system the information that a planetary gear stage is involved and that the added shaft group is epicyclic.

Bearings: An overview of all bearings is displayed here. If the bearing is connected to the housing on one side (inner ring or outer ring), only the shaft that has contact on the other side of the bearing is listed in the bearing element. If both the inner ring and the outer ring of the bearing are in contact with a shaft, both shafts are listed. The following conventions apply in this case: The upper shaft represents the internal or left shaft of the bearing and the lower shaft represents the outer or right shaft of the bearing.

Switchables: Select this option to display all the switching elements used in the model. Shafts that are connected to each other are listed in the switching elements. To add a shaft to an existing connection, click on a shaft in the Shaft view and drag it to one of these elements. The same link can also be created in the Sketcher. To do this, use the mouse to connect the switching element node with the appropriate second shaft.

Boundary conditions: All the model's kinematic boundary conditions are displayed in this area. Any shaft element that can transmit a torque, such as a coupling or a centrical load, can be dragged and dropped into a boundary. It can then be used as a power input or output, with given or calculated torque, power and/or speed.

Power flow: For each power split element, the user can drag and drop two gears or two couplings that are engaged in a gear contact or coupling connection. Each of these elements represents a power flow path. This means the power flow splits and then merges again at a different point in the gear unit.

Power loss: In this tab, you can define losses such as bearing losses, gear churning losses, gear meshing losses, planet carrier losses or seal losses in the model. Simply drag and drop the corresponding element or pair of elements into a power loss element. This element can then be specified as a torque, power or efficiency. It can also be calculated from the submodules.

Coupling connection: Two couplings can be dragged and dropped into a coupling connection. This can represent a physical connection such as a Cardan joint, hydrostatic coupling or others. Coupling connections can also be used to represent a virtual connection in which a speed, torque or power ratio between two boundaries can be specified.

Force transfer: Any support can be dragged and dropped as first element into a force transfer element. As second element, a centrical load or carrier element must be dropped. This connection is then used to transfer the forces and moments from the support located in one shaft calculation into the centrical load located in another shaft calculation. This is particularly useful for a pin representation on a carrier shaft.

Housing: A housing element can be added here. This element can be used to import a STEP file into the model or draw a rough box or cylinder. It can also be used for the housing deformation calculation.

13.1.3 Element Box

All the elements that can be used in the system are displayed in this window. These elements are grouped according to category: shafts, gears and couplings, forces and carriers, bearings and switchables, power flow, special elements such as coupling connection, force transfer and housing, and finally the gear and belt/chain calculation submodules. Click on an element to add it to the Shaft view and/or Element view (depending on what type of element it is). Use the icons in the Element Box to create a model from the beginning or add elements to an existing model.

13.1.4 Group view

All component relationships can be displayed in one view. A shaft calculation can therefore be added to this view and, for example, all gear calculations related to the gears mounted on its shafts will then also be added to the same view.

13.1.5 Sketcher

Use the Sketcher to "draw" the gear unit's kinematics with the mouse. All the definitions you make in the Sketcher are also displayed simultaneously in the Shaft view and Element view. As a result, you can create or modify the model directly in the Sketcher. You can switch between the windows at any time.

13.1.6 3D Viewer

The modeled gearbox is displayed in 3D in the 3D Viewer. The gearbox model can be moved in space, animated or recorded as a video. Use this view to check how shaft groups are positioned in space. Select **File > Export... > Complete 3D model** to save the contents of the 3D viewer as a STEP, Parasolid text (X_T) or Parasolid binary (X_B) file.

13.1.7 Kinematics

You can define the boundary conditions and power splits in this tab. You can then specify the speeds and torque/power values in the model's boundary conditions. The conditions defined here should ensure that the system is kinematically defined. There are no other functions for calculating kinematics. The system checks automatically after every entry you make, to see if a solution can be found. If a solution with the predefined conditions is found, all the values are updated accordingly. You can add any number of boundary condition sets with given torques and speeds to the model. Combined with the switchable sets, if there are any in the model, this represents a so called operating mode. In the control panel, you can then check if those combinations are kinematically well defined and select the operating mode to be used to display the current kinematics of the model.

13.1.8 Switching matrix

In this tab, all speeds can be defined and the status of all switching elements can be set for each speed. A Sizing button is also available to provide a unique shifting speed, with particular closed switching elements, for each potential power flow path. Each speed can also be renamed. The engaged gear can be changed on the control panel. The power flow is then updated directly in the Sketcher.

13.1.9 Ratio

The ratios for individual stages can be defined in this tab. In this case, enter the number of teeth on the meshing gears to determine the ratio. Alternatively, you can predefine the stage ratio and the number of teeth on a gear. In this case, the number of teeth on the other gear is calculated by the system.

If you predefine the number of teeth: The number of teeth on a gear with external toothing must be a positive value and the number of teeth on a gear with internal toothing must be a negative value.

If you predefine the ratio: If two gears with external toothing are meshing, enter the ratio as a negative value, because the gears rotate in different directions (have a different "sense ofrotation"). However, if a gear with external toothing meshes with a gear with internal toothing, enter the ratio as a positive value, because the gears rotate in the same direction (have the same sense of rotation).

13.1.10 Power loss

It is possible to select for the different types of losses (gears, bearings, seals) whether all losses or only a selection of losses should be calculated, or whether they should be entered by the user. In the case of gear losses settings can be further distinguished into load-dependent and load-independent losses. Based on these settings, the calculation of the losses is carried out. For gears all currently available calculation methods are incomplete. These gaps are filled with adapted methods.

In this tab, correction factors can also be defined that are used to adjust the calculated values to experimental results.

13.1.11 System data

This table can be used to connect several parameters that are common to all submodules in one system table. For example, a global oil level can then be defined in the system and automatically transferred to all shaft calculations.

13.1.12 Load spectrum

A system load spectrum can be created, in which a user-defined number of parameters can be varied. Boundary conditions for loads, losses, gear factors, shaft temperatures, and many other values can be defined as varying parameters in the spectrum. Different application cases can then be created with multiple variations of requested service lives. A load spectrum can also be imported directly. A simple kinematic load spectrum analysis or the entire strength analysis can then be performed. An evaluation of each individual load bin in the load spectrum can also be calculated. The main report shows the results of the corresponding calculation.

13.1.13 Variants

Several variants of submodule calculations can be defined for all shafts and gears, within a single table. A global variant can then be selected to import all submodule calculation files into the model. The results can then be compared with another variant.

13.1.14 Module specific settings

The settings for the elements to be used in the model are made in the "Module specific settings".

You can change the names of the elements that are to be used in the model in the **Naming of the elements** tab. The term <autoInc> after each element name means that a number is added automatically after that name. This number represents the number of added elements of the same sort.

In the **Default dimensions** tab, you can preset the dimensions for the toothing geometry and bearing geometry. When these elements are used in the model, the gear unit is displayed with these predefined sizes in the initial phase.

13.1.15 Modeling assistant

You will find predefined gear stages in this window. You can select different cylindrical gear stages, bevel gear stages and planetary stages. In addition to these stages, individual or paired shifting gear stages are present, which are primarily used for gearboxes with shifting elements. The size of the components is directly linked to the default sizes defined in the module specific settings.

13.1.16 System rough sizing

A rough sizing can be performed on all gears and shafts present in the model. Several parameters can be varied to get the most suitable sizing solution.

13.2 Modeling

Different methods for creating a gear unit model are provided in the system module. All these methods can be used in each phase in the model. Each step can be performed with different methods. You can use a different method from one step to the next at any time. The possibilities are described below.

13.2.1 Creating a model with the Element Box

You can use the icons in the Element Box to create the model in Shaft view. Click on an icon in the Element Box to insert it in the model tree structure, in Shaft view. The first element to start with is the shaft calculation. A shaft element is always created automatically when the shaft calculation is added. If a shaft calculation has several shafts with the same axis, you can insert them by clicking on the shaft icon in the Element Box. You can click on the other appropriate icons in the Element Box to insert gears, bearings or other connection elements such as couplings, synchronizers (switching elements) or the carrier element in these shaft elements. To define a new shaft axis, you can integrate a different shaft calculation by clicking on the relevant element in Shaft view.

After the shaft calculations with the shafts and their elements are finished, you can define the gear stages as transmissions. To do so, click the gear calculation elements in the Element Box. Each time you click on an element, the selected calculation element is listed in the corresponding selection, in the Element view.

Now you need to define the references between the gears and the calculations. To do so, click on the gears in Shaft view to select them, and then drag them into the associated gear calculation. Drag over two, three or four gears, depending on the calculation type (two-gear, three-gear, planet or four-gear calculation). Here, the convention that the sequence of gear elements from top to bottom matches the sequence of gears in the corresponding KISSsoft calculations applies.

Once you have linked all gear calculations with the gears, you can define the system's input and output conditions.

To define a boundary condition in the model, simply click on the appropriate element in the Element Box. To create the link with the couplings: in Shaft view, left-click on a coupling element, that is to be referenced, to select it, and then drag it over to the boundary condition element.

The model is displayed as a schematic sketch in the **Sketcher** tab at the same time. You can see how the shaft groups are arranged in the **3D Viewer** tab. You can define the boundary conditions in detail in the **Kinematics** tab.

13.2.2 Creating a model with the Sketcher

You can create the model in the Sketcher, using the mouse as a "drawing" tool, from the very start, or later in the modeling process. When you are "drawing" with the mouse, note these points:

To start and end a shaft, perform a double-click. To insert nodes on the shafts, click once. When you have finished drawing the shaft, a dashed blue line is displayed on the end of the mouse pointer. Click on the required grid line to set the position of the associated shaft axis. You can also right-click on the nodes on a shaft to select them, and then convert them into the appropriate element (bearing, gear, coupling, etc.) via the selection list. If more nodes are needed at a later time, you can insert them by double-clicking on a grid point of the shaft. You can delete a node at any time by right-clicking on it and then selecting "Delete".

You can double-click to begin defining another shaft and create it as described above. In the final step, click the left mouse button to assign the shaft to the axis of a previously defined shaft. The shaft can also be assigned to its own new axis in the drawing level. The model in the Sketcher is built up simultaneously in the Shaft view. When you create a new shaft axis in the Sketcher, a new shaft calculation is simultaneously generated in Shaft view.

Once you have drawn the shaft axes with the shafts and their elements, you can now link the gears with each other. To do so, drag and drop the gear element center points to link them with each other. Synchronizers and bearings are initially defined as individual elements on a shaft. However, these connection elements are often completely defined between two shafts. To assign these elements to a new shaft, use the mouse to drag and drop the associated node. Bearings that are not assigned to any other shaft are handled as bearings that have an outer or inner ring that is in contact with the housing. You can also assign the carrier icon to the axis on your planet shaft(s) by dragging and dropping the carrier icon.

13.2.3 Creating a model with groups

Besides creating a model using the Element Box and the Sketcher, you can use the Groups Assistant to generate individual finished sub-assemblies. You can then merge the individual shafts in these assemblies by dragging and dropping them in the model tree structure. This creates a single gearbox concept from the individual groups.

13.3 Special Calculations

13.3.1 Thermal rating

You can use the thermal rating to calculate the heat level in a particular gear unit. Several different methods have been implemented to enable you to select how the calculation is to be performed, according to the ISO/TR 14179 standard, Part 1 and Part 2. Thermal analysis can be split into two sections: power loss and heat dissipation. An external cooler can also be taken into account. Power loss and heat dissipation can be split up into several sections, to enable the effect of all the individual gear unit components to be taken into consideration. Power loss can be split into two main components: load-dependent and non-load-dependent. Both types of loss are usually present when a gear unit is running. Power loss can also be subdivided into gear unit elements such as gears, bearings and seals. Meshing and churning losses are taken into account for gears whereas rolling and sliding friction are taken into account for bearings and seal friction is taken into account for seals. In some cases, the results must be treated with caution, because the calculation methods used may not fully support the geometry type. Heat dissipation can be categorized as heat dissipation through the housing, foundation and rotating parts (input/output shafts and couplings) and cooling oil flow. A gear unit's total efficiency and total heat dissipation capacity for a given lubricant temperature, cooler power and input power can be calculated with ease. You can also specify two of these three entries and calculate the optimum value for the third parameter, which is the value with which you achieve the best heat level for the gear unit. In other words, this is the value at which the dissipated heat equals the heat generated through the power loss. Part 1 and Part 2 of the standard differ in the way the calculation values are input. The main benefit of Part 1 is that it enables you to enter your own heat transfer coefficients for heat dissipation through the housing (if it has a very specific shape), whereas, in Part 2, this coefficient is calculated using an approximation of the shape of the housing. The main benefit of this part is that it also takes fins, foundations and rotating parts into consideration when calculating heat dissipation.

13.3.2 Modal analysis

This special calculation has been designed to calculate the eigenfrequencies and eigenmodes of a complete shaft system, including the effect of gear connections between shafts.

To start running this calculation, click on **Calculation> Modal analysis** in the menu. You must define the number of eigenfrequencies to be calculated, and specify whether only torsional or all vibration types are to be included, and whether gyroscopic effects are to be taken into account (does not apply to torsional vibrations). You must also define which calculation method is to be used to calculate tooth contact stiffness. The following selections are available for this last option:

- As defined in ISO 6336 Method B, if the tooth contact stiffness used here matches the description in this standard.
- Using the contact analysis algorithm (for each gear pair), where a full contact analysis is performed in the gear connections. If KISSsoft does not have a contact analysis calculation for a particular gear pair type, or if the gear pair does not transfer power, the ISO 6336 process is used for that specific pair.
- Infinite: the tooth contact stiffness is assumed to be infinite. Select this option if you want to check limiting conditions.
- Ignore: the tooth contact stiffness is assumed to be zero and there is therefore no connection between the vibrating shafts (each shaft is vibrating independently).

When the calculation is finished, the results can be accessed in the report or in the graphics.

Note that, when you perform a modal analysis for a planetary system, the calculation does not take into account the effect of the positions of the rotating planets on the system's bending stiffness. This is similar to the quasi-static calculation procedure usually followed in eigenfrequencies analysis.

13.3.3 Campbell diagram

A Campbell diagram can be used to investigate the effects of shaft speed on the eigenfrequencies. This calculation can be used to define the critical eigenfrequencies for each speed.

To start running this calculation, click on the "Campbell diagram" menu option in the "Calculation" menu. In this dialog, you can also specify the method for calculating the gear mesh stiffness (as described in the Modal analysis section), the reference boundary for the calculation and the speed range. You can also specify the number of eigenfrequencies that are to be taken into account in the Campbell diagram. Finally, you also have the option of entering the number of resonance curves that are to be included when the Campbell diagram is displayed as a graphic. A kinematic analysis of the system is also performed for every speed of the reference boundary condition as part of the calculation. The speeds of the shafts are updated and then a modal analysis is performed for each of these reference speeds.

When the calculation is finished, the results can be accessed in the report or in the graphics.

13.3.4 Forced response

The forced response analysis can be used to calculate the real dynamic behavior of a shaft system that is subjected to dynamic loads because of unbalance masses. Deformations, rotation, forces and torques are taken into account in the calculated behavior.

To analyze the unbalance response, select **Forced response** in the **Calculation** menu. Select the appropriate option to take unbalance masses into account. There are four available methods for defining meshing stiffness (a description of the selection options is given (see chapter 13.3.2, Modal analysis)). You can also select the reference boundary that is to be used for speed control in the system. In addition, you can select the speed range and the number of calculation steps.

Finally, you can also define the material damping for torsional, axial and bending vibrations in this dialog. Note that the viscous damping of bearings must be defined separately for each bearing in the shaft calculation. When the calculation is finished, the results can be accessed in the report or in graphics.

13.3.5 Extended forced response

In the forced response analysis, the dynamic behavior of a powertrain system subjected to the dynamic loads from different sources of excitations is calculated. In this regard, the vibration characterization of the system under periodic excitations is performed. To do so, the harmonic excitation forces are applied to the system and the underlying responses are calculated. Three sources of excitation can be modeled. This includes the varying gear meshing forces and meshing stiffness, externally applied torque ripples and unbalanced mass excitations. The forced response analysis based on the unbalanced mass excitation and torque ripples is integrated in the extended force response module.

13.3.5.1 Theory

The theory of the forced response analysis is based on the research work done by Beermann [5]. In this approach, the excitation forces of the meshing gears are calculated and then are applied to the system according to their excitation orders. The total procedure is based on the frequency response analysis, where all the excitations and responses are represented in terms of the excitation frequencies together with their corresponding amplitudes and phase angles.
The forced response analysis in KISSsoft consists of three main steps:

- The first step is a pre-processing step. The shaft calculations are performed to calculate all relevant kinematic and geometric parameters of the system. Then, the contact analysis of all involved gear stages is carried out. The most important output data calculated are the transmission error and the nonlinear stiffness of gear pairs. These values are required to calculate the gear mesh excitation forces.
- The basic calculation of the forced response analysis of the system is performed in the second, and most important step. This step includes the generation of the coupled system of vibration equations and the calculation of the excitation forces. As the result of the system solution, dynamic contact forces at meshing gear pairs are calculated.
- The third step is a post-processing step in which the gear mesh outputs, bearing forces and shaft outputs at the documentation points are calculated. The calculation of the output data is based on the results of the main step.

The main part of the forced response algorithm consists of the following steps:

- Finding the system base frequency Ω_0 of all excitation frequencies and adjusting the time variables.
- Removal of duplicated frequencies for meshing gear pairs, and variable torque inputs.
- Building the stage matrices for a specific excitation frequency. A stage matrix contains all the dynamic forces and system responses caused by the effect of a unit load on gear pairs for a specific excitation frequency.
- Building the Fourier matrix [5] containing all translational and rotational forces of the system at meshing gear pairs.
- Calculating the angles between force exertion planes of successive meshing gear pairs: the bending displacement of a gear pair as the result of force exertion at an adjacent gear pair is projected on the direction of that force.
- Calculating the gear stiffness and mean transmission error of meshing gear pairs corresponding to the user-defined excitation frequencies.
- Defining boundary conditions for the shafts.
- Calculating the bending, axial, and torsion influence factors as the result of a unit load at gear pairs for all shafts at different excitation frequencies.
- Solving the system of dynamic equations to calculate Fourier coefficients.

- Calculating the static and dynamic forces at meshing gear pairs over time at each input shaft speed.

13.3.5.2 User interface

The powerful and user-friendly user interface of the forced response analysis allows analysts and engineers to perform the dynamic analysis of powertrain systems quickly and efficiently. Within this tool, a comprehensive list of different settings and options are provided which enable the user to precisely investigate the vibration characteristics of the system. With the current implementation, the forced response analysis of powertrains with helical and bevel gears mounted on normal and coaxial shafts with switchable or coupling connection elements can be accomplished.

The main analysis settings can be defined in the first part of the user interface where forced responses are analyzed:

Type of calculation: Three options are available. In the first option, **Unbalanced masses**, only the excitations from unbalanced masses are considered. The relevant parameters of the unbalanced masses including mass and eccentricity have to be defined in the corresponding shaft calculation module. At least one unbalanced mass with nonzero mass and eccentricity values has to be defined. Otherwise, an error message is shown. The position of the unbalanced mass has to be given within the corresponding shaft limits. It should not exceed the shaft length. There are four methods available for defining meshing stiffness (a description of the selection options is given (see chapter 13.3.2, Modal analysis)).

The second option, **Gear meshing forces**, excludes the unbalanced mass excitation and includes the gear mesh forces resulting from the transmission error and the externally applied torque ripples. In this case, the externally applied torque ripples can be input as another source of excitation in a separate window. The third option, **Unbalanced masses and gear meshing forces**, considers the three sources of excitation mentioned above.

Reference boundary: This is a list of all the boundaries on which the input speeds are defined in the system module's **Boundary** tab. With this input, one boundary is selected as the reference to assign the input speed for the forced response analysis. According to the selected reference boundary, a kinematic calculation is performed in the pre-machining step to calculate the speed of all other shafts.

Number of speeds: It specifies the number of speeds to run repeatedly the forced response analysis. The minimum and maximum numbers of speeds are, respectively, 1 and 10000.

Speed range (min/max): This is where the minimum and maximum values of the speed range of the reference boundary are specified, in units of 1/min. If the number of calculation steps is set to one, only one input box for specifying the speed of the reference boundary is shown. It is important to note that, to avoid numerical problems, you cannot specify a minimum speed of less than 0.01 [1/min].

13.3.5.3 Material damping

In powertrain systems, all deformable elements can dissipate energy when subjected to dynamic deformations. The internal damping of these elements represents the energy dissipation. Consequently, damping is an important factor in the design of structures to minimize noise, structural instability, and fatigue failure of components. Three different damping sources can be given: damping of bearings and supports which have to be defined in the shaft calculation module, structural damping of shafts, and gear meshing damping. The material damping of shafts in torsional, axial (tension/compression), and bending directions are specified by some loss factors. The default value is $1x10^{-5}$s. This type of damping is based on the Kelvin-Voigt model and is well-suited for the structural vibration analysis of continuous systems [5].

$$\sigma = E(\in + k_w \dot{\in})$$

where k_w is the damping coefficient. In this approach, the damping is introduced in the equations of motion through complex-valued quantities with the underlying assumption that the vibrations are harmonic. Thus, this damping model can be used in the frequency domain analyses.

13.3.5.4 Gear pairs for excitation

In the implementation of the forced response analysis, the excitation of all active gear pairs is considered by default for the **Gear meshing forces** excitation. However, sometimes it is required to consider only one or several gear pairs among all active pairs and to excite the system. Neglecting the gear pairs that are not of interest or may contribute less to the system excitation, can lead to a faster forced response analysis. In this case, one can easily characterize the effect of each gear pair in a separate analysis. The calculation algorithm remains the same. However, for gear pairs that are excluded from the excitation, the meshing stiffness can also be calculated using “ISO 6336 Method B”. Note that a contact analysis is still required to calculate the transmission error for those gear pairs. As another important aspect, at least one active gear pair must be selected.

13.3.5.5 Torque ripple

Torque ripple is a periodic excitation in torque which can result in vibrations and noise. This effect can usually be observed in many electric motor and combustion engine designs, referring to a periodic fluctuation in the output torque as the motor shaft rotates. The externally applied torque ripples, as another source of excitation, are integrated in the extended force response module. The main approach is very similar to the one used to calculate gear mesh excitations. However, the excitations resulting from torque ripple are added to those resulting from the transmission error, so that the forced responses in the system are calculated as the result of both excitation types.

The table for the torque ripple excitation input data is not active if **Unbalance masses** is selected as the type of calculation. The boundary conditions for the torque ripple input data have to be specified in this table. Click the + sign in the lower right-hand corner of the table to insert new rows for entering the input data for each boundary condition. After selecting the boundary condition, the corresponding input coupling with torque ripple has to be defined. Under **Input**, specify one of the input types for the time or frequency domain. Finally, select the input folder into which the input torque ripple data is to be uploaded.

Required specifications for the torque ripple input data in the time domain:

- Torque ripple input data in the system module must have the following file format: *.dat, *.txt or *.csv. The file must contain two columns with the torque ripple amplitudes.
- The first column must contain the angle of rotation for the corresponding boundary condition. The second column must contain the torque ripple amplitudes.
- The torque ripple amplitudes must be given for a duration of one (or several) full rotation(s) of the shaft on which the input coupling is loaded by the torque. This implies that the frequency of one torque variation cycle must be an integer factor of the corresponding shaft speed.
- The torque amplitudes must be reported at equally distanced rotation angles.
- The mean value of the torque ripple amplitudes must be zero. It means that the constant part (its mean value) must be already given as a constant torque through boundaries in the system module.

Required specifications for the torque ripple input data in the frequency domain:

- In the first line, the excitation orders are given.
- Enter the coupling speeds or the constant torques in the second and third lines.
- Then, enter the torque ripple amplitudes that correspond to each excitation order and to each speed and torque in the ranges.
- The coupling speeds and constant torques must be ascending.
- The coupling speeds and constant torques of the system module model must be in the range given in the input data file.
- The coupling speeds and constant torques of the system module model must correspond to a defined value of the torque ripple amplitudes.

In the table, a sample of the input data type for the torque ripple in the frequency domain for three excitation orders, four speeds, and three contact torques is shown.

Excitation orders	$f_1(1^{st})\,,f_2(2^{nd})$
Speeds range (rpm)	$\omega_1, \omega_2, \omega_3, \omega_4$
Constant torque	T_1, T_2, T_3

range (Nm)				
Torque amplitudes (1st order)	$A_{f_1\omega_1T_1}$	$A_{f_1\omega_2T_1}$	$A_{f_1\omega_3T_1}$	$A_{f_1\omega_4T_1}$
	$A_{f_1\omega_1T_2}$	$A_{f_1\omega_1T_2}$	$A_{f_1\omega_1T_2}$	$A_{f_1\omega_1T_2}$
	$A_{f_1\omega_1T_3}$	$A_{f_1\omega_1T_3}$	$A_{f_1\omega_1T_3}$	$A_{f_1\omega_1T_3}$
Torque amplitudes (2nd order)	$A_{f_2\omega_1T_1}$	$A_{f_2\omega_2T_1}$	$A_{f_2\omega_3T_1}$	$A_{f_2\omega_4T_1}$
	$A_{f_2\omega_1T_2}$	$A_{f_2\omega_1T_2}$	$A_{f_2\omega_1T_2}$	$A_{f_2\omega_1T_2}$
	$A_{f_2\omega_1T_3}$	$A_{f_2\omega_1T_3}$	$A_{f_2\omega_1T_3}$	$A_{f_2\omega_1T_3}$

Table 13.1: Example of the input data type for torque ripple in the frequency domain

13.3.5.6 Gear meshing forces

When the type of calculation includes **Gear meshing forces**, a plus sign appears for setting the following parameters that are relevant to this type of calculation.

Meshing stiffness: According to the contact analysis module, both the **Tangent stiffness** and **Secant stiffness** approaches are available. If the contact analysis algorithm per gear pair is used , a full contact analysis is performed separately for each gear pair. The results of transmission error and meshing stiffness are then used to calculate the excitation forces. The accuracy of the excitation forces in the forced response analysis is related to the accuracy level of the contact analysis. Therefore, prior to the forced response analysis, it is recommended to check the transmission error, meshing stiffness, and excitation force of each gear pair calculated in the contact module.

Excitation force type: Calculation of the excitation forces can be done in two different ways. The **Meshing stiffness times transmission error** option is based on the multiplication of the meshing stiffness and the transmission error. This assumes that the meshing stiffness and transmission error are represented as the summation of a number of harmonic functions in the form of

$$c(t) = \sum_{k=-N}^{N} \gamma_k e^{ik\Omega_0 t}, T\,E(t) = \sum_{n=-M}^{M} \mu_n e^{in\Omega_0 t}$$

The excitation forces can now be calculated from

$$F(t) = T\,E(t)\,c(t) = \sum_{n=-M}^{M} \mu_n e^{in\Omega_0 t} \sum_{k=-N}^{N} \gamma_k e^{ik\Omega_0 t} = \sum_{n=-M}^{M} \sum_{k=-N}^{N} \mu_n \gamma_k e^{i}(n+k)\Omega_0 t$$

The second option, **Calculated in contact analysis**, is based directly on the excitation forces calculated in the contact analysis module.

Dynamic modeling approach: The **Bending and torsional responses** approach uses the full degrees of freedom (including translational and rotational degrees of freedom) of the flexible shafts to calculate the system responses. Consequently, it captures all possible responses which might be important for the system analysis. In contrast, the **Torsional responses** approach only considers the rotational degree of freedom along the shaft axis. Compared to the first approach, it requires much less computational effort and significantly speeds up the forced response analysis. In this case, only the outputs related to the torsional DOF of the shafts can be seen. For more details (see chapter 13.3.5.11, Graphics).

Gear meshing damping: Gear meshing damping is known as an important determinative factor in dynamic response of the system under certain running conditions. With the viscous damping, the effect of lubricant film as well as the internal damping of the gear teeth in dissipating the kinetic energy into frictional heat and constraining the motion amplitudes is modeled. On the other hand, without damping, undesirable induced vibrations generate noise, increase dynamic loads and potentially damage gears and bearings.

Gear meshing damping can be defined as **Constant**, **Proportional to average meshing stiffness** or **Proportional to nonlinear meshing stiffness**. Typical values for the damping coefficient are between 102 and 104 Ns/m. In the proportional damping model, the damping coefficient is calculated from:

$$C = 2\xi\sqrt{\frac{J_pJ_g}{(1/J_pr_p^2) + (1/J_gr_g^2)}}k$$

where ξ is the damping ratio, k is the meshing stiffness, J_p and J_g are the mass moment of inertia of the pinion or gear, and r_p and r_g are the base circle radius of the pinion or gear.

Frequency response approach: The forced response analysis is based on the frequency response approach. Therefore, it uses the excitation force amplitudes represented in different frequencies and phases. Two different approaches are available: **Multiple harmonics** and **Single harmonic**. In the first case, all excitation frequency orders, up to the number of excitation harmonics, are taken into account when calculating the excitation forces. With the **Number of excitation harmonics**, the same number of frequencies of the excitation forces for all active gear pairs is considered. If this number exceeds the maximum number of available harmonics of any gear pair, then the lowest maximum number of harmonics among all gear pairs is considered in the excitation. The possible maximum number of harmonics which can be set is 100. When the torques ripple excitation is also given, then the given number of harmonics is considered for both excitation types. It is particularly of great importance to select a suitable number of harmonics to make a trade-off between the demanded level of the results accuracy and the analysis effort. Increasing number of harmonics results in higher number of dynamic equations, and consequently, can result in higher calculation time. Before performing the forced response analysis, we recommend you check the influence of transmission error amplitudes, meshing stiffness or torque ripple to decide the extent to which excitation orders should be considered.

In the **Single harmonic** approach, only one specific excitation frequency order, which equals the **Harmonic number**, is taken into account. Therefore, the effect of each excitation frequency order of active gear pairs on the system response can be investigated separately.

Solution approach: The frequency response approach results in a set of coupled dynamic equations. The solution to these equations calculates the dynamic contact meshing forces. Similar to the excitation forces, the system responses are also represented as the summation of some harmonic functions in terms of response frequencies. The solution approaches can be divided into **Direct** and **Iterative**. In the direct approach, the coefficient matrix of the system equations is a square matrix. Therefore, the number of response harmonics is equal to the number of excitation harmonics. However, in the iterative approach, the equations are solved iteratively. As the result of this approach, the number of response harmonics can be higher than the number of excitation harmonics.

In each step of the iterative approach, the **Convergence ratio** is used to check if the calculated results are fulfilling the convergence criteria. The smaller the value of the convergence ratio is chosen; the higher number of iteration steps is required to fulfill the convergence criteria and to stop the iteration process. Consequently, the accuracy of the results is enhanced.

13.3.5.7 Settings in the shaft calculation module

There are some important settings in the shaft calculation module which must be taken into account in detail before performing a forced response analysis. These settings can have a significant effect on the results. Some of them are briefly mentioned here.

Gear on shaft: Four types of definition for gears on a shaft are available in the **Basic data** tab:

1. Gears as load applications only
2. Gears as mass
3. Gears as mass and stiffness
4. Gears as mass and stiffness acc. to ISO 6336-1

According to the selected type of gear definition, the corresponding settings are considered internally in the forced response analysis to define the gears. If the definition of the gear in any shaft calculation differs from the definition in other shaft calculations, a warning message is displayed at the end of the forced response calculation. This definition can also be set once in the **System data** tab in the system module to ensure it is consistent for all gears.

Rolling bearings calculation: In the **Rolling bearing** tab in the shaft calculation module, you can select one of the following settings for the calculation method:

1. Stiffness: Not calculated
2. Stiffness: ISO/TS 16281

This item can also be set once in the **System data** tab in the system module to ensure it is consistent for all rolling bearings. If **Stiffness: Not calculated** is selected, a very large stiffness coefficient is considered in KISSsoft. A warning message with this information is displayed at the end of the forced response calculation. In addition, selections of the stiffness of the rolling bearings in a shaft calculation different from other shaft calculations results in another warning message.

General: The following options contribute to the results of the forced response analysis:

1. Consider weight
2. Consider gyroscopic effect
3. Consider deformation due to shearing (Timoshenko beam)

The first two items can also be set in **System data** tab in the system module to ensure they are consistent for all shaft calculations.

Supports and Bearings: When defining bearings, the stiffness coefficients, the damping coefficients and the imposed boundary conditions can have significant effect on the forced response analysis results. In addition, in the bearings, the damping coefficients can be manually specified by the user.

Documentation points: Some output types of the forced response analysis require a documentation point on a shaft. In fact, documentation points specify prescribed measuring points on the shafts where certain shaft outputs are demanded. This fact will be explained in greater detail in the **Graphics** section 13.3.5.11. If a documentation point is located outside of a shaft, an error message is shown. The same rule holds also for the coupling locations on the shafts.

13.3.5.8 Settings in KISSdesign

The speeds given in the **Boundary** tab are the nominal speed values of the boundaries. These speeds are only used if the **Reference boundary** speed in the user interface does not fulfill the forced response requirements. The torques of the coupling elements are used as the nominal constant torques. In the torque ripple excitation, these constant torques are added to the dynamic torque amplitudes to apply the variable torque excitation to the gears.

Notes:

1. The sense of rotation in the forced response analysis is based on the data in the **Boundary** tab
2. The contact analysis is based on the constant torque values. This means that even in the case of the torque ripple, the forced response uses the transmission error and stiffness of gear pairs calculated from the Contact analysis module, based on the constant torques.

When the type of calculation includes **Gear meshing forces**, there should be no zero-mesh frequencies present in the system. This can happen, for example, in a planetary system when the sun, planet carrier and ring gear have the same speed in the **Reference boundary** tab. In this case, the planets are turning in such a way that their teeth that contact the sun and ring gears are not changed. The planets are turning in such a way that their contact teeth do not change. In this way, all gears are rotating with the same angular velocity and therefore, an error message is shown.

13.3.5.9 Module specific settings

As it is already mentioned before in the theory of the forced response, after running the calculation, the basic dynamic analysis is done to calculate the dynamic gear mesh forces in the frequency domain. For this purpose, the system equations are generated and are solved. To generate all other results of the gear pairs, bearings and shaft outputs at documentation points, select the relevant elements in **Module specific settings** in the **Forced response** tab. The following settings can be set there.

13.3.5.9.1 Time domain inputs

Resolution: The quality of the results in the time domain can be adjusted by setting the resolution to low, medium, high or very high. This option does not affect the quality of the results in the frequency domain. In the forced response analysis, the end time for generating the time domain data is adjusted so that it can capture all possible excitations and complete full periods of the vibrations for all excitation frequencies. A base excitation frequency Ω_0 is determined for this purpose. This is related to the common divisor of all the excitation frequencies. This means the excitation frequencies are integer factors of Ω_0. The end time is therefore calculated as

$$T_{end} = 2\Pi/(\Omega_0)$$

The step time is then calculated:

$$\Delta T = 2\Pi/(R_f \cdot f_{max})$$

where R_f is the resolution factor which is adjusted according to the resolution level (low: R_f = 10, medium: R_f = 25, high: R_f = 50, very high: R_f = 100). Furthermore, f_{max} is the highest excitation frequency of the system.

Plot step factor: The plot step factor P defines the number of sampling plot data for generating the outputs in the following manner:

1. According to the resolution set by the user, the time step ΔT for calculating the results in the time domain is calculated.
2. The time step ΔT_{plot} for calculating and plotting the results is then calculated from $\Delta T_{plot}=P\cdot\Delta T$.
3. The highest possible time step, $\Delta T_{max}= 2\pi/(10\cdot f_{max})$, which corresponds to the lowest resolution, is calculated.
4. If the time step is $\Delta T_{plot} > \Delta T_{max}$, ΔT_{max} is the final time step for calculating and plotting the results.

The plot step factor is only active if the resolution is not "low".

End time: The default setting is that end time T_{end}, which is used to calculate the time domain results, is calculated as described above. However, if a specific time T_{end} is required or if the default end time is high and too much time is needed to generate the time domain results, the end time can be modified here.

13.3.5.9.2 Output data specification

After running the basic calculation in the forced response, the gear pairs, bearings, and documentation points can be selected here to calculate the output results. For each group, selection of all elements together as well as selective definition of each element is available. In addition to generating the output data in **Graphic**, the results can be saved in “txt” files and exported for further use. Therefore, it is required to select the demanded outputs by checking the corresponding boxes. To export these files, select **File > Export > Forced response > General**.

Dynamic factor: The dynamic factor characterizes the overall dynamic response of each gear pair at different running speeds as the result of transmission error excitation. Special attention must be paid to those speeds that have high dynamic factor values. The dynamic factor can be generated if the **Gear meshing forces** option was selected as the type of calculation. Consequently, it is not available if the **Unbalance mass** option was selected. To calculate the dynamic factor of gear pairs, system equations are generated and solved, to calculate the dynamic contact forces of each gear pair in the frequency domain. These forces are then converted to the time domain:

$$F(t) = \sum_{n=-h}^{h} F_n \ e^{in\Omega_0 t}$$

The maximum dynamic force F(t) is determined and divided by the corresponding static force F_s to determine the dynamic factor:

$$D(\Omega) = \frac{\underset{0 \le t \le T_{end}}{max} F(t)}{F_s}$$

Frequency and time domain outputs: Calculating the results of the bearings and shafts in the frequency domain is carried out based on the dynamic meshing forces calculated from the basic calculation step. Compared to dynamic factor calculation, it requires higher computational effort and therefore, it is recommended to select the demanded outputs.

For calculating the results in the time and speed domains, first the postprocessing is done for the selected outputs in the frequency domain. Then, in subsequent steps, they are represented in time. Since generating the output data in the time domain is carried out at all time steps in the time range, it is more time consuming than the case when only the frequency domain outputs are required.

13.3.5.9.3 KISSsoft-RecurDyn interface

Before exporting the bearing forces in time domain and creating the KISSsoft-RecurDyn interface file, it is necessary to define the required fields. This includes the inputs for generating the FEM mesh, the number of normal modes of the housing, the definition of the step files of the gears and shafts (if relevant), and bushing elements for supporting the housing in RecurDyn. To export these files, select **File > Export > Forced response > RecurDyn**.

13.3.5.10 Results window

When the calculation is finished, as shown in the table, the following data is generated in the results window for each gear pair at all the running speeds specified in the speed range:

Speed (1/min)	Base excitation frequency (Hz)	Time step (ms)	End time (ms)
n	Ω_0	ΔT	T_{end}
Speed (1/min)	Gear meshing frequency (Hz)	Max. dynam. force (N)	Dynamic factor
n	f	F_{max}	D

Table 13.2: Result window of the forced response analysis

13.3.5.11 Graphics

After entering the settings for the forced response in **Module specific settings**, click the **Run calculation** icon to update the settings to ensure the calculations are consistent. The results and outputs of the forced response analysis can be accessed in the graphics menu, **Graphic > Forced response**. This is based on the following requirements:

1. At least one documentation point is located on a shaft.
2. The number of speeds is at least two.
3. **Gear meshing forces** or **Unbalance mass and gear meshing forces** was selected as the type of calculation.

General data (plot requirements No. 1, 2):

Variation of the maximum value of the following parameters at documentation points versus speed of the reference boundary is shown:

- major half axis of whirl orbit
- minor half axis of whirl orbit
- phase of whirl orbit

The outputs are related to the plots of the whirl orbit to assess the deformation of flexible shafts at different sections. Figure 1 shows a typical elliptical path of the shaft center which can be characterized by the whirl orbit parameters.

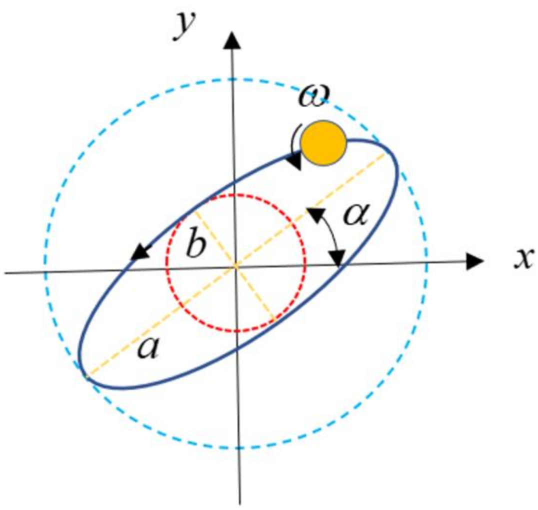


Figure 13.1: Whirl plot of a flexible shaft

3D deformation: Select this to display the system elements' vibrations and movement caused by the excitations. This feature can provide more insights to the response of the system`s elements by visualizing their movements in a 3D view. For this purpose, the forced response creates many sections on each shaft and runs the postprocessor to calculate all the relevant responses at these sections. Consequently, it can be very time consuming if complex systems that contain many shaft objects are involved.

Dynamic factor (plot requirement No. 2, 3):
The dynamic factor is plotted for all active gear pairs, with reference to the reference boundary speed. The plots are available in both “per gear pair” and “all gear pairs” formats. The reference speed of the reference boundary from the **Boundary** tab is also displayed here.

Bearing forces and moments:
time domain: Variation of the bearing forces and torques versus time or rotation angle of the shaft can be plotted at all running speeds of the reference boundary.

Frequency domain: The bearing forces and torque amplitudes in the excitation frequencies can be plotted for all reference boundary running speeds. Number of amplitudes is equal to the total number of distinguishable excitation frequencies. The amount of static force and moment are additionally shown in the comment window.

Speed range (requirement No. 2): The variation of the maximum value, minimum value, and peak to peak value of the bearing forces and torques can be plotted at all reference boundary running speeds.

Shaft outputs:

Time domain (requirements No. 1): Variation of the deformation, velocity, acceleration, force, rotation, and bending moment at documentation points versus time or rotation angle of the shaft can be plotted at all running speeds of the reference boundary.

Speed range (requirement No. 1, 2): Variation of the maximum value, minimum value, and peak to peak value of the deformation, velocity, acceleration, force, rotation, and bending moment at documentation points can be plotted versus speed of the reference boundary.

Transmission error (local and common axes, requirement No. 3):

The transmission error (TE) is plotted for all active gear pairs. The transmission error is calculated from the contact analysis module by considering the effect of the shift phase angle of successive gear pairs. For a more detailed explanation, see [6]. In the local axis plot, variation of the TE for each gear pair is plotted versus the rotation angle of the corresponding main gear. However, in the global axis plot, the TEs of all active gear pairs are plotted versus the angle of rotation of one gear as the reference.

Dynamic transmission error:

Time domain (requirements No. 3): Variation of the dynamic transmission error for all active gear pairs at all running speeds of the reference boundary is shown. For each gear pair, the X axis is the rotation angle (or its equivalent time duration) of the shaft which contains the main gear of that gear pair.

Speed range (requirement No. 2, 3): The variation of the maximum value, minimum value, and peak to peak value of the dynamic transmission error for all active gear pairs is displayed, with reference to the reference boundary speed.

Meshing contact force:

Time domain (requirements No. 3): Variation of the dynamic meshing contact force for all active gear pairs at all running speeds of the reference boundary can be plotted. For each gear pair, the X axis is the rotation angle (or its equivalent time duration) of the shaft which contains the main gear of that gear pair.

Frequency domain (requirements No. 3): The amplitudes of the dynamic meshing contact force for all active gear pairs at all running speeds of the reference boundary are plotted. Number of amplitudes is equal to the total number of distinguishable excitation frequencies. The amount of static contact force is additionally shown in the comment window.

Speed range (requirement No. 2, 3): Variation of the maximum value, minimum value, and peak to peak value of the dynamic meshing contact force for all active gear pairs is shown versus speed of the reference boundary.

Meshing stiffness (requirement No. 3):

Variation of the meshing stiffness versus rotational angle (or its equivalent time duration) for all active gear pairs is shown.

Campbell diagram (requirement No. 2):

At all running speeds of the reference boundary (X axis), the bearing forces and moments amplitudes (in color-legend format) and their associated frequencies (Y axis) are plotted. The following properties can be seen in the Campbell diagram:

- amplitude of bearing forces (max. value of all bearings)
- amplitude of each bearing force
- display in log10 and normal scale

More information about the maximum and minimum values of the bearing force amplitudes and their components in x, y, and z directions is provided in the comment window in the Campbell diagram.

Notes:

1. The results of Campbell diagram are based on the frequency domain analysis.
2. The resolution of the results is directly related to the **Number of speeds**. For this reason, it is important to note that the number of calculation steps should high enough to reflect the effect of all possible critical speeds in the speed range, so that the results can be shown more clearly.
3. The number of lines in the Campbell diagram is equal to the number of distinct excitation frequencies, from gear pairs (depending on the number of harmonics), unbalanced masses, and torque ripples.
4. The lines are shown in the same order of the excitation frequencies.
5. For identical excitation frequencies, the results overlapped.
6. The number of excitation harmonics influences the number of excitation frequencies due to gear mesh forces and torque ripple.

General remarks:

Depending on the **Dynamic modeling approach** selected in the forced response UI, some outputs cannot be generated. To clarify more this point, the following points have to be considered.

Output data for bending and torsional responses:

- All responses which correspond to the bending and torsional DOFs are calculated.
- The dynamic factor shows the ratio of the maximum dynamic gear mesh force along the PL to their static values.
- The dynamic bearing forces and moments, dynamic shaft outputs, and dynamic meshing contact forces are calculated and added to their corresponding static values.
- The dynamic transmission error shows the spring compression value at gear mesh positions which results in the calculated meshing contact forces.

- The meshing stiffness values, which are calculated from the contact analysis, are independent of the shaft speeds. Depending on the demanded number of harmonics, their amplitudes for each gear pair can be different.

Output data for torsional responses:

- Only the dynamic responses which correspond to the torsional DOF are calculated.
- The dynamic factor shows the ratio of the maximum dynamic gear torques to their corresponding static torque values.
- The dynamic bearing forces and moments are zero. Only the static bearing forces andmoments are shown in the Graphics.
- The dynamic shaft outputs, which are related to the bending DOFs (deflection, velocity, acceleration, and force), are zero. Only their static values are shown in the Graphics.
- The dynamic shaft outputs which are related to the torsional DOF (rotation and bending moment) are shown in the Graphics.
- The gear mesh outputs (meshing contact force, dynamic transmission error, meshing stiffness) are based only on the torsional excitation of the gears as well as torque ripples. No contact force excitation along the Pressure Line (PL) is considered.

13.3.5.12 Current limitations

The forced response analysis has the following limitations:

- A force response analysis cannot be performed for bevel gear differentials.
- The dynamic response of the housing is not considered in the analysis.

13.3.6 Housing deformation in static calculations

The way that housing deformation is included in static calculations is based on the use of a reduced stiffness matrix for the housing, as calculated by the Finite Element Method (FEM). This reduced stiffness matrix should include the nodes that refer to the center point of the bearings that connect the gear unit shafts to the housing.

13.3.6.1 Main calculation steps

The calculation steps for performing this kind of analysis are summarized below. The actual process used to generate the reduced stiffness matrix is not described, because it is different for each FEM computer program. Please refer to your FEM program manuals for more information.

Step 1: Setting up the calculation

The housing deformation calculation is a special system module calculation. To enable this special calculation to be used, at least one housing element must be added to the model. In order to use a housing element in this calculation, the appropriate checkbox must be set in the housing element properties.

In the **Calculation > Housing deformation** tab, you can define which housing element is to be used and enter the reduced stiffness matrix file name, required convergence accuracy and maximum number of iterations. The system of units used in the stiffness matrix file can either be read from the file, or set manually, in the **Calculation** tab. The tolerance used for mapping bearings to FEM master nodes can be also set there. More information about using this input data is provided below.

As the FEM model and the system module model might not have the same coordinates system, it is important to position the housing element correctly in the model. If you want to perform this positioning by manually mapping three FEM nodes to their corresponding bearings, you must select the appropriate checkbox in the housing element properties. To perform the positioning, you should select three non-colinear bearings in the table in the **Calculation** tab and then manually set the FEM master node IDs that correspond to them. After positioning has finished, it is advised to validate the position of the housing element in the 3D viewer by viewing either the master nodes or the 3D representation of the housing (displayed by right-clicking on the housing element). Note that when a STEP model is used for the geometrical representation of the housing, this model should be in the same coordinate system as the reduced FEM stiffness matrix to ensure the model is displayed correctly after positioning.

You will find more detailed information about the FEM programs supported by the software, and the file format requirements, in the relevant instructions (available on request).

Step 2: Performing the analysis

The first step in the calculation is to check the mapping of the FEM nodes to bearings. If any nodes are unmapped, the program reports them all and displays their distance to the closest bearing. At this point, you need to know if the positions of the specified nodes actually correspond to the bearings. You can then decide whether to continue with the calculation or cancel it. One possible reason that specific bearings do not correspond to nodes is that the housing is positioned incorrectly in the system module's coordinates system. If this is the case, the positioning of the housing must be validated. If not, and the difference between FEM nodes and bearings (as reported in the mapping message) is not too big, you can change the tolerance used in the mapping process. This might happen, for example, if the FEM node was positioned at the edge of a bearing instead of in the middle. If you continue the calculation, the program reduces the stiffness matrix for the part that corresponds to the mapped nodes, and therefore ignores all nodes that were not mapped to bearings. The calculation also ignores any predefined offsets and tilting values previously specified in the bearings and sets them to zero. The algorithm runs all the KISSsoft calculations and derives the forces on the bearings from their results. The program then uses these forces to calculate the offsets and tilting on the bearings (using the FEM stiffness matrix). The KISSsoft calculations are then run again with the new offsets, which might result in new bearing forces and offset values. This procedure is repeated iteratively until there is convergence between the forces and offset calculations. During the calculation process, it might sometimes happen that the maximum permitted number of iterations is reached, if a housing has low stiffness.

In any case, the system will display the results window, containing the percentage ratio between the maximum difference from the last two iterations to the maximum value from the last iteration, and store and apply the results from the last iteration. Note that, to ensure the algorithm finds a useful solution, you should input a number that is greater than 4 in the Maximum number of iterations field.

After the calculation has finished, you can perform further analyses. For example, you can perform contact analysis on gears to see the effect of housing stiffness on the gear unit design parameters. You can also use multiple housing elements and load different reduced stiffness matrices for each of them. This can be very useful if you want to compare the effect of different housing designs on the gear unit design. The results for each housing calculated using this method are then stored in the housing element. These results can then be viewed again by clicking on the housing element's "Restore results of bearings' offsets" function (right-click on the housing element).

The following functions are also available, for handling the offsets of mapped bearings (the tolerances remain the same):

- Reset bearings' offsets: set bearing offset values to zero.
- Save bearings' offsets: save the current offset values.
- Restore bearings' offsets: restore the saved offset values.

13.3.7 Characteristic frequencies

This special calculation enables the collection of the characteristic frequencies of rolling bearings and gear pairs (cylindrical gears, three and four gear trains, planetary gears, and bevel gears) of a modeled system. The values are shown in two diagrams in the results window and in the special report. In addition, several tools for detailed analysis are available as well as the user input of the frequencies from other components.

The user interface of this special calculation is divided into two parts: **General** and **Additional data**.

The **General** area contains the basic settings:

Reference speed: For selecting the reference speed. You can choose from boundary conditions or from shafts.

Reference shaft or Reference boundary: Presents the shaft/boundary from the existing model to which all other frequencies are referenced. The reference speed is displayed in the report and in the results window. It can also be displayed in the **Frequency domain** diagram under **Graphic > Characteristic Frequencies**. The reference speed is also used if the **Unit selection** field is set to **Order – normalized by the reference speed**. If constant frequencies are present in the analysis (results from the modal analysis, gear critical speed, frequencies from other components) the report will output the critical frequencies which are marked in red color. These critical frequencies are equal to the reference speed. If the **Consider reference speed tolerance range** option is selected, the reference speed tolerance range will be taken into account for outputting the critical frequencies.

Speed range (min/max) n: Is used in the plot of the X-axis in the **Frequency domain** diagram, and for defining the interval in which the critical or similar frequencies are output.

Number of harmonics: Is considered in the report and in both diagrams.

Unit selection: With this option you can change the default unit (Hertz – Hz) to cycles per minute (cpm). Cycles per minute is equivalent to 60 Hz. The third option is, **Order normalized by the reference speed**. If you select this option, all the frequencies are normalized by the reference speed. The Reference speed can be from shaft or from boundary condition.

You can also consider **Gear characteristics frequencies** during the calculation. Here, you can specify whether all gear characteristics frequencies to be are calculated or only a selection of them. Additionally, gear characteristic frequencies for different gear stages can be selected. For more information about characteristic gear frequencies (see chapter 15.25, Gear mesh frequencies).

The same calculation options are available for Bearing characteristic frequencies. Also, frequencies for specific bearings can be considered.

The **Shaft frequencies** option represents only the rotational frequency of a shaft. The consideration of the frequency of an individual shaft can be selected.

The following analysis options are available in **Additional data**:

Output similar frequencies: If this option is activated, frequencies are output that are inside the user-defined similar frequencies tolerance range. The results are given in the **Similar frequencies** chapter in the report. The frequencies considered in this option are the characteristic frequencies of gears, bearings, rotational frequencies of shafts, and user-defined frequencies from the **Include frequencies from other components** option. The frequencies at higher harmonics defined in **Number of harmonics** taken into account.

Consider reference speed tolerance range: This option considers the reference speed tolerance range, either as a percentage of the reference speed or as an absolute value. This enables the tolerance range plot in the **Frequencies domain** diagram and outputs the range in the result window and in the report. The tolerance range is also used to mark the critical frequencies shown in the report.

Include results from modal analysis: Takes into account the results of the **Modal analysis** special calculation (see chapter 13.3.2, Modal analysis). The frequencies from the modal analysis are shown as horizontal lines in the **Frequencies domain** diagram. The values are also shown in the report. In the report, the values of the reference speed at which the specific characteristic frequencies intersect the eigenfrequencies are shown. If this value is close to the reference speed, it is marked red, as this might be a critical point.

Include gear critical speed according to ISO 6336-1/ISO 10300-1: If this option is activated, this value is included as horizontal line in the **Frequencies domain** diagram. The value is also shown in the report and the resonance analysis is conducted, showing the reference speed value at which the specific characteristic frequencies intersect the critical speed frequency. If this value is close to the reference speed, it is marked red.

Include frequencies from other components: By choosing this option, you have the possibility to input additional frequencies. This input can be as a reference value to specific shaft or boundary conditions. In this case, the order or multiplier is input and the frequency is automatically calculated at the reference speed. The frequencies input as **Reference value frequencies** are dependent on the shaft speed or the boundary condition. The values are shown in the diagrams and in the results. The **Output similar frequencies** option also takes these values into account. For example, the input of the pole or slot number of an electric motor can be entered as a reference to an input–motor shaft. The values can also be input as absolute value frequencies which are constant with the reference speed. The absolute values are shown as a horizontal line in the **Frequency domain** diagram and are not considered in **Output similar frequencies**.

III

Gears

Chapter 14 - 24

14 Introduction

KISSsoft provides calculation modules for different toothing types, ranging from cylindrical gears in different configurations to bevel gears and face gears to worm wheels. The input windows for the different gear calculations are very similar. There are also calculation options for multiple modules. The table below shows you all the input windows in the individual calculation modules.

Input window	Section											
Basic data	15.1	is supported by all calculation modules										
Load	8.5											
Factors	15.3											
Reference profile	15.4											
Tolerances	15.6											
Modifications	15.7											
Tooth form	15.9											
Contact analysis	15.11		✔	✔	✔	✔	✔	✔				
Operating backlash	15.13		✔	✔	✔	✔	✔			✔	✔	
Master gear	15.14	✔	✔	✔	✔	✔	✔				✔	
AGMA 925	15.15		✔	✔	✔	✔	✔					

Table 14.1:

- Single gear, - Cylindrical gear pair, - Pinion with rack, - Planetary gear stage, - Three gears, - Four gears, - Bevel and hypoid gears, - Face gears, - Worms with double enveloping worm wheels, - Crossed helical gears, - Splines (Geometry and Strength)

14.1 Underlying principles of calculation

The geometry of straight or angled cylindrical gears is calculated according to ISO 21771 or DIN 3960. Many manuals and standards use very similar methods to calculate this geometry. In addition to calculating the geometry, it is very useful to have information about how to check for defects (undercut, insufficient active profile, etc.). Technical documentation provided by tooling manufacturers or machine tool manufacturers may also include information about this.

Measurements for tooth thickness allowances and backlash can be selected according to different standards, such as ISO 1328 (1970 edition) or DIN 3967. Manufacturing tolerances can also be defined using standards such as ISO 1328 (DIN ISO 1328), AGMA 2000, AGMA 2015, DIN 3961:1978 or DIN 58405 to suit the particular situation.

Strength is calculated in accordance with, for example, ISO 6336 or DIN 3990, by verifying common defects (tooth root fracture, pitting, scuffing, micropitting). These standards include the most comprehensive and detailed calculation methods currently available. There are two methods that can be used to calculate safety against scuffing. The integral temperature method of calculating scuffing safety is mainly used in the automobile industry, whereas the flash temperature method is used when turbo gear units are being manufactured. It has not yet been established which of these two methods is the more reliable.

Micropitting is calculated according to ISO/TS 6336-22 (formerly ISO/TR 15144-1), Method B. This method is very reliable for gears without profile modifications. However, in the case of gears with profile modifications, it has been specified that the tip relief C_a must correspond to the optimum tip relief C_{eff} (as proposed in the standard). If not, the verification must be performed without taking the modification into account. This is a significant disadvantage, because modifications have a considerable effect on micropitting. In this case, you should use Method A (see chapter 23.5.4.4, Safety against micropitting).

In the USA, the AGMA 2001 standard must be applied when calculating resistance. This calculation method differs so much from that required by DIN 3990 that the results cannot be compared. In addition, there are numerous different methods that are used to calculate the resistance of plastic gears.

One of the problems with applying DIN 3990 is the wide range of different calculation methods it contains. There are around 10 different calculation variants that can be applied between Method A (exact calculation involving measurements) and Method D (the simplest, rough calculation). It is therefore no surprise that very different results can be obtained from applying calculations according to DIN 3990 or ISO 6336 to the exact same gear wheel. Whenever possible, KISSsoft uses the most detailed formulae for dimensioning and analyses during this calculation procedure. This procedure corresponds to Method B. However, calculations performed using different programs may also give very different results. It also takes a lot of time and effort to investigate the precise reasons for this. It is therefore much more effective and efficient to use a reference program to perform the comparison. One such program is the ST+ cylindrical gear program package developed by the FVA (Forschungsverein Antriebstechnik, (Research Society for Transmission Techniques, Germany)), at the Technical University in Munich. For this reason, KISSsoft includes the **As in FVA program (DIN 3990)** option, which supplies the same results as calculation with the FVA code (see chapter 15.2.1, Calculation methods). The differences between results obtained by KISSsoft and the FVA program are negligible. They are due to the minor differences between the FVA program and the regular version of DIN 3990. If requested, we can provide you with a number of different documents to help you compare these methods.

Other interesting results are taken from Niemann's book [7]:

- Gear power loss with gear loss grade H_V according to equation (21.11/4)
- Mean coefficient of friction μ_m according to equation (21.11/6) with $1 \leq v_t \leq 50$ m/s
- Gear power loss P_{VZ} according to equation (21.11/3)

15 Cylindrical gears

You can use KISSsoft's cylindrical gear calculation software to calculate a range of different configurations.

- The single gear calculation has been developed to calculate the geometry and test dimensions of individual gears
- The cylindrical gear pair is the most important configuration for geometry and strength. You can also use it for additional calculations and several individual calculations at the same time.
- The planetary gear software checks the practical aspects of the configuration and monitors both pairs of gears while they are being sized. The Fine Sizing function enables you to optimize the center distance quickly and efficiently. You can usually input your own values here. However, you must take into consideration that, as torque cannot be applied to the planet, it is not possible to perform a strength analysis on a Wolfrom drive or on a Ravigneaux gear set.
- The configurations for three and four gears enable you to calculate a gear wheel chain, in which torque is applied only to the first and last gear.
- Double planetary stages: You can also use the 4-gear chain to calculate a double planetary stage. However, if gear 4 is an internal gear (negative number of teeth) a check is performed to see whether it could be a double planetary stage (planetary repositioning stage with the sun in the center). A note about this is added to the report in the "Supplementary data" section. If gear 4 is a double planetary stage, the center point is calculated under the assumption that M1 and M4 coincide.
- The calculation used for a rack and pinion includes one rack in the geometry calculation and one cylindrical gear with an infinite number of teeth for the strength calculation.

As the input screens for the different configurations are very similar, they are described together in the sections below.

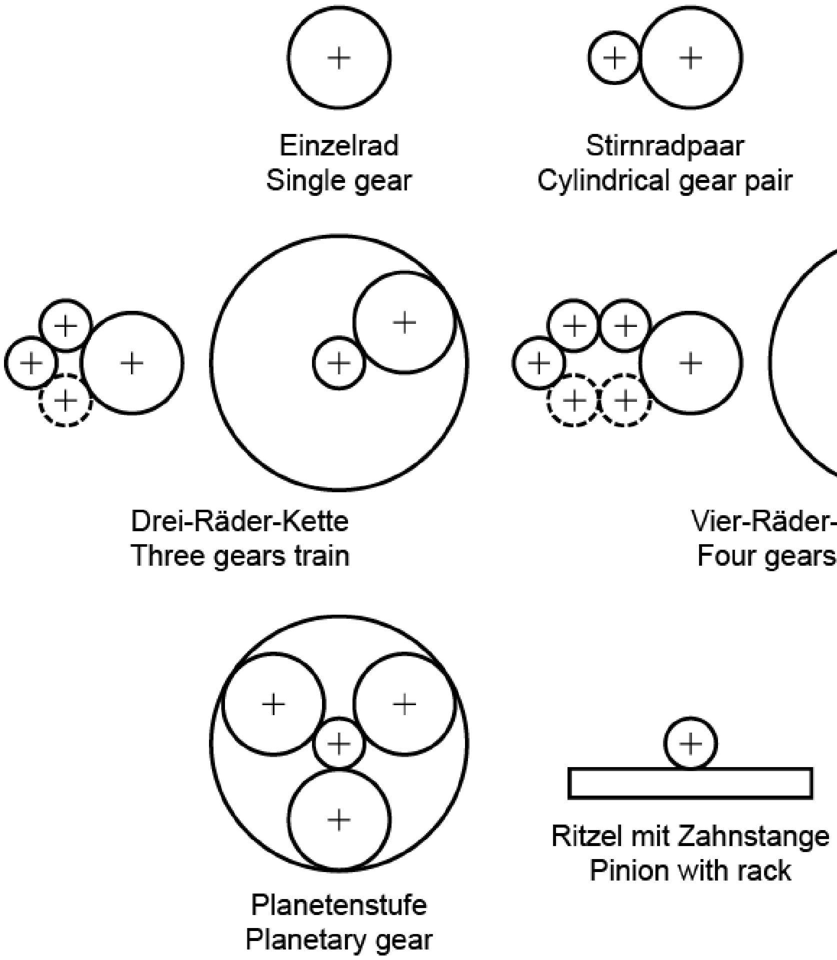


Figure 15.1: Cylindrical gear configurations

15.1 Basic data

The **Basic data** input window is one of the standard tabs (see chapter 5.1, Standard and special tabs) and is subdivided into the two groups: **"Geometry"** and **"Material and Lubrication"**.

15.1.1 Hand of gear for gear teeth

The direction of the helix angle of the gear (see Figure 17.3) defines the direction of the axial forces. Helical gear teeth usually produce less noise than spur gear teeth, but generate an additional bending moment and an axial force. A gear with continuous double helical teeth consists of two halves of a helical gear with a different hand of gear. Although it does not generate any axial forces, it must be possible to adjust the gear along its axis and it is more difficult to manufacture. In a herringbone gear, click the Plus button to set the gap width *b*n.

15.1.2 Normal module

Enter the normal module. The normal module defines the size of the teeth. A standard series is for example defined in DIN 780 or ISO 54. However, if you know the pitch, transverse module or diametral pitch instead of this, click on the Convert button to open a dialog window, in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select Input normal diametral pitch instead of normal module (see chapter 15.22.1.1, Input of normal diametral pitch instead of normal module) by clicking on **Calculation > Settings > General**.

15.1.3 Pressure angle at normal section

The normal pressure angle at the reference circle is also the reference profile flank angle. For standard gears, the pressure angle is α_n = 20°. Smaller pressure angles can be used for larger numbers of teeth to achieve higher contact ratios and insensitivity to changes in center distance. Larger pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

15.1.4 Helix angle at reference circle

Enter the helix angle in [°]. Click the Convert button in the **Convert helix angle window** to calculate this angle from other values such as, for example, the overlap ratio and axial force.

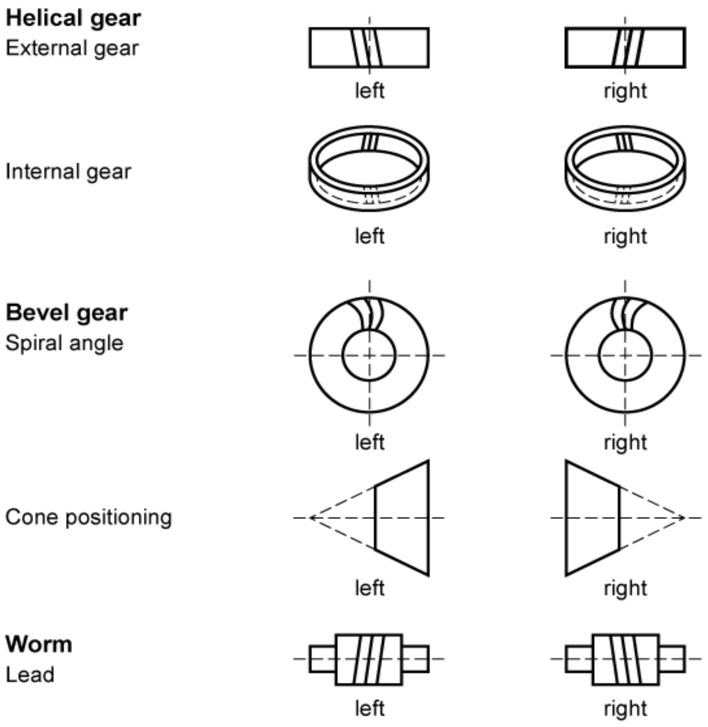


Figure 15.2: Helix angle at reference circle.

15.1.5 Center distance

As stated in ISO 21771, the center distance for external and internal toothings is positive for two external gears and negative for an external gear paired with an internal gear. The number of teeth on the internal gear and the axis center distance are always negative for internal toothing.

If you select the checkbox to the right of the axis center distance unit, the value used in the calculation will remain constant. Otherwise, the center distance will be calculated from the profile shift total.

Click the Sizing button to select one of the following sizing options:

- Fixed sum of profile shift coefficients. The center distance is calculated on the basis of a predefined profile shift sum. Click the Sizing button to display a suggested value for the profile shift sum, as defined in DIN 3992. The sum of profile shift influences the profile shift coefficients of both gears as well as the operating pitch circle and the operating pressure angle.
- Fixed profile shift coefficient Gear 1 (or 2), balance specific sliding. Optimize axis center distance with respect to balanced sliding: This option calculates the axis center distance in such a way as to balance gear pair specific sliding (for cylindrical gears) for a specified profile shift of a (selectable) gear. If the **Own input** menu option is not selected from the Own Input drop-down list in the **Reference Profile** input window, this calculation is performed with automatic tip alteration as specified in DIN 3960. You can also enter your own tip alteration value in the **Basic data** input window by clicking the **Details...** button. In the **Define geometry details window**, select the checkbox next to the **Tip alteration** input field.

15.1.6 Number of teeth

The number of teeth is, by default, a whole number. You can also enter the number of teeth as an amount with values after the decimal place (see chapter 15.22.1.2, Input of number of teeth with decimal places). For internal toothed gears, you must enter the number of teeth as a negative value as stated in ISO 21771. For a pinion-ring internal gear pair, the center distance must also be entered as a negative value (e.g. z_1 = 20, z_2 = -35, a = -7.5, m_n = 1).

The minimum number of teeth is limited by geometric errors such as undercut or tooth thickness at the tip. For example, if there are fewer than 17 teeth undercut will occur on spur gears without profile shifts.

15.1.7 Facewidth

Normally the facewidth shouldn't be greater than 10 to 20 times the normal module, or also not greater than the reference circle of the pinion. The contact pattern deteriorates if the facewidth is too large. Axial offset b_v. Click on the Plus button to the right of the facewidth input field (see also Figure). The axial offset reduces the effective width for the strength calculation. The common width is used to calculate the pressure. A certain amount of overhang is taken into account for the Tooth root strength. The selected pinion width is often somewhat greater than the gear width.

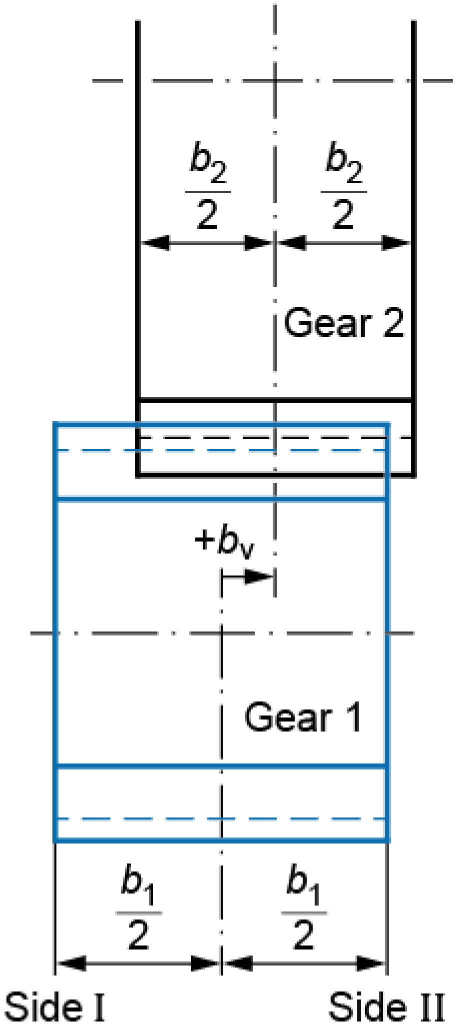


Figure 15.3: Axial offset bv

In double helical gears* you must specify the total width of the gear teeth (i.e. the width of both halves together with the gap). Click on the Plus button to the right of the toothing hand of gear drop-down list and enter the gap width *b*n.

*Double helical gears are gears that consist of two gear halves; the first half is angled to the left and the second half is angled to the right.

15.1.8 Profile shift coefficient

Preliminary note: If the profile shift sum has not yet been specified, click the Sizing button, to the right of the center distance input field (see chapter 15.1.5, Center distance), to display a suggested value for the distance in the **Size center distance** window. The suggested value is based on DIN 3992 recommendations for well balanced toothing (Area P4/P5). You will find more information about this in DIN 3992 or in Niemann [8], Fig. 22.1/6.

The tool can be adjusted for manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift, the tool is pushed further into the material, with the result that the tooth is narrower and undercutting may occur sooner. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

The distribution of the total profile shift affects the tooth thickness, sliding movements and strength values. It can be performed according to a range of different criteria. To achieve this, use the various sizing options provided by clicking the Sizing button in the **Size profile shift coefficient** window:

- For optimal specific sliding
  The value suggested here shows the profile shift, for a cylindrical gear pair, whose specific sliding between the pinion and the gear has been balanced. When more than two gears are involved, the profile shift coefficient is set to the smallest value that corresponds to the specific sliding movement at the root.
- For minimum sliding velocity
  The minimum sliding velocity at the tip of the two gears is often used for speed-increasing ratios. In a cylindrical gear pair, this means both gears have the same sliding velocity and that the access and recess length of the path of contact are also the same.
- For maximum root safety
  The profile shift coefficient is determined iteratively for the range x^*_{min} to x^*_{max}.
- For maximum flank safety
  The profile shift coefficient is determined iteratively for the range x^*_{min} to x^*_{max}.
- For maximum scuffing safety
  The profile shift coefficient is determined iteratively for the range x^*_{min} to x^*_{max}.
- For gear 1 without undercut and point at tip (min)
  The minimum value of the profile shift coefficient for gear 1 is calculated from the undercut boundary for gear 1 and the minimum top land for gear 2.

- For gear 1 without undercut and point at tip (max).
  The maximum value of the profile shift coefficient for gear 1 is calculated from the minimum topland of gear 1 and the undercut boundaries of gear 2.
- For undercut boundary per gear.
  The proposed value only refers to the selected gear. No check is performed to see whether the resulting profile shift is also permitted for the other gear in the pair. For more information, please refer to the explanations above.
- For minimum topland per gear.
  The proposed value only refers to the selected gear. No check is performed to see whether the resulting profile shift is also permitted for the other gear in the pair. You can specify the minimum thickness of the topland under **Calculation > Settings > General > Coefficient for minimum tooth thickness at the tip**. For more information, please refer to the explanations above.

► **Note:**

The Sizing button is disabled if the "Maintain tip circle when changing profile shift" or "Maintain root circle when changing profile shift" checkbox has been selected.

Click the Convert button and KISSsoft will determine whether the profile shift coefficients are to be taken from measured data or from values entered in drawings.

The following options are available:

- **Base tangent length**
  Enter the base tangent length and the number of teeth spanned here. This option cannot be used for (internal) helical gear teeth because their base tangent length cannot be measured.
- **Measurement over balls**
  To do this, enter this dimension and the diameter of the ball. In a gear with helical gear teeth and an odd number of teeth, the measurement over balls is not the same as the measurement over two pins. See Measurement over pins.
- **Measurement over 2 pins**
  To do this, enter this dimension and the diameter of the pin. You must also enter a minimum span for helical gear teeth and gears with an odd number of teeth, so the measurement can be performed. The measurement over pins cannot be measured in internal helical gears.
- **Measurement over 3 pins**
  Here, enter the measurement over pins and the pin diameter. For helical gear teeth and gears with an odd number of teeth, this is equivalent to the measurement over two rollers. You cannot use this option for internal and helical gear teeth, or gears with an even number of teeth.

- **Tip circle**
  This is a rather imprecise calculation because the tip diameter does not always depend solely on the profile shift.
- **Tooth thickness at reference circle**
  Here, enter the tooth thickness. You can also enter the arc length or chordal length, and specify whether the value is in transverse or normal section.

► **Note**

If one of the two profile shift values is displayed in gray, this means it will be calculated by KISSsoft. This is what happens when you select the checkbox for entering the center distance. If you overwrite a gray field, it will become active and KISSsoft will calculate the value for the other gear.

15.1.9 Tooth thickness modification coefficient

A tooth thickness modification coefficient xs is not usually used for cylindrical gears. This is why you cannot enter this factor in the user interface. However, in exceptional situations, you can use a tooth thickness modification coefficient, e.g. via the DLL interface, in KISSsys or by editing the saved .Z12 file. The RechSt.xs_Active = 1 and RechSt.xs_OwnInput = 1 variables must be set before you can set xs. Then xs can be applied in the ZkegR[0].XS variables for gear 1 etc. More information about the tooth thickness modification coefficient is provided (see chapter 18.2.8, Tooth thickness modification coefficient).

15.1.10 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, click on the Settings button and then select the **Input quality** option. The accuracy grade according to ISO 1328 (DIN ISO 1328) is very similar to the same quality in BS 436/2.

The manufacturing qualities that can be achieved are displayed in this table:

Manufacturing process	Quality according to ISO		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 15.1: Accuracy grades for different manufacturing processes

When converting qualities according to AGMA:
as defined in AGMA 2015-1-A01, Annex B.2, the total of the quality figures in Version 2015 (comparable with ISO) and Version 2000 equals 17.

Quality as specified in ISO 1328 and AGMA 2015	**Q. according to AGMA 2000**
1	16
2	15
3	14
4	13
5	12
6	11
7	10
8	9
9	8
10	7
11	6

Table 15.2: Manufacturing quality values in different standards

If you want to define different tolerances, click the Settings button. then select the **Varying qualities** option in the dialog you see. This activates the Plus button next to Quality in the main screen. Click the Plus button to display a new window in which you can enter the tolerances you require.

You can input the tolerances in standard-specific tabs. The changes in the window are then applied to all the gears in the calculation module.

This is the table in which you input any deviation from the base manufacturing quality (specified in the **Basic data** tab). Example: The base manufacturing quality of gear 1 is 6. If you then input +2 for the runout, the runout will be calculated with an manufacturing quality of 8.

In every case, only tabs (standards) that are possible for the calculation module are displayed. The user entries remain in this window as long as you continue using the same calculation module. This enables you to import a different file, and select the checkbox. The same entries will still be displayed in the window next to the Plus button. You only need to input the data again if you change calculation module.

Note about axis alignment tolerance according to ISO 10064:

The quality level used to input the axis alignment tolerances specified in ISO 10064 is usually the same as the best accuracy grade for all gears. If, for example, gear 1 has Q6 and gear 2 has Q5, quality level 5 is used for ISO 10064. You can also input these values in the **Operating backlash** tab.

15.1.11 Geometry details

Click the **Details** button on the top right, in the Geometry area, to display other input options. Here you can change the values for:

- Drawing number
- Rim thickness coefficient S_R*
- Inside diameter d_i
- Inside diameter of gear rim d_{bi}
- Web thickness factor b_s/b*
- Web thickness b_s

The drawing number is only used for documentation purposes. You can enter any text here.

The internal diameter is needed to calculate the mass moment of inertia. For solid wheels, enter 0. For external wheels with webs, enter the relevant diameter d_i as shown in Figure 14.4. For internal wheels, enter the external diameter of the gear rim. The inside diameter can be defined by entering either d_i or the rim thickness coefficient S_R*.

According to ISO or AGMA, the gear rim thickness s_r, defined by the inside diameter of rim d_{bi}, affects the strength. If no gear rim thickness is present, you can enter a value of 0 for d_{bi}. In this case, the gear rim thickness s_r will be determined from the diameter d_i. If a diameter for gear rim d_{bi} has been entered, the effective gear rim thickness S_r is determined from $(d_f - d_{bi})/2$. The gear rim thickness S_r will be output in the report. Where thin gear rims are used, this factor can greatly influence the calculation of safety factors. For thin gear rims, this value can also be calculated in accordance with VDI 2737 (see chapter 15.2.6.4, Safety factor root with gear rim influence (VDI 2737)).

Web thickness factor: If the inside diameter is <> 0, the value input for the web thickness (b_s or b_s/b) is taken into account. If b_s/b = 1.0 is set, this means no web is present. In this case, the gear body coefficient CR is 1.0. The ratio b/b_s can vary between 0.2 and 1.2. In this case, CR is then < 1 (if b/b_s < 1) or > 1 (if b/b_s > 1). The coefficient CR is then used to calculate the tooth contact stiffness ($c\gamma\alpha$).

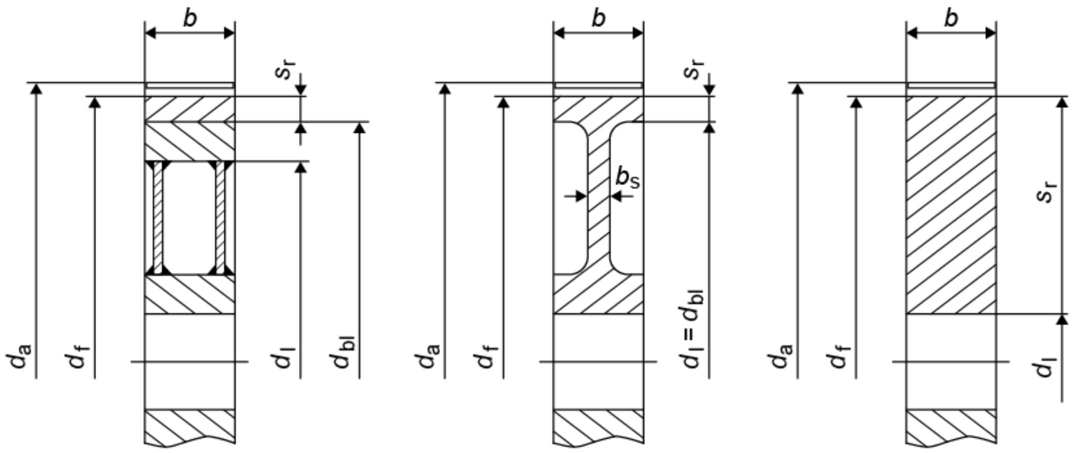


Figure 15.4: Dimensioning the diameter

15.1.12 Material and lubrication

15.1.12.1 Materials

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select **Own input** from the list or enter the material in the database (see chapter 9.4, External tables) first. Click the Plus button to the right of the materials drop-down list to display the **Define material, Gear 1(2)** window, in which you can select the material you require from the database list of available materials. Select the **Own input** option to enter specific material characteristics. This option corresponds to the **Create a new entry** window in the database tool.

Strength calculation with normal gear materials:

The cylindrical gear strength calculation formulae defined in ISO 6336, DIN 3990 or AGMA 2001 only involve specific (most commonly used) materials and treatment methods. These are:

- Through hardening steel
- Case hardening steel
- Nitriding steel
- Structural steel
- Grey cast iron with spheroidal graphite
- Cast iron with flake graphite

Strength calculation with unusual gear materials (not taken into account in standards):

- Stainless steel
- Free cutting steel

- Aluminum and bronze alloys

KISSsoft handles these materials in the same way as through hardening steels. This affects a range of less important values that are used to calculate the permitted tooth root and flank strength: factors Y_{NT}, Y_{drelT}, Y_{RrelT}, Y_X, Z_{NT}. The infinite life strength values σFlim and σHlim must either be measured or already be known. The S-N curve (Woehler lines) must be defined and used to achieve more accurate calculations.

- Sinter
  According to information from the company MIBA (A), sinter has similar properties to GC. For this reason, all the factors specified in DIN or ISO, which depend on the material type, are determined for sinter according to all the formulae that are applicable for GC.

Plastics

The strength of plastic gears can be calculated either according to Niemann VDI 2545 or VDI 2736. The material properties (Young's modulus etc.) and the permitted tooth root and flank stresses are greatly affected by the temperature and the type of lubrication. This is why calculating the characteristics for plastic gears requires so much time, effort and experience, especially if only very little material data is available. VDI guideline 2736 lists the tooth root and flank strengths for a number of basic materials:

- Tooth root strength: POM, PA 12, PA66, PET, PE, laminates
- Flank strength: PA 12, PA6, PA66, PBT, laminates
- Tensile fatigue strength for worms: POM, PA46, PA66, PEEK

Materials manufacturers also provide gear data that can be used to calculate the strength of plastic gears. If requested, KISSsoft can also provide the relevant material files.

KISSsoft users can also add their own material data to the plastics database. The appropriate DAT file contains specific data for each material. The user can then edit the DAT files to calculate plastic gears using the values for their own materials. They can then use the plastics manager (see chapter 61) to create new .dat files.

As defining the permitted root and flank limiting values takes so much time and effort, and because these values are often not present, KISSsoft can also perform the calculation using very basic material properties (e.g. a static calculation can be performed if tensile strength data is present).
As additional information, the name of the plastic includes an overview of the data that is available for calculating plastic gears.

The data used to calculate plastic gears is available in this format: **[S B Fog Wd]**.

Abbreviations used here:

S - The material's maximum or yield material strength is provided for calculating static root strength

B - S-N curves (Woehler lines) are provided for calculating the root infinite life strength

F - S-N curves (Woehler lines) for all lubrication types are provided for calculating the tooth flank infinite life strength

Fo - S-N curves (Woehler lines) for oil lubrication are provided for calculating the tooth flank infinite life strength

Fg - S-N curves (Woehler lines) for grease lubrication are provided for calculating the tooth flank infinite life strength

Fd - S-N curves (Woehler lines) for a dry run are provided for calculating the tooth flank infinite life strength

Fog - S-N curves (Woehler lines) for oil and grease lubrication are provided for calculating the tooth flank

W - Wear coefficients for all lubrication types are provided for calculating wear

Wo - Wear coefficients for oil lubrication are provided for calculating wear

Wg - Wear coefficients for grease lubrication are provided for calculating wear

Wd - Wear coefficients for a dry run are provided for calculating wear

C - S-N curves (Woehler lines) are available for calculating the infinite life strength of the tooth root in crossed helical gears.

► **Note:**

When a calculation method according to Niemann or VDI is selected, the tooth root, tooth flank and wear are calculated automatically, if the data for the calculation is present. If no data is present for one or more of these methods, only the calculations for which data is available are actually performed.

Converting hardness to infinite life strength values σHlim, σFlim

When you enter data for your own material, the hardness is converted into the infinite life strength values σHlim, σFlim. To open the conversion dialog, click the appropriate conversion button next to the input fields for the infinite life strength values σHlim, σFlim. The data is converted in accordance with the ISO 6336-5:2003 formula described in section 5.

(The data for forged steels is used for "unalloyed/through hardened" and "alloyed/through hardened" heat treatable steels.)

σHlim, σFlim=A*x+B

x: Hardness value in the unit used in the table (depending on the HV or HBW material type)

A,B: Factors for the particular material type and processing. (from Table 1, ISO 6336-5)

Values for σHlim and σFlim that are required for the conversion specified in ISO 6336-5 are displayed directly in the material screen under "Own input" if these values are possible for the specified hardness and material type.

In the next conversion dialog, click on another conversion button next to the hardness input field to start converting the hardness value. In the case of materials that are not alloys you can calculate the hardness from the tensile strength value or other hardness values.

Shot peened gear

Selecting **Shot peened** only affects the alternating bending factor *Y*M if this is calculated according to ISO 6336-3, Annex B. This information is primarily required for documentation purposes, so that this data can be added to drawings. It is well known that shot peening improves the root safety factor. Until now, the standards for gear strength have not included any suitable data. However, if measurements have shown the extent to which σFlim is increased by shot peening, this effect can be taken into account either with the technology coefficient *Y*T or by increasing σFlim.

15.1.12.2 Calculation of the wear coefficient kw for steel

According to Niemann [7], Table 21.6/5, and Plewe's dissertation [9], which calculates an approximate guide value for coefficient of wear, kw depends on the size of the lubricant gap in the operating pitch circle hc. The function defined by Plewe, kw = f(hmin), is valid for standard mineral oil without high pressure additives.

You should take care when using this guide value, because the existing information is far from complete. In particular, very little is known about the influence of surface roughness and the influence of lubricant additives. You should take careful measurements to check the wear factor to ensure reliable results from the calculations.

Lubricant influence factor: As stated in [7], adding suitable additives to a lubricant can significantly reduce the amount of wear. The lubricant's influence factor can therefore lie in a range between 0.01 and 1.

Material influence factor Plewe took measurements from various different material pairings: Gear made of through hardening steel paired with a hard or soft counter gear, gear pairs made of case-hardening steel, and gear pairs made of nitriding steel. The value of kw as defined by Plewe was then determined for these combinations. The influence factor can be used for other combinations, if known. For more information, see [7].

15.1.12.3 Lubrication

Select the lubricant from a list. If you select **Own input**, click the Plus button to specify your own lubricant.

If you see the note (with k_w info) after the lubricant description, this means a lubrication influence factor k_{wlub} is present for this lubricant. This factor can then be used to determine the wear coefficient k_w more accurately.

You can select oil bath or oil injection lubrication, or grease lubrication, or none at all (dry run). You can select dry run only when using a calculation method for plastics.

Click on the Plus button to the right of the lubrication drop-down list to display the **Define temperatures** window.

This is where you can define either the lubricant temperature or the root and flank temperatures for plastics and dry run. These temperatures are calculated for plastics. You can also deactivate the calculation and enter your own temperature values.

You can select either a closed, semi-open or open gear unit. This has an effect on the temperature calculation according to VDI 2545 and VDI 2736.

The power-on time is also taken into account for plastic gears when calculating the flank and root temperature. For worm gear units, this time is also included when calculating the thermal safety.

Note about calculating temperature:
It is assumed that heat is constantly dissipated and that heat is only generated during the specified power-on time. The precondition for this is that the gear unit is only run for a short period of time (maximum 15 minutes), and is then stopped again. If this is not the case, the power-on time must be set to 100%.

15.1.12.3.1 Calculating the required amount of lubricating oil

When the injection lubrication method is used, the required amount of lubricating oil is calculated as specified by Schlecht [10]. This assumes a difference of 10°C between the temperature of the oil at the inlet and outlet. The specific heat capacity c_p (Ws/(kg*K) and the specific weight at operating temperature are defined as specified by Niemann [8].

15.2 Load

The **Rating** (load) input window is one of the standard tabs (see chapter 5.1, Standard and special tabs) and is subdivided into the two groups **Strength** and **Load spectrum**.

15.2.1 Calculation methods

In the drop-down list, you can select the following strength calculation methods:

- 1. **Geometry calculation only:** If the **Rating** module is not selected in the **Calculation** menu, only the geometry is calculated.
- 2. **Static calculation**: Unlike DIN 743 which, for example, has a specific method for static shaft calculations, ISO 6336 does not have its own calculation method for static calculation. In a static calculation, the nominal stress is usually compared with the permitted material parameters (yield point and/or tensile strength). This performs a static calculation of cylindrical gears in KISSsoft. In this calculation, the nominal stress in the tooth root (calculated using the tooth form factor Y_F) is compared with the yield point and tensile strength. (see chapter 15.2.1.1, Static calculation).
- 3. **ISO 6336:2019 Method B** (Calculation of load capacity of spur and helical gears**:** Method B is used for this calculation.
- 4. **ISO 6336: 2006**: Earlier version of ISO 6336, no longer valid.
- 5a. **DIN 3990 Method B**: Y_F Method B(Calculation of load capacity of cylindrical gears). This calculation is also performed using Method B. However, either Method B or Method C can be used to calculate the tooth form factor (we recommend Method C for internal toothings. Otherwise, use Method B).
- 5b. **DIN 3990 Method B**: Y_F Method C
- 6. **DIN 3990, Part 41 (Vehicle Transmission), Method B** (Load capacity calculation for vehicle transmissions)**:** Method B is used for this calculation. Two application factors must be transferred to form the load spectra (see chapter 15.3.1, Application factor).
- 7. **AGMA 2001-C95**: The standard has been implemented in its entirety. The dynamic factor and the face load factor are calculated in accordance with AGMA recommendations. The geometry factors (for tooth root and flank) are calculated entirely in accordance with ANSI/AGMA 908-B89. In addition to all the relevant intermediate results, the following values are also supplied: Pitting Resistance Power Rating, Contact Load Factor, Bending Strength Power Rating, Unit Load for Bending Strength, Service Factor.
  This calculation can also be used for every other cylindrical gear configuration (including planetary stages). However, it must be noted that the AGMA standard does not permit tooth root strength for internal gear pairs to be calculated directly. In this case the calculation must be performed using the graphical method (see chapter 15.2.7, Strength details (AGMA)).
- 8. **AGMA 2001-D04**: Most recent version of AGMA 2001. Differs only slightly from the previous version C95.
- 9. **AGMA 2101-D04** (Metric Edition)**:** Equivalent to AGMA 2001-D04, but all values in SI units.

- 10. **Special AGMA standards (6004-F88, AGMA 6014-A06):** Special standards used in the USA to calculate the strength of open gear rims. These calculation methods are based on the AGMA 2001 or 2101 basic standards. However, some factors have been specifically defined for special applications. AGMA 6014 replaces the old AGMA 6004; but both methods are still available because AGMA 6004 is still requested.
- 11. **AGMA 6011-I03 for turbo drives (High Speed Helical Gear Units):** The AGMA 6011 standard is a special edition for high-speed gear units and is less complex than the AGMA 2001 (or metric AGMA 2101) basic standard. In this case, "less complex" means that some data is already predefined. For example, AGMA 2001 has the options "Open gearing", "Commercial gear unit" and "Precision gear unit" for defining the face load factor, whereas AGMA 6011 has "Precision gear unit" as a predefined requirement. AGMA 6011 also provides information to help you select the application factor K_a for specific turbo-driven applications and other useful notes about this type of gear unit (lubrication arrangement etc.). It is therefore always possible to perform the calculation according to AGMA 6011 using AGMA 2001 or 2101 without causing any problems. To input data correctly for AGMA 2001, as implemented in KISSsoft, that is also correct for AGMA 6011, you must be aware of the constraints, and take them into consideration when entering the parameters. Select the AGMA 6011 method to save the user having to do this. In this situation, the program checks whether all the constraints are set. If not, the system displays a message prompting the user to confirm that modifications are to be made.
- 12. **API 613:2021 Special-purpose Gears for Petroleum, Chemical, and Gas Industry Services:** According to API 613, the calculation must be performed according to AGMA 2001. Special limitations in terms of material type, hardness, quality, required life, and face load factor are considered in this strength calculation method. More details about how to perform a calculation correctly are provided in the Instructions that describe the necessary inputs and checks: kisssoft-anl-078-E-CylindricalGears API613.docx.
- 13. **AGMA 6015-A13 for rolling mill gears:** The AGMA 6015 standard is a special edition for rolling mill gears and is less complex than AGMA 2001 basic standard. In this case, "less complex" means that some data is already predefined. For example, AGMA 2001 has the options "Open gearing", "Commercial gear unit" and "Precision gear unit" for defining the face load factor, whereas AGMA 6015 has "Precision gear unit" as a predefined requirement. Other fundamental restrictions are listed in Chapter 1 of the standard. Select the AGMA 6015 method to save the user having to do this. In this situation, the program checks whether all the constraints are set. If not, the user sees a prompt asking them if they want to make any modifications.
  The permitted material properties for bending (sat) and pitting (sac) specified in AGMA 6015 for the same material are different from the properties given in AGMA 2001. The values must be defined by the user in accordance with Table 3 (sac) and Table 4 (sat) in AGMA 6015 and then input in the program (set material to **Own input**)!
  AGMA 6015 provides conditions for "Service factors" in Annex C. The conditions are for

information purposes only (not binding) and must be discussed and agreed with the customer. Input the coefficients by selecting **Module specific settings** > **required safeties** and then clicking **Service factors > Service factors for AGMA**.

- 14. **GOST-21354-87:** Calculation according to the Russian guideline (latest edition, 1987). Take the following notes into account, (see chapter 15.2.1.2, GOST 21354-87).
- 15. **Plastic according to Niemann:** Please refer to [7] and Table 13.3 to see the differences.
- 16. **Plastic according to VDI 2545 (Y_F Methode B)** (thermoplastic materials used in gears)**:** This method has been withdrawn and replaced by the new method, according to VDI 2736. This regulation defines how calculations are performed on gears made of plastic or combinations of plastic and steel. (see chapter 15.2.1.3, Plastics according to Niemann, VDI 2545 or VDI 2736).
- 17. **Plastic according to VDI 2545 (Y_F Methode C)**: In this calculation method, the tooth form factor $Y_F$is calculated according to Method C.
- 18. **Plastic according to VDI 2545-modified (Y_F Method B):** This method was recommended for use by KISSsoft before VDI 2736 was published. VDI 2736 contains all the modifications recommended according to Tables 13.3 and 13.4. This method is recommended for plastics with normal toothing. Transverse contact ratio $\varepsilon_\alpha < 1.9$. See Table 14.4 for differences between VDI und VDI-modified.
- 19. **Plastic according to VDI 2545-modified (Y_F Method C):** This method is recommended for plastics with deep tooth forms. Transverse contact ratio $\varepsilon_\alpha > 1.9$. See Table 14.4 for differences between VDI und VDI-modified. See Table 14.4 for the differences between VDI und VDI-modified. In this calculation method, the tooth form factor Y_F is calculated according to Method C.
- 20. **Plastic according to VDI 2736:** We recommend you use this calculation method, VDI 2736, which was published for the first time in 2014/15. It includes all methods described in Sheet 2 of VDI 2736 (empirical calculation, tooth root, tooth flank, deformation, wear).
- 21. **As in FVA program (DIN 3990)**: Supplies the same results as the German Research Association for Power Transmission Engineering (FVA) Reference Program. Based on DIN 3990 Method B with minor differences. KISSsoft regularly performs comparisons between calculation examples using the FVA's STplus cylindrical gear calculation method. The first comparison was performed in 2002, with STplus edition 1988. The next was in 2003 (with STplus 3.2). As new investigations mean that STplus differs slightly in some places from DIN 3990, it was decided to implement this calculation approach as "similar to FVA" in KISSsoft. The most recent comparison was performed with STplus 6.0 in 2016. As the DIN 3990 standard has remained unchanged since 1985, the results for the various different programs have also remained much the same.
- 22. **BV/Rina FREMM 3.1 Naval Ships and Rina 2010 (ISO 6336):** calculation guidelines for ships' engines.

- 23. **DNV41.2, calculation guideline for ships' engines:** In principle, the Det Norske Veritas calculation guideline [11] for ships' engines corresponds to ISO 6336 (root, flank) and ISO/TS 6336-20/21:2022 (scuffing). However, it does have some significant differences, especially where S-N curves (Woehler lines) are concerned. These differences are detailed in our kisssoft-anl-076-DE-Application_of_DNV42_1.pdf information sheet, which is available on request.
- 24. **Lloyd's Register, classification for ships:** Calculation guidelines for ships' engines.
- 25. **ISO 13691High-speed special-purpose gear units:** Calculation guidelines for high-speed gear units

15.2.1.1 Static calculation

Each coefficient (application factor, face load factor, transverse coefficient, dynamic factor) is set to 1.0. The load at the tooth root is calculated with the tooth form factor according to ISO 6336 Method B and the helix angle (without the stress correction factor).

$\sigma_{F0} = \frac{F_t}{b_{eff} \cdot m_n} \cdot Y_F \cdot Y_\varepsilon \cdot Y_\beta$	(12.1)
$\sigma_F = K_A \cdot K_V \cdot K_{F\alpha} \cdot K_{F\beta} \cdot \sigma_{F0}$	(12.2)

It also calculates the local tooth root stress multiplied by the stress correction factor YS . This stress is approximately the same as the normal stress calculated in an FEM model. This stress is then also output in the report:

$\sigma_{F0} = \frac{F_t}{b_{eff} \cdot m_n} \cdot Y_F \cdot Y_S \cdot Y_\varepsilon \cdot Y_\beta$	(12.3)

The stress correction factor YS is used to take into account a local stress concentration in the root radius.

In the static load case, the material can expand locally in this area, which does not yet lead to failure, but results in a stress redistribution, thereby reducing the highest stress.

In this situation of stress redistribution, the factor YS is no longer required because the stress concentration, which is taken into account by YS, has been eliminated.

Therefore, in general, static safety factors can be calculated without the factor YS. For this reason, the safety factors shown in the results window are the results calculated without the factor YS.

If you have a very critical application, e.g. an aerospace gearbox, the factor YS can still be included in order to obtain a more conservative result. For this case, the listed results do include YS in the main report.

15.2.1.2 GOST 21354-87

Quality according to GOST 21354-87
GOST only takes into account one quality, which is why the poorer quality of the two gears is used during the calculation.
Q = max (Q1, Q2)

Infinite life strength values for root and flank
The infinite life strength values σ_{Flim} and σ_{Hlim} are saved in the KISSsoft database, or you can enter them by selecting the **Own Input** option.

Infinite life strength for root σ_{Flim}
The infinite life strength σ_{Flim} is calculated as follows, according to GOST:
$\sigma_{Flim} = \sigma_{Flim0} * Y_z * Y_g * Y_d * Y_A * Y_T$
σ_{Flim0} – nominal infinite life strength at limit load cycle (GOST 21354-87 Tables 14-17).
Y_z - blank coefficient (GOST 21354-87 Table 13, Formula 10.3).
Y_d – takes into account the hardening of the root transition zone (GOST 21354-87 Tables 14-17).
Y_g – takes into account the grinding of the root transition zone (GOST 21354-87 Tables 14-17).
Y_T – technology factor (GOST 21354-87 Table 13, Formula 10.2). The default technology factor setting is 1.0, but you can change it in KISSsoft by selecting Factors > **Z-Y-Factors**.
Y_A – alternating bending factor (GOST 21354-87 Table 13, Formula 10.6). The default alternating bending factor setting is 1.0, but you can change it to **Own Input** in KISSsoft by selecting Factors > **Alternating bending factor**.
The Y_d, Y_g, Y_z factors cannot be entered in KISSsoft and must be included directly when the infinite life strength is entered.
In addition to the factors mentioned above, the infinite life strength defined in GOST must be divided by 2.0 before being entered in KISSsoft. In the calculation, σ_{Flim} is then multiplied by the stress correction factor $Y_{ST} = 2$, in a similar way to in ISO or DIN.

Consequently, the correct entry for σ_{Flim} in KISSsoft, for calculations according to GOST, is:
σ_{Flim} (KISSsoft entry) = σ_{Flim0} (according to GOST) $* Y_d * Y_g * Y_z / 2.0$

Required minimum safeties
GOST has the special property that the minimum safety set for tooth root fracture and the flank depends on the material type and the surface hardening. For this reason, you can enter minimum safeties for every gear individually, for GOST, under **Settings** > **Required safeties**.

Information about root rounding
In various GOST formulae, a distinction is made between whether the root rounding is ground or not. To make this distinction, you must select **Details for root and flank strength calculation** and enter a suitable value.

Face load factor flank $K_{H\beta}$
The face load factor (flank) is calculated according to GOST, Table 6, Formula 7. The considerations described in GOST 21354-87 Annex 6 are ignored.

Face load factor root $K_{F\beta}$
The face load factor (root) is calculated according to GOST 21354-87 Table 13, Formula 4.

Dynamic factor K_v
The dynamic factor is calculated according to GOST 21354-87 Table 6, Formula 6. If conditions (34) and (35) specified in formula 6 are not fulfilled, KISSsoft calculates the dynamic factor according to GOST 21354-87 Annex 5.

Load spectra
Calculations with load spectra are performed using the rules defined by Palmgren-Miner, according to ISO 6336-6.

Safety of the hardened layer
The safety of the hardened layer is calculated according to DNV 41-2.

15.2.1.3 Plastics according to Niemann, VDI 2545 or VDI 2736

The calculation methods used for plastics take special account of the fact that these materials are very sensitive to extremes of temperature. The types of lubrication used here include oil, grease or none at all (dry run). The acceptable load for each material is calculated from figures in data tables, in DAT format, while taking into consideration the local temperatures at the tooth flank and root, and the number of load cycles. The local temperature can be calculated when grease is used as the lubricant or during a dry run. However, when oil is used as the lubricant, the oil temperature is used as the local temperature. The calculation is performed for combinations of plastic/plastic and also steel/plastic. The acceptable deformation is also checked. KISSsoft supplies data for the following materials:

- Polyamide (PA12, PA6, PA66, PA46)
- Polyacetal (POM)
- Polyetheretherketone (PEEK)
- Polybutylene terephthalate (PBT)
- Polybutylene terephthalate (PET)
- Laminate
- Data about other materials is available on request

All the specific properties of each material are stored in text tables (DAT files) to enable the integration of own materials (see chapter 9, Database Tool and External Tables). The strength of plastics can be calculated either as defined by Niemann [12], or according to VDI 2545 (1981*) [13] or VDI 2736 [14]. You can also use the modified calculation method as detailed in VDI 2545. This calculates the stress using the tooth root stress correction factor Y_S. The major differences between the two methods are:

*Calculation method VDI 2545 has been withdrawn and replaced by VDI 2736.

Root	Niemann	VDI 2545	VDI 2545-mod.	VDI 2736
Y_F	C	B or C	B or C	C
Y_S	DIN 3990	1.0	DIN 3990	DIN 3990
Y_ε	1.0 [8)]	$1/\varepsilon_\alpha$ [7) 9)]	$1/\varepsilon_\alpha$ [7) 9)]	DIN 3990
Y_β	1.0	DIN 3990[10)]	DIN 3990 [10)]	DIN 3990
σFE	2 *σ_{Flim}	σ_{Flim}	2 *σ_{Flim}	2 *σ_{Flim}

15.3 table: Differences between the calculation methods used to calculate the root safety factor for plastics

Flank	Niemann	VDI 2545	VDI 2545-mod.	VDI 2736
Z_ε	1.0	DIN 3990	DIN 3990	DIN 3990
Z_V	DIN 3990 [5) 10)]	1.0	1.0	1.0
Z_R	DIN 3990 [6) 10)]	1.0	1.0	1.0

15.4 table: Differences between the calculation methods used to calculate the tooth flank load capacity for plastics

Tooth deformation: Very different calculation methods!
[5)] only for laminated wood, otherwise 1.0
[6)] only steel/plastic combinations, otherwise 1.0
[7)] For tooth form factor Y_F as defined in Method B: 1.0
[8)] the method sets the face contact ratio for the tooth root stress to the value 1.0. According to Niemann, this is because the material data is not always precise. The formulae used in VDI 2545 correspond to those used in ISO 6336:1996.
[9)] For crossed helical gears = $0.25 + 0.75/\varepsilon\gamma$
[10)] For crossed helical gears = 1.0

15.2.1.4 Calculation method for scuffing

Scuffing is calculated according to flash temperature and integral temperature method using the following selection options:

- **DIN 3990-4**

If the strength calculation is performed according to DIN, scuffing is always calculated using DIN 3990-4.

- **ISO/TS 6336-20/21:2022**

In all other cases, where DIN 3990-4 is not used, the calculation of scuffing is always performed according to ISO/TS 6336-20/21:2022.

Contrary to DIN 3990-4, the following formulae are used for the tooth bulk temperature (analogous to ISO/TS 6336-20/21:2022):

$$\lambda_{MC} = \lambda_{oil} + X_S \cdot 0.70 \cdot \lambda_{flaint}$$

$$\lambda_{MB} = \lambda_{oil} + X_S \cdot 0.47 \cdot \lambda_{flamax}$$

For injection lubrication, X_S=1.2 (otherwise 1.0). There is little point in multiplying the oil temperature (λ_{oil}) by the coefficient as specified in DIN 3990-4.

- **DIN 3990-4, similar to STplus**

STplus (Version 6.0) uses the original formulae according to DIN 3990-4 for the tooth bulk temperature. Contrary to DIN 3990-4, the dynamic oil viscosity η_M is calculated with the oil temperature (instead of the tooth bulk temperature).

15.2.1.5 Calculation method for micropitting

Micropitting is calculated according to ISO/TS 6336-22 (previously known as ISO/TR 15144-1). Further information,(see chapter 14.1, Underlying principles of calculation) and (see chapter 23.5.4.4, Safety against micropitting).

15.2.1.6 Calculation method for tooth flank fracture

Tooth flank fracture appears in the area of the active tooth flank instead of in the area of the highest bending stress at the 30° tangent.

Tooth flank fracture (TFF) can be calculated according to the draft ISO Technical Specification ISO/TS 6336-4. Earlier investigations performed by Dr Annast in Munich [15] have later been updated and expanded by others. Witzig [16] has put together a first draft of ISO/TS 6336-4. Important: TFF as specified in ISO/TS 6336-4 can only be applied for case-hardened materials.

Click on the Plus button to enter the data required for tooth flank fracture.

Hardening depth (HD)
You can input the intended hardening depth (for hardness HV400, for nitrided steels, or HV550 for all other steels). You can also input the hardness HV300. This value is then used to display the hardening curve as a graphic. The input applies to the depth measured during final machining (after grinding).

When you input this data, the safety of the hardened surface layer is calculated automatically according to DNV 41.2 [11]. A minimum value of t400 (nitrided steel) or t550 (all other steels) is used here. If only the value for HV300 is known, this value is then used. However, the calculation should then only be seen as an indication. The calculation is performed as described in the "Subsurface fatigue" section in [11]. The values required to define the CHD hardening depth coefficient Y_C, as specified in DNV 41.2, are also needed. The calculation does not use the same approach as the calculation for the proposal for the recommended hardening depth, but still returns similar results. To obtain a proposal for a sensible hardening depth, we recommend you call the relevant calculation by selecting **Report -> Proposals for hardening depth**. A maximum value for the hardening depth is only used to check the hardening depth at the tooth tip. It is mainly used for documentation purposes.

There are three calculation options:

- Use a hardness file for the gear material, if this file already exists in the database
- Select an independent file with the hardness information,
- Directly enter the core hardness and a method for generating a theoretical hardness curve according to Lang or Thomas (as in ISO/TS 6336-4)

Using measured hardening curves for tooth flank fracture according to ISO/TS 6336-4

Evaluations of measurements taken at wind power installations by Vestas (2017) in the ISO TC60-WG6 committee have shown that reliable results cannot be obtained using measured hardness curves according to the method specified in ISO/TS 6336-4 (due to the scatter of the individual measuring points). We recommend that you use the theoretical hardness curve defined by Thomas (or Lang) to approximate the measured hardness curve and then use this value in the calculation.

```
-- index number     depth (mm)  hardness (HV)
-- these lines are comments
DATA
1   0.2 700
2   0.3 675
3   0.4 650
4   0.5 625
5   0.6 600
*   *   *
*   *   *
*   *   *
END
```

Figure 15.5: Structure of the hardness file (important: depth values must be entered in mm)

The results of the tooth flank fracture calculation are given in a report (select "Report" -> "Tooth flank fracture").

15.2.2 Service life

Enter the required service life directly in the input field.
Click the Sizing button to size this value. This process uses the minimum safety value for the tooth root and flank strength to calculates the service life (in hours) for every gear and for every load you specify. The service life is calculated according to ISO 6336-6:2006 using the Palmgren-Miner Rule. The program displays the system service life and the minimum service life of all the gears used in the configuration. Click the Sizing button to size the service life either with or without defining a load spectrum (see chapter 15.2.8, Define load spectrum).

15.2.2.1 Number of load cycles

KISSsoft calculates the number of load cycles from the speed and the required service life. If you want to influence the value, you can define it in the **Number of load cycles for gear** n **window**. Click the Plus button to access this. Here, you can select one of five different calculations for calculating the number of load cycles.

1. **Automatically** The number of load cycles is calculated automatically from the rating life, speed, and number of idler gears.
2. **Number of load cycles** Here, you enter the number of load cycles in millions. You must select this option for all the gears involved in the calculation, to ensure this value is taken into account.
3. **Load cycles per revolution** Here you enter the number of load cycles per revolution. For a planetary gear unit with three planets, enter **3** for the sun and **1** for the planets in the input field.

   Note:
   If the **Automatically** selection button in the calculation module is selected, KISSsoft will determine the number of load cycles, taking into account the number of planets, in the **Planetary stage** calculation module.
4. **Load cycles per minute** Here you enter the number of load cycles per minute. This may be useful, for example, for racks or gear stages where the direction of rotation changes frequently, but for which no permanent speed has been defined.
5. **Effective length of rack** The rack length entered here is used to calculate the number of load cycles for the rack. The rack length must be greater than the gear's perimeter. Otherwise, the calculation must take into account the fact that not every gear tooth will mesh with another. You must enter a value here for rack and pinion pairs. Otherwise the values N_L(rack) = N_L(pinion)/10 are set.

► **Note**

This calculation method is used for transmissions that only travel over one oscillation angle.

Assume a scenario in which a reduction is present,

$$i = \frac{z_2}{z_1}$$

and an oscillation angle w in [°] from gear 2, where gear 2 constantly performs forwards and backwards movements with the angle value w_2. The effective endurance is given as the service life. The two coefficients f_{NL1} and f_{NL2}, which modify the absolute number of load cycles, N_L, are now calculated. To do this:

- a) Set the alternating bending factor of the pinion and gear to 0.7, or calculate it as defined in ISO 6336-3:2006. In this case, one complete forwards/backwards movement is counted as one load cycle.
- b) Coefficients f_{NL1} and F_{NL2} for pinion and gear are defined as follows:

$$f_{NL1,2} = \frac{ROUNDU\,P(\frac{W_{1,2}}{360})}{2 * \frac{W_{1,2}}{360}}$$

- w_2 = oscillation angle gear 2
- w_1 = W2*i
- ROUNDUP = round up to a whole number

The value in the counter displays the actual number of loads that occur during a complete cycle (forward and backward oscillation) on the flanks (not teeth) that are most frequently subjected to load. By rounding up this number to the next whole number, every rotation recorded is counted as a load.

Then, to determine the required $f_{NL1,2}$ factor, the actual number of loads that occur per flank is divided by the number of loads that would occur per cycle, if rotation were to continue without a backward rotation at the angle of rotation (1 load for each 360°).

Example calculation for $f_{NL1.2}$:

Gear 1 rotates through a half cycle at **540°** while gear 2 oscillates by **90°** (i = 6).

In a complete cycle, the oscillation angle moves forwards once an backwards once.

The actual number of load cycles that occur in a complete cycle on the flanks that are most frequently subjected to load (only one side of the tooth is taken into consideration) is then:

For gear 1:

$$ROUNDU\,P(\frac{540}{360}) = 2$$

For gear 2:

$$ROUNDUP\left(\frac{90}{360}\right) = 1$$

Without adjusting the coefficients, the number of counted load cycles in a complete cycle would then be:

For gear 1:

$$2 * \left(\frac{540}{360}\right) = 3$$

For gear 2:

$$2 * \left(\frac{90}{360}\right) = 0.5$$

The coefficients are therefore f_{NL1} and f_{NL2}:

$$f_{NL1} = \frac{2}{3} = 0.667$$

$$f_{NL2} = \frac{1}{0.5} = 2$$

- c) Then, input coefficients fNL1 and fNL2 in the **Load cycles per revolution** input field.

The strength calculation can now be performed for the correct number of load cycles, on the basis of the data entered in steps a through d.

15.2.3 Reliability

Reliability is calculated according to Bertsche's study [17], in which the possible methods have been described in great detail. The most commonly used approach, and one which is well suited to the results that can be achieved in "traditional" mechanical engineering calculations, is "Weibull distribution". In this case, Bertsche recommends the use of 3-parameter Weibull distribution. The reliability R of a machine element is calculated as a function of the number of load cycles t using the following equations.

$$R(t) = e^{-\left(\frac{t-t_0}{(T-t_0)}\right)^{\beta}} * 100\%$$

Parameters T and t_0 can be derived from the mathematically achievable service life of the component, H_{att}, as follows (with F_O according to the calculation method, Table 1, β and f_{tB} from Table 2 according to Bertsche):

Calculation method	**Damage probability Fo**			
	1%	10%	Other	Comment
Shaft, DIN 743			2.5%	Assumed, not documented

Shaft, FKM guideline			2.5%	
Shaft, AGMA 6001	*			If kC = 0.817
Rolling bearing, ISO 281		*		If coefficient a1 = 1.0
Tooth flank, ISO 6336; DIN 3990	*			
Tooth root, ISO 6336; DIN 3990	*			
Tooth flank, AGMA 2001	*			If randomness factor KR=1
Tooth root, ISO 6336; AGMA 2001	*			If randomness factor KR=1

Table 15.5: 14.6a: Damage probability for different calculation methods when determining material properties

$$T = \left(\frac{H_{att} - f_{tB} \cdot H_{att10}}{\sqrt[\beta]{-\ln\left(1 - \frac{F_o}{100}\right)}} + f_{tB} \cdot H_{att10} \right) \cdot fac$$

$$t_0 = f_{tB} \cdot H_{att10} \cdot fac$$

with

$$H_{att10} = \frac{H_{att}}{(1 - f_{tB}) * \sqrt[\beta]{\frac{In(\frac{1 - F_o}{100})}{In(0.9)}} + f_{tB}}$$

Factors and parameters for calculating reliability according to Bertsche [17].

	Coefficient ftB	**Weibull shape parameter** β
Shafts	0.7 to 0.9 (0.8)	1.1 to 1.9 (1.5)
Ball bearing	0.1 to 0.3 (0.2)	1.1
Roller bearing	0.1 to 0.3 (0.2)	1.35
Tooth flank	0.4 to 0.8 (0.6)	1.1 to 1.5 (1.3)
Tooth root	0.8 to 0.95 (0.875)	1.2 to 2.2 (1.7)

Table 15.6: 14.6b: Coefficients for a Weibull distribution according to Bertsche. The mean values used in KISSsoft are given in brackets.

► **Note:**

Calculating reliability using Weibull distribution uses the calculated service life, and so also takes into account the required safeties. To calculate reliability without taking required safeties into account, set the safeties to 1.0.

15.2.4 Peak overload factor

The peak overload factor K_{AP} according to DNV 41.2 is used when calculating scuffing, when a short-period high overload, which is not covered by the application factor K_A, occurs. In this case, the root and flank are calculated with K_A, but the scuffing is calculated with K_{AP}. This procedure is especially useful in the case of special applications. It enables the methodology to also be used for calculations according to the ISO standard.

15.2.5 Power, torque and speed

Click the Sizing button next to the power input field (for the torque) to calculate the power (torque) so that a predefined minimum safety (see chapter 15.22.5, Safety factors) can be maintained.

Power, torque and speed must always be defined with a positive value. Enter the working flank to predefine the sense of rotation.

The Plus button next to the speed input field now becomes visible for planetary stages. You can then input a second speed value (in addition to the speed of the reference gear). You can enter the speed as either a positive or negative value. A positive value means that the second gear rotates in the same direction as the selected reference gear. A negative value means it rotates in the opposite direction.

► **Note:**

The Plus button is also active for gear pairs when you are modeling epicyclic gears with gear pairs in KISSsys, as it is when you call KISSsys. You can then enter the speed of the planet carrier [nSteg]. The main speed [n] of the reference gear with n - nweb is then used in the calculation. This returns the exact number of load cycles.
You cannot change the reference gear, torque, speed or power if the load spectrum with values is defined.

15.2.6 Strength details

Click on the **Details** button for the root and flank strength calculation to display a dialog in which you can make additional settings for the strength calculation. Note that a different window layout is used for calculation methods according to AGMA (see chapter 15.2.7, Strength details (AGMA)). Additional settings are available in the corresponding Plus buttons for scuffing, tooth flank fracture, micropitting and subsurface fatigue.

15.2.6.1 Allow simplified calculation according to DIN 3990/ISO 6336

If you select this option, you can use the calculation methods for steel gears to calculate plastic gears. This calculation is performed according to the infinite life strength values listed in the materials database. The values for the plastics given in this database apply for oil lubrication, a temperature of 70°C and number of load cycles 10^8. In contrast to the calculation according to VDI 2545 or VDI 2736, the strength values do not depend on the temperature and lubrication type.

The calculation is performed in the same way as for through hardening steel, with the appropriate S-N curve (Woehler line) as defined in ISO 6336.

15.2.6.2 Checking contact stress σHB and σHD for both gears

According to ISO 6336 (or DIN 3990), the Hertzian pressure is only monitored for the driving gear at the single tooth contact point B, and in D the pressure is only monitored for the driven gear. This option can be used to check both gears at points B or D, depending on which one is subject to the greater Hertzian pressure.

15.2.6.3 Strength calculation using average tolerance field (of tooth form)

By default, values for theoretical gear teeth (without allowances) are referenced for calculation. When you select this checkbox, KISSsoft performs the calculation with the average allowances for the center distance, root circle and tooth thickness. This option is suitable for use where large tolerances are present.

15.2.6.4 Safety factor root with gear rim influence (VDI 2737)

The strength calculation for internal toothings is not very accurate. A significant improvement is needed. Gear rims are often subject to stresses that can affect their load capacity. At present, VDI 2737 is the only guideline that includes gear rim stress and the influences associated with this. The calculation is performed in two steps

1. Tooth root fracture safety (static and endurance) without taking the gear rim influence into account.
2. Tooth root fracture safety with gear rim influence. In this case, the maximum shear stress in the tooth root outside the meshing can in some conditions be greater than the actual bending stress in the tooth that is under load.

As in ISO 6336:2006, the notch factor Y_S is determined at the place at which the tangent on the flank and the tooth center line form an angle of 60°.

The results of the calculation specified in VDI 2737 are detailed in their own section in the normal report.

Factor for maximum load (VDI 2737)

To calculate static safety in accordance with VDI 2737, input a maximum load factor. This is then multiplied with the nominal torque. To calculate the infinite life strength, the nominal torque is, as usual, multiplied with the application factor K_A.

15.2.6.5 Profile modification

You can modify the theoretical involute in high load capacity gears by grinding the toothing. You will find suggestions for sensible modifications (for cylindrical gears) in KISSsoft module Z15 (see chapter 15.7, Modifications).

The type of profile modification has an effect on transverse coefficients $K_{H\alpha}$ and $K_{H\beta}$ and on the way scuffing safety is calculated. The load sharing factor $X\gamma$ is calculated differently depending on the profile modification. The main difference is whether the profile has been modified or not. However, the differences between the versions for **high load capacity gears** and for **smooth meshing** are relatively small. The strength calculation standard presumes that the tip relief C_a is properly sized, but does not provide any concrete guidelines. The load sharing factor $X\gamma$ is calculated as follows, depending on the type of profile modification according to DIN 3990:

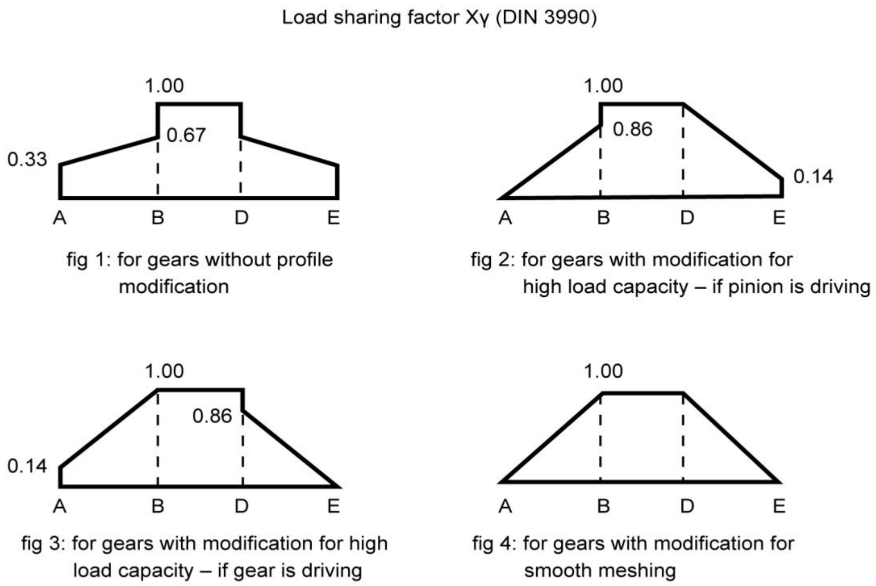


Figure 15.6: Force distribution factor Xγ for different profile modifications

15.2.6.6 Life factors as defined in ISO 6336

Set the life factor Z_{NT} to reduce the permitted material stress according to ISO 6336-2:2006:

$$\sigma_{H\,\lim red} = Z_{NT} \cdot \sigma_{H\,\lim}$$

$$\sigma_{F\,\lim red} = Y_{NT} \cdot \sigma_{F\,\lim}$$

As stated in ISO 6336, this value is important for cylindrical gear calculations and is the reason for the lower safeties for the range of endurance limit, compared with DIN 3990.

1. **normal (reduction to 0.85 at 10^{10} cycles):** The permitted material stress in the range of endurance limit (root and flank) is reduced again. The life factors Y_{NT} and Z_{NT} for $\geq 10^{10}$ load cycles are set to 0.85.
2. **increased if the quality is better (reduced to 0.92):** Y_{NT} and Z_{NT} are set to 0.92 for $\geq 10^{10}$ load cycles (in accordance with the data in ISO 9085).
3. **with optimum quality and experience (always 1.0):** This removes the reduction and therefore corresponds to DIN 3990. However, this assumes the optimum treatment and monitoring of the materials.

15.2.6.7 Modification of S-N curve (Woehler lines) in the range of endurance limit

In a standard Woehler diagram, the range of endurance limit is reached at a particular number of load cycles. From this point onwards, the dynamic strength no longer changes even when the number of load cycles increases. This behavior is called "according to Miner".

However, more recent investigations have revealed that there is actually no such thing as an infinite life strength and that the S-N curve (Woehler line) should be modified in the infinite life strength range.

In the range of endurance limit, you can therefore select the following modified forms:

- Miner (corresponds to DIN 3990, Parts 2, 3 and 6). Pitch ∞ (horizontal)
- According to Corten/Dolan. Pitch p
- According to Haibach modified. Pitch 2*p
- According to Haibach original. Lead 2*p-1 (according to [18])

  The lead p mentioned here matches the S-N curve (Woehler line) according to ISO, AGMA or DIN in the fixed period range, determined from YNT or ZNT. See also ISO 6336-6 [19].

The figure below (see Figure 15.7) shows the corresponding characteristics. Experience has shown that performing a service life calculation with load spectra using the Miner method returns results that are far too optimistic. We recommend you use the Haibach method of approach.

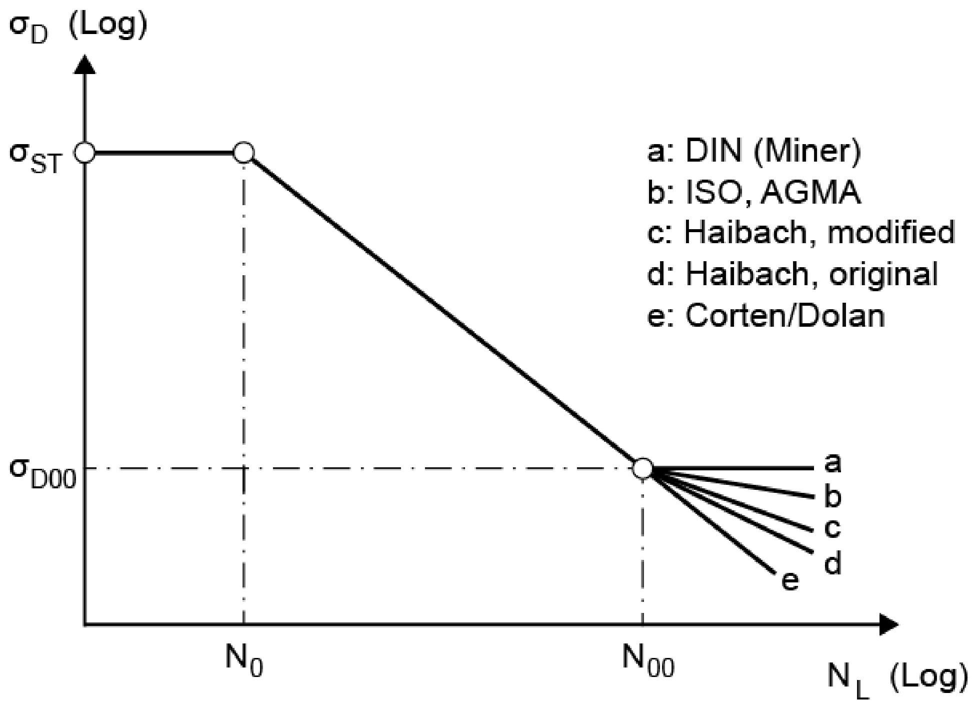


Figure 15.7: Infinite life strength models

Note concerning calculations according to ISO or DIN:

The pitch (slope) of the S-N curve (Woehler line) for tooth bending in the time-dependent domain (between N0 and N00) is defined using the YNT, YdrelT, YRrelT and YX coefficients for the static and endurance cases, but in the endurance domain (NL > N00), only the YNT coefficient is used for the static and endurance cases. The same applies for pitting with the ZNT, ZL, ZV, ZR and ZW coefficients. This corresponds to the procedure used in ISO 6336 for the endurance domain. However, this does mean that buckling occurs on the S-N curve (Woehler line) at N00, according to the Corten/Dolan rule.

As an example: for case-carburized steel, the pitch (slope) of the S-N curve (Woehler lines) in the endurance domain is 13.2, but in the range of endurance limit, it is approximately 10, depending on the precise values for YdrelT, etc.

If all the coefficients, YdrelT, etc., are set to 1.0 using "Own Input", the S-N curve (Woehler line) will be constant.

► **Note:**

The saved *.z?? files and the STANDARD.z?? file contain the ZS.CortanDolanFactors variable. This can be set to = true. This can force the program to also extrapolate the YdrelT, YRrelT, YX, ZL, ZV, ZR and ZW coefficients in the endurance range, in contrast to the ISO definition.

15.2.6.8 Define tooth form factors

The tooth form factor Y_F takes into account how the tooth form affects the nominal tooth root stress σ_{F0}. The stress correction factor Y_S takes into account the effect of the notch on the tooth root. These two factors can be calculated in three different ways:

1. **According to the formulae in the standard (normal)**
   As defined in ISO 6336 or DIN 3990, the tooth form and the stress correction factors are calculated at the tooth root at the point at which the tangent and the tooth center line form an angle of 30°. However, it is generally acknowledged that this method is rather imprecise, especially for deep tooth forms.
2. **Using graphical method**
   According to Obsieger [20], there is a more precise approach in which the product of the tooth form factor Y_F and the stress correction factor Y_S is calculated and the maximum value is determined. This method is based on the manufacturing process used for a specific tooth form and is applied to all points in the whole root area. This maximum value is then used to calculate the strength. Factors Y_F and Y_S are calculated according to the formulae in ISO 6336 or DIN 3990.

   This is the recommended method, particularly for unusual tooth forms and internal toothings. If required, this calculation procedure can also be applied in strength calculations as defined in ISO 6336 and DIN 3990, as well as in fine sizing.

   Note:
   If you use the graphical method here, KISSsoft will calculate the tooth form before it calculates the strength, each time. It takes its parameters either from the cutter data you have entered previously, in the Tooth form input window (see chapter 23.2.1, Tooth form), or from the default settings in the **Reference profile** input window. The maximum value of the product of the tooth form and stress modification factor is calculated at the same time and included in the stress calculation.

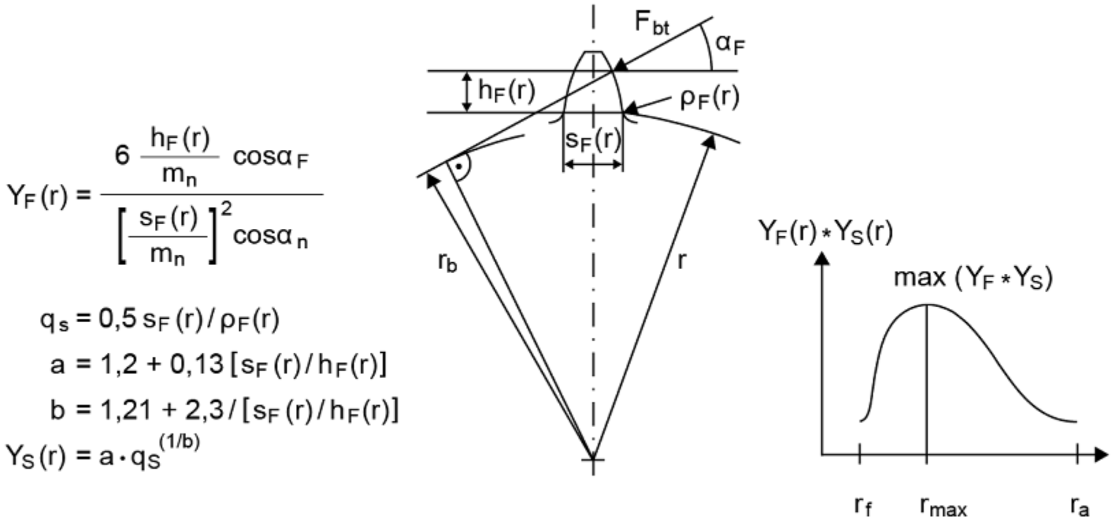


Figure 15.8: Figure: Tooth form factors using the graphical method

3. **for internal toothing, according to proposal 2737**
   When calculating strength according to ISO 6336 or DIN 3990, select this option to use the tooth form factor as defined in VDI 2737, which is more precise for internal toothing, because it evaluates the stress at the point of the 60° tangent and derives the tooth form from the manufacturing process with the pinion type cutter.

   The calculation specified in ISO 6336 for calculating tooth root stress is more accurate than the one implemented in DIN 3990. However, the calculation applied to the root rounding in the critical point (for a 60° tangent) is still incorrect. The method defined in VDI 2737, Annex B is much more accurate, which is why we recommend you use this method. If you select this option, only the root rounding ϱF and the root thickness sFn in the critical cross-section is calculated in accordance with the formulae in VDI 2737. All other sizes are calculated according to ISO 6336.

   The table (below) uses 4 examples to show the large variations that still arise in root rounding between the result defined in ISO 6336 and the effective values measured on the tooth form. However, the calculation method stated in VDI 2737 is very suitable.

Gear x=	**Pinion Cutter x0=**	**ϱF in ISO 6336-3 2006 and 2007-02**	**ϱF in the current edition of ISO 6336-3 2007-04**	**ϱF measured on the tooth flank**	**ϱF with VDI 2737**
-0.75	0.1	0.201	**0.426**	**0.233**	**0.233**
-0.75	0.0	0.175	**0.403**	**0.220**	**0.220**
0.0	0.1	0.298	**0.364**	**0.284**	**0.286**
0.0	0.0	0.274	**0.343**	**0.265**	**0.264**

Table 15.7: Comparison of root roundings

Note about the calculating Y_F:

- The theoretical profile shift is used in the calculation if the allowance is As < 0.05*mn (in accordance with ISO 6336-3). Otherwise the larger manufacturing profile shift xE.e (where the theoretical contact ratio is applied) is used. This corresponds to the procedure used in the STplus program (from Munich, Germany). An exact definition is not provided in the ISO standard. However, a specific tolerance field can be predefined in the **Details** tab for the root and flank strength calculation. This value is then always used to calculate strength and for the transverse contact ratio.
- According to the ISO standard, the reference profile for the entire toothing is to be used for the calculation. For this reason, if you input the reference profile for pre-machining with protuberance, and a manufactured profile with remaining protuberance is left after deduction of the grinding allowance, the reference profile for final machining is used for the calculation. A grinding notch is produced in the reference profile for pre-machining without a protuberance (or a protuberance that is too small). To ensure that this situation can be correctly taken into consideration, the pre-machining reference profile (with pre-machining manufacturing profile shift) is used to calculate Y_F. The final machining reference profile is also used to calculate the grinding notch and therefore to define Y_{Sg} (section 7.3 in ISO 6336-3).

15.2.6.9 Tooth contact stiffness

Tooth contact stiffness is required to calculate the dynamic factor and the face load factor. You can use one of these calculation options:

1. **In accordance with the formulae in the standard (normal)**
   In the standard calculation, the tooth contact stiffness c_g is calculated using empirical formulae (in ISO 6336, DIN 3990, etc.).
2. **Using the tooth form**
   Using this option, the tooth form stiffness c' is calculated according to Weber/Banaschek's dissertation [21]. This takes into consideration tooth bending, basic solid deformation, and Hertzian pressure. The last condition determines the load dependency of c'. The contact stiffness is determined using the effective tooth form (see Meshing stiffness (Z24)). The mean value of the stiffness curve calculated using this method is then included in the calculation. If required, this calculation procedure can also be applied in strength calculations as defined in ISO 6336 and DIN 3990, as well as in fine sizing (Z04). The single spring stiffness c' is calculated from the c_g, by extrapolating c' from the formula for c_g (ISO or DIN).
3. **constant**
   If you select this option, the tooth contact stiffness is set to a constant of 20 N/mm/µm.

15.2.6.10 Small amount of pitting permissible

In specific cases, the appearance of a slight amount of pitting on the flank may be permissible. In a range of materials, this results in higher flank safeties in the limited life range due to the changed S-N curve (Woehler lines), as can be seen in either ISO 6336-2, Figure 6, curve 1 or DIN 3990-2, Figure 8.1.

15.2.6.11 Consider pressure angle factor YSa

The tooth root stress calculation according to ISO 6336 is valid for a reference profile with a normal pressure angle of 20° according to ISO 53. For an extended consideration of the influence of the normal pressure angle on the tooth root stress, the pressure angle factor $Y_{S\alpha}$ as defined in Langheinrich [22] can be used.

External toothing, rack:

$Y_{Sa,\ sym} = 1+0.012(20-\alpha_n)$

Internal toothing:

$Y_{Sa,\ sym} = 1+0.001(20-\alpha_n)$

15.2.6.12 Lubricant factor

The lubricant factor is needed to calculate the coefficient of friction, loss, micropitting and scuffing. It can be set using the appropriate Settings button.

As specified in ISO/TS 6336-22:

- 1.0 for mineral oils
- 0.6 for water-soluble polyglycols
- 0.7 for non-water-soluble polyglycols
- 0.8 for polyalphaolefins
- 1.3 for phosphate esters
- 1.5 for traction fluids

15.2.6.13 Structural factor XwrelT or structural factor Xw (scuffing)

The structural factor takes into account differences in materials and heat treatment at scuffing temperature. The factor can be set using the appropriate Settings button. Structural factor X_{wrelT} (in DIN 3990 and in ISO/TS 6336-21:2022) or structural factor X_w (in ISO/TS 6336-20:2022) is used, depending on which standard is selected. However, in this case, $X_{wrelT} = X_w/X_{wT}$ and $X_{wT}= 1$ apply.

This results in $X_{wrelT} = X_w$. The two factors are identical.
However, the standards do not provide any details about how to proceed when different types of material have been combined in pairs. You must input this factor yourself, because it is not set automatically by KISSsoft.

Through hardened steels	1.00
Phosphated steels	1.25
Coppered steels	1.50
Nitrided steels	1.50
Case-hardened steels	1.15 (with low austenite content)
Case-hardened steels	1.00 (with normal austenite content)
Case-hardened steels	0.85 (with high austenite content)
Stainless steels	0.45

Table 15.8: Structural factor according to DIN 3990-4

The standard does not provide any details about how this factor is to be applied when the pinion and gear are made of different types of material. In this case it is safer to take the lower value for the pair.

15.2.6.14 Grinding notch

As defined in DIN 3990 or ISO 6336, the effect of the grinding notch can be taken into account by the coefficient Y_{Sg} . Here, you input the ratio t_g to the radius of grinding notch ϱ_g in accordance with the figure in DIN 3990-3, section 4.4 or ISO 6336-3, Figure 5. KISSsoft then calculates a coefficient $_g$ = Y_{Sg}/Y_S (a coefficient with which Y_S is multiplied).

The grinding notch depth t_g is calculated using the distance of the 30° tangents from preliminary contour and the finished contour. If an allowance for pre-machining has already been input in KISSsoft, (see Figure. 14.11), then the ratio t_g/ϱ_g can no longer be entered by the user. In this case, it is defined by the program. A grinding notch occurs when a grinding depth (see chapter 15.7, Modifications) has been entered, and no protuberances remain, either because no protuberance tool was used, or the selected allowance was too small. The rounding radius ϱ_g is then defined by generating the grinding wheel on the 30°- tangent (on the 60° tangent for internal toothings).

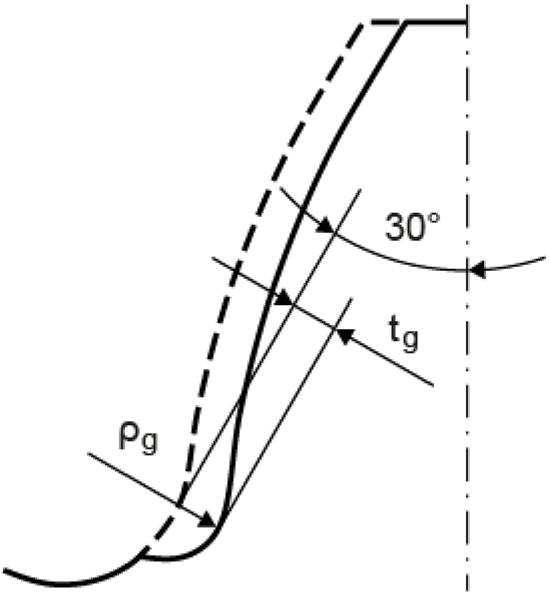


Figure 15.9: Grinding notch

15.2.6.15 Pretension

The influence of a press fit or other processing methods that influence tooth root stress can be taken into account with the pretension σ_P . This value influences the calculated tooth root stress as well as the safety according to the following formulae:

For static strength:

$$\sigma'_F = \sigma_F + \sigma_P$$

$$S'_S = \frac{R_P}{\sigma'_F}$$

$$S'_B = \frac{R_m}{\sigma'_F}$$

For fatigue strength:

$$\sigma'_{FG} = \sigma_{FG} \cdot \left(1 - \frac{\sigma_P}{R_m}\right)$$

$$S_F' = \frac{\sigma_{FG}'}{\sigma_F}$$

The pretension σ_P merely generates additional results in the reports. The results in the results window remain unchanged. You define this under "Strength" > "Details".

▶ **Note 1**

This rule is not documented in the ISO standard. **For this reason, we recommend extreme caution if the pretension effect is to be taken into account.** The formulae are proposed by Alstom Ecotecnia. KISSsoft only shows this effect in the report.

▶ **Note 2**

If the main calculation (single load or load spectra) requires the use of this rule, the value σ’Flim must be changed as follows, according to the equation for ’FG:

$$\sigma' F \lim = \sigma F \lim * \cdot \left(I - \frac{\sigma_P}{R_m} \right)$$

σ’Flim has to be introduced instead of σFlim in the material values; then the main calculation is performed using this pretension rule.

15.2.6.16 Moments of inertia

If required, you can enter the moment of inertia of the gears. This value is used when calculating the dynamic factor. If you enter an invalid value (deviation > 100% from the expected value), the system displays a warning. Usually, in such cases, the incorrect unit has been used for the input.

15.2.6.17 Optimal tip relief

To calculate safety against micropitting as specified in Method B in ISO/TS 6336-22, you must specify whether or not the profile modification is to be assumed to be optimal. The same applies to calculating the safety against scuffing. The software checks whether the effective tip relief (Ca) roughly corresponds to the optimum tip relief (Ceff). If this check reveals large differences, i.e. Ca < 0.333*Ceff or Ca > 2.5*Ceff, a warning is displayed. In this case, the value you input is ignored and is documented accordingly in the report.

15.2.6.18 Root rounding, ground

The setting specifying whether the root rounding is ground is only used in calculations according to GOST.

15.2.6.19 Information about material hardness

For more information about material hardness, refer to part of **Tooth flank fracture calculation method** (see chapter 15.2.1.6).

15.2.7 Strength details (AGMA)

Click on the **Details** button for the root and flank strength calculation to display a dialog in which you can make additional settings for the strength calculation.

► **Note**

Only values in the input window that differ from those defined in ISO are described here.

15.2.7.1 Limited life coefficients

The limited life coefficients determine which material values can be entered in the field for limited time and strength. In standard applications, infinite life strength values up to 10^{10} load cycles are reduced from 100% to 90% for the root and to 85% for the flank. According to AGMA, the reduction in strength also extends beyond 10^{10} load cycles. In critical applications, where a gear unit breakdown must be prevented at all cost, the material values are reduced even more, in comparison to those used in standard application areas.

15.2.7.2 Tooth form factors

For cylindrical gears with small helix angles, or cylindrical spur gears, you can specify that the load is to be applied either at the tip or at the single tooth contact point (the more precise option). For cylindrical gears with a large helix angle ($\varepsilon_\beta \geq 1$) according to AGMA, the load is always applied in the single tooth contact point.

Calculating with the single tooth contact point results in a lower load at the tooth root because the load is divided between the two teeth. However, this load distribution does not take place if large single normal pitch deviations occur and therefore the force should be assumed to be placed at the tooth tip.

As stated in AGMA, the contact point between the tooth form and the Lewis parabola is selected as the critical root cross section. The stresses are determined here. AGMA does not provide a formula for calculating internal toothings. Instead, it recommends you use the graphical method to calculate the tooth form. The required data is to be taken from measurements. If you click the checkbox to select the graphical method for calculating the tooth form factor, the software automatically calculates the tooth form at the point where the K_f or I factor is greatest. In contrast to the method defined by Lewis, where the calculation is only performed at the contact point of the parabola, the calculation using the cross section with the greatest stresses gives more precise results, and is therefore the method we recommend for external gears too.

15.2.7.3 Transmission accuracy level number

A_V (or Q_V for AGMA 2001-C95 or earlier) is calculated according to the formulae defined in AGMA 2001 or 2101 and is extremely dependent on the accuracy grade (manufacturing quality). A_V is permitted to be one level higher or less than the accuracy grade (manufacturing quality) and is needed to calculate the dynamic factor. You can overwrite this value if required.

15.2.8 Define load spectrum

In this group, you can also access load spectra that have been stored in the database. You can also define the load spectra directly.

If you select **Read**, you can import a file (in either .txt or .dat format) with a load spectrum.

The 'Example_DutyCycle.dat' file in the **dat** sub-folder in the KISSsoft installation directory is an example of a file that shows how a load spectrum can be defined. For load spectra on carriers, the special format shown in 'Example_Carrier_DutyCycle.dat' must be applied with a header.

If you want separate factors ($K_{H\beta}$, K_γ, etc.) to be taken into account in the calculation with load spectra for each load bin, open the **Factors** tab. In it, make the appropriate settings for the load distribution coefficient K_γ (see chapter 15.3.4, Mesh load factor), the alternating bending factor Y_M (see chapter 15.3.5, Alternating bending factor) and the face load factor $K_{H\beta}$ (see chapter 15.3.6, Face load factor). An example of how a load spectrum can be defined with factors ($K_{H\beta}$, K_γ, etc.), can be seen in the 'Example_DutyCycleWithFactors.dat' file in the **dat** sub-directory in the KISSsoft installation directory.

A load spectrum can also be generated from a series of measurements with torque/speed/time if you select the **torque measurement** option. Click the Convert button below the Load spectrum table to call this option, (see chapter 15.8, Torque measurement).

15.2.8.1 Type of load spectrum

The service life for load spectra is calculated as specified in ISO 6336, Part 6, and is based on the Palmgren-Miner rule.

Three load spectra are predefined here, as shown in DIN 15020 (Lifting Appliances), along with many other standard spectra. You can also input your own load spectra.

A load spectrum consists of several elements (up to 50 in the database or an unlimited number if imported from a file). Each element consists of the frequency, speed, and power or torque. The data always refers to the reference gear you selected when you input the nominal power (Performance-Torque-Speed screen). The program stores these values as coefficients so that they are modified automatically when the nominal power changes.

If two speeds that are not equal to zero have been predefined for planetary stages, you can select two load spectra. In this case, only the speed factor is important for the second load spectrum.

► **Note**

The calculation takes into account the load dependency of the K coefficients (dynamic, face load and transverse coefficients). If you want to examine the result in greater detail, you will find the most interesting interim results in the Z18-H1.TMP text file (in the TMP directory).

15.2.8.2 Load spectra with negative bins

Load spectra with negative load bins (T < 0 and/or n < 0) can be calculated as follows.

IMPORTANT:

- A load bin is considered to be negative if the non-working flank is placed under load.
- This does not apply for the calculation of the pitting safety for idler gears (in the case of planetary gear stages, it only applies to the sun and internal gear; in the case of planets, it is assumed that both flanks are always under load).
- When calculating the root safety, this is only applied to bins where the alternating bending factor is Y_M=1.0.

Torque factor	Speed factor	Flank under load	Load bin is
+	+	Working flank (*)	evaluated as positive
+	-	Working flank (*)	evaluated as positive
-	+	Non-working flank	evaluated as negative
-	-	Non-working flank	evaluated as negative

Table 15.9: Evaluation of a load bin, depending on the prefix operator

(*) Working flank as entered in the **Rating** tab

You can select the following in "**Details**" in the **Strength** group, in the **Rating** tab:

- For calculating pitting safety
    - Evaluate all negative load bins as positive (as up to now)
    - Only consider positive load bins
    - Only consider negative load bins
    - Calculate both cases and document the less favorable case
- For calculating root safety

- Evaluate all negative load bins as positive (as up to now)
- Increase tooth root stress for negative load bins by 1/0.7
- Increase tooth root stress for positive load bins by 1/0.7
- Calculate both cases and document the more realistic case

15.2.8.3 Classification of load spectra according to F.E.M. Guideline

The "Supplementary Data" section in the report for a load spectrum contains the spectrum factor k_m, and the machine classes L, T and M for a load spectrum, according to F.E.M 1.001 [23].

The spectrum factor k_m lies between 0 and 1 and describes the load on a machine caused by active torque. If the load spectrum is 100% torque for 100% of the time, k_m equals 1. The spectrum class L, according to table T.2.1.3.3 in F.E.M. 1.001, is calculated from the calculated spectrum factor k_m , and increases from L1 to L4 with the active torque.

The application class T is determined from the entire duration of the load spectrum. To do this, the machine is assigned to one of the application classes T0 to T9, according to a duration of between zero and 50,000 operating hours (see Tab. T.2.1.3.2 in F.E.M. 1.001). In each case, the predefined and achievable application classes T are output in the report.

A machine class M is determined according to the determined application class T and spectrum class L, according to table T.2.1.3.2 in F.E.M. 1.001. In each case, the predefined and achievable machine classes M are output in the report. Machine classes M1 (short operating time, low loads) to M8 (long operating time, high loads) are assigned, depending on the application class T and spectrum class L.

15.3 Factors

The **Factors** input window is one of the standard tabs (see chapter 5.1, Standard and special tabs).

15.3.1 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. The next table provides information about the factor values. You will find full details in ISO 6336, DIN 3990 and DIN 3991.

When deciding which application factor to select, you must take into account the interrelationship between the required safeties, assumed loads and application factor.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**

uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 15.10: Assignment of operational behavior to application factor

DIN 3990, Part 41 (car gearboxes), distinguishes between application factors for flank strength K_{AH} and for tooth root strength K_{AF}. Except for flank strength calculations, all other calculations (e.g. resistance to scoring) use application factor K_{AF}.

However, according to DIN 3990 Part 41, the application factor can also be less than 1.0. This is intended to avoid the need to perform a calculation involving load spectra. For example, DIN 3990, Part 41, Annex A, suggests the following values for a 4-speed car gearbox:

Gear	**R**	**1**	**2**	**3**	**4**
N_L	10^5	2 * 10^6	1.5 * 10^7	3 * 10^7	2 * 10^8
K_{AH}	0.65	0.65	0.65	0.65	
K_{AF}	0.70	0.70	0.80	0.80	

Table 15.11: Application factor as defined in DIN 3990, Part 41

15.3.2 Dynamic factor

The dynamic factor takes into account additional forces caused by natural frequencies (resonance) in the tooth meshing. It is usually calculated using the calculation method you selected, however you can also input the value if it has already been derived from more precise measurements. To change the value, click the checkbox to the right of the input field.

15.3.3 Transverse load factor

The transverse load factor $K_{H\alpha}$ is calculated according to the selected calculation method. The transverse load factor takes into account uneven contact characteristics across a number of teeth. When the contact ratio increases, the transverse load factor also becomes larger depending on the predefined manufacturing quality. A high contact ratio will result in a reduction of the root stresses. The transverse load factor will compensate for this effect for large single normal pitch deviations.

In unusual cases, the transverse load factor will be unrealistically high. If you want to reduce the transverse load factor in this situation, simply click the checkbox to the right of the input field. You can then change this value.

15.3.4 Mesh load factor

The mesh load factor takes into consideration the uneven load distribution across multiple planets or idler gears. In this case, the load is multiplied by this coefficient. Dimensioning suggestion according to AGMA 6123-C16:

	Number of planets								
Application level	2	3	4	5	6	7	8	9	**Flexible Mounting**
1	1.16	1.23	1.32	1.35	1.38	1.47	1.52	-	without
2	1.00	1.05	1.25	1.35	1.38	1.47	1.52	1.61	without
3	1.00	1.00	1.15	1.19	1.23	1.27	1.30	1.33	without
4	1.00	1.00	1.08	1.12	1.16	1.20	1.23	1.26	with

Table 15.12: Load distribution coefficient Kγ defined by the number of planets

Application level	**Description**
1	Typical of large, slow-turning planetary gear units
2	Moderate quality, typical of industrial gears
3 & 4	High quality gear units, e.g. for gas turbines

Table 15.13: Meaning of the application level

The mesh load factor can also be calculated in accordance with ISO/TS 6336-20 (mesh load factor K_{mp}) or DNVGL-CG-0036 with

$$K_\gamma = 1.0 + 0.25 \cdot \sqrt{n_p - 3}$$

n_p – number of planets ≥ 3

▶ **Note**

Level 2, or higher, requires at least one floating element.
Level 3, or higher, requires a flexible gear rim.
In a flexible assembly, the planets must be mounted on flexible pins/shafts or on bearings with couplings.

Depending on the toothing quality and the number of planets, use the **Calculated according to AGMA 6123** method to determine the distribution coefficient K_γ for application levels 1 to 3.

If a different load distribution coefficient is input for each element, when load spectra are in use, you should select the **Own input, per load stage** method.

15.3.5 Alternating bending factor

The tooth root strength calculation is used solely to calculate pulsating load on the tooth root. However, in some cases, the tooth root is subject to alternating bending loads (e.g. a planet gear in planetary gear units). In this scenario you can change the alternating bending coefficient of individual gears by selecting either the **Own input** or **Own input, per load spectrum element** methods. As an alternative to transferring these values directly, select the **Calculate in accordance with ISO 6336-3 Annex B** method to calculate the coefficient. To do this, you must then open the **Rating** tab, go to the Load spectrum section, and input the f_{low} and f_{high} parameters for each gear. f_{high} must always have the fixed default value of 100%.

ISO 6336-5:2003, section 5.3.3 and DIN 3990-5, section 4.3, state 0.7 as the value Y_M for pure cyclic load. In ISO 6336-3:2006, Annex B, the stress ratio *R* for idler and planetary gears is taken into account by using this formula:

$Y_M = \dfrac{1}{1 - R \cdot \dfrac{1-M}{1+M}}$	(12.16)
$R = -1.2 \cdot \dfrac{f_{low}}{f_{high}}$	(12.17)

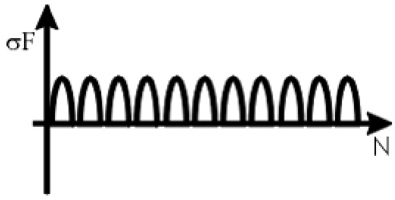
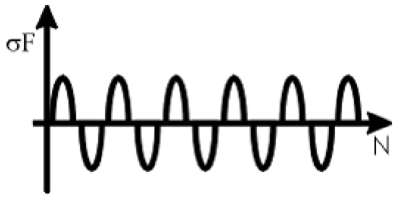
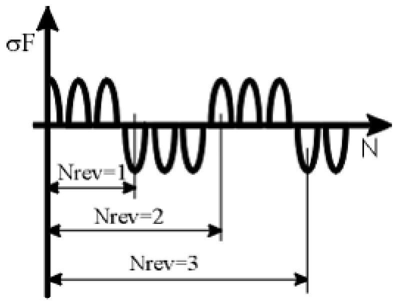
f_{high}	Load on the flank side that is subject to the higher load (must always have the fixed default value of 100%)
f_{low}	Load on the flank side that is subject to the lower load
M	Dimensionless number depending
	on the type of treatment and load type
	(see Table B.1 in ISO 6336:2006-3, Annex B)
R	Stress ratio
Y_M	Alternating bending factor

Treatment	Endurance strength	Coefficient for static proof
Steels		

case-hardened	0.8 - 0.15 Y_S	0.7
Case-hardened and shot peened	0.4	0.6
Nitrided	0.3	0.3
Flame/induction-hardened	0.4	0.6
Not surface-hardened steel	0.3	0.5
Cast steel	0.4	0.6

Table 15.14: Mean stress ratio M as specified in Table B.1 - Mean Stress Ratio - in ISO 6336:2006-3

According to Linke [24], the alternating bending factor (described there as Y_A) is determined as shown in Figure 14.19. For plastics, Niemann recommends [7] 0.8 for laminated fabric and 0.667 for PA (polyamide) and POM (polyoxymethylene).

Operating Mode	Alternating Bending Factor (Mean Stress Influence Factor) Y_M	Load Direction
Pulsating	1	
Alternating	0.7 (1) 0.65 (2)	
Oscillating	$0.85 - 0.15 \cdot \frac{\log N_{rev}}{6}$ (1) $0.85 - 0.20 \cdot \frac{\log N_{rev}}{6}$ (2) $(1 \leq N_{rev} \leq 10^6)$ 0.7 (1) 0.65 (2) $(N_{rev} \geq 10^6)$	

(1) Linke, H.: Stirnradverzahnung, Carl Hanser Verlag, 1996.
(2) Linke, H.: Stirnradverzahnung, Carl Hanser Verlag, 2010.

Figure 15.10: Alternating bending factor in accordance with Linke [24]

► **Note**

Y_M is not standardized and is not part of ISO 6336-3 Method B. You should ensure that the values used are backed up by experience or tests.

15.3.5.1 Load spectrum with alternating torque

Load bins can also be entered with negative torques.

The problem:

until now, no calculation guidelines have been drawn up to describe how to calculate gears with alternating load spectra.

The only unambiguous case is when a change in alternating torque takes place during every cycle and in each bin of the load spectrum. At this point, a load change corresponds to exactly one double-load with +torque and then with -torque. This instance can be calculated correctly by entering the load spectrum of the +torques and the alternating bending factor Y_M for the tooth root. The flank is also calculated correctly, because the +torques always apply to the same flank.

If, in contrast, the drive runs forwards for a specific period of time and then runs backwards, the experts agree that the tooth root is not subjected purely to an alternating load (and possibly this is the only point at which an alternating load change takes place). However, discussions are still raging as to how this case can be evaluated mathematically. It is even more difficult to define how mixed load spectra with unequal +torques and -torques for the tooth root are to be handled. For this type of case, only the +torques are considered for the flank (with the prerequisite that the +torques are equal to, or greater than, the -torques).

Note about handling load spectra with reversing torque:

A load progression as represented in the figure below, where the tooth is subjected to a load a few times on the left flank, and then a few times on the right flank, can be converted into a load spectrum as shown below. This is represented in an example here.

Load progression (example):

- 13 loads with 100% of the nominal load (100 Nm) on the left flank, then
- 9 loads with 80% of the nominal load (80 Nm) on the right flank, etc.

This results in the following process:

- 11 load cycles with 100% load, positive torque, pulsating; then
- 1 load cycle with 100% load on the left and 80% load on the right; then
- 7 load cycles with 80% load, negative torque, pulsating; then
- 1 load cycle with 80% load on the right and 100% load on the left;

then repeated again from the start.

This can be represented as a load spectrum as follows:

Frequency	torque	Left flank load	Right flank load
11/20 = 0.55	100 Nm	100%	0%
7/20 = 0.35	80 Nm	0%	100%
2/20 = 0.10	100 Nm	100%	80%

Table 15.15: Load progression shown as a load spectrum

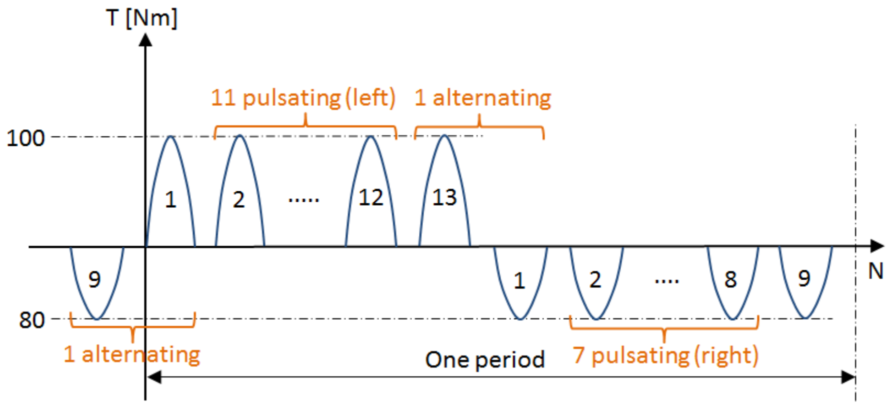


Figure 15.11: Load progression

15.3.6 Face load factor

The face load factors $K_{H\beta}$,$K_{F\beta}$,$K_{B\beta}$ take into consideration the influence of an uneven load distribution over the facewidth on the contact stress, tooth root stress and scuffing stress. You can specify that the face load factor is either to be set as a constant value or calculated from other values. If you already know the face load factor $K_{H\beta}$, select the **Own input** method and input this value. Click the Plus button to display the **Define face load factor** window. In it, the calculation according to DIN/ISO is displayed: in that calculation, you can use a number of parameters to calculate the value you require.

The usual setting here is "Calculation according to calculation method". The face load factor is then calculated according to the formulae used in the strength calculation standard (ISO, AGMA or DIN). You will need to input some values for this. These values are displayed on the right of the window (flank line modification, etc.) and are described in the sections that follow. You can input other values by clicking the Plus button in the **Define face load factor** window.

For the **Factor K with stiffening effect** entry: the pinion has the effect that it stiffens the shaft (stiffening effect) if d_1/d_{sh} >1.15 and the pinion is created with a fixed interference fit or shaft/pinion on the piece.

The formulae proposed in the standards for defining the face load factor $K_{H\beta}$ enable you to determine $K_{H\beta}$ very quickly (but only empirically, and therefore not very accurately). The $K_{H\beta}$ factor calculated using these formulae is usually higher than it actually is, so the calculated value is therefore on the conservative side. If the face load factor is increasing (> 1.5), it makes sense to perform a more accurate calculation. To do this, use the "Calculation without manufacturing deviation according to ISO 6336-1 Annex E" method.

The “Calculation according to ISO 6336 Annex E” method is very accurate, but time consuming. As described in [19], it calculates any gaping in the meshing, and therefore defines the load distribution over the entire facewidth. To perform this calculation, you will need to know the exact dimensions of the shafts and support. Click the **Define axis alignment** button to input the shaft values stored in the shaft calculation module for the relevant shafts.

The "Calculation with manufacturing deviation, as defined in ISO 6336 Annex E" method is the most accurate method. It also requires you to specify the toothing tolerance $f_{H\beta}$ (total tooth trace deviations over the bearing facewidth) and set the axis alignment tolerance f_{par} (angular deviation of the axis position in the plane of action). As described in [19], the load distribution over the facewidth is calculated 5 times: First without deviation, then with ($+f_{H\beta}$,$+ f_{par}$), ($+f_{H\beta}$,$- f_{par}$), ($-f_{H\beta}$,$+ f_{par}$), ($-f_{H\beta}$,$- f_{par}$). The largest face load factor $K_{H\beta}$, determined in the process, represents the final result.

Click the Sizing button next to the entry for $|\Sigma f_{H\beta}|$ to display a subwindow that contains a range of suggestions about how to take manufacturing errors into account.

The maximum suggestion shows the possible highest value for the tolerance interval for f_{par} and $\Sigma f_{H\beta}$. The statistically evaluated proposal shows tolerances that correspond to a statistically evaluated tolerance interval with 99.7% probability.

The following formulae are used to define the total tolerance f_{total}:

$f_{par} = f_{\Sigma\beta\text{-ISO}} * \cos(\alpha_{wt}) + f_{\Sigma\delta\text{-ISO}} * \sin(\alpha_{wt})$ (value in the plane of action, effect of housing manufacturing errors as specified in ISO/TR 10064-4 or DIN 3964))

$f_{total\text{-}maximal} = \Sigma f_{H\beta\text{-}max} + f_{par\text{-}max} = f_{H\beta 1} + f_{H\beta 2} + f_{par}$

$f_{total\text{-}statistic} = \Sigma f_{H\beta\text{-}stat} + f_{par\text{-}stat} = 3 * \sqrt{[(f_{H\beta 1}/3)^2 + (f_{H\beta 2}/3)^2 + (f_{par}/3)^2]}$;

where the values are subdivided as follows:

$\Sigma f_{H\beta\text{-}max} = +|f_{H\beta 1}| + |f_{H\beta 2}|$;

$f_{par\text{-}max} = |f_{par}|$

$\Sigma f_{H\beta\text{-}stat} = f_{total\text{-}statistic} * (f_{H\beta 1} + f_{H\beta 2}) / (f_{H\beta 1} + f_{H\beta 2} + f_{par})$;

$f_{par\text{-}stat} = f_{total\text{-}statistic} * f_{par} / (f_{H\beta 1} + f_{H\beta 2} + f_{par})$;

► **Note**

If you want to calculate the face load factor by applying a load spectrum for each element, select either the **Own input, per load stage**, **Calculation according calculation method** or **Calculation with/without manufacturing deviation according to ISO 6336-1 Annex E, per load stage** method.

In the cylindrical gear pairs, three- and four-gear chains, and planetary systems, calculation module, shaft calculation files can be called and used to calculate the relative displacement between the tooth flanks more accurately, based on the corresponding shaft bending lines (see chapter 15.3.7, Taking into account shaft bending (face load factor and contact analysis)). Torque, power and force for all the load bins involved in the shaft calculation are then modified according to the partial load factor w_t .

Torsion of the gear body can be taken into account. Here the calculation assumes a solid cylinder or hollow cylinder (external diameter = root circle + 0.4*normal module or operating pitch circle, depending on what has been predefined under "Settings", bore = inside diameter) is involved. In other words, the internal diameter is taken into account and the torque on one side is zero. The torque is distributed in a linear fashion along the facewidth (parabolic course of deformation by torsion). You can select which side is to be subjected to torsional moment. In this case, I and II refer to the same side, as is also the case when you enter the toothing modifications. The increase in torque for a sun in planetary stages is taken into account by using multiple meshing (several planets). Multiple meshing is not taken into consideration in any other configuration (e.g. for gear pairs). In such situations, the correct torque curve can be used if the deformation is taken from the shaft calculation.

The facewidth is divided into slices to help you calculate the face load factor as defined in ISO 6336, Annex E:

You can set the accuracy of the face load factor calculation according to Annex E in the "Define number of slices" dialog. Click the Plus button next to the calculation method to open this dialog.

15.3.6.1 Face load factor settings for calculation according to ISO, Annex E

Load factors: Define how load factors K_V, K_A and K_γ are used. They can be taken into account when calculating load distribution and axis alignment according to ISO 6336-1, Annex E.

Iterating the load distribution of the meshings (only affects planetary stages): If shaft data is used to define the axis alignment, a constant load distribution over the facewidth is initially assumed when bending is calculated in the shaft calculation. This is a satisfactory approximation if the load distribution is fairly well distributed, and the face load factor is therefore not greater than 1.3 (maximum 1.5). If the load distribution is less favorable, return the load distribution value from the gear calculation to the shaft calculation, and calculate bending again with the modified (and not linear) load distribution. This produces a more accurate, modified load distribution. This iterative determination of the load distribution across all the meshings is then performed until the load distribution stops changing in all the meshings. Be aware that this option only shows an effect if at least one of the deformation components is linked with the shaft calculation.

Tooth contact stiffness: This defines whether tooth contact stiffness is calculated according to ISO 6226 (Cγβ), (default setting) or whether it is constant with Cm = 11 N/mm/μm as defined in AGMA 927-01.

Calculating the moment of resistance in torsion: If the calculation of torsion due to deformation in the "**Define axis alignment**" dialog is set to "Side I/II", the diameter specified here is used in the calculation.

15.3.6.2 Flank line modification

You can achieve more balanced contact characteristics if you perform targeted flank line modifications. The figure below shows the two most frequently used modifications (see Figure 15.12).

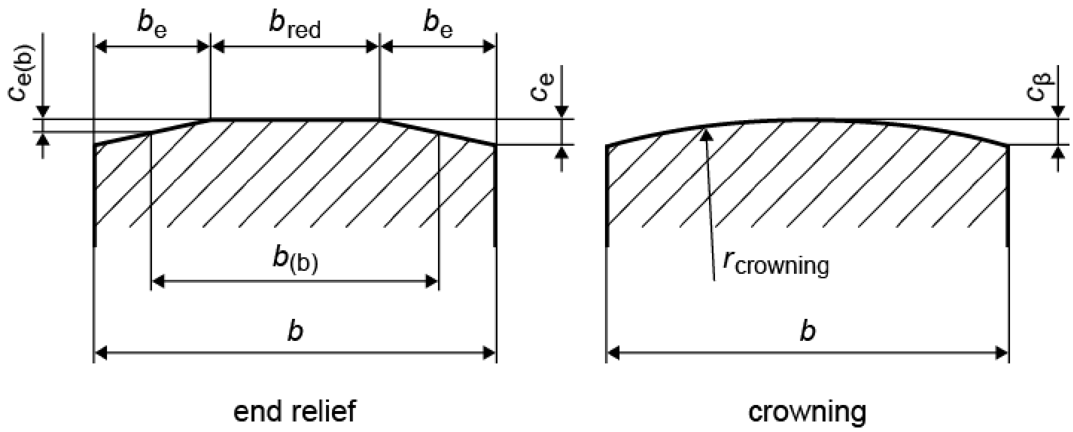


Figure 15.12: End relief and flank line crowning

15.3.6.3 Cylindrical gear pairs

The calculation, as specified in ISO 6336, is based on an approximate estimate of the pinion deformation. In many cases, this is extremely inaccurate and usually results in face load factors that are much too high.

The face load factor is the ratio between the maximum and average line load. The basic equation used for the face load factor corresponds to equation (41) in the standard:

*The equation numbers used in this section refer to ISO 6336:2006

$K_{H\beta} = \frac{(F_m / b)_{\max}}{F_m / b} = 1 + \frac{F_{\beta y} \cdot c_{\gamma\beta}}{2 \cdot (F_m / b)}$	(14.4)

The effective tooth trace deviation $F_{\beta y}$, see equation (52) in the standard, is defined with the inclusion of a deformation component f_{sh} that is specified in a linearized manner. The multiplier 1.33 in the equation stands for the conversion of the linearized specific deformation progression into the real parabolic progression - see equation. (14.5).

$$F_{\beta y} = F_{\beta x} \cdot \kappa_\beta = \left(1.33 \cdot f_{sh} + f_{ma}\right) \cdot \kappa_\beta$$	(14.5)

The manufacturer component of the tooth trace deviation f_{ma} is derived from tolerances specified by the manufacturer. If a standard procedure for checking the manufacturing quality is used, you can apply this formula (equation (64) in the standard):

$$f_{ma} = \sqrt{f_{H\beta 1}^2 + f_{H\beta 2}^2}$$	(14.6)

If you have used KISSsoft's shaft calculation software to calculate the exact tooth trace deviation due to deformation (torsion and bending) in the plane of action, you can correct the approximately calculated value f_{sh}, derived from the standard, and therefore calculate the width factors much more precisely! The formula as specified in ISO 6336 only applies to solid shafts or hollow shafts that have an internal diameter that is less than half of the external diameter.

In Method C2, the face load factor is calculated using these equations:

Size	**Drop-down list**	**Selection**	**Equation**	**No.**
$K_{H\beta}$				(8.04)/ (8.06)
F_β				(8.08)
F_β	position of the contact pattern	not verified or inappropriate favorable optimal		(8.26) (8.27) (8.28)
f_{sh}				(8.39)
f_{sh0}	flank line modification	none	0.023 • γ	(8.31)
		flank line crowning	0.012 • γ	(8.34)
		end relief	0.016 • γ	(8.35)
		full flank line modification	0 • γ	a)
		slight flank line crowning	0.023 • γ	b)
		helix angle modification	0.0023 • γ	b)

		flank line crowning + helix angle modification	0.0023 • γ	b)
γ	toothing	straight/helical double helical		(8.32) (8.33)
f_{ma}	flank line modification	none	1.0 • $f_{H\beta}$	(8.51)
		flank line crowning	0.5 • $f_{H\beta}$	(8.53)
		end relief	0.7 • $f_{H\beta}$	(8.52)
		full flank line modification	0.5 • $f_{H\beta}$	a)
		slight flank line crowning	0.5 • $f_{H\beta}$	b)
		helix angle modification	1.0 • $f_{H\beta}$	b)
		flank line crowning + helix angle modification	0.5 • $f_{H\beta}$	b)

Table 15.16: Overview of equations used according to DIN 3990:1987

a) same as DIN 3990, Equation (6.20)
b) same as ISO 9085, Table 4

Size	**Drop-down list**	**Selection**	**Value**		**No.**
$K_{H\beta}$					(39)/ (41)
F_β					(43)
		not verified or inappropriate			(52)
F_β	position of the contact pattern	favorable			(53)
		optimal			(56)
f_{sh}					(57)/ (58)
f_{ma}					(64)
		none	1 /	1	
		flank line crowning	0.5/	0.5	Table 8
		end relief	0.7/	0.7	
B_1/B_2	flank line modification	full flank line modification	0 /	0.5	(56)
		slight flank line crowning	1 /	0.5	

		helix angle modification	0.1/	1.0	Table 8
		flank line crowning + helix angle modification	0.1/	0.5	

Table 15.17: Overview of equations used according to ISO 6336:2006

Type of pinion shaft
Load as defined in ISO 6336:2006, Figure 13 (DIN 3990/1, Figure 6.8) or the bearing positioning is shown in the figure below.

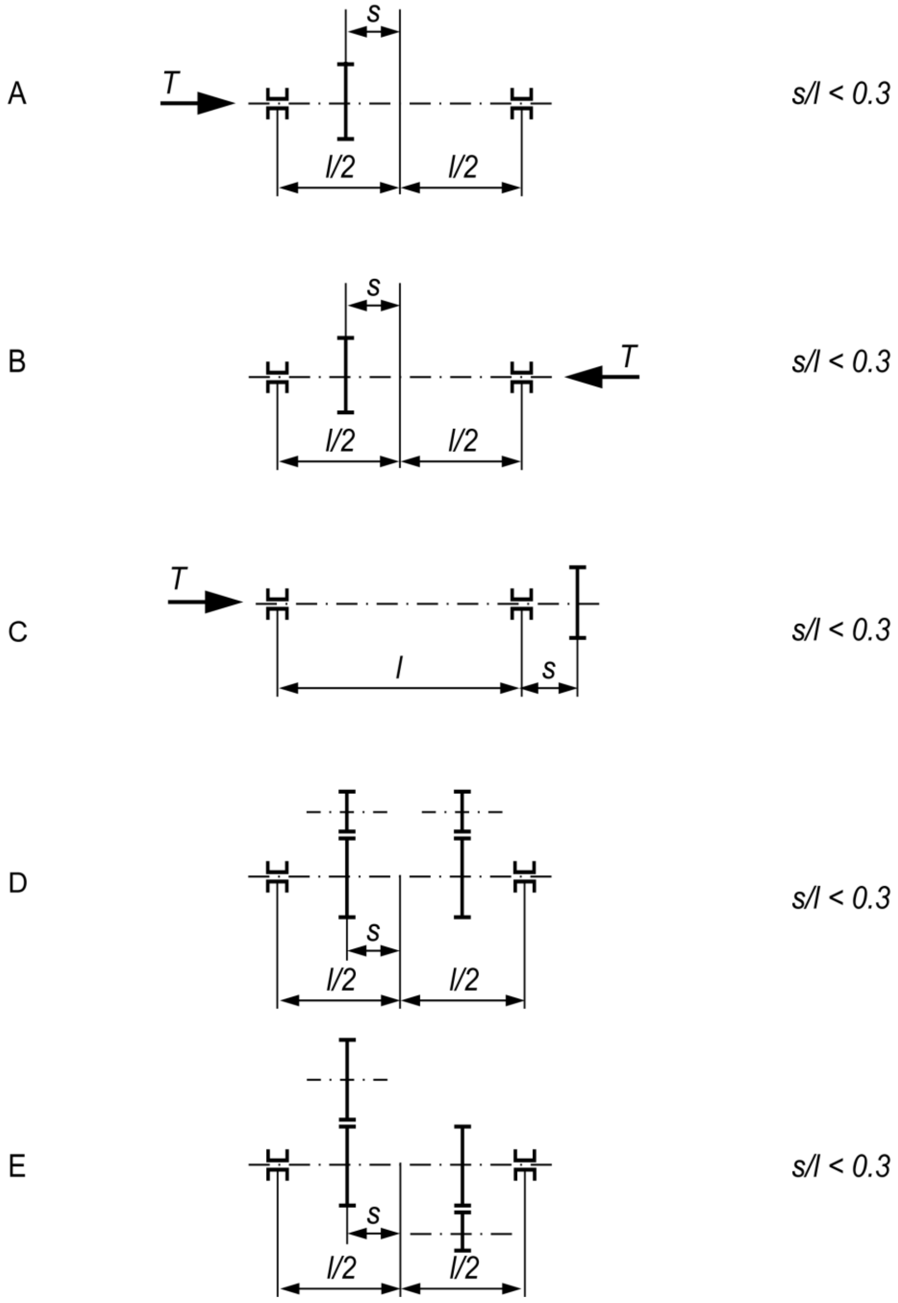


Figure 15.13: Load as defined in ISO 6336:2006, Figure 13.

Load according to AGMA 2001

Definition of s and s_1 according to AGMA 2001, Figure 13-3. Figure 14.23 shows the bearing positioning as described in AGMA 2001.

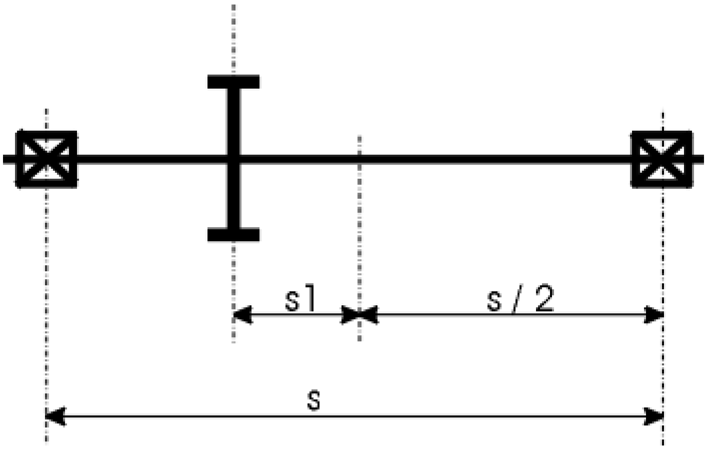


Figure 15.14: Load as defined in AGMA 2001, Figure 13-3

15.3.6.4 Planetary stages

The face load factors for planetary stages are calculated in a different way than for cylindrical gears. The deformation component f_{sh} is derived from the deformation of the mating gears on the shaft due to torsion and bending. In order to simplify the situation for a pinion-gear pair, only the pinion deformation (which is much greater) is taken into account.

Planetary stages are subject to the following significant deformations:

- Since the sun has several tooth meshings, all radial forces are canceled out. No bending takes place because deformation is caused solely by torsion. However, the multiple meshing which corresponds to the number of planets means this is greater than for normal integral pinion shafts.
- A planet gear has two meshings with opposed torques, which prevents deformation due to torsion. Bending may be calculated in the same way as for integral pinion shafts. However, the circumferential force must be doubled because of the sun/planet and planet/internal gear.
- In most cases, rim deformation can be ignored. As a result, the torsion at the pinion and the bending at the planet bolt must be taken into consideration for sun/planet meshing whereas, for planet/internal gear, only the bending at the planet bolt is important. For most support arrangements for planets, bending can be determined analytically using a procedure similar to that specified in ISO 6336. The 4 most common cases are displayed below (see Figure 15.15).

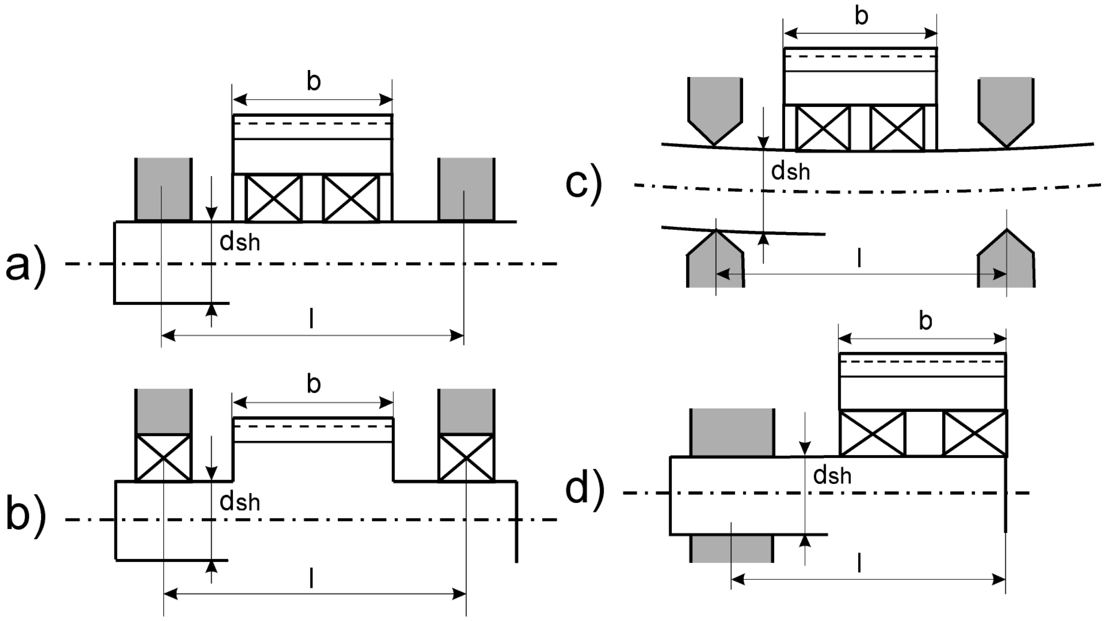


Figure 15.15: Support arrangement for planets

a) Planets mounted with fixed clamped bolts on both sides

b) Planets are on bolts, which have flexible bearings in planet carrier

c) Planets mounted with gently tightened bolts (flexible bearings) on both sides

d) Planets mounted with fixed clamped bolts on one side

Configuration	**ISO 6336**	**DIN 3990**	**AGMA 2001**
a	Part 1,	Formulae	Chapter 15, (37)
	Annex D	6.20/6.21/6.24/6.25/	
b	Part 1,	Formulae	Chapter 15, (37)
	Annex D	6.24A/6.24B/6.25A/6.25B	
c and d	Part 1,	Formulae as defined in Part 1,	Chapter 15, (37)
	Annex D	Annex C, see [25].	

Table 15.18: Configuration of planetary stages as defined in ISO, DIN and AGMA

For ISO 6336, see also the explanation in [25].

Equations 14.7a to 14.7d show the bending components in relationship to the distance x from the start of the planet's bearing facewidth. As we are only interested in bending variation across the facewidth, the constant term was left out of the equations so that $f_b(x = 0)$ is zero. Similar formulae can be found in other technical documentation [26]. These equations apply for cases a through d:

$f_{bpla} = 2\frac{64}{\pi}\frac{F_m/b}{d_{sh}^4 E_p} * [\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2(3l - 6b + b^2/l)}{48} + \frac{b^2x(3l - 4b + b^2/l)}{48}]$	(14.7a)
$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m/b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2(l-b)}{8} + b^2x\left(\frac{l}{8} - \frac{b}{12}\right)\right]$	(14.7b)
$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m/b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2(l-b)}{8} + b^2x\left(\frac{l}{8} - \frac{b}{12}\right)\right]$	(14.7c)
$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m/b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{6} - \frac{b^2x^2}{4} + \frac{blx(l-b)}{2}\right]$	(14.7d)

The sun's deformation due to torsion, as described in the equation (14.8), can be calculated from Annex D (f_t according to formula D.1).

$f_{tso} = p \cdot \frac{8}{\pi} \cdot \frac{F_m/b}{0.39 \cdot E_{so}}\left(\frac{b}{d_{so}}\right)^2 \cdot \frac{x}{b}\left(1 - \frac{x}{2b}\right)$	(14.8)

In order to stay as close as possible to the methods used in ISO 6336 (and be able to apply formula 2), the average deformation components f_{bmpla} (bending at the planet) and f_{tmso} (torsion at the sun) will be determined.

$f_{tso} = \frac{1}{b}\int_0^b f_{tso}(x)dx \qquad f_{bmpla} = \frac{1}{b}\int_0^b f_{bmpla}(x)dx$	(14.9)
$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m/b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16}\left(-\frac{b}{5} + \frac{l}{6} + \frac{b^2}{18l}\right)$	(14.10a)
$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m/b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16}\left(\frac{l}{3} - \frac{b}{5}\right)$	(14.10b)
$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m/b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16}\left(\frac{l}{3} - \frac{b}{5}\right)$	(14.10c)

$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^2}{4} \left(\frac{b^2}{5} - bl + l^2 \right)$	(14.10d)
$f_{tmso} = p \cdot \frac{8}{3\pi} \cdot \frac{F_m / b}{0.39 E_{so}} \cdot \left(\frac{b}{d_{so}} \right)^2$	(14.11)

According to ISO 6336:2006, equation D.8, the linearized deformation components of the tooth trace deviation f_{sh} (in mm) will be:

$f_{sh}(Paarung\ Sonne - Planet) = 2000 \cdot (f_{tmso} + f_{bmpla})$	(14.12)
$f_{sh}(Paarung\ Planet - Ring) = 2000 \cdot f_{bmpla}$	(14.13)

This can then be used with equations (14.4) and (14.5) to calculate face load factors for the sun/planet and planet/gear rim.

Formula symbol	Unit	Meaning
b	mm	Meshing width
$c_{\gamma\beta}$	N/(mm µm)	Meshing stiffness
d_{pla}	mm	Planet reference circle
d_{sh}	mm	Planet shaft diameter
d_{so}	mm	Sun reference circle
E_p	N/mm2	Young's modulus for planet bolt/shaft
E_{so}	N/mm2	Young's modulus for sun
f_{bpla}	mm	Planet shaft deflection
$f_{H\beta}$	µm	Helix slope deviation according to ISO 1328
$f_{m\alpha}$	µm	Tooth trace deviation
		manufacture error
f_{sh}	µm	(Linearized) deformation components of the
		tooth trace deviation
f_{tso}	mm	Sun torsion deviation
F_m/b	N/mm	Average line load
$(F_m/b)_{max}$	N/mm	Maximum local line load

Fβy	*μ*m	Actual tooth trace deviation
KHβ	[-]	Face load factor
l	mm	Planet bolt/shaft length
p	mm	Number of planets
x	mm	Distance to the left side of the facewidth
κ*β*	[-]	Run-in factor

Table 15.19: Overview of formula symbols

15.3.6.5 Calculation of KHß with manufacturing errors

According to ISO 6336-1(E), the lead variation (fHb) and shaft misalignment (fma) errors are also taken into account in the plane of action. In such a case, their combined effect is taken into account for the flank gap in five cases:

- Case 1: fma = fHb = 0, i.e. no error
- Case 2: fma = |fma|, fHb = |fHb|, so positive values for both errors
- Case 3: fma = +|fma|, fHb = -|fHb|
- Case 4: fma = -|fma|, fHb = +|fHb|
- Case 5: fma = -|fma|, fHb = -|fHb|, so negative values for both errors

The face load factor $K_{Hß}$ is calculated for all five cases, and the maximum value is selected as the face load factor of the gear pair.

The positive direction always lies in the direction of the pinion's material, seen from a common point of contact.

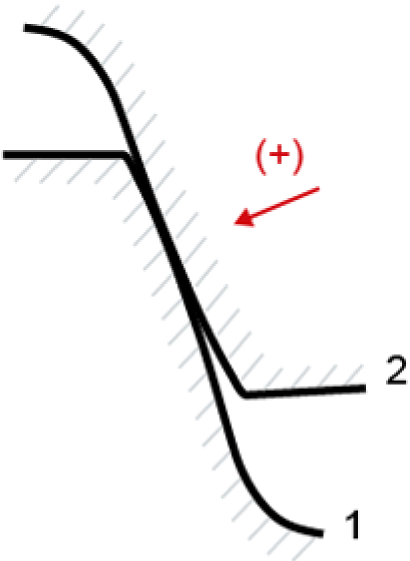


Figure 15.16: Definition of the positive direction

In all five cases, the manufacturing error is documented in the report and in the gaping and load distribution graphics.

Proposed value for fhb and fma

Click on the Sizing button next to the input field for |fHb| to display suggestions of usable data for fHb and fma.

"Maximum" shows the largest possible values for fHb and fma. The values are derived from the fHbT (helix slope deviation) tolerances of the two gears and from the axis alignment tolerance ($f_{\Sigma\beta}$ and $f_{\Sigma\delta}$).

The "statically evaluated" proposal displays the probable maximum values (99.7% probability). This proposal is calculated as follows:

$$f_{tot} = 3 * \sqrt{(f_{HbT1})^2 + (f_{HbT2})^2 + (f_{\Sigma})^2}$$
$$f_{Hb} = 3 * \sqrt{(f_{HbT1})^2 + (f_{HbT2})^2}$$
$$f_{ma} = f_{tot} - f_{Hb}$$

15.3.6.6 Defining the misalignment for individual parts

The following parts are assumed in a planets system:

- Sun wheel
- Planet carrier

- N planet gears with the corresponding n pins
- Internal gear

You can specify the position of these parts in the gear unit and the corresponding misalignment in the **Define axis alignment** dialog. To display this dialog, click on the **Axis alignment** button in the **Factors** or **Contact analysis** tab. All values refer to the shared facewidth.

You can define more parameters in the "**Axis alignment, proportional**" tab for load-specific alignment of system elements:

- Tilting of the sun to the gear axis (see Figure 15.17). If no shaft file is used, the sun can be handled as a "floating sun".
- Tilting of the planet carrier to the gear axis (see Figure 15.18)
- Tilting of the planet pin relative to the planet carrier in circumferential direction dt and in radial direction dr (see Figure 15.19). To model a carrier deformation due to torsion, you must first set a value for dt. This value refers to the planet's facewidth.
- The tilting of the planet gear is relative to the planet bolt axis. The positive misalignment (in circumferential direction dt and radial direction dr) is defined according to the convention (see Figure 15.19).
- The tilting of the internal gear relative to the gear axis (see Figure 15.17). The conical extension of the internal gear can also be taken into consideration.
- The deformation of the planet bolt is caused by the twisting of the planet carrier. If the direction of torque has been input in the "Torsion" tab, the software checks the values and issues a warning message if the prefix for dt has not been entered correctly.
  If the direction of torque has been input in the "Torsion" tab, the software assumes that dt represents the twisting of the carrier due to torque. For this reason, the sign for dt is changed when $K_{H\beta}$ is calculated for load bins with a negative load factor.

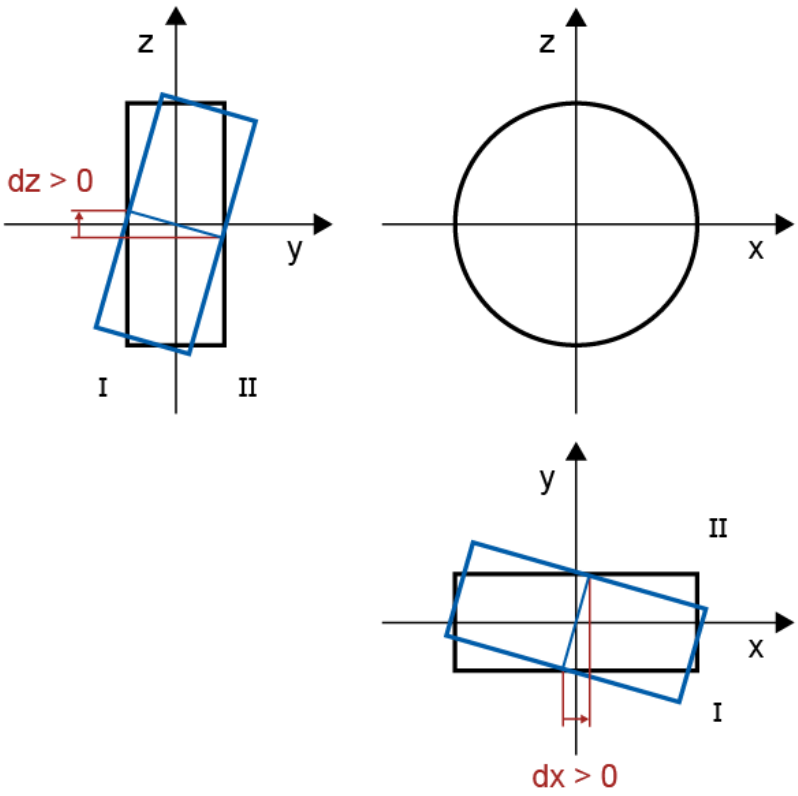


Figure 15.17: Tilting of the sun and internal gear to the gear axis

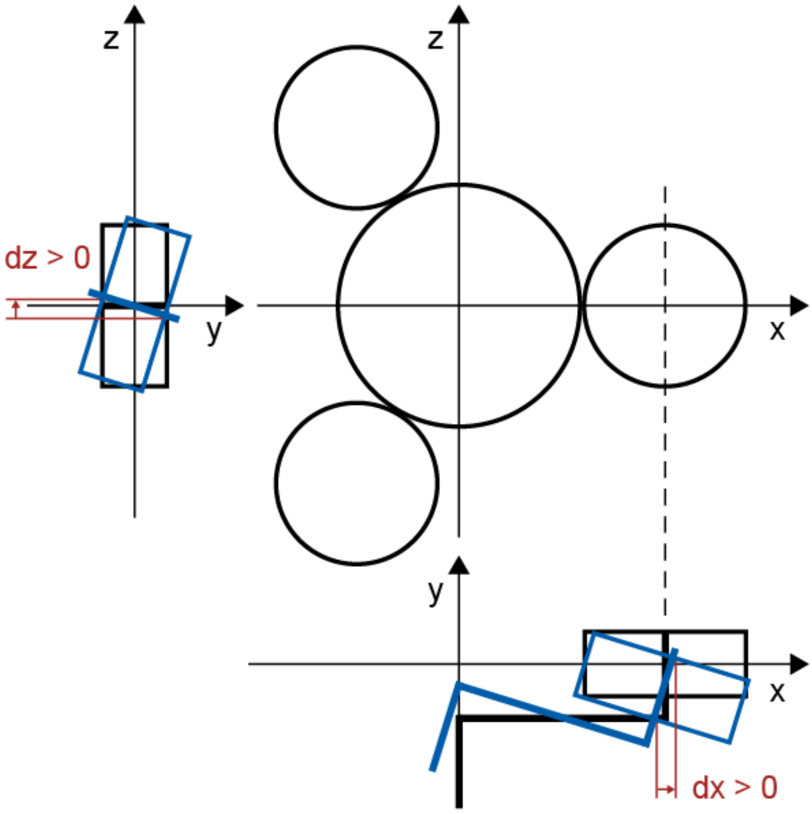


Figure 15.18: Tilting of the planet carrier to the gear axis

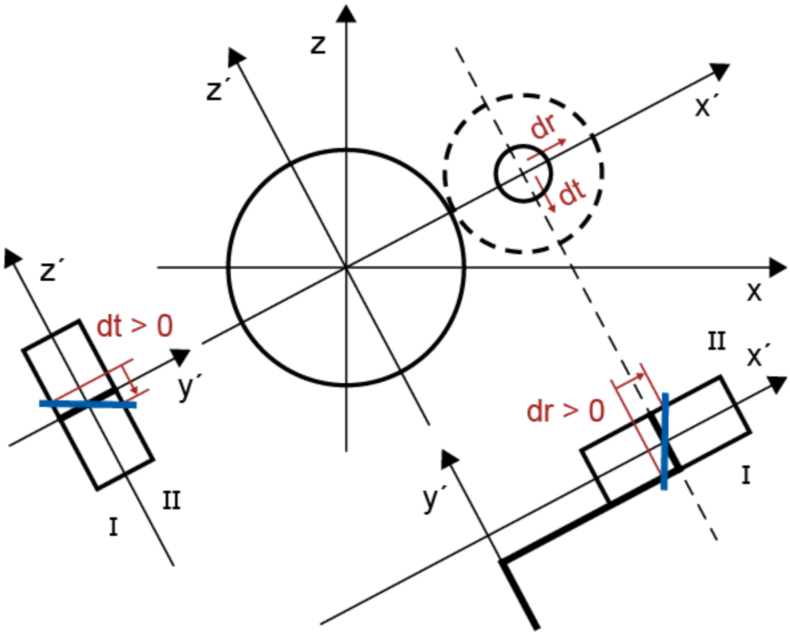


Figure 15.19: Tilting of the planet pin to the planet carrier

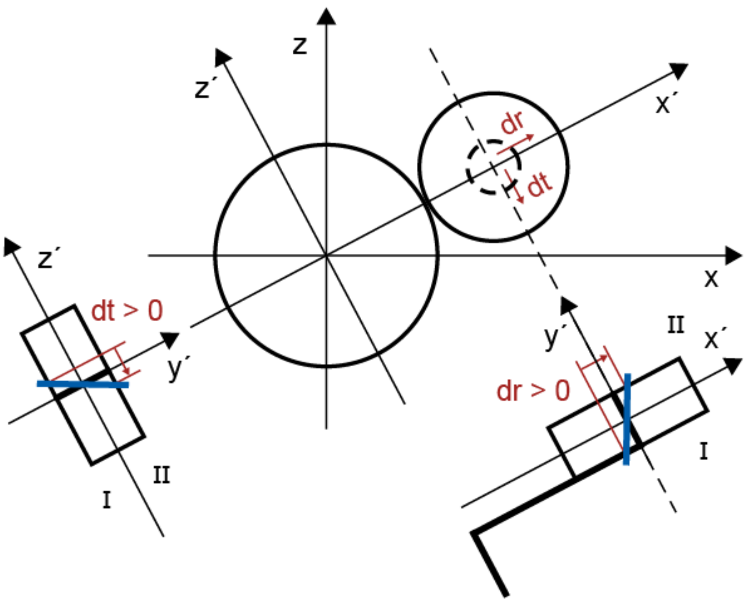


Figure 15.20: Tilting of the planet to the planet pin

You can also use shaft files to define the alignment of all the shafts, except the planet pin. The shaft files undergo the same checks as a gear pair. For example, the value input for gear torque in the shaft calculation files must match the value entered for the gears in the calculation module. The carrier shaft is characterized by its two couplings: one coupling transfers the torque to the sun wheel and the other transfers the torque to the internal gear. The "effective diameter" for both couplings must be the same as the sun-planet center distance. The "length of load application" must also be appropriate for the facewidth of the planet gear. If a shaft file is used for the sun, planet or internal gear, you must click on an additional Plus button to select the meshing that must be taken into consideration.

The proportional axis alignment is scaled with the partial load wt (for contact analysis), or with the ISO factors K_V, K_A and K_γ.

The angle to the first planet Θ defines where the first planet gear must be located for each system definition. Every one of the subsequent planetary gears must have an angular offset of 2π/N to the previous gear. The load distribution on the planet for the specified planet carrier misalignment is dependent on the position of the planets. Modifying Θ will also change $K_{H\beta}$, which is why this entry enables you to calculate the "worst case".

You can set the non-load-specific inclination/deviation error of axis in the **"Axis alignment, constant"** tab.

In the "**Torsion**" tab, you set the side from which torque is introduced to the system or the side from which it is produced (depending on whether the element is a driving or driven element). You can select one of the following 3 options for inputting the direction of torque:

- Not taken into account
- Torque is applied/produced on side I

- Torque is applied/produced on side II

Each configuration is also displayed as a graphic so that the user can check their entries.

If a shaft file is used to define the shaft deformation, the torque is calculated automatically from the results of the shaft calculation.

The planet carrier is usually more complicated than is specified in the shaft calculation. For this reason, the carrier torsion is often greater than determined in the shaft calculation. Consequently, you can either take the torsion deformation value from the shaft calculation or enter it under dt for "planet bolt" (or use an FEM calculation to determine it).

15.3.6.6.1 Calculating planet carrier deformation with FEM

The deformation of the planet carrier causes the planet pin to become misaligned (the pin tilts at dt and dr relative to the planet carrier axis). Use the Finite Elements Method (FEM) to calculate the exact planet carrier deformation. A range of different options are available here:

- The calculated FEM results can be input directly as point coordinates and point deformations (one node for each of side I and side II on a two-sided planet carrier; two nodes on one side for a one-sided planet carrier, (see Figure)

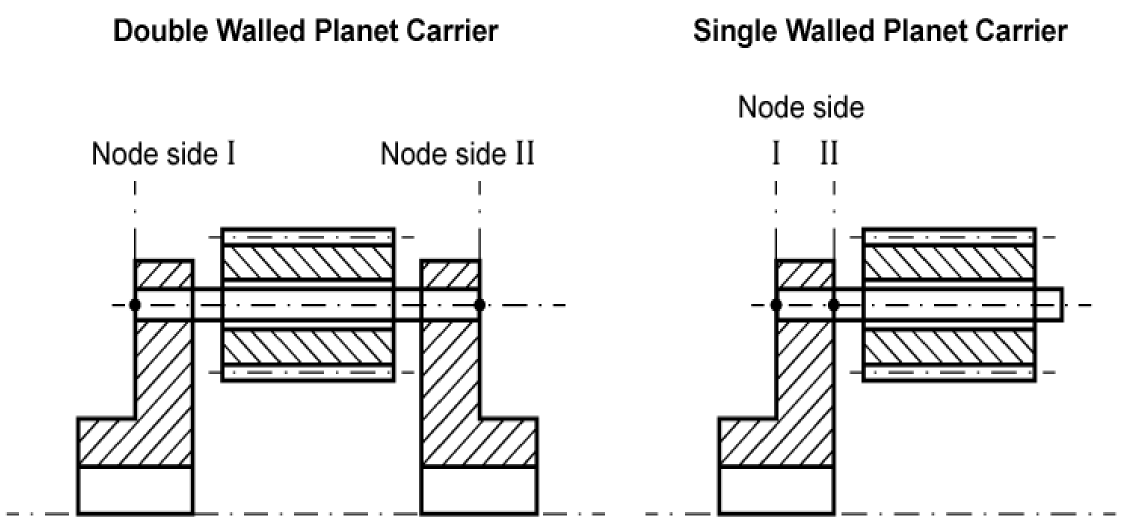


Figure 15.21: Planet carrier tab

- Import the file with the FEM results for the planet carrier deformation. The deformations in both nodes are then extracted from this file. The node coordinates do not need to be specified exactly. The deformation data of the adjacent node is transferred.

Input some of the planet carrier's fundamental dimensions. KISSsoft then generates the carrier in 3D and uses the relative torque to define the planet carrier's deformation.

Input this data:

Single- or two-sided planet carrier
Pin diameter (d)
Coefficient for the external diameter of the planet carrier (f_{wa})
Coefficient for the inside diameter of the planet carrier (f_{wi})
Coefficient for the wall thickness of the planet carrier, which may be different for side I and side II (f_{swI} and f_{swII})
Planet carrier width factor (f_{bpc})
Coefficient for planet carrier's connector (f_{dcon})
Coefficient for the planet carrier's internal connector (f_{dicon})
External flange diameter on side I (d_{faI})
Flange length on side I (L_{fI})
Flange wall thickness on side I (S_{wfI})
External flange diameter on side II (d_{faII})
Flange length on side II (L_{fII})
Flange wall thickness on side II (S_{wfII})
Planet carrier material (select this from the database).

These coefficients can all be input under "Details", either as coefficients or directly, as dimensions. You can also click on the "Dimension planet carrier" and "Dimension flange" buttons to display standard entries for this data.
Remember that you can also set the mesh fineness.

The coefficients and dimensions are shown in greater detail in the next figure. Depending on how the direction of torsion is entered, side I or side II may not be required for a one-sided planet carrier.

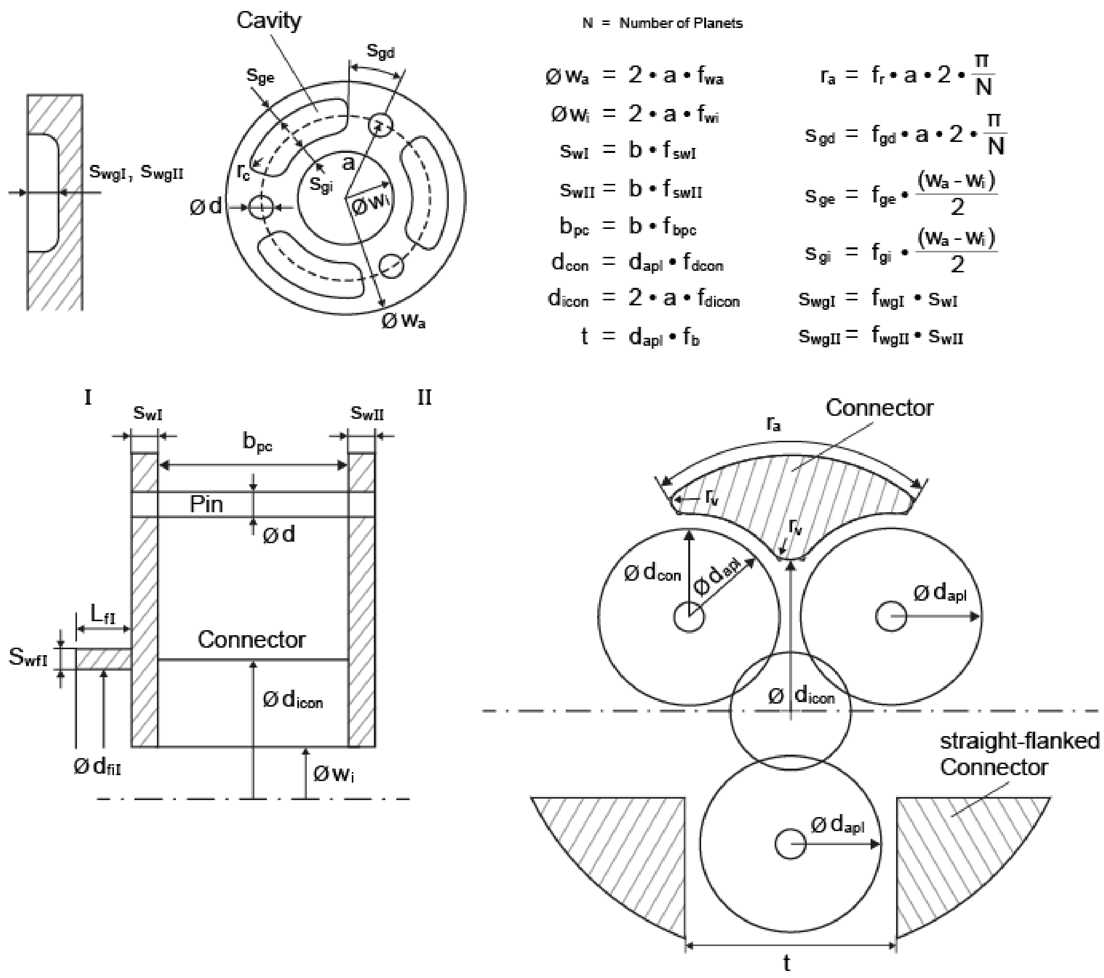


Figure 15.22: Coefficients and dimensions for planet carriers

In addition to the carrier variants shown above, you can also input a step model of the carrier directly.

The point to remember here is that the carrier is clamped on the internal diameter of the flange. If no flange is present, it is clamped on the internal diameter of the planet carrier. If both these diameters are identical, it is clamped along the entire length of the flange and the internal diameter of the carrier. If a step model is used, it is clamped at the specified flange diameter.

15.3.7 Taking into account shaft bending (face load factor and contact analysis)

Shaft bending can be taken into account using the "Define axis alignment" dialog. You can access this dialog either from the "Factors" tab (provided that either the "Calculation according to ISO 6336 Annex E" or "Calculation with manufacturing allowance according to ISO 6336 Annex E" option is selected, in the "Face load factor" field) or the "Contact analysis" tab.

15.3.7.1 Main settings

The **Define axis alignment** dialog is where you define the proportional and constant deviation error of axis ($f_{\Sigma\beta p}$, $f_{\Sigma\beta c}$) and the inclination error of axis ($f_{\Sigma\delta p}$, $f_{\Sigma\delta c}$). The proportional deviation/inclination error of axis is defined at the nominal torque and scaled with the corresponding ISO coefficients strategy (see Load factors, Module specific settings in the **Face load factor/Contact analysis** tab).

Instead of defining the deviation and inclination of the axes directly (linear deformation model), you can also use shaft calculation files for a more precise definition of the effect of bending and torsion on the shafts on which the gears are mounted.

The "**Define axis alignment**" dialog is described below. This is where you determine the axis alignment by using the shaft calculation files. In the "File Shaft Gear 1/Gear 2" fields, enter the file name for the shafts to which the pinion (1) or the gear (2) belong. You must input the file name with its entire path (for example C:\MyCalculations\ContactAnalysis\pinion_shaft.W10). However, if the shaft files are stored in the same folder as the gear calculation file Z12, you only need to input the name of the shaft calculation file (as shown in the figure).

The resulting scaling of the load is displayed in % in the upper part of the dialog.

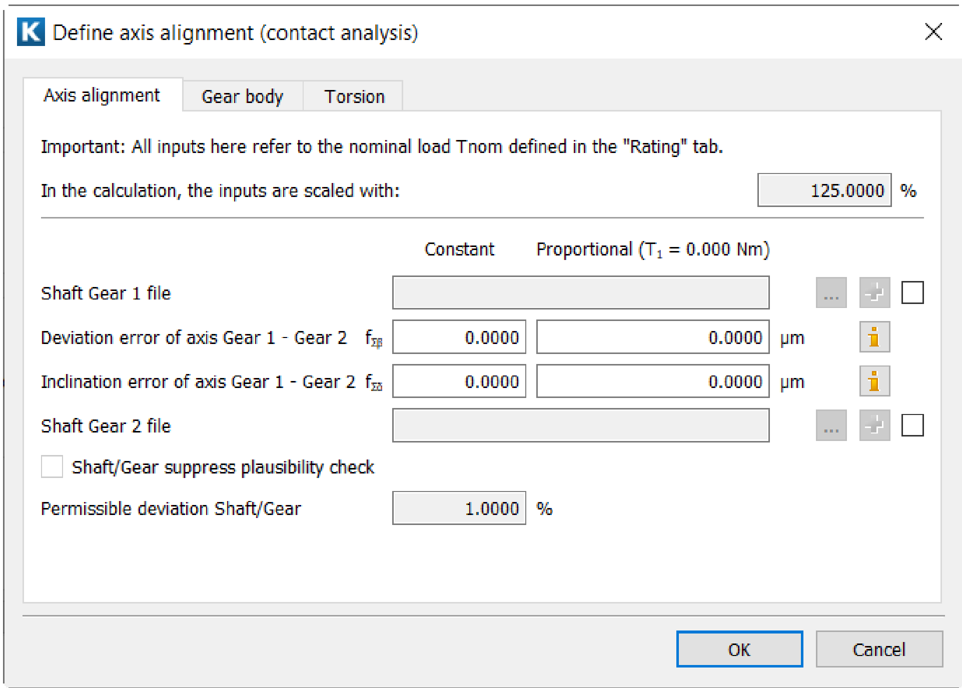


Figure 15.23: Define axis alignment (planets and gear pair)

If a gear body is to be used in the axis alignment calculation, it must be defined in the appropriate special calculation and selected as active in the **Define axis alignment** dialog, in the **Gear body** tab.

If a shaft file is used, select the additional Plus button to select the meshing that is to be taken into account.

Conical expansion can be taken into account for internal gears.

15.3.7.2 Conditions for using shaft calculation files

If you are working with shaft files, the sizing parameters in the gears module must match those in the selected W010 files. More specifically:

1. The pinion geometry must match the geometry defined for the pinion in shaft file 1. The selection is based on the operating pitch circle, the direction (driving/driven) and the contact flank. The same applies to the gear shaft.
2. The gear pair performance must match the gear performance defined in the shaft files.
3. The shaft rotation for both the pinion and the gear (according to shaft files W10) must be consistent. For example, if the pinion rotates in a clockwise direction, the gear must rotate counterclockwise. However, if the gear is an internal gear, both the pinion and gear must rotate clockwise in this example.

From these conditions you can also easily see whether the shaft files can be used for the contact analysis. If one of these conditions is not met, no calculation can be performed.

In addition to the conditions listed above, a number of other conditions (warnings) concerning the helix angle, the facewidth, and the gear's working transverse pressure angle, are also checked.

All the conditions can controlled with the "Permitted deviation shaft/ gear" entry or switched off by clicking the "Suppress shaft/gear plausibility check" setting.

15.3.7.3 Effect of torsion on the body of the gear

You can take the effect of torsion on the body of the gear into account either by applying the results of the shaft calculation or by inputting your own data (the same applies to side I and II). Obviously, the results of the shaft calculation can only be referenced if shaft files have been used to define the axis alignment.

If you defined the gear's torsion in "Side I/Side II"", then the torsion moment of resistance is calculated from the root circle df and the internal diameter.

15.3.7.4 Handling bending and torsion using the results for the shaft

If a gear pair has been found and the shaft calculations performed successfully, the bending and the effect of torsion are determined from the results for the shaft.

The results for bending in each shaft file are all transferred to a single coordinate framework, where pinion contact occurs at 0° and gear contact occurs at 180°. The torsional angle of each gear is assumed to be 0° on the side that is furthest to the left (side I, i.e. the side with the smallest Y-coordinate in the shaft file) and every torsional angle for this particular gear then refers to this side.

15.3.8 Z-Y factors, C-K factors and technology factor

If necessary, you can modify any of the factors that affect the permitted material values (root and flank) as specified in ISO or DIN in the **Z-Y factors** window. When calculation according to AGMA is selected the C-K factors can be modified. The location of the Z-Y and C-K factors window can be found under the **Factors** tab.

The Z_L, Z_V, Z_R, Z_W and Z_X factors affect the pitting stress limit σ_{HG}, the Y_T, Y_{drelT}, Y_{RrelT}, Y_X factors influence the tooth root safety limit σ_{FG}.

Z-Y factors can be specified in predefined range from 0.5 to 2.0. If values outside this range are entered, they are set to 1.0. In the case of C-K factors (AGMA), higher values can be input, but a warning message to inform the user is shown.

The technology factor takes into account the change in tooth root strength caused by processing. In this situation, the material's permissible stress is multiplied by $Y_T \geq 1.0$. This factor is not specified in the DIN or AGMA standards and is therefore set to 1.0.

You can only input the gear rim factor Y_B for the calculation method according to ISO 6336. If you select a different method, this option is deactivated and the factor is set to 1.0.

Tooth root area processing type	**Technology factor Y_T**
Shot peening	
Case-hardened/carbonitrided toothing; not ground in the reinforced areas	1.2
Rollers	
Flame- and induction-hardened toothing; not ground in the reinforced areas	1.3
Grinding	
For case-hardened or carbonitrided toothing	0.7 (general); 1.0 (CBN grinding discs)
Cutting machining	
Not for profile ground toothing!	1.0

Table 15.20: Technology factor according to Linke

According to Bureau Veritas/RINA [27], these technology factors apply:

Tooth root area processing type	**Technology factor Y_T**

Shot peening	Case hardening steel	1.2
Shot peening	Through hardening steel	1.1
Shot peening	Nitriding steel	1.0

Table 15.21: Technology factors as defined by Bureau Veritas/RINA Directives

Y_T is not part of ISO 6336. However, chapter 6.7.2 provides values for increasing σ_{Flim} as a function of shot peening. These can be presented using the technology factor. The values only apply to tooth root bending stresses and shot peened case hardening steel.

Material class	**Technology factor Y_T**
ML	1.0
MQ	1.1
ME	1.05

Table 15.22: Technology factor according to the note in ISO 6336-5:2016, Section 6.7.2

15.3.8.1 Calculate lubricant factor with oil temperature

Unlike in ISO 6336 and DIN 3990, where the calculation is always performed using the oil viscosity at 40°C, the lubricant factor is calculated with the oil viscosity at operating temperature when you click the lubricant factor Z_L checkbox. If this option is selected, the material hardening factor Z_W is also calculated using the viscosity at operating temperature.

15.3.8.2 Calculating the size factors for small gears

Calculation of size factors for small gears similar to that stated in FVA report 410:

If $m_n < 1$, the size factors (according to DIN or ISO) are Z_X and $Y_X > 1$.

► **Note**

If you use this method, you may need to adjust the required safeties that have been used up to now. This is because much higher computational safeties are produced.

15.3.8.3 Calculating the case hardening depth factor according to FVA 271 report

According to FVA 271:2001 [28], two hardening depth-dependent factors Z_{CHD} and Y_{CHD} can be used if case hardening steels are selected for the gear material and if the calculation is not performed according to AGMA.

The factors are limited to the range $0.5 \leq Z_{CHD}, Y_{CHD} \leq 1.0$. The calculation according to FVA 271 considers the user-defined range of the hardening depth at the hardness of 550 HV. This range is then compared to the range of the optimal hardening depth, for which the limits ($CHDoptimal_{min}$ / $CHDoptimal_{max}$) are calculated according to FVA 271.

As there is the comparison of two ranges (user input through case hardening depth at 550 HV and the one calculated according to FVA 271), the following behavior is implemented:

1. If both user input values are smaller than $CHDoptimal_{min}$, the factors Z_{CHD} and Y_{CHD} are calculated according to equations 33 and 38 [28], considering the smaller of the two hardening depth values.
2. If both user input values are larger than $CHDoptimal_{max}$, the factors Z_{CHD} and Y_{CHD} are calculated according to equations 35 and 40 [28], taking into account the larger of the two hardening depth values.
3. If the user input values are between $CHDoptimal_{min}$ and $CHDoptimal_{max}$, the factors Z_{CHD} and Y_{CHD} are equal to 1.
4. If one user input value is smaller/larger than CHDoptimal, the factors Z_{CHD} and Y_{CHD} are calculated according to both equations 33 and 38 and equations 35 and 40 [28], using the smaller value in the strength calculation.

15.3.9 General calculation procedure for KHbeta as specified in ISO 6336-1, Annex E.

1. Import the shaft files, select the correct gears, and then perform the initialization
2. Calculate the shafts and determine the bending lines and torsion in the point of contact (if uniform load distribution is present, determine these values along the facewidth of the gear)
3. Take into account flank modifications from Z012 (not W010)
4. Calculate the gaps in the tooth contact, then the load distribution with tooth contact stiffness and finally calculate $K_{H\beta}$.
5. Use the calculated load distribution to correct the load distribution on the original gears
6. Divide the gears into "sections" whose load values are defined in the previous step
7. Use the flank contact ratio (as a vector) from the previous iteration g_{k-1} and the current flank contact ratio g_k to calculate the root of the sum of the square error.

$$\lambda = \sqrt{\sum \left(100 \cdot \frac{g_k^i - g_{k-1}^i}{g_{k-1}^i} \right)^2}$$

If λ>0.1%, go back to step 2 and perform further iterations. Otherwise exit.

This procedure exactly follows the method described in ISO 6336-1, Annex E, but uses a stricter iteration criterion.

15.4 Reference profile

In contrast to traditional mechanical engineering, where a predefined standard reference profile is most commonly used, the reference profile is often modified in precision mechanics. Input the gear tooth reference profile or the appropriate tool in the **Reference profile** input window. You can input this data either as coefficients, as lengths or as the diameter.

15.4.1 Configuration

The reference profile of the gear teeth is usually predefined. However, you can also define your own hobbing cutter or pinion type cutter. The pinion type cutter parameters are also used in the strength calculation to calculate the tooth form factor. You can also select **Constructed involute** for precision engineering. In this case, the involute is defined directly together with a root radius.

15.4.1.1 Tool: Hobbing cutters

Select the hobbing cutter you require from the selection list and then click the Plus button beside it:

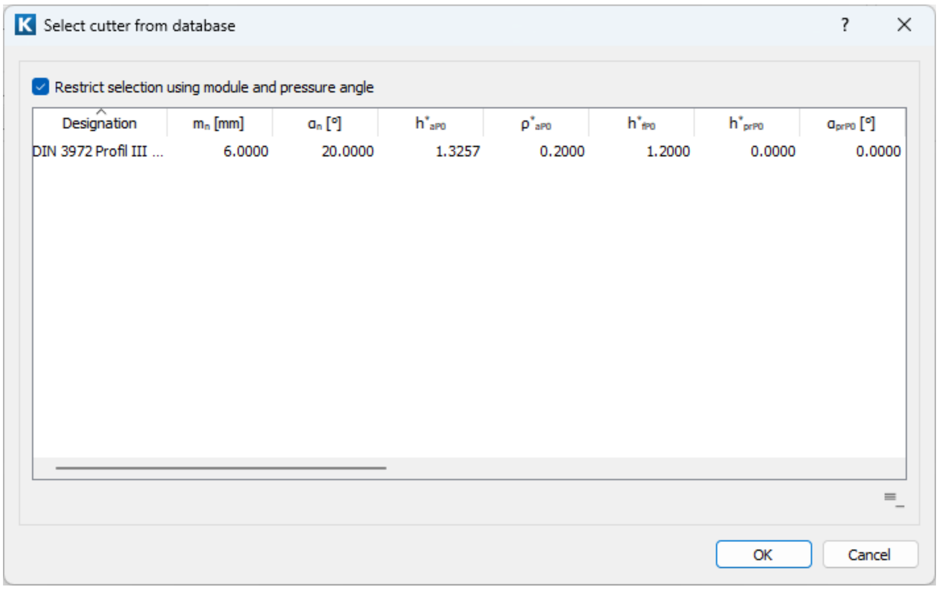


Figure 15.24: Hobbing cutter selection window

If you select a profile (e.g. DIN 3972 III), the list displays the tools that are present in the relevant cutter file. (The name of the cutter file is entered in the database.) Click on the **Restrict selection using module and pressure angle** checkbox to only display tools whose modules and pressure angles match those defined in the gear geometry. By default, only tools that match the selected module and pressure angle are displayed. If tools are selected from the **Tooth form** tab, cutters that meet the condition $\cos(\alpha_n)*m_n = \cos(\alpha_{n1})*m_{n1}$ are also displayed. The standard tolerance is set to + - 1°.

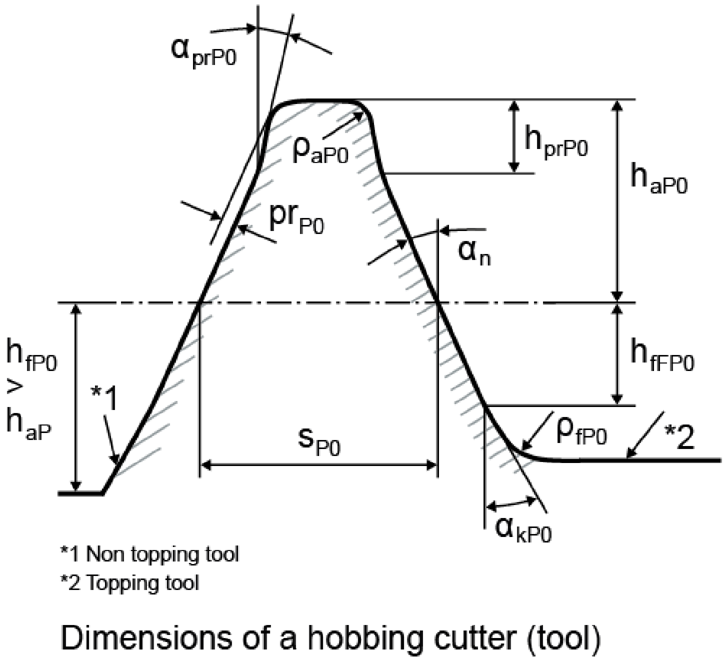


Figure 15.25: Reference profile for tool configuration: Hobbing cutters

Hobbing cutters for asymmetrical gears:

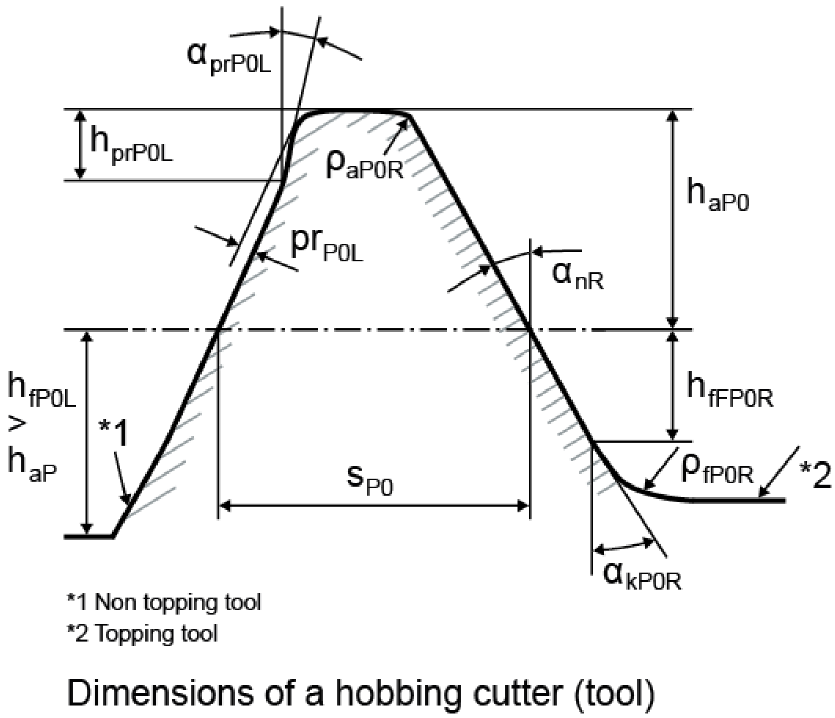


Figure 15.26: Reference profile for asymmetrical gears for the configuration tool: Hobbing cutters

Select **Own input** to directly define your own cutter:

- The cutter addendum coefficient h^*_{aP0} defines the cutter addendum, which then defines the gear root circle. A usual value is 1.25.
- The cutter tip radius coefficient ϱ^*_{aP0} defines the cutter tip radius, which then defines the gear root radius. The tip radius is limited by the maximum geometrically possible radius, depending upon the profile addendum and the pressure angle. This value usually lies in the range 0.2 to 0.38.
- The cutter's dedendum coefficient h^*_{fP0} defines the cutter's dedendum , which then defines the tip circle, for a topping tool. A usual value for this is 1. In a non-topping tool, there has to be a certain amount of clearance between the tool and the gear tip circle, which the software checks. 1.2 is a usual value for an addendum of the reference profile of 1.
- The root radius coefficient ϱ^*_{fP0} defines the root radius of the cutter. In a topping tool, the root radius cuts a tip rounding on the gear in most cases. Depending on the geometric conditions, a chamfer or corner may occur on the tip.
- The protuberance height coefficient h^*_{prP0} defines the protuberance length, measured from the addendum. The protuberance is used as an artificial undercut to prevent a grinding notch from being created. The protuberance height can be calculated from the protuberance size and angle.

- The protuberance angle α^*_{prP0} is usually smaller than the pressure angle. However, in the case of some special cutters, it may also be larger. In this case, no undercut is present, but the tooth thickness at the root of the gear is larger. The protuberance angle can be calculated from the protuberance size and height. If 0 is input, no protuberance is present.
- When calculating the contact ratio, protuberance is not taken into account until it reaches a certain value because a contact under load is assumed for profile modifications. To set the threshold value that takes into account protuberance and buckling root flank for active diameters, select the Calculation > Settings (see chapter 15.22.3, Calculations) menu option.
- The root form height coefficient h_{FfP0}* defines the end of the straight flank part of the tool with a pressure angle α_n. The height is measured from the tool reference line.
- The ramp angle a_{KP0}* defines a ramp flank or a profile modification that is present in the cutter. The length is determined using the root form height coefficient. The angle is greater than the pressure angle α_n. If you enter the value 0, this part will be ignored.
- The threshold value used for protuberance is also taken into consideration here when calculating the diameter and the contact ratio (see chapter 15.22.3, Calculations).
- For the usual tools, the tooth thickness coefficient of reference line s^*_{P0} equals s^*_{P0} = π/2. The value can be overwritten for special tools.
- The addendum coefficient of the gear reference profile h^*_{aP} for a non-topping cutter is defined with the usual value of h^*_{aP} = 1 of the gear reference profile or using the gear's tip circle. The value can be calculated by converting the tip circle value.

By clicking on the **Database** button (next to the data source input field), the hobbing cutter can be written into the database (only if the hob is defined as own input). Once it is in the database, the hob is available for selection under Data source.

15.4.1.2 Tool: Pinion type cutter

Click the Plus button next to the pinion type cutter designation to select a pinion type cutter for internal and external gears from a list. Pinion type cutters as specified in DIN 1825, 1826 and 1827 are listed here. You use this window in the same way as the **Select hobbing cutter** window, 15.4.1.1. By default, the list only displays tools that match the selected module, pressure angle and helix angle.

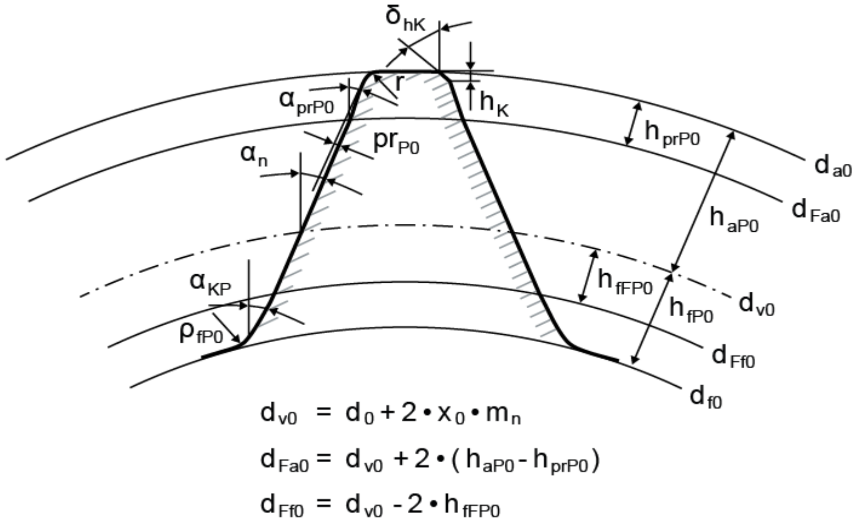


Figure 15.27: Reference profile for tool configuration: Pinion type cutter

Pinion type cutter for asymmetrical gears:

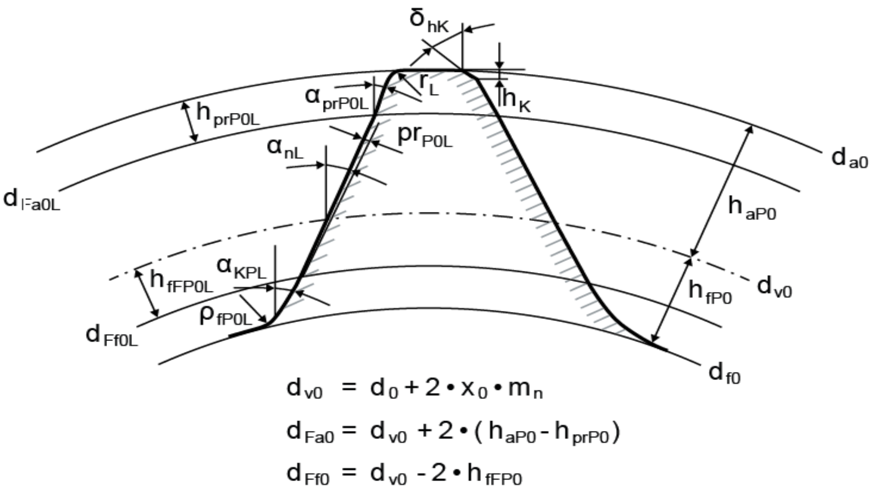


Figure 15.28: Reference profile for asymmetrical gears for the configuration tool: Pinion type cutter

Select **Own input** to directly define your own pinion type cutter:

- KISSsoft can prompt the number of teeth z_0 for the cutter. If the number of teeth is too small, it may not be possible to manufacture the cylindrical gear tip form circle and/or root form circle. If the number of teeth is too great, it may cause collisions during manufacture.
- The pinion type cutter profile shift coefficient x_0 is often unknown. However, it does influence the root circle of the resulting gear. This value is set automatically, together with the number of teeth.

- A pinion type cutter tip often takes the form of a radius or a chamfer. The tip form is not defined in the standards. To be on the safe side, a chamfer of mn/20 was set in the files. However, you should check this value if necessary.
- The pinion type cutter addendum coefficient h^*_{aP0} defines the pinion type cutter addendum that in turn determines the pinion type cutter tip circle and the gear root circle. A usual value is 1.25.
- The pinion type cutter dedendum coefficient h^*_{fP0} defines the pinion type cutter dedendum height that in turn determines the tip circle for a topping tool. A usual value for this is 1. In a non-topping tool, there has to be a certain amount of clearance between the tool and the gear tip circle, which the software checks. 1.2 is a usual value for an addendum of the reference profile of 1.
- The root radius coefficient of the pinion-type cutter ϱ^*_{fP0} defines the radius at the cutter root. In a topping tool, the root radius cuts a tip rounding on the gear in most cases. The input value is only displayed for a topping tool.
- The protuberance height coefficient h^*_{prP0} defines the protuberance length, measured from the addendum. The protuberance is used as an artificial undercut to prevent a grinding notch from being created.
- The protuberance angle α^*_{prP0} is usually smaller than the pressure angle. If 0 is input, no protuberance is present.
- When calculating the contact ratio, protuberance is not taken into account until it reaches a certain value because a contact under load is assumed for profile modifications. To set the threshold value that takes into account protuberance and buckling root flank for active diameters, select the Calculation > Settings (see chapter 15.22.3, Calculations) menu option.
- The root form height coefficient h_{FfP0}* defines the end of the tool involute with the pressure angle α_n. The height is measured from the tool reference line.
- The ramp angle α_{KP0}* defines a ramp flank or a profile modification that is present in the cutter. The length is determined using the root form height coefficient. The angle is greater than the pressure angle α_n. If you enter the value "0", this part will be ignored.
- The threshold value used for protuberance is also taken into consideration here when calculating the diameter and the contact ratio (see chapter 15.22.3, Calculations).
- The addendum coefficient of the gear reference profile h_{aP}* with the usual value of h_{aP}* = 1 defines the gear's tip circle for a non-topping tool. The value can be calculated by converting the tip circle value.

By clicking on the **database** button (next to the data source input field), the hobbing cutter can be written into the database (only if the hob is defined as own input). Once it is in the database, the hob is available for selection under Data source.

15.4.1.3 Reference profile

The reference profiles displayed here are taken from the database. If you can't find a suitable reference profile here, you must first enter it in the database (see chapter 9, Database Tool and External Tables) (**Z000.ZPROF**). Alternatively, select **Own input** from the drop-down list, to open a dialog in which you can edit all the input fields, and so change all the reference profile parameters. The **Label** input field is displayed under the **Reference profile** drop-down list. This is where you enter the name of your own profile, which will then appear in the calculation report.

► **Note**

You do not create a new entry in the database when you define your own profile in the **Own input** field.

The reference profile details are listed according to ISO 53, DIN 867 or DIN 58400. This is the reference profile data for the gear. You can calculate the corresponding values in [mm] by multiplying it with the normal module. Please note the following points:

- If the reference profile is set to **Own input**, the tip alteration is set to zero (see chapter 15.7, Modifications). For this reason, the addendum may change when you toggle from one window to another.
- If you are using reference profile **BS 4582-1:1970 Rack 2** to determine the correct tip and root diameters, you must input an appropriate tooth thickness tolerance of

  $$A_s = -0.1572 \cdot m_n$$

  directly. The tip and root diameter will then match the values defined in BS 4582-1(8).
- The ramp flank is usually used to generate a tip chamfer (also called "semi-topping"). Alternatively, you can also use a small buckling root flank value to generate a profile modification. However, profile modifications are usually defined in the **Modifications** window (see chapter 15.7, Modifications).
- If the angle of the ramp flank is only slightly different from the pressure angle, it is not taken into account in the contact ratio because the assumption for profile modifications is that the contact ratio will not decrease under load. In contrast, the contact ratio should be reduced accordingly for a chamfer. In Settings (see chapter 15.22.3, Calculations), you can specify the difference in angle that is to be used as the threshold in profile modifications and chamfers.
- If a pre-machining tool is used, the additional measure for the pre-machining must be entered separately. You must input the gear's reference profile for the pre-machining. The calculation of the reference profile for final machining then takes the grinding wheel into account and documents this in the report (see chapter 15.4.2, Pre-machining and grinding allowance).

- For profile modifications, where the angle difference < threshold value (see above), the tip form height coefficient h FaP* does not change between pre-machining and final machining.

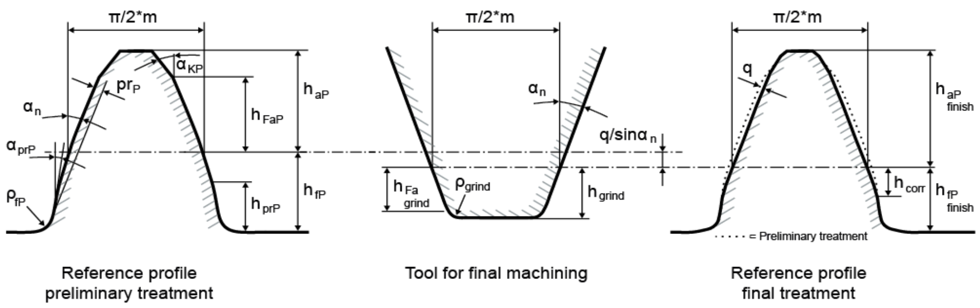


Figure 15.29: Reference profile for configuration: Reference profile gear

Reference profile for asymmetrical gears:

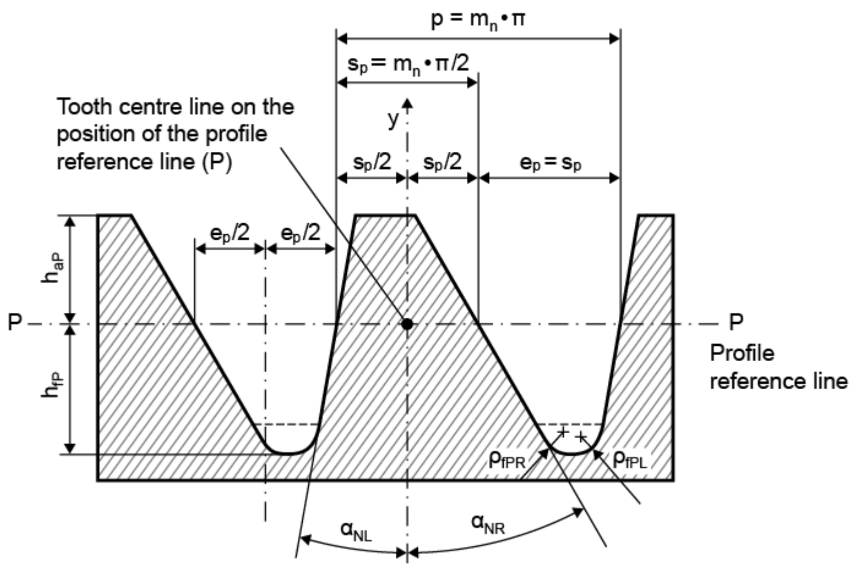


Figure 15.30: Reference profile for configuration: Reference profile for asymmetrical gears

- Click the Sizing button next to the Reference profile drop-down list to display a dialog which contains proposals for reference profiles according to the following criteria:
  - Both gears with (dNf-dFf) minimum
  - Both gears at minimum topland (x is optimized to suit sliding velocity)
  - Both gears at minimum topland (do not change x)
  - Deep tooth form according to the theoretical profile contact ratio defined in the Sizing tab, in the "Module specific settings" (**Calculation > Settings**)

- Click the Convert button next to the "Reference profile" drop-down list to display a dialog window in which you can select a gear, and then copy its reference profile properties.
- haP* always applies as the normal gear reference profile. The tooth thickness on the reference line is defined as

$s_P = \frac{\pi}{2} \cdot m_n$	(12.19)

15.4.1.4 Constructed involute

When you select **Constructed involute**, you do not need to enter as many parameters as you do when you select **Reference profile**. The essential difference is that the manufacturing process is not simulated, and the involute is generated directly. Normally the protuberance part of the constructed involute is defined as an involute with a different base diameter (calculated from α_{prP}). Additionally, protuberance can be defined also as a straight line.

In the gear root, the involute is closed by a radius that is defined by the root radius coefficient ρ_{fP} . In theoretical involutes, the root radius coefficient is usually greater than the coefficient for a reference profile, because the manufacturing process does not involve generation.

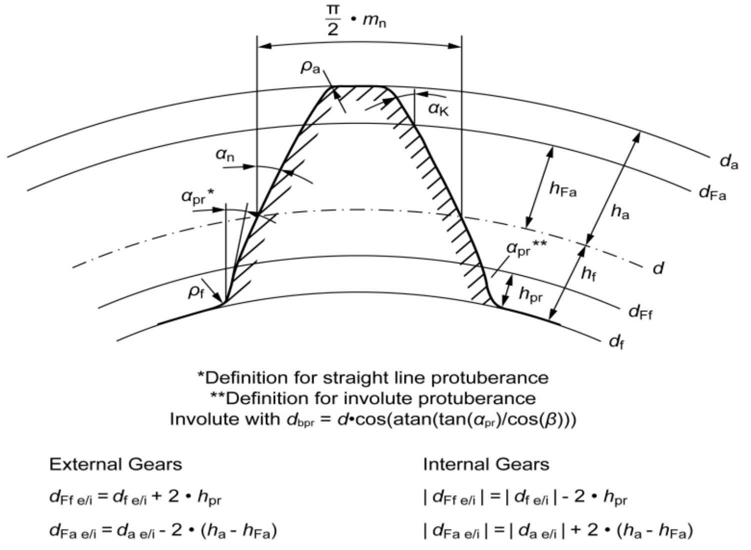


Figure 15.31: Reference profile for configuration: Constructed involute

For asymmetrical gears:

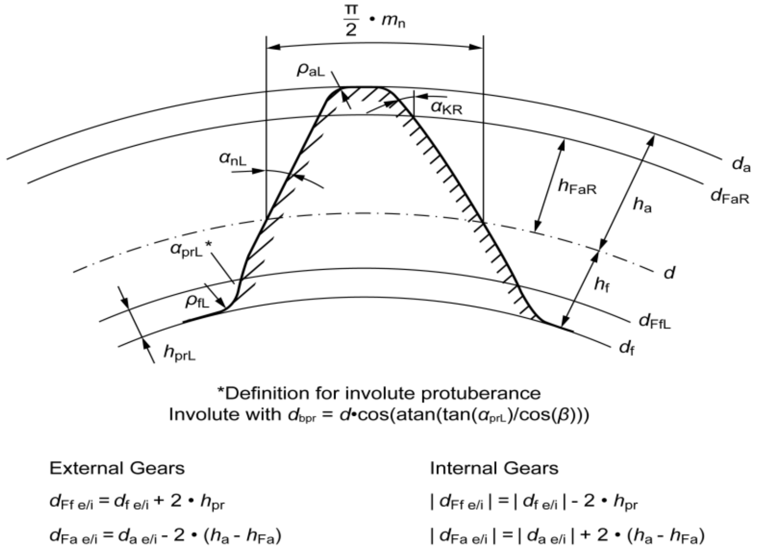


Figure 15.32: Figure: Reference profile for asymmetrical gears for the configuration: Constructed involute

15.4.2 Pre-machining and grinding allowance

Often gears are premachined with a grinding allowance, then hardened and then ground. The tooth flank, but not the tooth root, is usually involved in the grinding process.

Note: If a cutter, pinion type cutter or constructed involute is selected as the pre-machining tool, the gear reference profile for pre-machining is calculated internally from the tool data.

In this case, the root circle is created by the pre-machining cutter and the flank by the grinding process. To complete this process correctly, select either **Pre-machining** (with own input, or with reference profile for grinding allowance III or IV as specified in DIN 3972) or select **Final machining**. If you decide to use pre-machining, the **Grinding allowance** field is displayed. You can also add your own tolerances to the database. Enter the pre-machining tool's profile (exception: h_{aP} *) as the reference profile. For the tooth thickness deviations (tolerances), enter the tooth thickness allowance for the finished gear teeth (A_s). In KISSsoft, the grinding allowance is calculated for the finished toothing. The pre-machining is then performed using the following tooth thickness allowance:

$$A_s' = A_s + \frac{2 \cdot q}{\cos \alpha_n}$$

For special requirements, click on the Plus button in the **Define grinding allowance tolerance** window, to increase the tolerance. If a value is input for qmax-qmin, then qmax = q+(qmax-qmin)/2 and qmin = q-(qmax-qmin)/2 are used to define the pre-machining allowances.
The tolerance interval qmax-qmin is limited to the smaller value of either 200% of the tooth thickness tolerance interval (As.e-As.i), or 30%, or the grinding allowance (q).

KISSsoft then determines the reference profile that corresponds to the finished tooth form. This tooth form will also be used to calculate the factors Y_F and Y_S for the tooth root strength. The tooth form is then defined automatically by overlaying the pre-machining contour with the subsequent grinding process. The root diameters are derived from the reference profile for pre-machining. The control data (e.g. base tangent length) is calculated and printed out for both the premachined and the finished gear teeth.

► **Important exception**

The addendum coefficient h_{aP}* is the theoretical addendum coefficient that is used to calculate the theoretical tip diameter coefficient. The appropriate minimum root height of the hobbing cutter h^*_{fP0}, which is required to create the tooth form without topping, is output in the report. h_{aP}* always applies as the final machining reference profile for the gears. The tooth thickness on the reference line is π/2*mn.

15.4.3 Tip alteration

The tip alteration $k*m_n$ is usually calculated from the profile shift total to ensure that the tip clearance does not change. However, if the reference profile is set to **Own Input**, the tip alteration will not be calculated. In an external gear pair, a reduction in the tip alteration results in a negative value for the tip circle reduction. In contrast, in internal toothings, the result is a positive value for both gears, and therefore also an increase in the tooth height. In KISSsoft, the tooth height of internal toothing is not increased, and therefore the tip alteration is limited to 0.

Alternatively, you can specify your own tip alteration. However, this only has an effect on non-topping tools. Otherwise, the value is set to 0 when it is calculated. Click a Sizing button to calculate the proposed value for a constant tip clearance.

Click the Convert button to input a tip diameter (either d_a, d_{aE} or d_{ai}) which is used to convert the tip alteration, using the reference profile present.

15.5 Manufacturing

This tab is where you specify the manufacturing process for pre-machining and final machining. You can also check whether special manufacturing processes such as power skiving or honing can be used. Manufacturing deviations (measured deviations, natural twist, slope and form deviations) can be set in a table.
Setting manufacturing deviations enables the influence on the meshing properties to be determined, using the tooth contact analysis function. This provides a way to determine the effect of manufacturing deviations (according to the specified accuracy grade) on the theoretically optimally sized toothing.

15.5.1 Details about the grinding process

This is where you define the grinding process. These inputs are necessary if a grinding allowance is present in the **Reference profile** tab, or if profile modifications are added in the **Modifications** tab. The start of modification at the tip or root specifies the height at which the grinding process processes the gear. The radius of the grinding wheel tip must also be predefined. If the grinding process reaches the diameter that matches the selected start of modification at the root, the software simulates the complete roll-off of the grinding tool. The grinding notch that may result is calculated and taken into account in the strength calculation according to ISO/DIN. You can input the data as coefficients, as lengths or as the diameter. Where profile modifications are defined over a particular length (e.g. linear root relief), the length is measured from the selected start of the modification at the tip or root.
The manufacturing process with a tool and gear can only be checked in the **Manufacture** 2D graphic.

Usually, the tooth root area is not included in grinding. When you enter a value for **Start of modification at root** you can, if required, also specify that the root area is included in grinding. The grinding wheel addendum [h^*_{grind}] is also usually entered in this case. The profile modifications in the root then start from the tip form height [$h_{Fa}{}^*_{grind}$] of the grinding wheel, but not before the gear's base circle.

► **Note:**

Recommendation for the **Generation** or **Form grinding** setting:
if it is not known whether the grinding process is performed using the generation or form grinding process, we recommend you select the "Form grinding" process, if you input finished teeth without a pre-machining tool. We also recommend you select "Generation" if you input finished teeth with a pre-machining tool.

15.5.2 Power skiving

Select this option if you need to check whether power skiving can be used as the final machining process on a gear. Click the Plus button to open a window in which you can enter specific additional details.

Use the checks to generate a rough estimate of the limitations of the tool and the machine and also, optionally, to show possible collisions between the tool and the workpiece. You can use the tests to perform an initial evaluation, but this cannot be regarded as a replacement for a final analysis performed together with the tool manufacturer.

Tool selection

The **Check for Power skiving** dialog is where all the entries for the checks are defined. The maximum and minimum possible skiving wheel diameters are the key values for selecting the appropriate tool. These values are already stored for specific machines, and can also be entered manually.

The default number of teeth on the tool is set to 20. Click on the Sizing button to calculate a suitable number of teeth which takes into account all the currently active tests.

Meshing tool with work piece

Click on **Meshing tool with work piece** to define a tool/workpiece pairing with reference to the helix angle. You can enter this value either as the axial crossing angle or as the helix angle of the tool.

The system then uses these values to check whether power skiving is actually possible for the tooth geometry of this particular tool/workpiece combination.

Collision check

The system can also check the configuration for possible collisions between the workpiece and the tool. To do this, select the corresponding scenarios in **Collision check**. In each case, enter the relevant distance to the gear teeth, the "Groove width", and the relevant diameter, the "Groove diameter".

Results and documents for tool manufacture

The results are listed in the report. Companies that purchase tools from suppliers can use the "Offer" button to generate a special document that contains all the details needed for obtaining an offer for a power skiving tool.

Fine sizing

The **Check for Power skiving** function is also available as part of the fine sizing process. The check is performed automatically once the conditions have been set. In the tab that contains the fine sizing results, the user can right-click on the "Display columns" function to activate it and display the "PSKx" columns.

Interface to the Gleason power skiving program

We recommend you use Gleason's power skiving software to monitor the manufacturing process more accurately, if Gleason machines are used for the power skiving process. To do this, select the **Generate input data for Gleason's in-house program** checkbox. This automatically generates the appropriate " GleasonPowerSkivingInput-?.cuc" file every time a calculation is performed.

15.5.3 Honing

Select this option if you need to check whether honing can be used as the final machining process on a gear. Click the Plus button to open a window in which you can enter specific additional details.

Use the checks to generate a rough estimate of the limitations of the tool and the machine and also, optionally, to show possible collisions between the tool and the workpiece. You can use the tests to perform an initial evaluation, but this cannot be regarded as a replacement for a final analysis performed together with the tool manufacturer.

Collision check

The system can also check the configuration for possible collisions between the workpiece and the tool front or rear side. To do this, select the corresponding scenarios in "Collision check". The required data is displayed in the individual help screens (Info button).

Results and documents for tool manufacture

The results are listed in the report. Companies that purchase tools from suppliers can use the "Offer" button to generate a special document that contains all the details needed for obtaining an offer for a honing tool.

Fine sizing

The **Check for Honing** function is also available as part of the fine sizing process. The check is performed automatically once the conditions have been set. In the tab that contains the fine sizing results, the user can right-click on the **Display columns** function to activate it and display the "Honing" columns.

15.6 Tolerances

Gear teeth geometry is calculated for a backlash-free state. A slightly smaller tooth thickness is manufactured, to prevent the gears jamming in practice. This reduction in tooth thickness (in contrast to the backlash-free state) is known as the tooth thickness allowance. The upper tooth thickness allowance is the upper limit of the tooth thickness. The lower tooth thickness allowance is the lower limit of the tooth thickness.

► **Example:**

Tooth thickness in a backlash-free state:	4.560 mm
Upper tooth thickness allowance:	-0.050 mm
Lower tooth thickness allowance:	-0.060 mm
This results in the actual tooth thickness:	4.500 to 4.510 mm

15.6.1 Tooth thickness tolerance

This drop-down list contains the tolerances listed below. You can also include your own tolerance tables. You will find more detailed information about this in the section about the KISSsoft Database tool (see chapter 9.4, External tables).

15.6.1.1 DIN 3967

Selection of a tolerance as specified in DIN 3967 (for a gear unit with a module greater than 0.5 mm). Suggestions as defined by Niemann [7] (page 84):

Cast ring gears	a29, a30
Ring gears (normal clearance)	a28
Ring gears (narrow clearance)	bc26
Turbo gears (high temperatures)	ab25
Plastic machines	c25, cd25
Locomotive gears	cd25
General mechanical engineering,	
Heavy machines, non-reversing	b26
General mechanical engineering,	
Heavy machines, reversing	c25,c24,cd25,cd24,d25,d24,e25,e24
Vehicles	d26
Agricultural vehicles	e27, e28
Machine tools	f24, f25
Printing presses	f24, g24
Measuring gear units	g22

15.6.1.2 ISO 1328

The current edition of ISO 1328 no longer includes fit (tolerance) classes for tooth thickness allowances. For this reason, many companies have continued to use the fit (tolerance) classes specified in the old 1975 edition.

15.6.1.3 DIN 58405

Proposals as specified in DIN 58405, Part 2: Allowances for precision mechanics;
usual gear modifications as defined in DIN 58405 Sheet 2

Material	**Processing**	**Center distance tolerance**	**Base tangent length tolerance**
Hardened steel	Ground	5J	5f
Heat treatable steel	finely milled	6J	6f

Light metal	finely milled	7J	7f
Light metal	finely milled	8J	8f
Steel/laminate	finely milled	6J	6e
Steel/laminate	finely milled	7J	7d/7c
Light metal	finely milled	8J	8d/8c
Plastic	milled	9J	9e/9d
Plastic	injection molded	10J	10e

15.6.1.4 Own Input

Select this option to input your own data. However, you should note that the values for tooth thickness allowance, the normal or circumferential backlash (per gear) and the base tangent length allowance all depend on each other. The (negative) base tangent length allowance corresponds to the normal backlash.

15.6.2 Tip diameter allowances

You can specify the tip diameter allowances if a non-topping tool has been defined. In contrast, the tip diameter allowances for a topping tool are defined from the tooth thickness allowances. These allowances influence the effective contact ratio due to the effective tip circle.

Click the Plus button to specify a tolerance field according to ISO 286. The tolerances prefix operator is changed in internal toothings because the tip circle is used as a negative value in the calculation. The tolerance class is saved internally and modified when the tip circle changes.

Click the Convert button to specify the minimum and maximum tip diameter from which the allowances are to be calculated.

15.6.3 Root diameter allowances

Root diameter allowances are usually calculated from the tooth thickness allowances. In the gear cutting process, the gear backlash is produced by reducing the manufacturing distance of the tool. This is why the root diameter allowances depend on the tooth thickness allowances.

A different manufacturing process is used in special cases, e.g. for sintered gears or extruded plastic gears. The user can then input their own root diameter allowances.

Click the Convert button to specify the minimum and maximum root diameter from which the allowances are to be calculated. Click the Plus button to specify a tolerance field according to ISO

286. This defines the allowances, which only need to be entered once in the input screen. The tolerance class is not saved for later use.

15.6.4 Center distance tolerances

The center distance tolerances are defined either by a standard tolerance taken from the database or the value you enter in the **Own Input** field. They influence the gear backlash and the contact ratio.

15.6.5 Settings

The base tangent length and the mass across balls and rollers for the most suitable number of teeth over which the measurement is to be taken, or the roller diameters, are specified in the report. If you want to use a different number of teeth spanned, or a different diameter of ball/pin in an existing drawing, this is where you can overwrite the values selected by the software.

However, no results are output if you enter values for which a measurement cannot be performed. If the **do not cancel when geometry errors occur** (see chapter 15.22.1.3, Don't abort if geometry errors occur)option is selected, test masses are also output for cases in which they could not be measured, for example, for points of contact above the tip circle.

► **Note**

The proposed ball/roller diameters are taken from the **Z0ROLLEN.dat** file. These values are taken from the **Z0ROLLENANSI.dat** file for splines as defined in ANSI 92.1. This file corresponds to the recommended diameters specified in DIN 3977. You can then use an Editor to modify the existing ball/pin. You will find more detailed information about how to handle external datasets in the External tables section (see chapter 9.4, External tables).

15.7 Modifications

Tooth form modifications can be defined in the **Modifications** tab. Tip chamfer/rounding and face chamfer can also be defined. Profile and flank line modifications can be defined in the **Modifications** table.

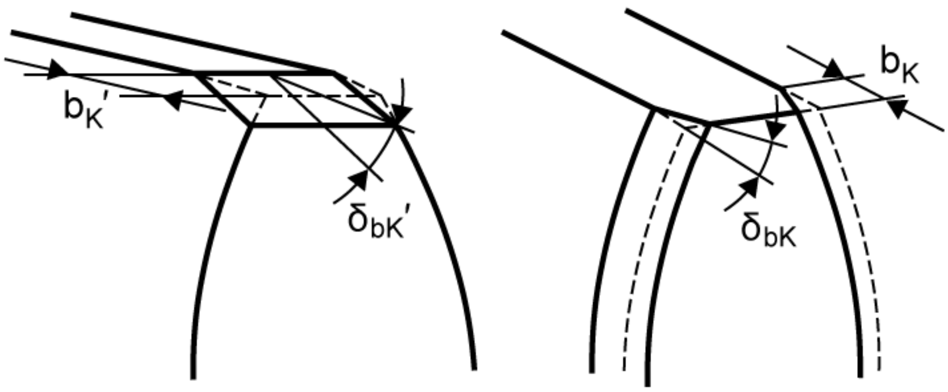


Figure 15.33: Defining face chamfer

► **Note:**

The face chamfer on the tip is not entered for gear calculations as it does not reduce the strength. However, if an unusually large chamfer is involved, h_k' and δ_{bK}' can be simulated by a tip chamfer or face chamfer. The standards do not offer any guidance for this.

15.7.1 Modification type

To create a new entry in the list of modifications, click the Plus button. Double-click on a cell in the **Type of modification** column to display a drop-down list if you want to change the value in that cell.

The next two sections, (see chapter 15.7.3, Profile modifications) and (see chapter 15.7.4, Flank line modifications), describe the method for performing modifications according to ISO 21771.

Inputting different modifications for right or left flank: In the **Flank** drop-down list, you can specify whether a modification is to be applied to the right flank, the left flank or both flanks.

Definition of the right-hand/left-hand tooth flank (according to ISO 21771):

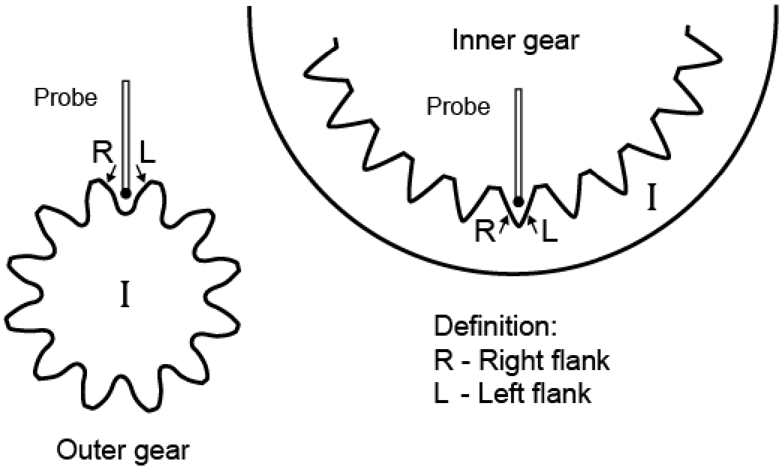


Figure 15.34: Definition of the right-hand/left-hand tooth flank according to ISO 21771

15.7.2 Individual modifications per tooth

Each tooth in a cylindrical gear can be modified individually. The individual modification per tooth option can be activated by clicking the Settings button (if the strength calculation is not activated). The modifications for each tooth can be specified in the "Modifications" table. The applied modifications change the tooth form graphic for the specific tooth (only in transverse section) and the 3D model.

15.7.3 Profile modifications

Profile modifications are deviations from the involute, known as height modifications. The sections that follow detail the possible profile modifications you can make in KISSsoft.

► **Note 1:**

Before you can define height modifications, you must first input the length factor L_{Ca^*}. The length factor is the roll length L_y (from the start of the modification to the tip form circle or root form circle) divided by the normal module: $L_{Ca^*} = (L_{dFa} - L_{dC})/m_n$ or $L = (L_{dC} - L_{dFf})/m_n$. The roll length L_y is calculated according to ISO 21771, Equation 17, or DIN 3960, Equation 3.3.07.

The theoretical diameter d_a or d_{Fa} is always used to calculate the start of the modification at the tip.

► **Note 2:**

Measuring tip relief C_a directly on the tip circle may be inaccurate. If tip reliefs have been defined, the report states the tip relief on a special measuring circle called d_{check}, for measuring purposes. Measuring circle $d_{check} = d_{Fa.i} - 0.02 \cdot m_n$.

► **Note 3:**

Like tip reliefs, different profile modifications can also be predefined with negative C_a parameters in exceptional circumstances. As the grinding process always removes material, a negative tip relief results in a tooth form where the tooth root removes material at a constant rate (according to C_a). In the predefined modification length range, the amount of material removed is reduced so that it is zero at the tip ((see Figure 15.35).

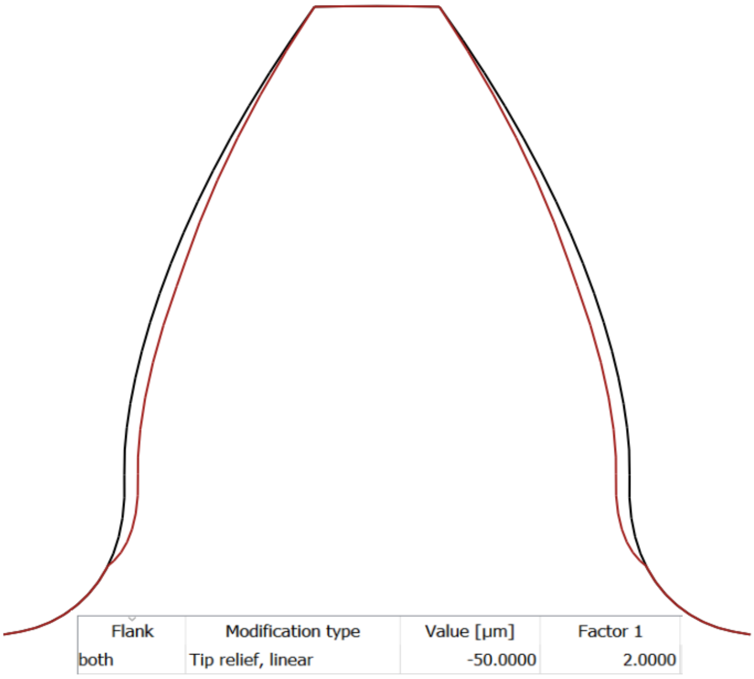
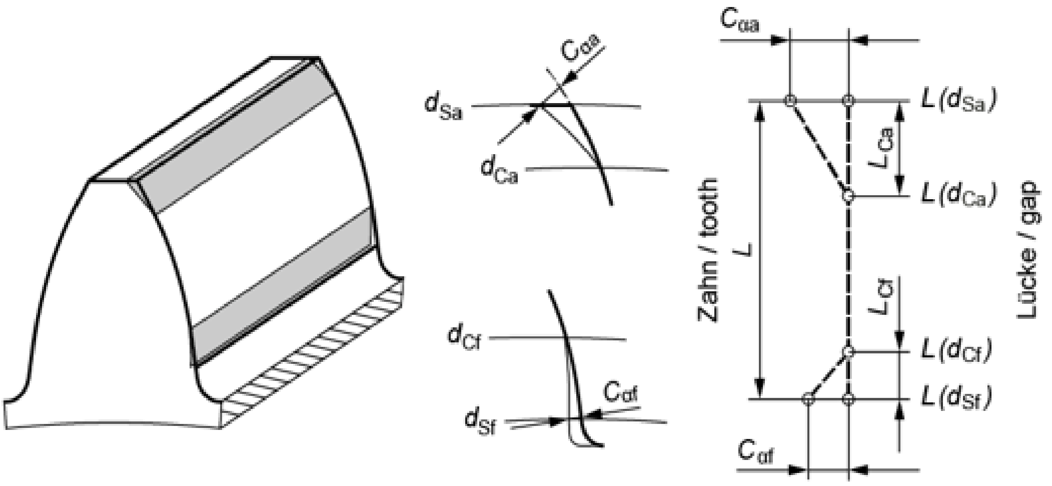


Figure 15.35: Profile modification with negative Ca parameters

15.7.3.1 Tip and root relief, linear

Figure (see Figure 15.36) shows a linear tip and root relief with transition radiuses as specified in the profile diagram. The **Value** and **Factor 1** settings are also shown in this figure. The start of the modification at the tip and root can be defined in the **Manufacturing** tab. The transition from the relief to the involute is not uniform, which can cause increased local contact stresses.



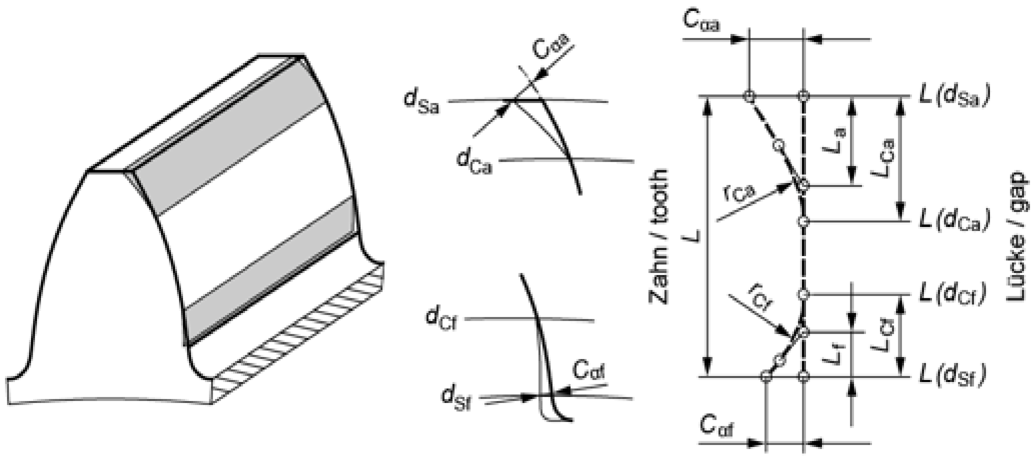
d_{Sa}	tip relief, start, at tip	d_{Sf}	root relief, start, at root
d_{Ca}	tip relief, end, at tip	d_{Cf}	root relief, end, at root
$C_{\alpha a}$	tip relief, value, normal to involute	$C_{\alpha f}$	root relief, value, normal to involute
L_{Ca}	tip relief, roll length	L_{Cf}	root relief, roll length
L	roll length		

	Value	Factor 1	Factor 2
Inputs	$C_{\alpha a}$, $C_{\alpha f}$	L_{Ca}/m_n, L_{Cf}/m_n	-
Conditions	$\neq 0$	> 0	-

Figure 15.36: Linear tip and root relief

15.7.3.2 Tip and root relief, linear with transition radius

Figure (see Figure 15.37) shows a linear tip and root relief with a transition radius as defined in the profile diagram. The **Factor 1** and **Factor 2** settings are also shown in this figure. The start of modification at tip and at root can be set in the **Manufacturing** tab. The transition between the relief and involute is tangential.



d_{Sa}	tip relief, start, at tip	d_{Sf}	root relief, start, at root
d_{Ca}	tip relief, end, at tip	d_{Cf}	root relief, end, at root
$C_{\alpha a}$	tip relief, value, normal to involute	$C_{\alpha f}$	root relief, value, normal to involute
L_{Ca}	tip relief, roll length	L_{Cf}	root relief, roll length
r_{Ca}	transition radius, at tip	r_{Cf}	transition radius, at root
L	roll length		

	Value	Factor 1	Factor 2
Inputs	$C_{\alpha a}$, $C_{\alpha f}$	L_{Ca}/m_n, L_{Cf}/m_n	r_{Ca}/m_n, r_{Cf}/m_n
Conditions	> 0	> 0	> 0

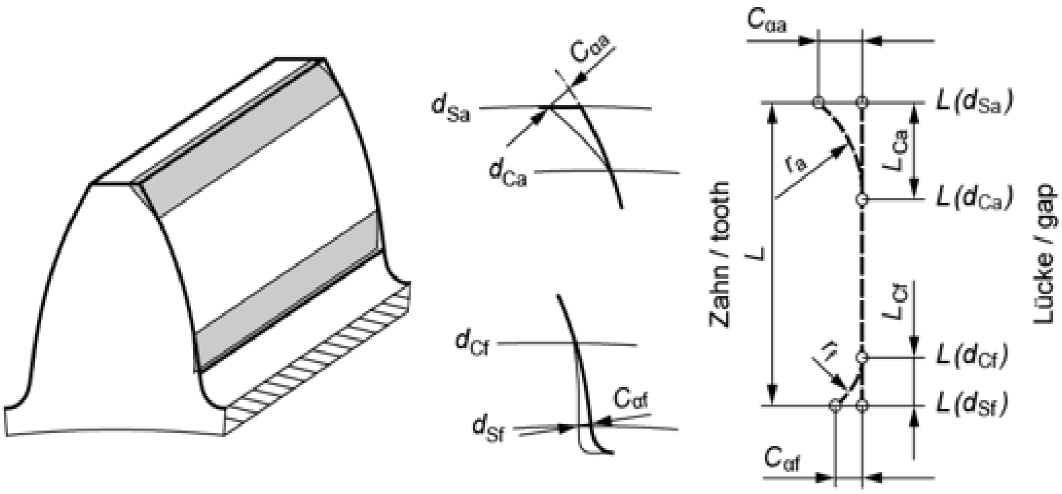
Figure 15.37: Linear tip and root relief with transition radius

If **Factor 2** = 0, then r_{Ca} is calculated in such a way that $L_a = 0.8 \cdot L_{Ca}$ applies. The corresponding **Factor 2** is calculated and applied. If **Factor 2** is so large that $L_a < 0.75 \cdot L_{Ca}$ applies, then r_{Ca} is calculated in such a way that $L_a = 0.75 \cdot L_{Ca}$ applies. The corresponding **Factor 2** is calculated and applied.

Similarly, to represent root reliefs, input the values for $C_{\alpha f}$ and the quotient from L_{Cf} and m_n, and the quotient from r_{Cf} and m_n.

15.7.3.3 Tip and root relief, arc-like

Figure (see Figure 15.38) shows the arc-like tip and root relief as defined in the profile diagram. The **Value** and **Factor 1** settings are also shown in this figure. The start of the modification at the tip and root can be set in the **Manufacturing** tab. The transition between the relief and the involute is tangential, so there is a smooth transition between the involute and the tooth form relief part.



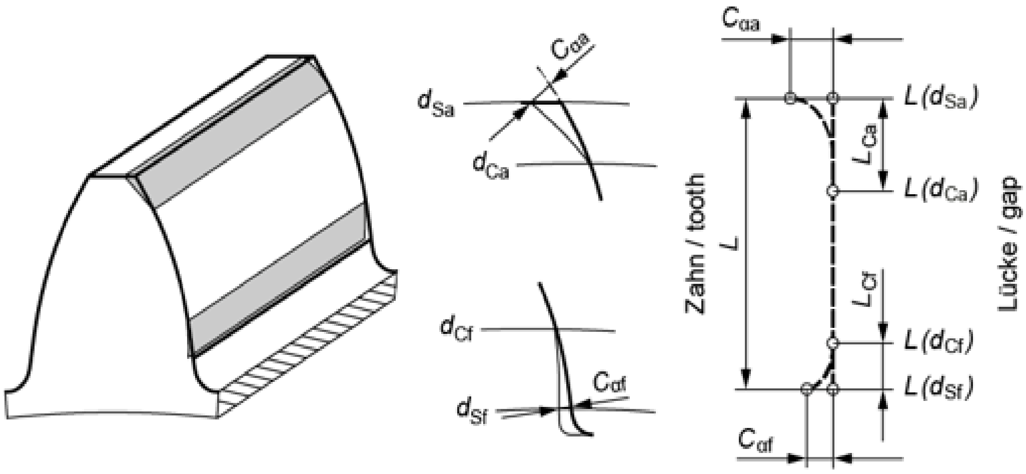
d_{Sa}	tip relief, start, at tip	d_{Sf}	root relief, start, at root
d_{Ca}	tip relief, end, at tip	d_{Cf}	root relief, end, at root
$C_{\alpha a}$	tip relief, value, normal to involute	$C_{\alpha f}$	root relief, value, normal to involute
L_{Ca}	tip relief, roll length	L_{Cf}	root relief, roll length
r_a	tip relief, radius	r_f	root relief, radius
L	roll length		

	Value	Factor 1	Factor 2
Inputs	$C_{\alpha a}$, $C_{\alpha f}$	L_{Ca}/m_n, L_{Cf}/m_n	-
Conditions	$\neq 0$	> 0	-

Figure 15.38: Arc-like tip and root relief

15.7.3.4 Tip and root relief, progressive

Figure (see Figure 15.39) shows the progressive tip and root relief as defined in the profile diagram. The **Value** and **Factor 1** settings are also shown in this figure. For setting **Factor 2** (see chapter 15.9.2.11, Progressive profile modification). The start of the modification at the tip and root can be defined in the **Manufacturing** tab. The transition between the relief and involute is tangential.



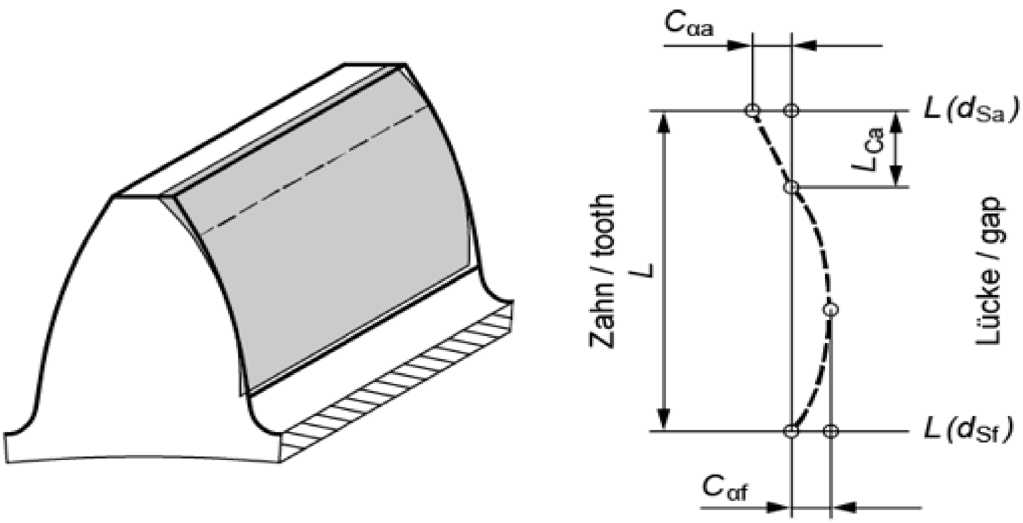
d_{Sa}	tip relief, start, at tip	d_{Sf}	root relief, start, at root
d_{Ca}	tip relief, end, at tip	d_{Cf}	root relief, end, at root
$C_{\alpha a}$	tip relief, value, normal to involute	$C_{\alpha f}$	root relief, value, normal to involute
L_{Ca}	tip relief, roll length	L_{Cf}	root relief, roll length
L	roll length		

	Value	Factor 1	Factor 2
Inputs	$C_{\alpha a}$, $C_{\alpha f}$	L_{Ca}/m_n, L_{Cf}/m_n	see manual
Conditions	$\neq 0$	> 0	$5 < \mathit{Factor\ 2} < 20$

Figure 15.39: Progressive tip and root relief

15.7.3.5 Tip relief, linear with profile crowning

Figure (see Figure 15.40) shows the linear tip relief with profile crowning as defined in the profile diagram. This is a combination of a linear tip relief and a connected profile crowning. The **Value**, **Factor 1** and **Factor 2** settings are also shown in this figure. The start of modification at tip and at root can be set in the **Manufacturing** tab.



d_{Sa}	modification, start, at tip	d_{Sf}	modification, end, at root
$C_{\alpha a}$	tip relief, value, normal to involute	L_{Ca}	tip relief, roll length
$C_{\alpha f}$	profile crowning, value	L	roll length

	Value	Factor 1	Factor 2
Inputs	$C_{\alpha f}$	L_{Ca}/m_n	$1000 \cdot C_{\alpha a}/m_n$
Conditions	$\neq 0$	> 0	> 0

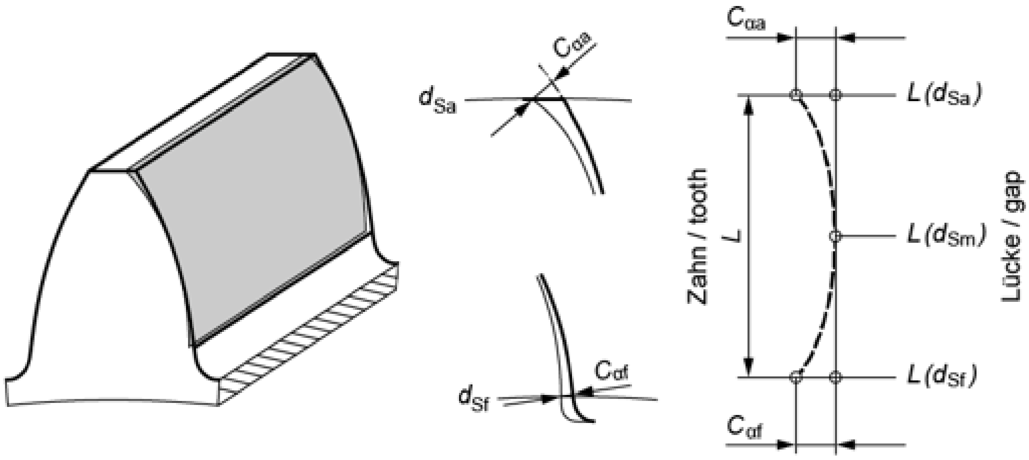
Figure 15.40: Linear tip relief with profile crowning

This modification is usually applied to attempt to merge the linear tip relief without bending tangentially into the profile crowning. A value for Factor2_opt=... is output in the Info field for this purpose. If you input this value in the **Factor 2** field, you will achieve exactly this.

15.7.3.6 Profile crowning, roll length-centered

Figure (see Figure 15.41) shows the (roll length-centered) profile crowning as defined in the profile diagram. The Value setting is also shown in this figure. The start of the modification at the tip and root can be set in the **Manufacturing** tab.

Roll length-centered profile crowning occurs where material is removed constantly, and increasingly, in the direction of the tip and root circle, in the transverse section, starting from the middle of the calculated tooth flank length. Points A and E and the value C_a define the arc-like progression. C_a = $C_{\alpha a}$ = $C_{\alpha f}$ applies for profile crowning. Eccentric profile crowning (see chapter 15.7.3.8) can be used for different crowning at the tip and root.
The Roll length-centered crowning definition is the same as the definition in ISO 21771 and produces an arc-like modification in the profile diagram.



d_{Sa}	profile crowning, start, at tip	d_{Sf}	profile crowning, end, at root
$C_{\alpha a}$	profile crowning, value, at tip, normal to involute	$C_{\alpha f}$	profile crowning, value, at root, normal to involute
L	roll length		

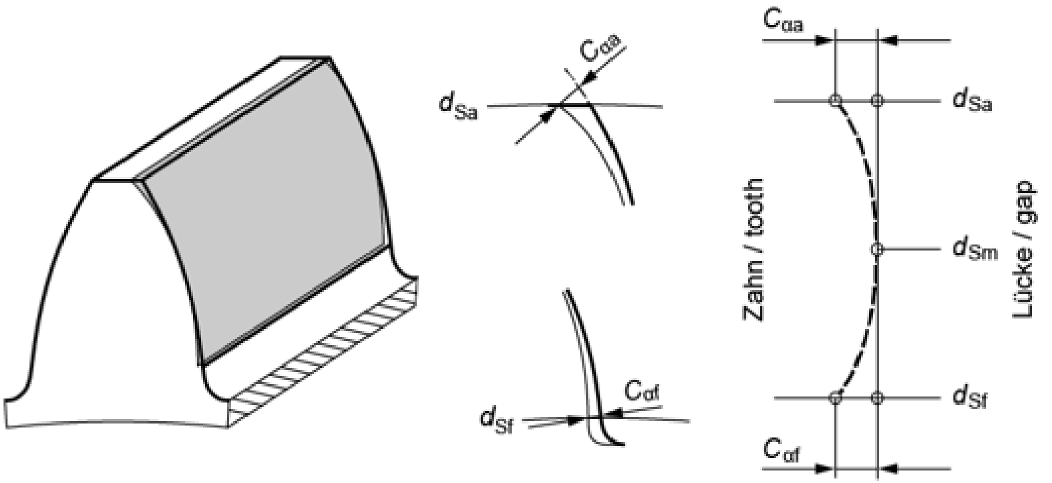
	Value	Factor 1	Factor 2
Inputs	$C_{\alpha a} = C_{\alpha f}$	-	-
Conditions	$\neq 0$	-	-
Equation	$L(d_{Sm}) = 0.5 \cdot \left(L(d_{Sa}) + L\left(d_{Sf}\right)\right)$		

Figure 15.41: Roll length-centered profile crowning

15.7.3.7 Profile crowning, diameter-centered

Figure (see Figure 15.42) shows the (diameter-centered) profile crowning as defined in the profile diagram. The Value setting is also shown in this figure. The start of the modification at the tip and root can be set in the **Manufacturing** tab.

Diameter-centered profile crowning occurs where material is removed constantly, and increasingly, in the direction of the tip and root circle, in the transverse section, starting from the middle of the calculated tooth height. The arc-like curve is set using points A and E and the value C_a. $C_a = C_{\alpha a} = C_{\alpha f}$ applies for profile crowning. Diameter-centered profile crowning results in an arc of circle in the direction of the tooth height.



d_{Sa}	profile crowning, start, at tip	d_{Sf}	profile crowning, end, at root
$C_{\alpha a}$	profile crowning, value, at tip, normal to involute	$C_{\alpha f}$	profile crowning, value, at root, normal to involute

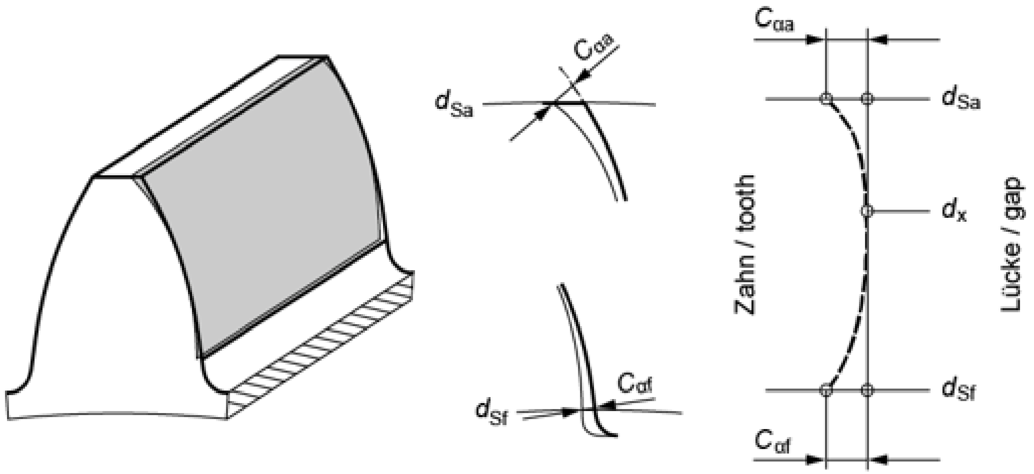
	Value	Factor 1	Factor 2
Inputs	$C_{\alpha a} = C_{\alpha f}$	-	-
Conditions	$\neq 0$	-	-
Equation	$d_{Sm} = 0.5 \cdot (d_{Sa} + d_{Sf})$		

Figure 15.42: Diameter-centered profile crowning

15.7.3.8 Profile crowning, eccentric

Figure (see Figure 15.43) shows the eccentric profile crowning as defined in the profile diagram. The **Value**, **Factor 1** and **Factor 2** settings are also shown in this figure. The start of the modification at the tip and root can be set in the **Manufacturing** tab.

Eccentric profile crowning is similar to diameter-centered profile crowning. **Factor 1** is used to adjust the peak of the crowning and **Factor 2** is used to adjust the value of the crowning at the root. Eccentric profile crowning results in the generation of 2 arcs of circle, one for the tip and one for the tooth root, in the direction of the tooth height.



d_{Sa}	profile crowning, start, at tip	d_{Sf}	profile crowning, end, at root
$C_{\alpha a}$	profile crowning, value, at tip, normal to involute	$C_{\alpha f}$	profile crowning, value, at root, normal to involute
L_x	tip to profile crowning vertex, roll length	L	roll length

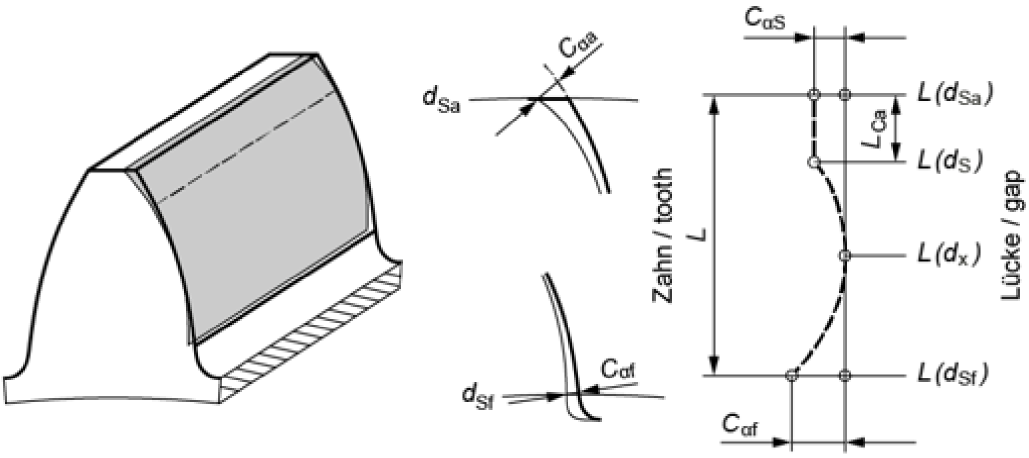
	Value	Factor 1	Factor 2
Inputs	$C_{\alpha a}$	$(d_{Sa}-d_x)/(d_{Sa}-d_{Sf})$	$C_{\alpha f} / C_{\alpha a}$
Conditions	$\neq 0$	< 1	≥ 0

Figure 15.43: Eccentric profile crowning

15.7.3.9 Profile crowning, shortened

Figure (see Figure 15.44) shows the shortened profile crowning as specified in the profile diagram. The **Value**, **Factor 1** and **Factor 2** settings are also shown in this figure. The start of modification at tip and at root can be set in the **Manufacturing** tab.

Shortened profile crowning is used in combination with a tip relief. The tip relief should start at ds.



d_{Sa}	modification, start, at tip	d_{Sf}	modification, start, at root
$C_{\alpha f}$	profile crowning, value, at root, normal to involute	L_{Ca}	tip to profile crowning start, roll length
L_x	tip to profile crowning vertex, roll length	L	roll length

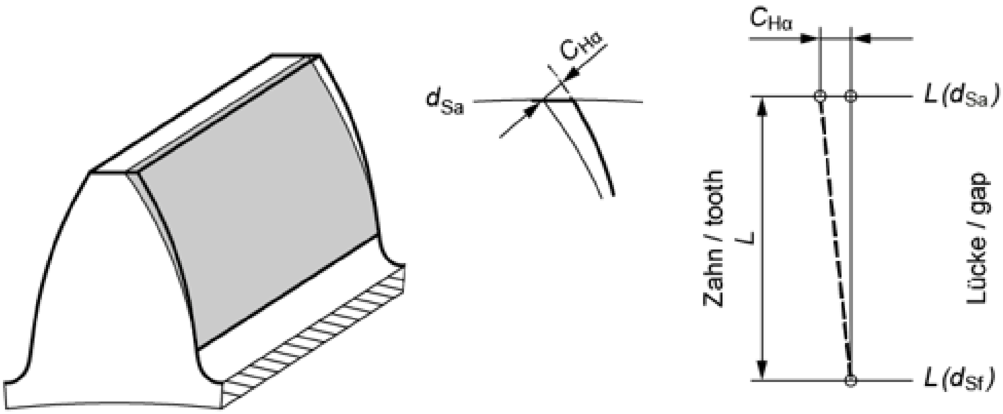
	Value	Factor 1	Factor 2
Inputs	$C_{\alpha f}$	$(d_{Sa}-d_x)/(d_{Sa}-d_{Sf})$	$(d_{Sa}-d_S)/(d_{Sa}-d_{Sf})$
Conditions	≠ 0	0 < *Factor 1* < 1	0 < *Factor 2* < 1

Figure 15.44: Shortened profile crowning

15.7.3.10 Pressure angle modification (value)

Figure (see Figure 15.45) shows the pressure angle modification (value) as defined in the profile diagram. The Value setting is also shown in this figure. The start of the modification at the tip and root can be set in the **Manufacturing** tab.

The pressure angle modification (value) is set as a linear tip relief, in a similar way (see chapter 15.7.3.1, Tip and root relief, linear). The difference is, however, that the pressure angle modification is applied across the entire roll length.



d_{Sa}	transverse profile slope modification, start, at tip	d_{Sf}	transverse profile slope modification, end, at root
$C_{H\alpha}$	transverse profile slope modification, value, normal to involute	L	roll length
$\alpha_{n\ eff}$	effective pressure angle		

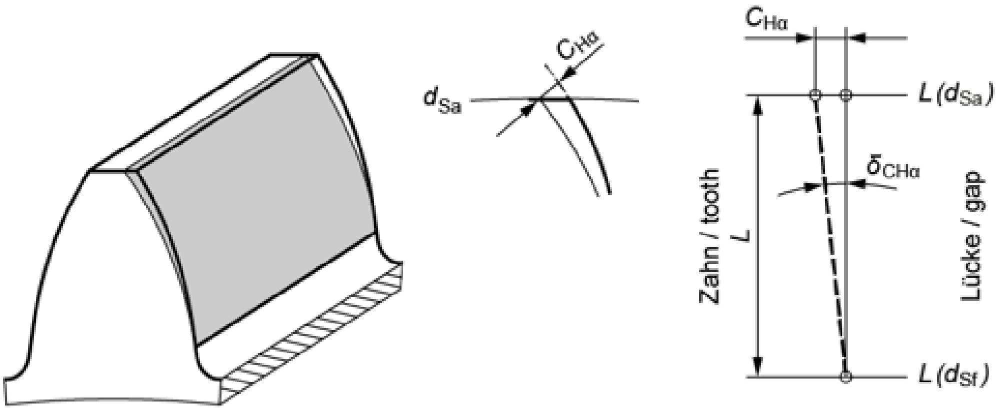
	Value	Factor 1	Factor 2
Inputs	$C_{H\alpha}$	-	-
Conditions	$\neq 0$	-	-
Equation	$\alpha_{n\ eff} = atan\left(cos(\beta) \cdot tan\left(\alpha_t + 0.001 \cdot C_{H\alpha}/Abs\left(L(d_{Sa}) - L(d_{Sf})\right)/\tan\alpha_t\right)\right)$		

Figure 15.45: Pressure angle modification (value)

15.7.3.11 Pressure angle modification (arc minutes)

Figure (see Figure 15.46) shows the pressure angle modification (angle) as defined in the profile diagram. The **Factor 1** setting is also shown in this figure. The start of the modification at the tip and root can be set in the **Manufacturing** tab.

The pressure angle modification (angle-minutes) is set as a linear tip relief, in a similar way (see chapter 15.7.3.1, Tip and root relief, linear). The difference is, however, that the pressure angle modification is applied across the entire roll length. Enter the pressure angle modification value in minutes of an angle in Coefficient 1.



d_{Sa}	transverse profile slope modification, start, at tip	d_{Sf}	transverse profile slope modification, end, at root
$C_{H\alpha}$	transverse profile slope modification, value, normal to involute	$\delta_{CH\alpha}$	transverse profile slope modification, value, angle
L	roll length	$\alpha_{n\,eff}$	effective pressure angle

	Value	Factor 1	Factor 2
Inputs	-	$\delta_{CH\alpha}$ (in arc minutes)	-
Conditions	-	$\neq 0$	-
Equation	$C_{H\alpha} = 1000 \cdot tan\,\alpha_t \cdot (Factor\ 1/60/\cos\beta) \cdot Abs\left(L(d_{Sa}) - L(d_{Sf})\right)$ $\alpha_{n\,eff} = atan\left(cos(\beta) \cdot tan\left(\alpha_t + 0.001 \cdot C_{H\alpha}/Abs\left(L(d_{Sa}) - L(d_{Sf})\right) / tan\,\alpha_t\right)\right)$		

Figure 15.46: Pressure angle modification (arc minutes)

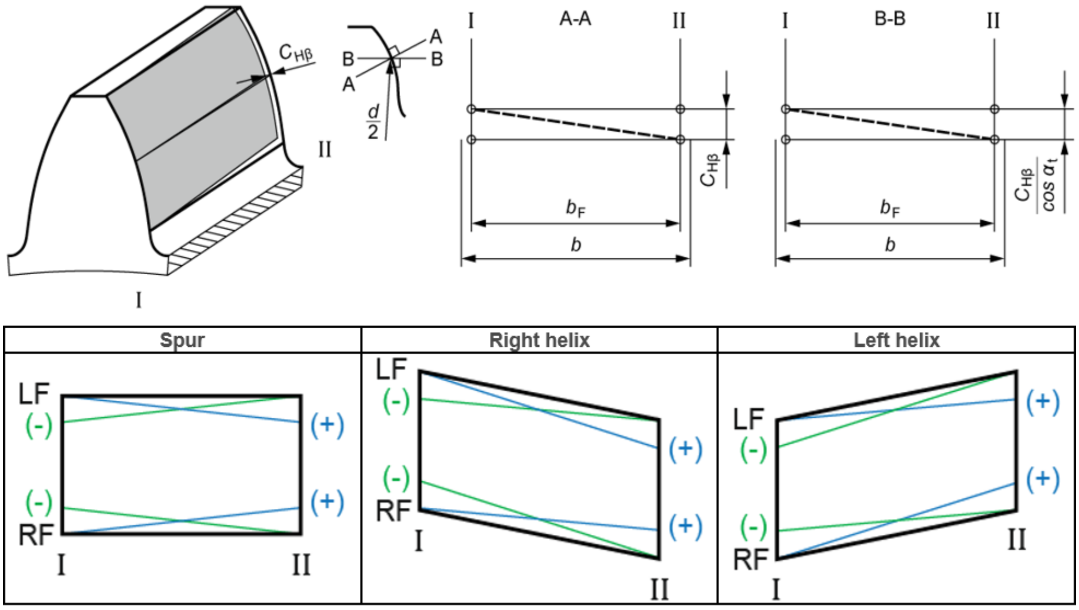
15.7.4 Flank line modifications

Flank line modifications are deviations across the facewidth. The following sections explain the flank line modifications that can be done using KISSsoft.

15.7.4.1 Helix angle modification, conical

Figure (see Figure 15.47) shows the conical helix angle modification as defined in the flank line diagram. The **Value** setting is also shown in this figure. The modification is applied over the effective facewidth of the gear.

The helix angle modification is set as a linear end relief, in a similar way (see chapter 15.7.4.7, End relief, linear, sides I and II). The difference is, however, that the helix angle modification is applied across the entire effective facewidth of the gear.



b	facewidth	b_F	effective facewidth
$C_{H\beta}$	helix angle modification, value	β_{eff}	effective helix angle

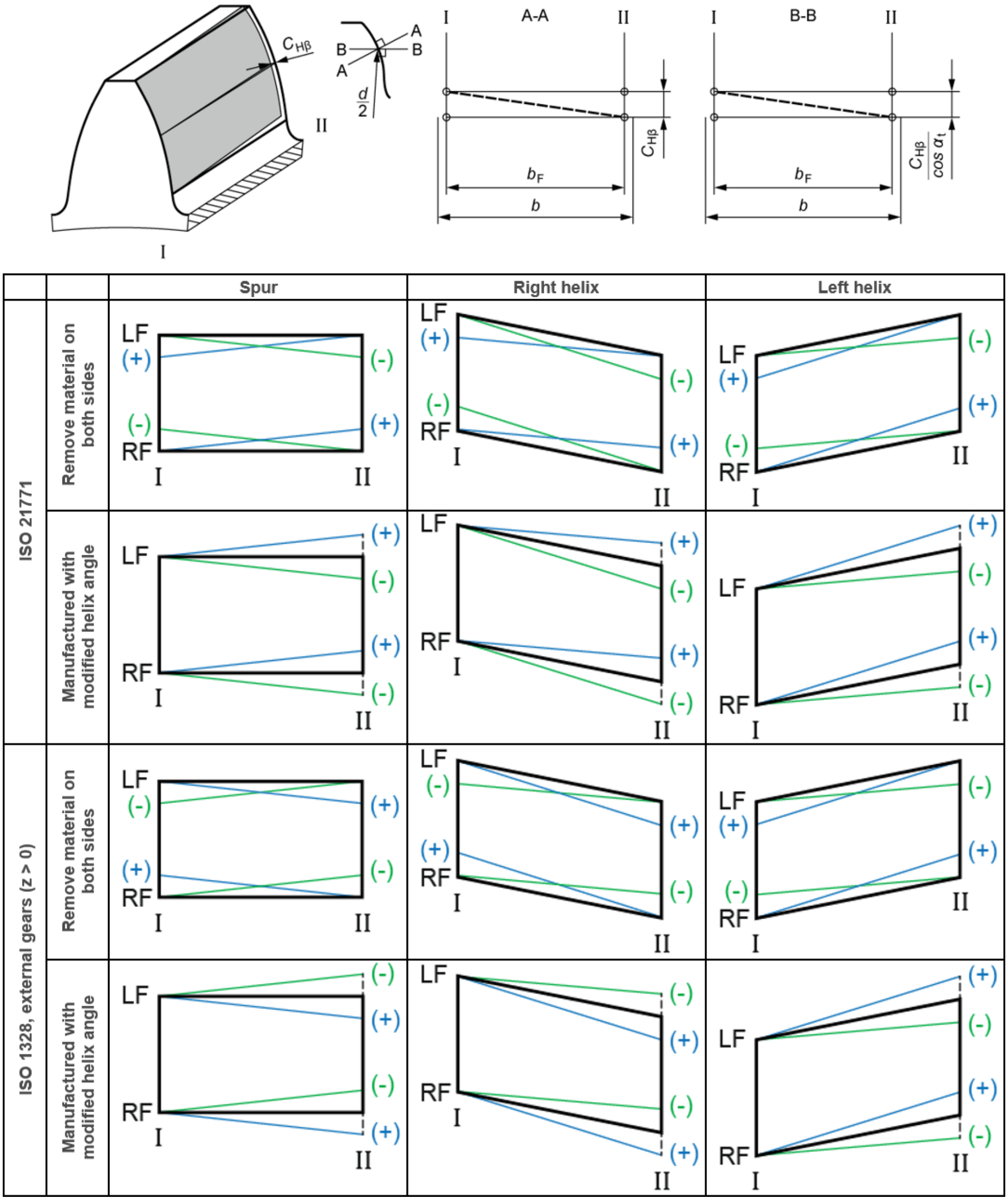
	Value	Factor 1	Factor 2
Inputs	$C_{H\beta}$	-	-
Conditions	≠ 0	-	-
Equation	$C_{H\beta} = 1000 \cdot cos\,\alpha_n \cdot b_F \cdot (tan(\beta - Factor\ 1/60) - tan(\beta))$ $\beta_{eff} = atan\left((b_F \cdot tan\,\beta \pm Abs(0.001 \cdot C_{H\beta}/\,cos\,\alpha_t))/b_F\right)$		

Figure 15.47: Conical helix angle modification

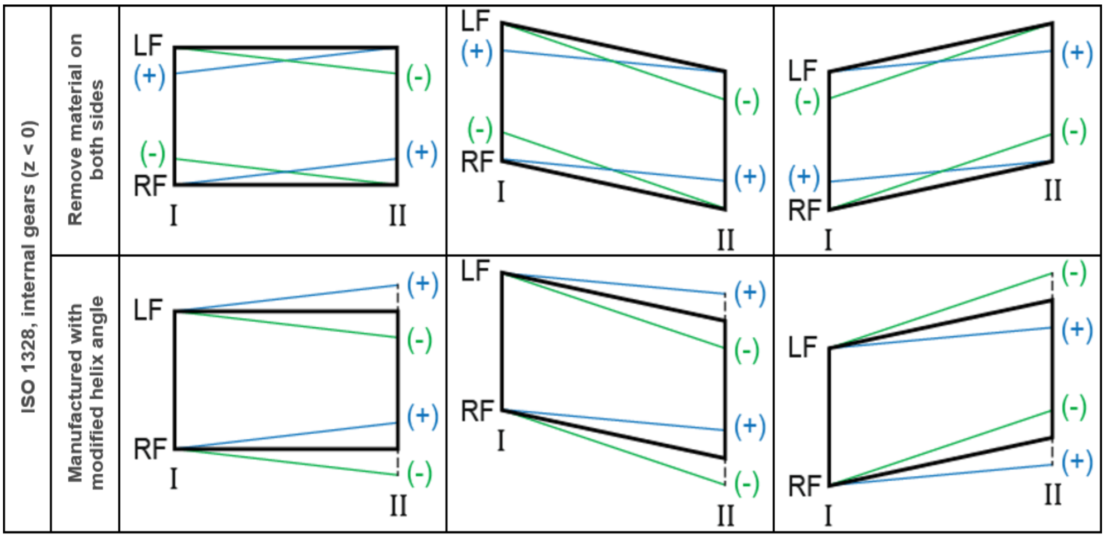
15.7.4.2 Helix angle modification, parallel (value)

These figures (see Figure 15.48) show the parallel helix angle modification (value) as defined in the flank line diagram. The **Value** setting is also shown. The modification is applied over the effective facewidth of the gear.

The way the helix angle modification is specified differs in ISO 1328 and ISO 21771 and for internal and external gears. Other KISSsoft settings (**Remove material on both sides** or **Manufacture with modified helix angle**) also have an effect on the modification (see Figure 15.48).



$C_{H\beta}$
A
B
$\frac{d}{2}$
I
II
A-A
B-B
b_F
b
$\frac{C_{H\beta}}{\cos \alpha_t}$
Spur
Right helix
Left helix
ISO 21771
Remove material on both sides
Manufactured with modified helix angle
ISO 1328, external gears (z > 0)
LF
RF
(+)
(-)



b	facewidth	b_F	effective facewidth
$C_{H\beta}$	helix angle modification, value	β_{eff}	effective helix angle

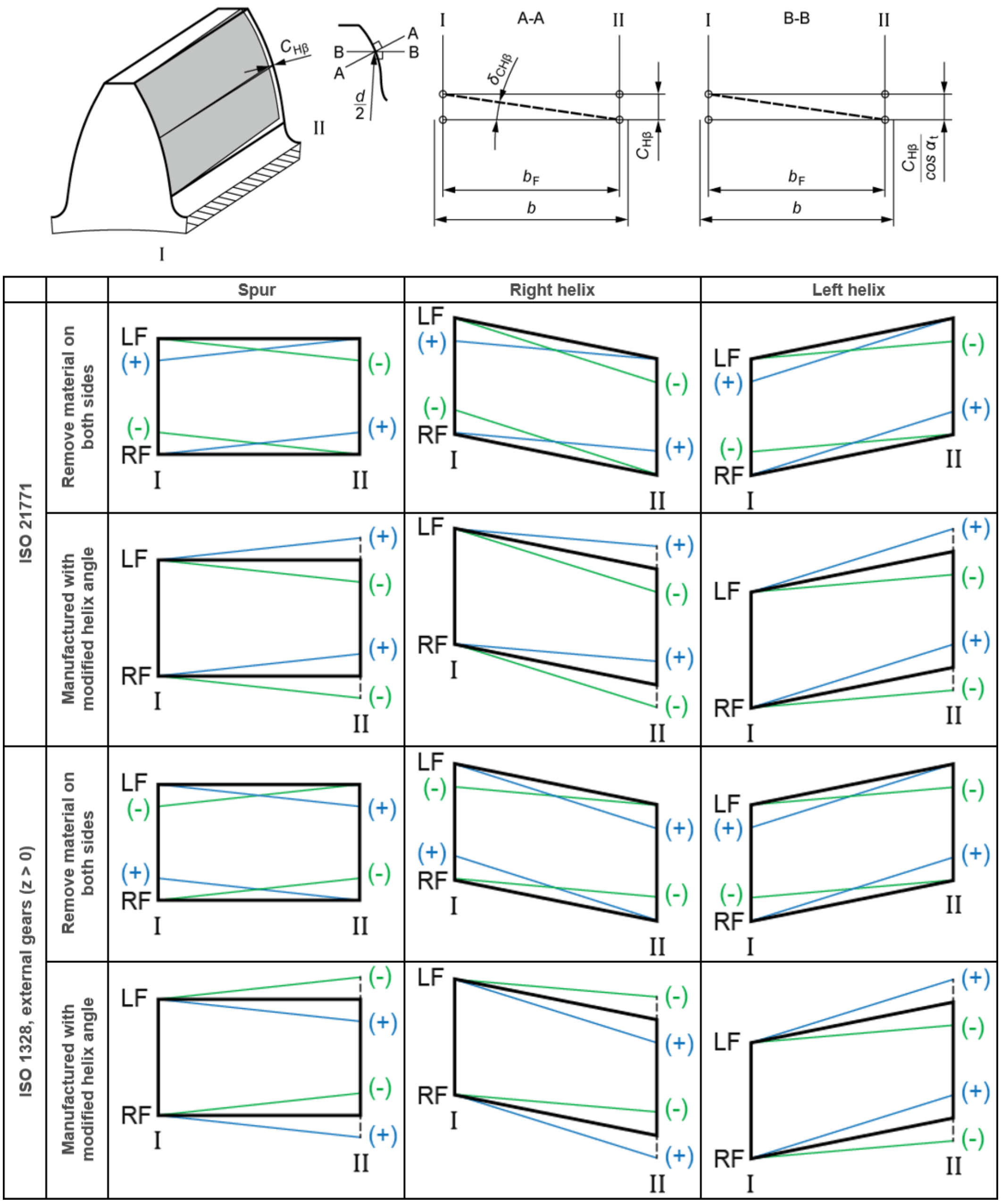
	Value	Factor 1	Factor 2
Inputs	$C_{H\beta}$	-	-
Conditions	≠ 0	-	-
Equation	$C_{H\beta} = 1000 \cdot \cos\alpha_n \cdot b_F \cdot (\tan(\beta - Factor\ 1/60) - \tan(\beta))$ $\beta_{eff} = atan\left((b_F \cdot \tan\beta \pm Abs(0.001 \cdot C_{H\beta}/\cos\alpha_t))/b_F\right)$ More information in Instruction 117: Definition of helix angle modification		

Figure 15.48: Parallel helix angle modification (value), part 1 and 2

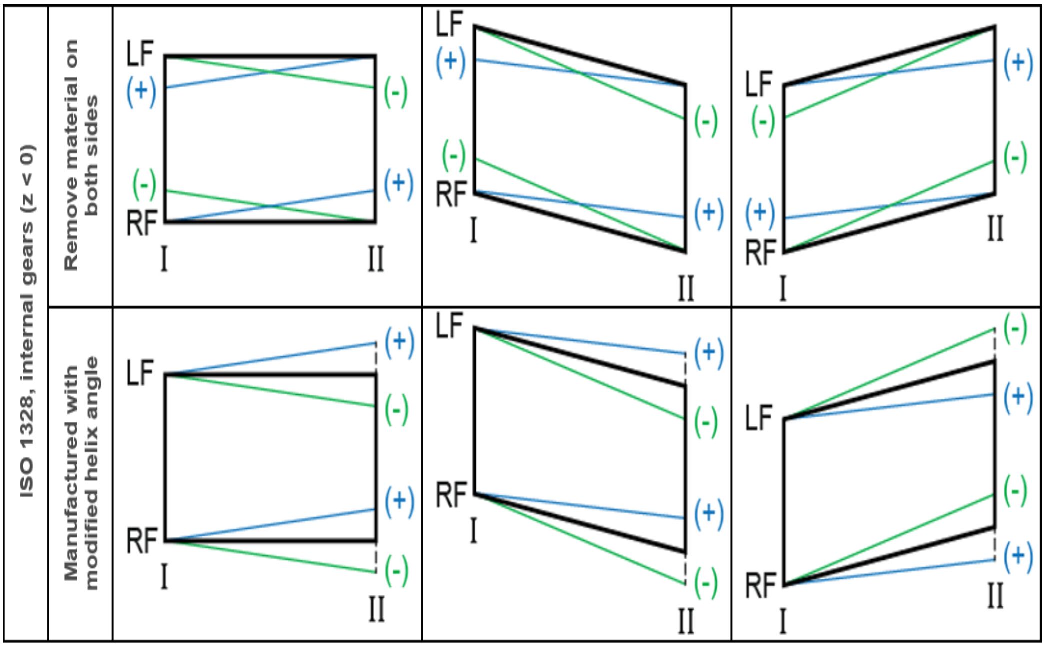
15.7.4.3 Helix angle modification, parallel (arc minutes)

These figures (see Figure 15.49) show the parallel helix angle modification (arc) as defined in the flank line diagram. The **Factor 1** setting is also shown. The modification is applied over the effective facewidth of the gear.

The way the helix angle modification is specified differs in ISO 1328 and ISO 21771 and for internal and external gears. Other KISSsoft settings (**Remove material on both sides** or **Manufacturing with modified helix angle**) also have an effect on the modification (see Figure 15.49).



$C_{H\beta}$
B
A
A
B
$\frac{d}{2}$
I
II
A-A
B-B
$\delta_{CH\beta}$
$C_{H\beta}$
$\frac{C_{H\beta}}{\cos \alpha_t}$
b_F
b
Spur
Right helix
Left helix
ISO 21771
ISO 1328, external gears (z > 0)
Remove material on both sides
Manufactured with modified helix angle
LF
RF
(+)
(-)



b	facewidth	b_F	effective facewidth
$C_{H\beta}$	helix angle modification, value	$\delta_{CH\beta}$	helix angle modification, value, angle
β_{eff}	effective helix angle		

	Value	Factor 1	Factor 2
Inputs	-	$\delta_{CH\beta}$ (in arc minutes)	-
Conditions	-	$\neq 0$	-
Equation	$\beta_{eff} = atan\left(\left(b_F \cdot tan\,\beta \pm Abs(0.001 \cdot C_{H\beta}/\cos\alpha_t)\right)/b_F\right)$ More information in Instruction 117: Definition of helix angle modification		

Figure 15.49: Parallel helix angle modification (arc minutes), part 1 and 2

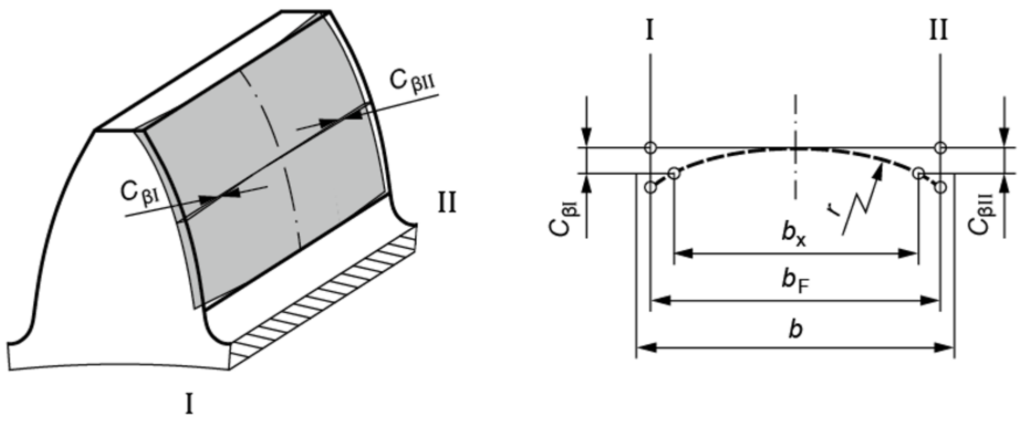
15.7.4.4 Flank line crowning

Figure (see Figure 15.50) shows the flank line crowning as defined in the flank line diagram. The **Value** and **Factor 1** settings are also shown in this figure. The modification is applied over the effective facewidth of the gear.

Flank line crowning occurs where material is removed constantly, and symmetrically, in the direction of the facewidth, starting from a common point at which the tooth profile remains unchanged. The material is removed in an arc-like manner, with its maximum located at point $b_F/2$. $C_\beta = C_{\beta I} = C_{\beta II}$ applies.

► **Note**

Flank line crowning, with its maximum to the right of point b_F /2, is often used in practice 15.7.4.5.



b	facewidth	b_F	usable facewidth (b-$2 \cdot b_k$)
b_X	reference width	$C_{\beta I}$	flank line crowning, value, at side I
$C_{\beta II}$	flank line crowning, value, at side II	r	flank line crowning, radius

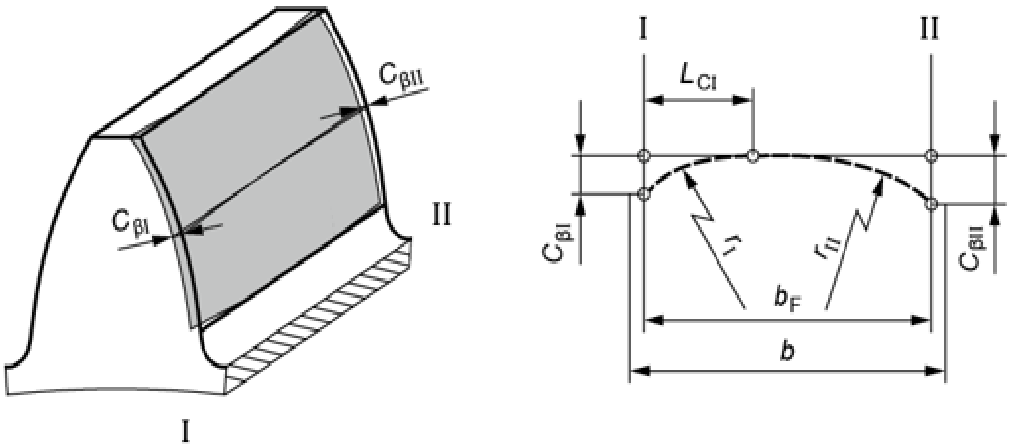
	Value	Factor 1	Factor 2
Inputs	$C_{\beta I} = C_{\beta II}$	b_X/b_F	-
Conditions	≠ 0	0 < *Factor 1* ≤ 1	-

Figure 15.50: Flank line crowning

15.7.4.5 Flank line crowning, eccentric

Figure (see Figure 15.51) shows the eccentric flank line crowning as defined in the flank line diagram. The **Value**, **Factor 1** and **Factor 2** settings are also shown in this figure. The modification is applied over the effective facewidth of the gear.

Eccentric flank line crowning occurs where material is removed in an arc-like manner, in the direction of the facewidth, starting from a common point at which the tooth profile remains unchanged. Factor 1 can be used to set the crowning center point. The crowning value on side I and side II can differ from the crowning value for eccentric flank line crowning.



b	facewidth	b_F	usable facewidth (b-2·b_K)
$C_{\beta I}$	flank line crowning, value, at side I	$C_{\beta II}$	flank line crowning, value, at side II
r_I	flank line crowning, radius, at side I	r_{II}	flank line crowning, radius, at side II
L_{CI}	side I to crowning vertex, length		

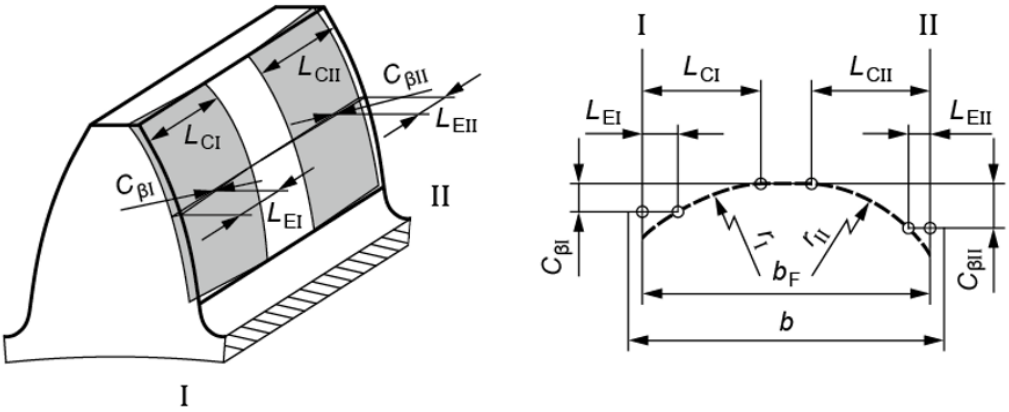
	Value	Factor 1	Factor 2
Inputs	$C_{\beta I}$	L_{CI}/b_F	$C_{\beta II}/C_{\beta I}$
Conditions	≠ 0	0 < *Factor 1* < 1	> 0

Figure 15.51: Eccentric flank line crowning

15.7.4.6 Flank line crowning, sides I and II

Figure (see Figure 15.52) shows the flank line crowning, side I and side II, as defined in the flank line diagram. The **Value**, **Factor 1** and **Factor 2** settings are also shown in this figure. The modification is applied over the effective facewidth of the gear.

This modification resembles the modification for flank line crowning (see chapter 15.7.4.4), but it can be set individually for each side. Factors 1 and 2 can be used to set the crowning start point and the position of $C_ß$.



b	facewidth	b_F	effective facewidth
$C_{\beta I}$	flank line crowning, value, at L_{EI}, at side I	$C_{\beta II}$	flank line crowning, value, at L_{EII}, at side II
r_I	flank line crowning, radius, at side I	r_{II}	flank line crowning, radius, at side II
L_{CI}	flank line crowning, length, at side I	L_{CII}	flank line crowning, length, at side II

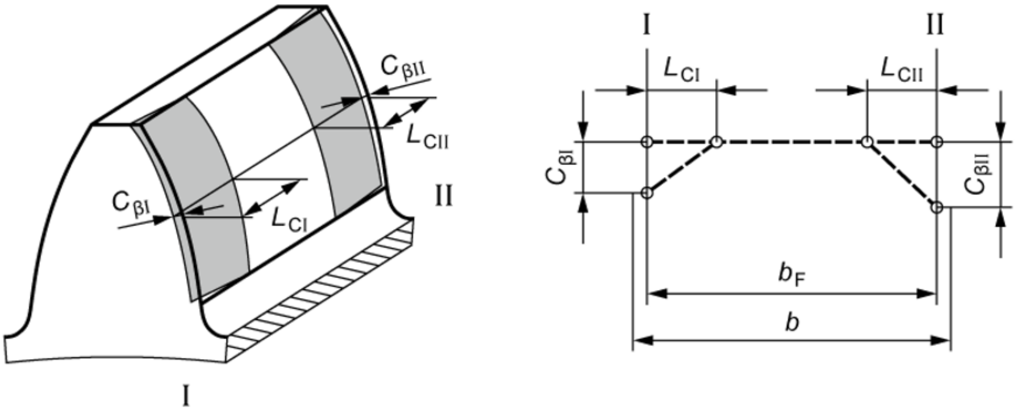
	Value	**Factor 1**	**Factor 2**
Inputs	$C_{\beta I}$, $C_{\beta II}$	L_{CI}/b_F, L_{CII}/b_F	L_{EI}/b_F, L_{EII}/b_F
Conditions	> 0	*Factor 2 < Factor 1* < 1	0 < *Factor 2 < Factor 1*

Figure 15.52: Flank line crowning, sides I and II

15.7.4.7 End relief, linear, sides I and II

Figure (see Figure 15.53) shows the linear end relief, side I and side II, as defined in the flank line diagram. The **Value** and **Factor 1** settings are also shown in this figure. The modification is applied over the effective facewidth of the gear.

Linear end relief occurs where material is removed constantly, and increasingly, from the flank line, starting from particular points, in the direction of the front and rear face surface. In this case, the numbers for I and II relate to both face surfaces (see Figure 15.53).



b	facewidth	b_F	effective facewidth
$C_{\beta I}$	end relief, value, at side I	$C_{\beta II}$	end relief, value, at side II
L_{CI}	end relief, length, at side I	L_{CII}	end relief, length, at side II

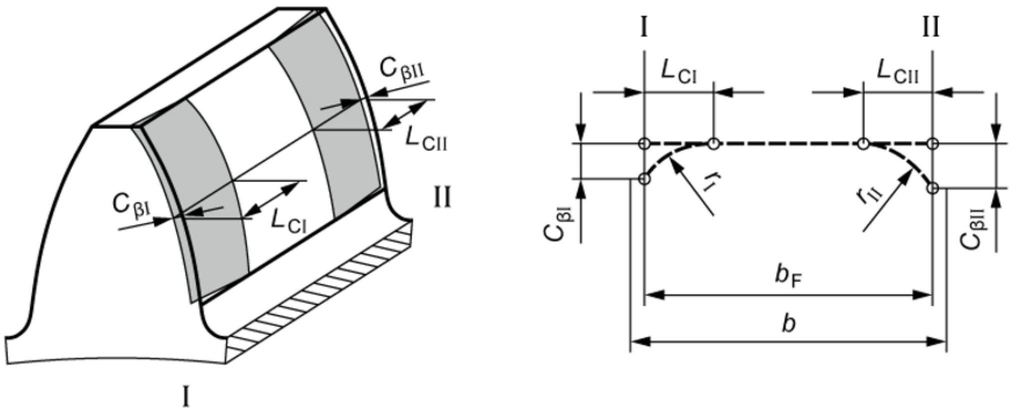
	Value	**Factor 1**	**Factor 2**
Inputs	$C_{\beta I}$, $C_{\beta II}$	L_{CI}/b_F, L_{CII}/b_F	-
Conditions	> 0	0 < *Factor 1* < 1	-

Figure 15.53: Linear end relief, sides I and II

15.7.4.8 End relief, arc-like, sides I and II

Figure (see Figure 15.54) shows the arc-like end relief, side I and side II, as defined in the flank line diagram. The **Value** and **Factor 1** settings are also shown in this figure. The modification is applied over the effective facewidth of the gear.

Arc-like end relief occurs where material is removed constantly, and increasingly, from the flank line, starting from particular points, in the direction of the front and rear face surface. In this case, the numbers for I and II relate to both face surfaces (see Figure 15.54).



b	facewidth	b_F	effective facewidth
$C_{\beta I}$	end relief, value, at side I	$C_{\beta II}$	end relief, value, at side II
r_I	end relief, radius, at side I	r_{II}	end relief, radius, at side II
L_{CI}	end relief, length, at side I	L_{CII}	end relief, length, at side II

	Value	Factor 1	Factor 2
Inputs	$C_{\beta I}$, $C_{\beta II}$	L_{CI}/b_F, L_{CII}/b_F	-
Conditions	> 0	0 < *Factor 1* < 1	-

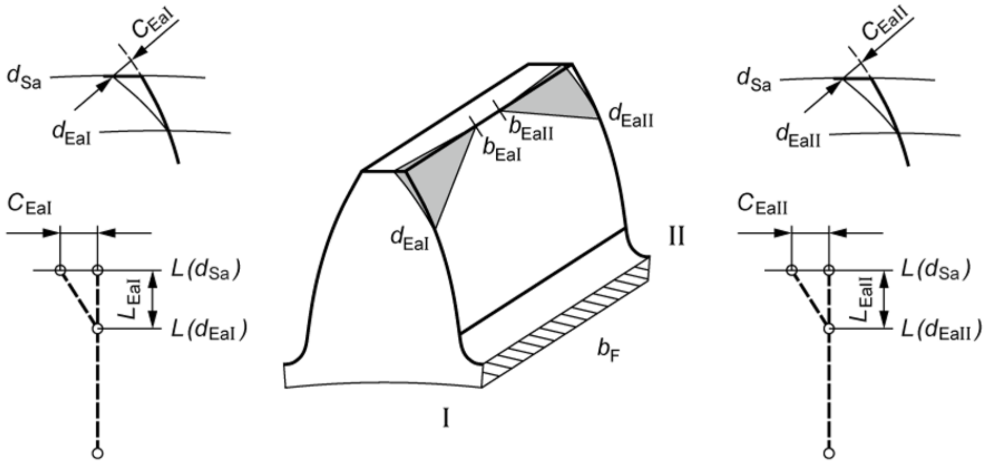
Figure 15.54: Arc-like end relief, sides I and II

15.7.5 Combined profile modifications and flank line modifications

15.7.5.1 Triangular end relief, sides I and II

Figure (see Figure 15.55) shows the triangular end relief, side I and side II, as defined in the profile diagram. The **Value**, **Factor 1** and **Factor 2** settings are also shown in this figure. The modification is applied over the effective facewidth of the gear.

Triangular end relief occurs where material is removed constantly, and increasingly, from the profile line and flank line, starting from particular points, in the direction of the front and rear face surface. In this case, the numbers for I and II relate to both face surfaces (see Figure 15.55).



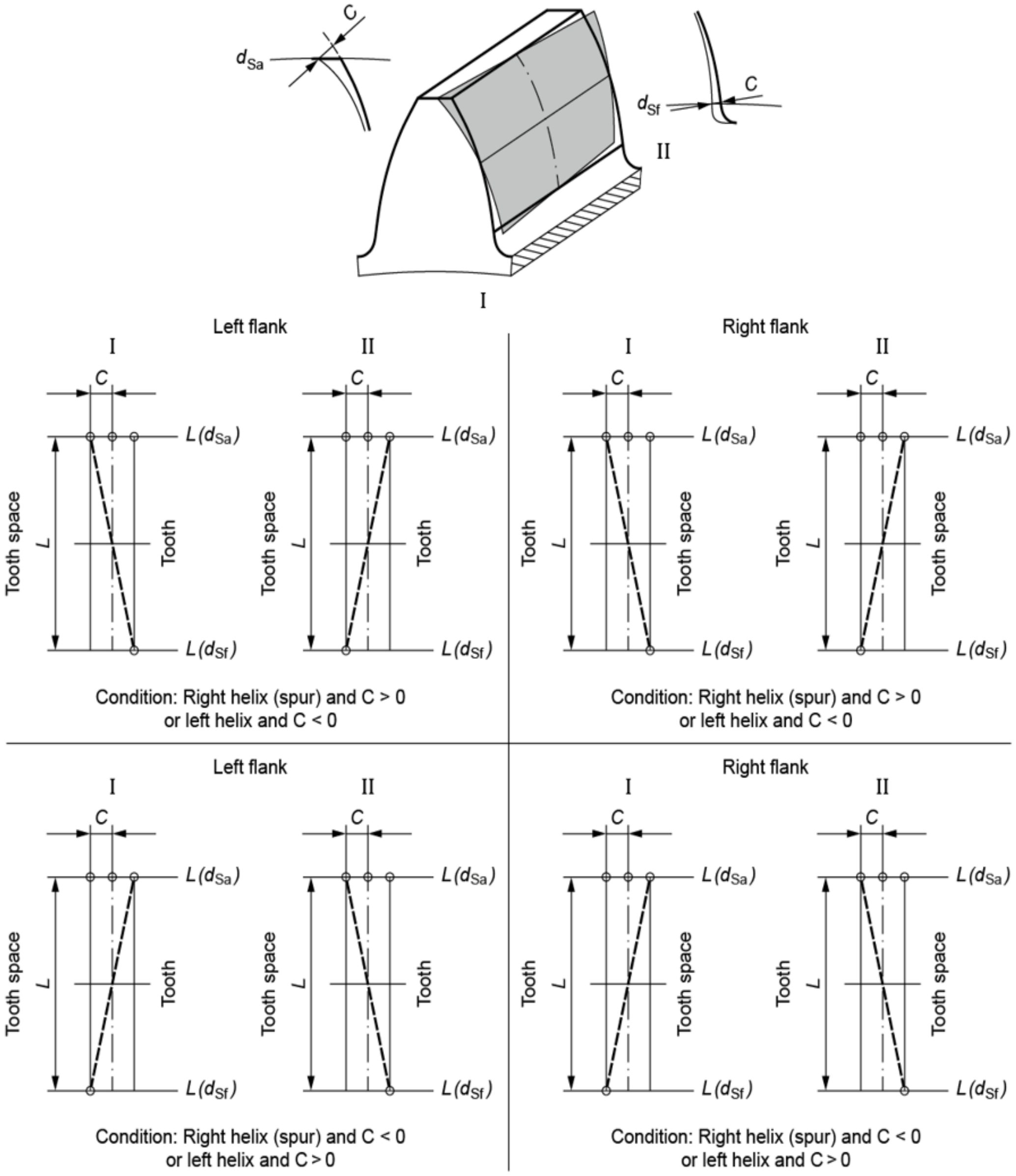
b	facewidth	b_F	effective facewidth
d_{Sa}	triangular relief, start, at side I	d_{Sa}	triangular relief, start, at side II
d_{EaI}	triangular relief, end, at side I	d_{EaII}	triangular relief, end, at side II
C_{EaI}	triangular relief, value, at side I	C_{EaII}	triangular relief, value, at side II
b_{EaI}	triangular relief, length, at side I	b_{EaII}	triangular relief, length, at side II

	Value	**Factor 1**	**Factor 2**
Inputs	C_{EaI}, C_{EaII}	L_{EaI}/m_n, L_{EaII}/m_n	b_{EaI}/b_F, b_{EaII}/b_F
Conditions	> 0	> 0	0 < *Factor 2* < 1

Figure 15.55: Triangular end relief, sides I and II

15.7.5.2 Twist

Twist is the torsion of the transverse section profile along a helix. Usually, the angle increases in a linear progression from the start of the effective flank to its end. The definition in ISO 21771 is incomplete because it only describes twist on the right flank. The definition according to GFT (Getrag-Ford-Transmissions) is more complete and is therefore the standard solution used in industry. Modification C can be either a positive or negative value.



d_{Sa}	twist, start, at tip	d_{Sf}	twist, end, at root
C	twist, value		

	Value	**Factor 1**	**Factor 2**
Inputs	$4{\cdot}C$	-	-
Conditions	$\neq 0$	-	-

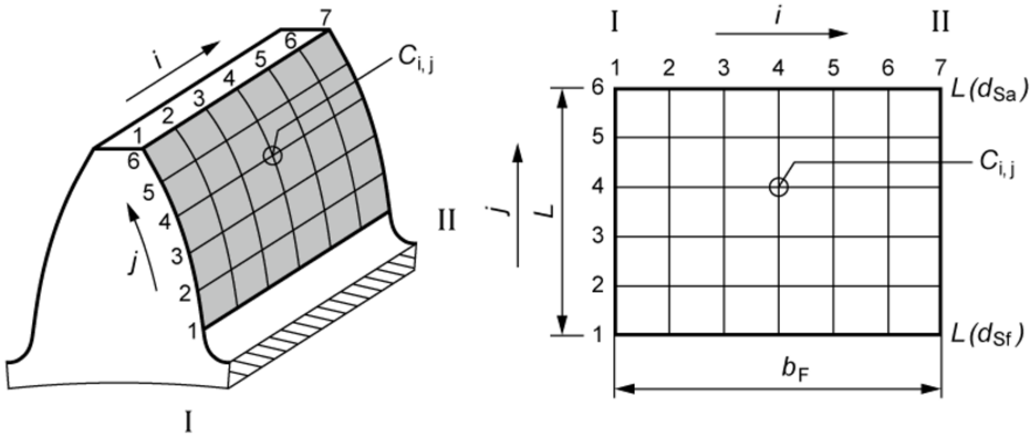
Figure 15.56: Twist

The notation used here is also shown in the helix angle modifications sections (see chapter 15.7.4.1, Helix angle modification, conical) and following sections and pressure angle modifications (see chapter 15.7.3.10, Pressure angle modification (value)) and following section.

15.7.5.3 Topological modification

Use topological modification to define any type of modification. The actual modification is described in the file that is to be imported. You will find an example of this type of entry in the "topological_template.dat" file in the dat directory. The file's name indicates its purpose. You can define coefficients in any section and for any rolling depth. When the file is imported, these coefficients are multiplied by the value input under C_a. To display and check the modification, click on **Graphic -> 3D Geometry -> Modifications**.

You will find an Excel application, "Topological Crowning.xlsx", in the \dat directory. In that file, you can edit the table in which the topological modification is defined and then copy it to a .dat file. This Excel file also has an example of how to define a negative profile crowning.



d_{Sa}	topological modification, start, at tip	d_{Sf}	topological modification, end, at root
b_F	effective facewidth	$C_{i,j}$	topological modification, value, at position *i,j*
L	roll length		

	Value	Factor 1	Factor 2
Inputs	see manual	-	-
Conditions	> 0	-	-

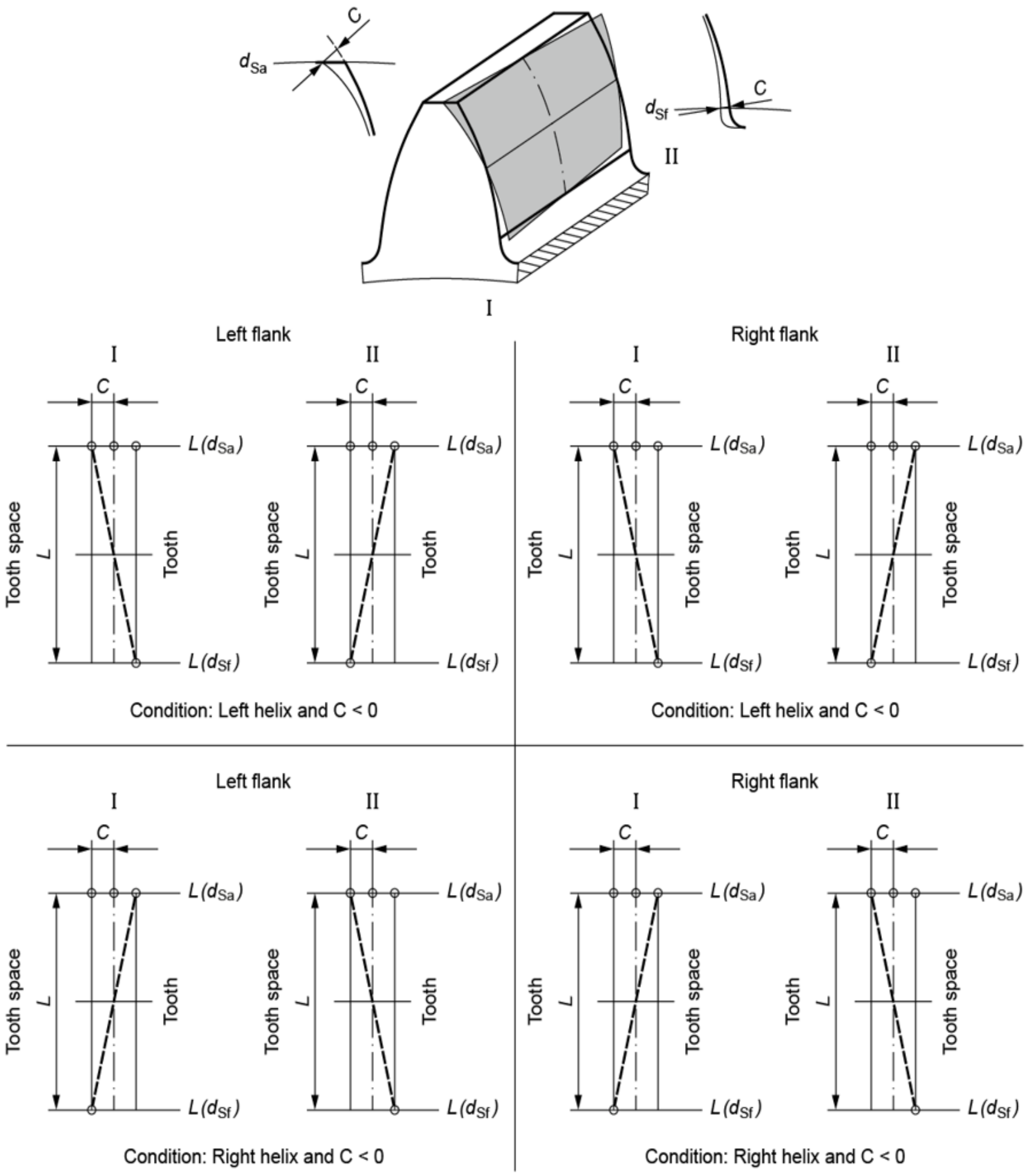
Figure 15.57: Topological modification

15.7.6 Manufacturing deviations

15.7.6.1 Twist due to manufacturing: natural twist due to flank line crowning Cβ (generation grinding)

In the **Modifications** tab, you can select **Twist due to manufacturing** as a modification. This is a natural twist that occurs when flank line crowning is created on helical gears as part of the generation process on standard grinding machines. The resulting twist depends on the value C_β of the flank line crowning, the helix angle and also the involute length. The calculation is performed using data provided by the company Gleason-Pfauter, in Ludwigsburg, Germany. The formula used here corresponds to equation 5.16 in Hellmann's dissertation [29]. Enter the value of the crowning to be ground, C_β, in the **Value** column. The resulting twist is then determined during the calculation process, and is documented under **Information**. The generation grinding process always creates a negative twist.

Twist due to manufacturing can only be calculated if a generation grinding process is used. If a form grinding process is involved, different methods that are suitable for the particular process must be used to determine the resulting twist. Form grinding always generates a positive twist.



d_{Sa}	natural twist, start, at tip	d_{Sf}	natural twist, end, at root
C	natural twist, value, $f(C_\beta)$	C_β	flank line crowning

	Value	Factor 1	Factor 2
Inputs	C_β, see manual	-	-
Conditions	> 0	-	-
Info	Natural twist value C is always negative.		

Figure 15.58: Twist due to manufacturing

15.7.6.2 Measured manufacturing deviation

Use the **Measured manufacturing deviation** to import Gleason GAMA CMM data directly and convert it to the format used for topological modifications in KISSsoft. As a result, the template format for the modification to the measured manufacturing deviation is the same as the template for the topological modifications. It is assumed that the values in the template are given in micrometers. To display the modifications, click on **Graphic -> 3D Geometry -> Modifications**.
By defining the modification, you can then analyze how manufacturing deviation affects gear performance, transmission error, contact stress and other properties. This functionality is essential in the **Design - Manufacturing - Measuring Closed Loop** because it ensures that the manufactured gears have the required properties.

The modification can also be defined manually, but instead, we recommend you do define it in the **Import measured manufacturing deviation** in the **Manufacturing** tab. Click on the Convert button to open this dialog.

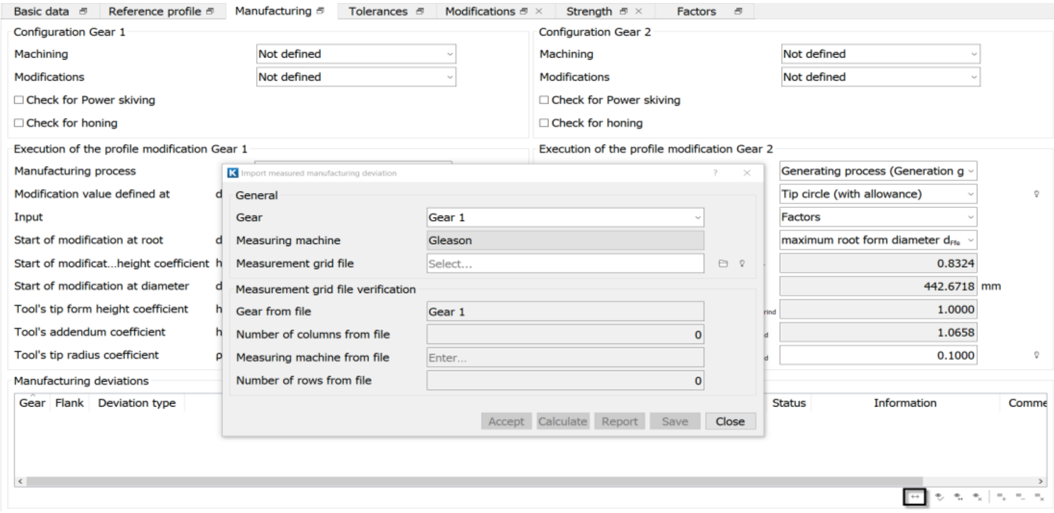


Figure 15.59: Importing the measured manufacturing deviation

The CMM data with the manufacturing deviation is assigned in the window. At present, only the Gleason GAMA data format is supported. The manufacturing deviation data here should be in the same format as displayed in the FN column.

```
*********************************************************************************
*                              MEASUREMENT DATA - LIST                          *
*                              PINION CONVEX                                    *
*-------------------------------------------------------------------------------*
*DRAWING NUMBER :                                                               *
*ANGULAR TOOTH-THICKNESS ERROR % ZDIF !-16.2653 [DEG] % (J,I) ! (5,3)           *
*-------------------------------------------------------------------------------*
*COLUMNS % NSPG ! 9 ;  LINES % NZLG ! 5                                         *
*-------------------------------------------------------------------------------*
*DATE :  15 Jun 2020                                                            *
*********************************************************************************
*   J  I          XP          YP          ZP                             FN     *
*234567890123456789012345678901234567890123456789012345678901234567890123456789*
     1  1      15.2990     -2.7665     -9.3186                           -.00691
     1  2      16.2254     -3.4014     -9.3189                           -.00569
     1  3      17.0949     -4.2238     -9.3188                           -.00589
     1  4      17.8968     -5.2083     -9.3186                           -.00722
     1  5      18.6213     -6.3386     -9.3186                           -.00699
     2  1      15.4391     -1.8344     -6.9890                           -.00496
     2  2      16.4024     -2.4126     -6.9892                           -.00422
     2  3      17.3205     -3.1807     -6.9892                           -.00384
     2  4      18.1808     -4.1149     -6.9890                           -.00501
     2  5      18.9727     -5.1993     -6.9891                           -.00444
```

Figure 15.60: Gleason GAMA CMM data

After you have assigned the file, click on the **Calculate** and **Save** buttons to save the template files to the correct folder. After you have saved the file, click on the **Apply** button. The program then automatically generates the modifications entry, as shown below.

Gear	Flank	Modification type	Value [μm]	Data file	Status
Gear 1	right	Measured manufacturing deviation		C:/Users/data/Template_Gear1_RF.dat	active
Gear 1	left	Measured manufacturing deviation		C:/Users/data/Template_Gear1_LF.dat	active

Figure 15.61: Inputting the measured manufacturing deviation

15.7.6.3 Profile and flank line deviations

The form deviation in the profile or flank line direction can be set using a sinus-shaped waviness. The double amplitude should be smaller than or equal to the corresponding form deviation according to ISO, AGMA or DIN ($f_{f\alpha T}$ or $f_{f\beta T}$). You can also set the waviness length and the start at the tip or side I. The slope deviation should be less than or equal to the corresponding slope deviation according to ISO, AGMA or DIN ($f_{H\alpha T}$ or $f_{H\beta T}$).

You can set a total deviation by specifying a form deviation (double amplitude) and an additional slope deviation.

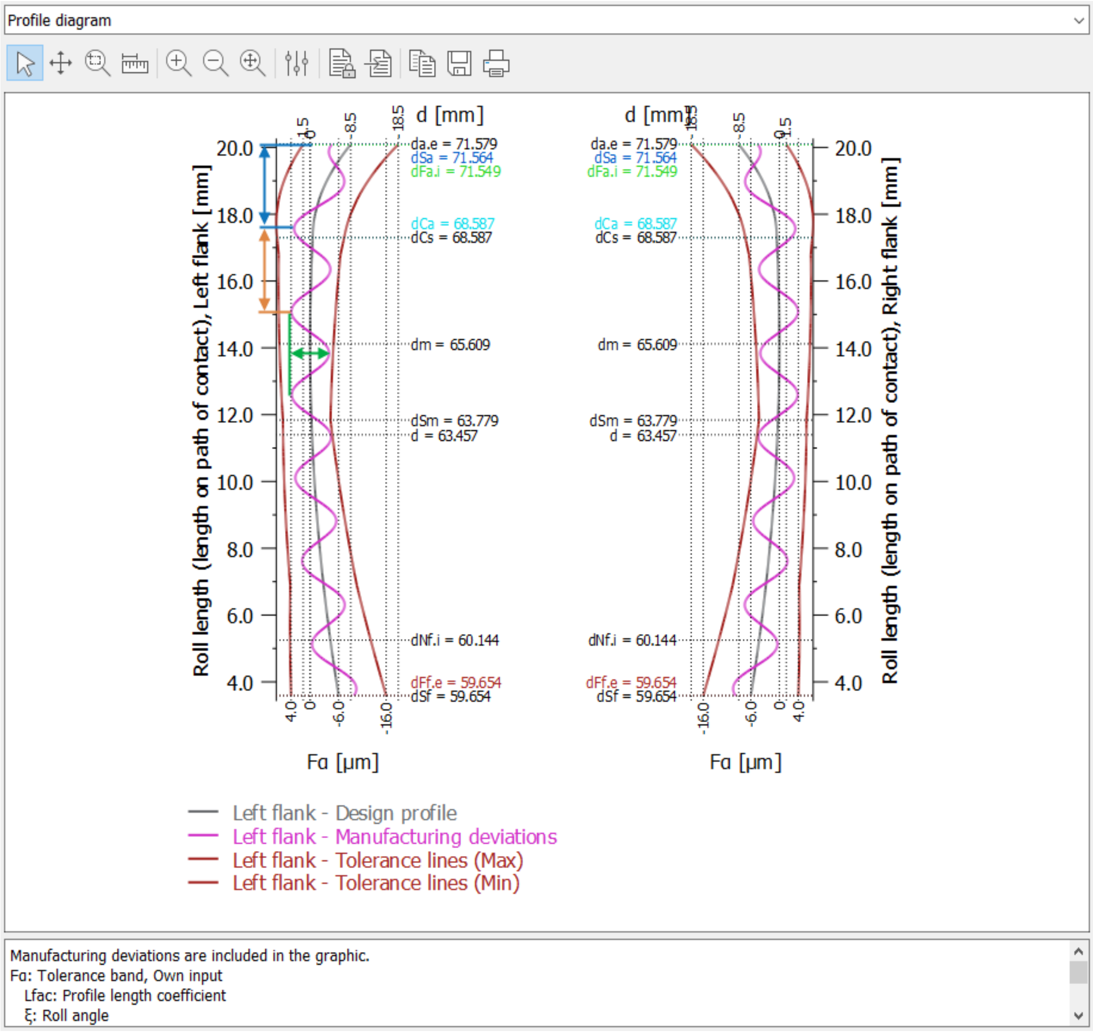
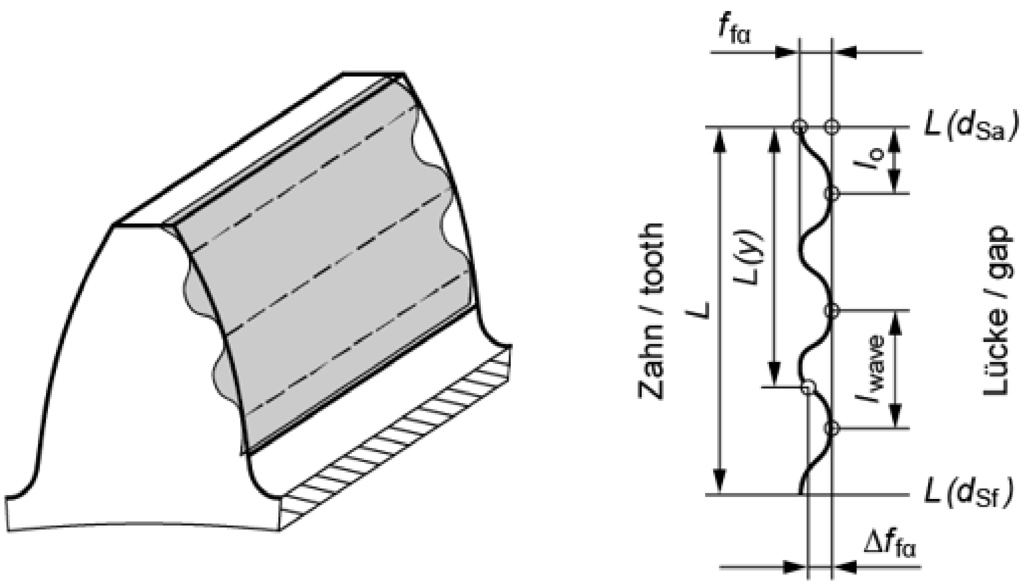


Figure 15.62: Simulation of a profile form deviation shown in the profile diagram

15.7.6.3.1 Profile form deviation

The double amplitude (μm), wave length (in module) and the start, distance from the tip until the wave maximum (in module) are set, as shown in (see Figure 15.62). The wave length is constant in the direction of the roll length, as the waviness typically results from machining (in the direction of the machining path of contact).



d_{Sa}	profile form deviation, start, at tip	d_{Sf}	profile form deviation, end, at root
L	roll length	l_0	phase shift
l_{wave}	wave length	$f_{f\alpha}$	profile form deviation, value

	Value	Factor 1	Factor 2
Inputs	$f_{f\alpha}$	l_{wave}/m_n	l_0/m_n
Conditions	> 0	> 0	-
Equation	$\Delta f_{f\alpha} = \frac{f_{f\alpha}}{2} \cdot \left[1 - \sin\left(90^\circ + 360^\circ \cdot \left(\frac{L(y)}{Factor\ 1 \cdot m_n} - \frac{Factor\ 2}{Factor\ 1}\right)\right)\right]$		

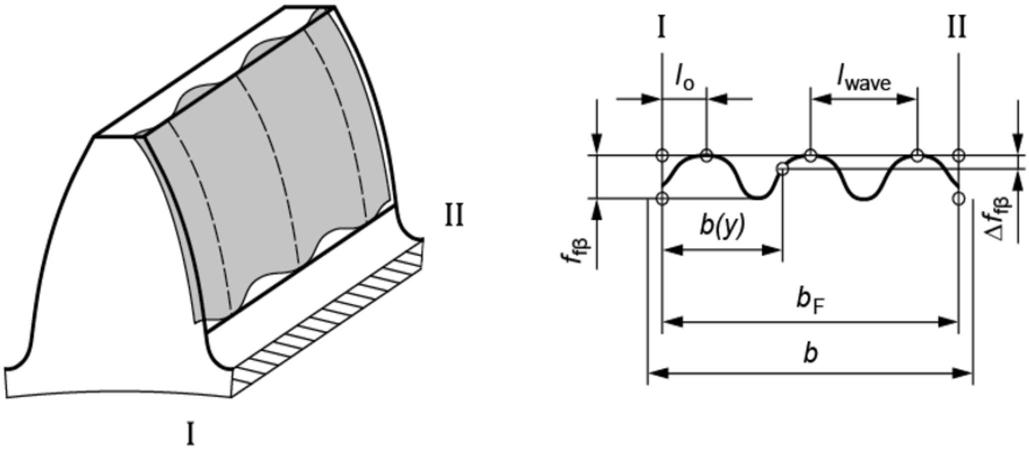
Figure 15.63: Profile form deviation

15.7.6.3.2 Profile slope deviation

Inputs and definition are the same as in the details in (see chapter 15.7.3.10, Pressure angle modification (value)).

15.7.6.3.3 Helix form deviation

The double amplitude (µm), wave length (in module) and the start, distance from side 1 until the wave maximum (in module) are set.



b	facewidth	b_F	usable facewidth (b-2·b_k)
l_0	phase shift	l_{wave}	wave length
$f_{f\beta}$	helix form deviation, value		

	Value	Factor 1	Factor 2
Inputs	$f_{f\beta}$	l_{wave}/m_n	l_0/m_n
Conditions	> 0	> 0	-
Equation	$\Delta f_{f\beta} = \frac{f_{f\beta}}{2} \cdot \left[1 - \sin\left(90^\circ + 360^\circ \cdot \left(\frac{b(y)}{Factor\ 1 \cdot m_n} - \frac{Factor\ 2}{Factor\ 1}\right)\right)\right]$		

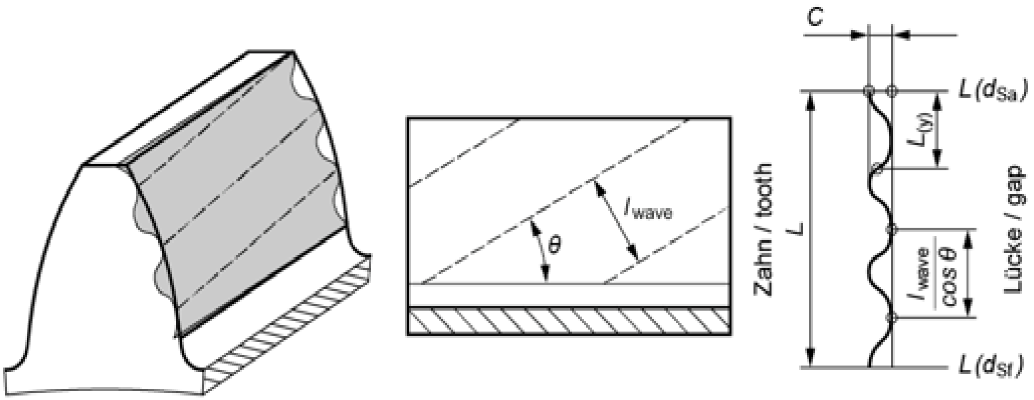
Figure 15.64: Helix form deviation

15.7.6.3.4 Helix slope deviation

Inputs and definition are the same as in the details in (see chapter 15.7.4.2, Helix angle modification, parallel (value)).

15.7.6.3.5 Waviness due to manufacturing

A waviness that is vertical, relative to angle θ (input as **Factor 2**) is simulated. If the angle is set to 0, this means that base helix angle β_b is used.



d_{Sa}	waviness from manufacturing, start, at tip	d_{Sf}	waviness from manufacturing, end, at root
L	roll length	l_{wave}	wave length
C	waviness from manufacturing, value		

	Value	Factor 1	Factor 2
Inputs	C	*Factor 1*	θ (°)
Conditions	> 0	> 0	> 0
Equation	$l_{wave} = Factor\ 1 \cdot m_n / \cos(\theta)$ $\Delta C = \frac{C}{2} \cdot \left[1 - \sin\left(180^\circ + 360^\circ \cdot \left(\frac{L(y)}{l_{wave}}\right)\right)\right]$ If *Factor 2 = 0*, the waviness is applied in the direction of the base helix angle β_b.		

Figure 15.65: Waviness due to manufacturing

15.7.7 Rough sizing, modifications (microgeometry)

Click on the Sizing button, in the **Modifications** tab (below the table), to open the **Rough sizing modifications (microgeometry)** dialog.

This is where you can size a profile modification or a flank line modification. These functions are described in the following sections. Click on the **Apply** button to transfer the sizing results to the **Modifications** tab. When a profile modification is transferred, all profile modifications that are already present are automatically deleted. When a flank line modification is transferred, all flank line modifications that are already present are deleted. A new function in Version 2021 enables you to specify whether or not this data should be deleted.

15.7.7.1 Profile modification

1. Tip relief on the driven gear reduces the entry impact, whereas tip relief on the driving gear reduces the exit impact. Tip reliefs are therefore usually applied to both gears. They are only applied to the driven gear alone in exceptional circumstances.

2. When calculating the profile modification, you must always specify the tip chamfer. If not, the active involute will not be included in the calculation.
3. Tooth contact stiffness is always calculated according to the selected calculation method. Alternatively, the contact stiffness can also be determined from the tooth form (see chapter 15.2.6.9, Tooth contact stiffness).
4. The points along the length of path of contact are labeled according to ISO 21771. In the case of a driving pinion, a tip modification must be applied on the pinion from H -DE to E (or D to E) and on a gear, from A to H -AB (or from A to AB). For a driven pinion, the labels are swapped according to ISO 21771 (A becomes E, E becomes A).
5. KISSsoft calculates the tip relief value for a nominal torque that has been changed by the modification value. In the case of gears that do not always have the same operating torque, the modification value is assumed to be approximately 50 to 75% of the maximum moment, evenly distributed across the pinion and the gear. The default value for tip relief C_α is defined using the mean value of the data as defined by Niemann. A (somewhat greater) value (C.I) is set as the meshing start at the tip of the driven gear. The value C.II is set as the value for the meshing end at the tip of the driving gear. When you select the **For smooth meshing** profile modification, the value C.I is also set at the meshing end.
   For deep tooth forms, where $\varepsilon_\alpha > 2$, the load-dependent portion of tip relief is reduced, depending on accuracy grade (manufacturing quality), to 12.5% (for quality level 8 and poorer) and up to 50% (for quality level 5 and better).
6. KISSsoft also calculates the modification length, also known as the "long modification", which extends from point A to point B of the length of path of contact. However, the "short modification" only goes to point H-AB (the midpoint between A and B). Usually, the short modification is selected. However, the modification length (from A to AB) should not be too short. A minimum length (related to the tooth height) of $0.2m_n$ should always be present. This value is checked during sizing. If the length from A to AB is too short, the program prompts you to use a minimum height of $0.2m_n$. However, the result of this is that the contact ratio in the unmodified part will be less than 1.0 (< 2.0 for deep tooth forms where $\varepsilon_\alpha > 2$). The program then displays an message telling you of this.

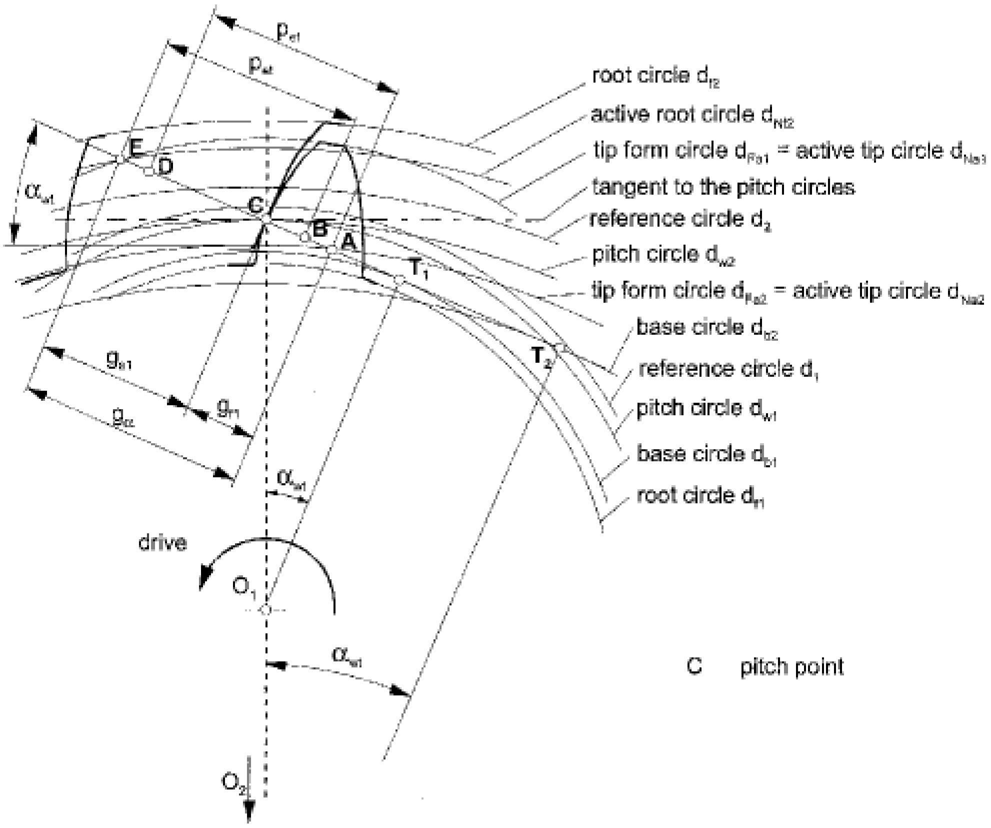


Figure 15.66: Figure 14.34: Length of path of contact for a cylindrical gear

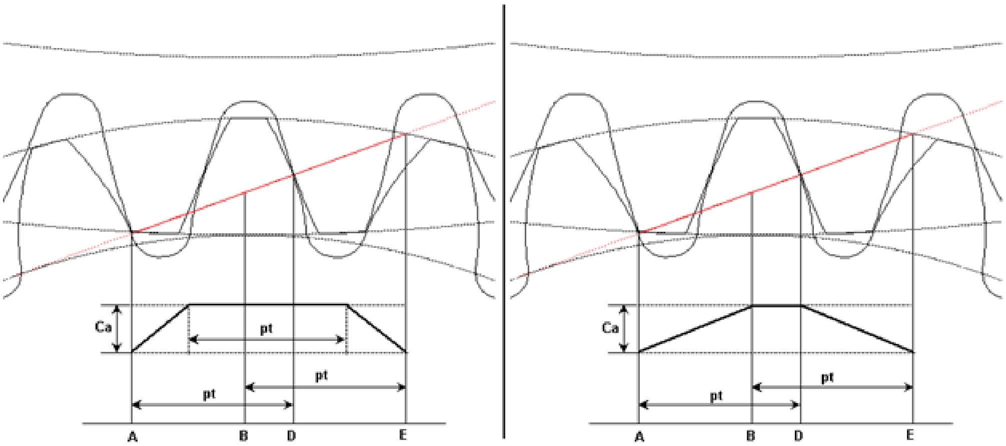


Figure 15.67: Short (left) and long profile modification

7. The type of **profile modification** has an effect on how safety against scuffing is calculated.
   If you select **For high load capacity gears** according to the suggestion given in Niemann, the profile modification at the end of the contact (point E on the path of contact) is somewhat less than the profile modification at the beginning of the contact.

If you select **For smooth meshing**, the profile modification at the end of contact is set to the same values as the profile modification at the beginning of contact.

15.7.7.2 Flank line modification

A flank line modification, such as end relief (see chapter 15.7.4.7, End relief, linear, sides I and II) or crowning (see chapter 15.7.4.4, Flank line crowning), is sized as specified in ISO 6336, Part 1, Annex B.

If you are working with planets systems, the proposed flank line modification can be used to compensate for a misalignment of the planet and the sun. It can also take into account the effect of torsion on a particular gear. You will find more detailed information about the direction of torque and the axis alignment in the "Defining the misalignment of individual parts" section.

However, be aware that this sizing suggestion only applies to planets with a symmetrical misalignment because of the torsion that influences the carrier.

Then, the proposed modifications ($K_{H\beta}$ = 1) are only correct if the system has a single planet. If several planets are present, the program searches for the best compromise so that the proposed modification minimizes the maximum $K_{H\beta}$ for all the planet contacts.

If ISO 6336-1, Annex E, is applied, an additional precise sizing of the flank line modification is performed, as eccentric crowning or centrical crowning with a helix angle modification.

15.7.8 Notes about profile modification

If you select a **short profile modification**, the length of the modification at the tooth tip (or at the tooth root) for both gears is defined such that the contact ratio of the part of the tooth flank that has not been changed by the modification is still exactly 1.0 (for deep toothing with ε_α > 2 is still exactly 2.0). This type of profile modification is the one most frequently used because it always ensures a sufficiently large transverse contact ratio (no matter what load is involved).
This short profile modification runs from point A of the path of contact up to the point H-AB (the midpoint between point A and point B), or from E to H-DE. However, the result of this is that the contact ratio in the unmodified part is 1.0.

If you want to design a gear unit that runs as quietly as possible, it is usually better to select the long profile modification, because the transmission error is usually much lower in this case. To properly evaluate the effect of a profile modification, we recommend you calculate the meshing under load (see chapter 15.11, Contact analysis).

15.7.9 Using diamond dressing wheels and grinding worms

Click on the Convert button in the **Manufacturing (Modifications > Grinding worm/dressing wheel)** tab to call an option which enables you to find out whether suitable grinding worms (with their associated diamond dressing disc) are present for manufacturing the gear. A list of all suitable dressing wheels is generated from the "DressingWheel.dat" file. Dressing wheels which are not suitable for the currently entered module and pressure angle are ignored when the file is read.

The file to be loaded must be in the …\ext\dat\ or …\dat\ sub-folder, in the KISSsoft installation directory (although KISSsoft will search for them in \ext\dat\ first). When the file is imported, lines that start with a backslash are ignored.
In a line, all entries after the first are separated by semicolons (starting from the left):
1. Text, is ignored when the file is read
2. Text, is ignored when the file is read
3. Normal module [mm]
4. Pressure angle αn [°]
5. Profile crowning (depth crowning) radius rc [mm] (when "straight" is read, this radius is set to 1010.
6. Length of the linear tip relief LRELIEF [mm]
7. Angle φ [°] or radius rRELIEF [mm] of the linear tip relief (if the angle value is in degrees and arc minutes: x°xx or xØxx. If it is the radius: Rxxx)
8. Text, is ignored when the file is read
9. Text, is ignored when the file is read
10. Text, is ignored when the file is read
11. Position hp of the high point of the profile crowning on the dressing wheel [mm] (input along the flank, starting from the tip)
12. Dressing wheel addendum hfpd [mm]
13. Dressing wheel dedendum hapd [mm]
14. Gap AL*ref between the flank of the dressing wheel and the grinding worm [mm] (measured along the datum line)
15. Tooth root radius ϱ [mm] of the dressing wheel
16. Dressing wheel article number/label
17. Text, is ignored when the file is read
18. Text, is ignored when the file is read
19. Number of threads, grinding worm
20. Reference circle [mm] of the grinding worm

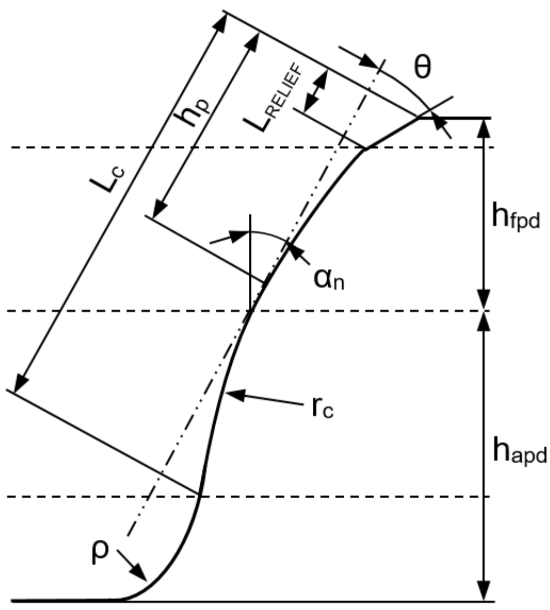


Figure 15.68: Dressing wheel with linear tip relief

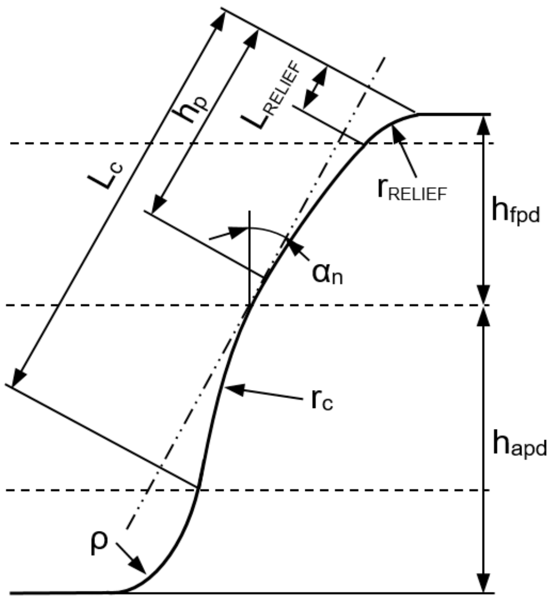


Figure 15.69: Dressing wheel with radial tip relief

This enables you to load the same dressing wheel several times, in different lines, with different grinding worms, for example. Do not enter a semicolon at the start or end of the line. The last line of the file must not be empty.

The system then displays the achievable tip and root modifications, according to the selected gear, with the loaded dressing wheels, in the first window. Only gears for which pre-machining (without a topping tool) has been defined, and a machining stock has been entered, are displayed.
The system then displays the suitability of the particular dressing wheel in the third column. Here, the following apply: "" mean suitable without modification of the gap *A*L*ref ("air between the flanks"), "" means suitable if the gap *A*L*ref is modified and "" means not suitable. The modifications and toothing diameter are calculated using a tooth thickness tolerance position as stated in the value in the **Tolerances** tab for the tooth form calculation. The system also displays the target tip relief Ca and the target modification length LCa* above the table, depending on the selected gear (according to the entries in the Modifications table in the **Modifications**) tab.

When a dressing wheel is selected, the system displays the basic data for the grinding process in a second window called "Selected grinding worm/dressing wheel". You can move the dressing wheel by Δh by modifying the gap *A*L*ref and also modify the grinding worm's lead angle and reference diameter. The modifications are then recalculated and displayed.

Once a grinding worm has been finally selected, the data in the profile modifications that most closely match the dressing wheel is adjusted. During this process, the data for the grinding worm and the dressing wheel is written to the "Dressingwheel.tmp" file. This file is stored in the Windows Temp folder.
These modifications are input in the Modifications table, and all previous profile modifications are deleted. The gear contour is now defined in the same way as it will be produced in the grinding process with the selected grinding worm. In addition, the depth of immersion of the grinding worm is set in the **Final machining** tab and the grinding process is set to "Generation grinding" with

"Grinding of flank and root".

The report also contains the diameter of a measuring circle dmess with the associated tip relief C_a. The measuring circle lies between the tip form circle d_{Fa} and the start of the tip relief d_{Ca} (dmess = (2*dFa + dCa) / 3).

► **Note:**

You can perform all the standard manipulations with dressing wheels described here in the **Selected grinding worm/dressing wheel** window.

a) **Sharpening the grinding worm**
. This is performed very frequently and regularly after a specified number of gears has been produced. Achieved by a radial feed of around 0.5 to 1.0 mm on the dressing device. Here, the gap *A*L*ref is automatically kept constant by the dressing device. Resharpening with an unchanged *A*L*ref only produces a minimal change in the worm lead angle.

b) **Modifying the gap AL*ref** (without changing the radial feed)
*A*L*ref can be increased on a grinding worm that has already been profiled (i.e. dressed). *A*L*ref can only be reduced on a grinding worm that is new, i.e. has not yet been profiled.

c) **Modifying the radial feed** ("center distance" dressing disc-grinding worm *d*worm)
The value for a can only be increased on a grinding worm that is new, i.e. has not yet been profiled. The "a" value can normally be reduced on a grinding worm that has already been profiled (i.e. dressed). Theoretically, in this case (radial reduction), the gap *A*L*ref could be reduced at the same time. This should prevent a minimal change to the grinding worm pitch (in a constant diameter dx), which would need to be corrected.

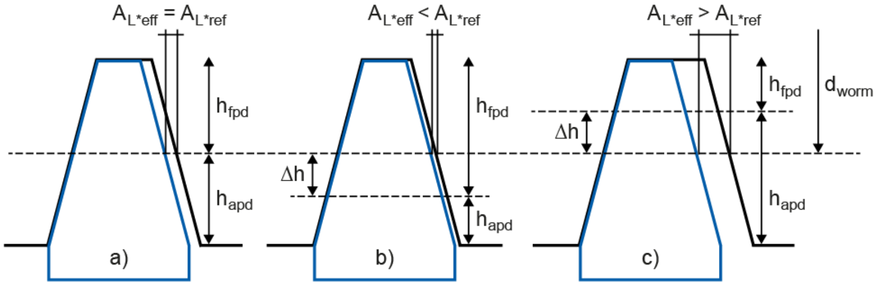


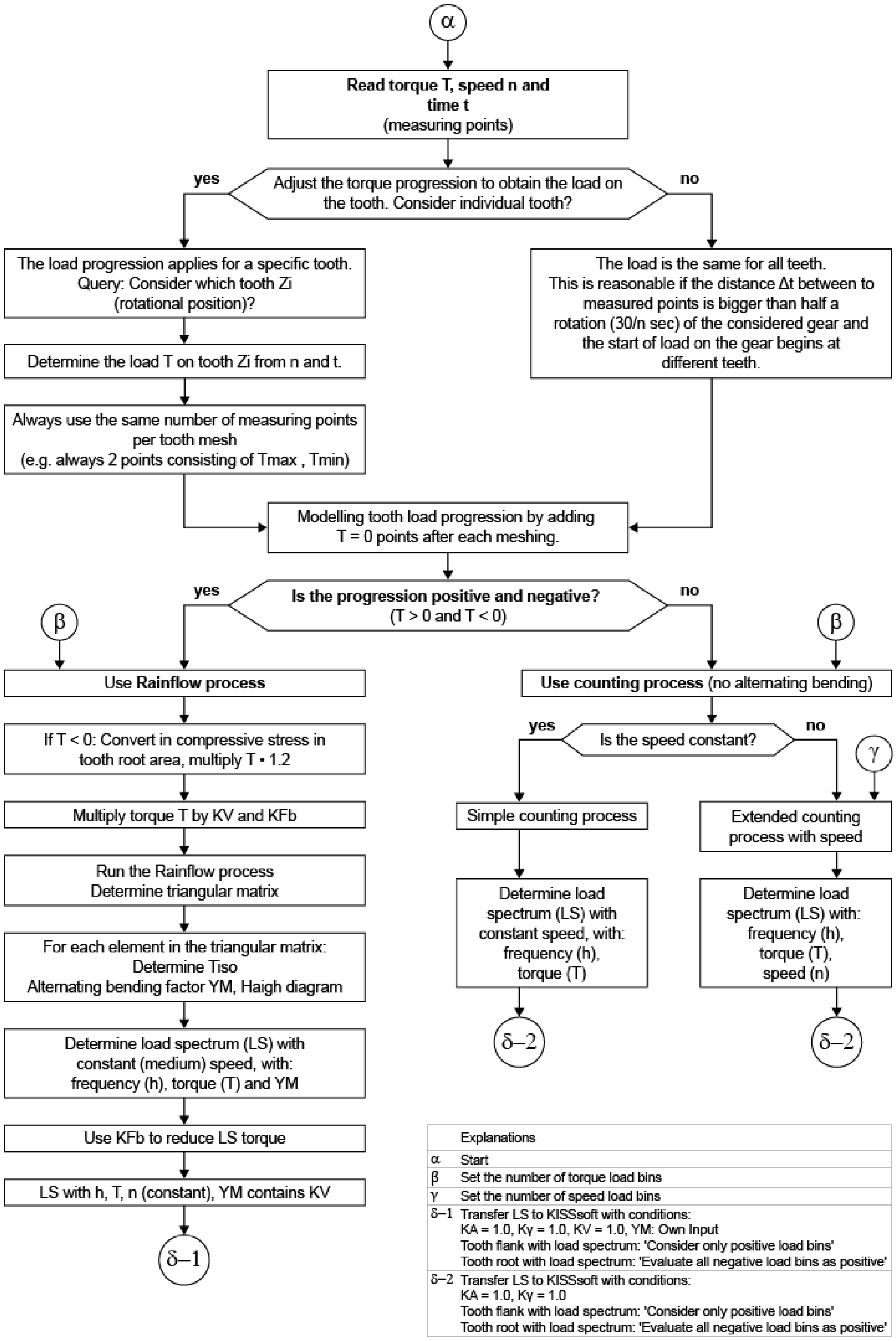
Figure 15.70: Grinding worm with dressing wheels (examples shown with and without displacement Δh)

15.8 Torque measurement

The calculation option for defining a load spectrum for gears using the measured torque curve enables you to generate a load spectrum from a measured torque curve. If all the torque measuring points are positive, an extended "simple count" method is used. In more complex torque curves that have positive and negative values, the Rainflow method is used and a load spectrum with alternating bending factors $\mathbf{Y_M}$ that takes alternating torque into account is determined.

This calculation option is available for all gear calculations that can perform calculations with load spectra.

A load spectrum can also be generated for rolling bearings and shafts. The simple count method is used for rolling bearings and the Rainflow method is used for shafts.



A load spectrum that can be used with KISSsoft is then determined from a measured torque curve. When using this method on a tooth, you must be aware that one tooth is subjected to load during meshing when the gear is rotated and then the load is removed again. The torque curve on the tooth is therefore changed by adding a point with torque zero after every measuring point (torque, speed, time).

To start the calculation, click on the selection list below **Calculation** in the title bar above **Torque measurement** or go to the **Strength** tab and click the Sizing button below the Load spectrum table.

15.8.1 Grid and spread

The maximum and minimum torque are defined when the torque points are imported. The number of required torque load bins you enter are then used to create the torque grid. The number of measuring points that fall within a particular torque load bin are counted and used to define the frequency of each load bin. The greater the number of torque load bins, the more accurate the resolution and the greater the number of load bins in the resulting load spectrum. Load spectra with a greater number of load bins also take significantly longer to calculate. You must think carefully about how accurate you want the evaluation to be (high load bin number) and how fast (low load bin number).

Usually, including in ISO 6336-6, the torque grid is predefined with a constant load bin width. However, as usually only the 2 to 10% load bins with the highest torques are damaging, spreading the torque distribution can improve accuracy without increasing processing time. Spreading means that the width of the load bins in the high torque range becomes narrower and the width in the lower range increases correspondingly. You can view the load bin width in the "Interim results" report.

15.8.2 Multiplier

The imported torque can be multiplied using the multiplier f_T. The imported speed is then multiplied with 1/ f_T accordingly. This is a good idea if the torque measurement is performed on the gear unit's input (or output) side and the load spectra are to be determined individually for each gear stage.

15.8.3 Torque curve

Two different torque curve cases can occur:

Torque is always positive (or zero): In this case, the "counting method" can be used to perform the conversion for the gears. The tooth root is only ever subjected to pulsating load. A matrix containing the torque and speed interval is formed and then each measuring point is put into the appropriate category ("counting"). This results in a load spectrum that has elements with different torque and speed (extended "simple count" method). The normal calculation ("all teeth") assumes that each measuring point on the torque curve occurs on each tooth. The "Determine load spectrum for a specific angular position" option is not activated. However, the torque curve is usually measured over a short time period and it is then assumed that this curve repeats constantly over the entire rating life. Every tooth therefore experiences every torque measuring point over time. The exception to this are actuators, where torque is always experienced in the same position. In this case, each tooth is only ever subjected to exactly the same torque.

Torque has positive and negative values: For the tooth flank, this is covered by only taking positive values into account. However, alternating load occurs at the root. This means that the Rainflow method must be used to determine significant occurrences of alternating load from the torque curve [1, 2]. The Rainflow method produces a matrix which shows how often a torque curve from T_{upper} to T_{lower} occurs. The matrix therefore has a torque interval in both axes: once for T_{upper} and once for T_{lower}. In the Haigh diagram, T_{upper} and T_{lower} can then be used to determine the alternating bending factor Y_M (ISO 6336-3) and the torque T_{ISO}. The Rainflow method is usually applied with stresses, not with torques. As the tooth root bending stress and torque are proportional, you can also use the torque. However, to ensure the correct values are determined, the torque must be multiplied with the dynamic factor KV and the face load factor for the root KFβ! This is because KV depends on the speed, which is no longer taken into account in the subsequent Rainflow calculation. And KFβ is not proportional to the torque, which is why it is different for T_{upper} and T_{lower}.

As the torque must be multiplied with KV*KFβ, this creates the problem that the load spectrum calculated using these values can only be applied to the root. KHβ must be used for the tooth flank. For this reason, once the load spectrum has been calculated using the Rainflow results, the torque of each load bin is divided by KFβ. The load spectrum then only contains KV and can therefore be used for the root and for the flank.

Either the Amzallag method or the ASME method can be used as the Rainflow method. Amzallag is used in ISO 12110-2 [3]. The calculation used in KISSsoft is checked using the example in Annex B of ISO 12110-2.

15.8.4 Calculation

The load spectrum calculation is performed for the reference gear and can usually also be performed for the gear pair (planetary gear stage, 3-gear, 4-gear). The normal calculation ("all teeth") assumes that each measuring point on the torque curve occurs on each tooth. This approach is correct if the torque's prefix operator never changes. However, if alternating torque occurs, this approach is only correct if the time interval between the individual measuring points is long enough to allow the gear being considered to perform one full rotation (or more).

If the **Determine load spectrum for a specific tooth** option has been selected, the speed and time information is used to calculate when a particular torque measuring point occurs on a selected tooth. The calculation is then performed. A load spectrum that has been determined in this way then only applies to the selected tooth according to its angular position on the reference gear. Despite this, it is possible to obtain a "generally" applicable load spectrum by selecting **Determine and use the angular position with average damage** to find a tooth which has experienced "average" damage when compared to all other teeth. This is a good option if the measured torque curve occurs repeatedly and the gear that is being analyzed has different angular positions at the start of the approach. In the case of actuators and similar mechanisms, where the angular position at the start always remains the same, we recommend you select the **Determine and use the angular position with maximum damage** option. This is because each tooth always experiences the same torque curve and the tooth that is subjected to the highest load is relevant for calculating the rating life. The damage experienced by an angular position is evaluated by using the Rainflow method to determine the corresponding load spectrum and then calculating the equivalent torque Teq as detailed in ISO

6336-6, Equation A.2.
Click the **Graphic** selection button to display the torque curve on a single tooth.

The number of measuring points per meshing must be constant to ensure that a load spectrum with the correct frequency distribution per element can be achieved. For this reason, the measuring point with highest and lowest torque is determined in each meshing, and then used throughout the calculation. All other measured points will be deleted. On request, the calculation can also be set so that only the measuring point with the highest torque is defined in each meshing, and then this value is used throughout the calculation.

If several measured points of a single rotation of the gear are measured, the number of torque changes increases progressively when the points that do not occur on the tooth under investigation are removed. The load spectrum of the individual tooth therefore includes a greater proportion of alternating bending loads, which results in lower tooth root safeties.

Example:
torque measurement with 100,000 measuring points, one measurement every 0.1 s. The torque prefix operator changes every 30 s. The gear with 20 teeth rotates once per second.

This results in 1 change per 300 measuring points, i.e. a change frequency of 1/300 = 0.333%.

In contrast, tooth X on the gear only "experiences" every 10th measuring point (10 points per second, 1 rotation per second). In other words, only 30 measuring points in 30 s, a change frequency of 1/30 = 3.33%!

As the calculation is complex, a very large number of interim results can be displayed. This helps you check its progress effectively. To display interim results, click on the appropriate flag in **Interim results**.

Once the calculation has finished, you can transfer the load spectrum to the **Strength** tab. Here, the system checks whether particular settings need to be changed, to ensure the calculation can be performed correctly. The necessary changes are displayed. Simply select "Yes" to confirm, if you want to apply them. For example, the application factor must be set to 1.0. If you are using the Rainflow variant, the dynamic factor must be set to 1.0 because KV is present in the torque.

15.8.4.1 Use in the script editor

The Import torque -> Determine load spectrum -> Service life calculation and Damage function work well in the script editor. A call to the CalcSafetyTooth_MeasuredTorque () function performs all 3 steps.

The **CylGearPair16** example includes a script. In this case, the number of load bins in the torque grid is increased incrementally from 50 to 250 and the damage to the root of Gear 1 is output.

15.8.5 Notes

Grid for torque resolution

The grid resolution has a major influence on the result. As the measuring points are arranged over the torque grid, their distribution to load bins with high torques, in particular, has a significant effect. On the other hand, defining a grid with a very high resolution will result in a correspondingly large number of load bins and calculations that take much longer to run.

Torque curve	Maximum variation	Suggested number of grid elements n_R
Only positive	$\Delta T/T_{max} <= 0.5$ ($\Delta T = T_{max}-T_{min}$)	50
Only positive	$\Delta T/T_{max} > 0.5$	50-100
Positive and negative		(50) 100-200

The values in this table apply for a constant load bin width.

Sampling rate

The sampling frequency (when recording torque measuring points) should not affect the result (unless it is too slow and load peaks are overlooked as a consequence). The sampling rate must be significantly higher than the torque signal frequency. See also DIN 45667 "Classification methods for evaluation of random vibrations".

Speeds

If the Rainflow method is used, only the torques at the measuring points are processed. Their associated speeds are ignored. Therefore, the average speed for all the measuring points is calculated. In the load spectrum, this value is then assigned to all load bins. For this reason, the dynamic factor of each measuring point is defined. The Rainflow method is then performed with T*KV. The speed values would also be ignored if the "Simple Count" method is used. However, this method can be expanded by distributing the measuring points in a torque-speed matrix to ensure the speed is included in the load spectrum.

Input files

In CSV format with the following information per row, optionally with:
a) Torque

b) Time; Torque

c) Time; Speed; Torque

Comment lines must start with //.

Click the "Data type CSV (delimiter-separated)" option when saving the CSV file in Excel.

If the value for "Speed" in variant a) or b) is missing, the nominal speed is used instead.

If the value for "Time" in variant a) is missing, a time of 1 second between two torque points is assumed.

You will find an example in the \Example directory: "TorqueData from Round Drive.csv" file.

15.9 Tooth form

In addition to the actual calculation, the tooth form calculation offers a number of other options, because it simulates manufacturing with a precisely defined cutter. These options include, for example,

- tooth form modifications with profile modifications and root contour optimization
- taking into account several steps in the manufacturing with different tools
- calculating the cutter (pinion type cutter or hobbing cutter) required to manufacture the gear teeth (for example, for tooth forms that have been imported from a CAD program or for modified tooth forms)
- tooth form modifications for injection molds or for use in manufacturing pinion type cutters

► **Note**

Special tutorials that specifically deal with tooth form modifications are available for use. These tutorials can be downloaded from the KISSoft website, https://www.kisssoft.com.

The **Tooth form** calculation module input window has of two columns. The left-hand column shows which operations are to be performed on the gears. The right-hand column consists of the **Tolerance field for calculation** and **Approximation for export groups** areas and the relevant operations group.

15.9.1 Context menu

Right-click in the operation directory structure group to display a context menu. This menu refers to the active element in the directory (shown with a blue background).

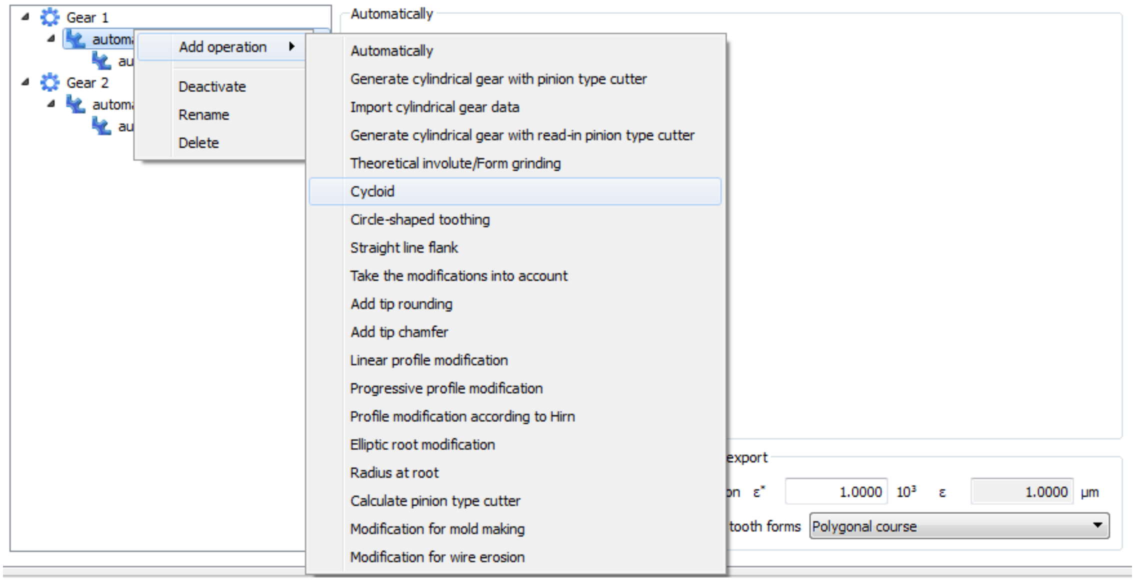


Figure 15.71: Context menu in the tooth form calculation

The context menu gives you these selection options:

- **Add operation** Select this menu option to open a sub-menu that lists the operations that can be performed on a particular gear (see chapter 15.9.2, Operations).
- **Choose as result** This result is usually displayed in the graphic and used in the strength calculations. The default setting is for the results of the last operation to be displayed here, unless the modification involves mold making, wire erosion, or a pinion type cutter.
- **Activate/Deactivate** Use this option to remove an operation that has been assigned to a gear from the list without deleting it. The icon is then marked with a red cross. The **Activate** menu option returns a deactivated operation to the list of active operations. The red cross then disappears.
- **Rename** Changes the name of an operation. Note that, if you change the name of an operation, this does not change the area name in the right-hand sub-window.
- **Delete** Permanently removes an operation entry, along with all its associated parameters.

15.9.2 Operations

You can use a combination of different operations to calculate the tooth form. You can apply one processing step after another, for example, using a hobbing cutter or a pinion type cutter and applying modifications such as roundings or profile modifications. You can label each operation to make it easy to identify at a later point in time.

15.9.2.1 Automatically

The default operation for the tooth form calculation is **Automatically**. The tooth form (with all its pre-machining and final machining) is then generated using the data entered in the standard tabs (see chapter 5.1, Standard and special tabs). Any modifications you have defined are taken into account when generating the tooth form. You can also disable this part of the operation in the context menu. The same applies to any tip chamfer or rounding you specify. If you select ZA as the flank shape, a ZA worm will be generated. Otherwise a ZI worm is created.

► **Note**

If the **Automatically** operation has been disabled, none of the data input in the **Reference profile** or **Modifications** input windows will be taken into consideration.

15.9.2.2 Generate cylindrical gear with hobbing cutter

To generate a cylindrical gear with a cutter, input the gear reference profile. When you add this operation, the window is filled automatically, based on the values you defined in the **Reference profile** input window. If the tool is a non-topping tool, the addendum of the reference profile is determined automatically from the tip circle, and not transferred from the values you input. For special applications (manufacturing a gear with a cutter with a different module), you can modify the module mn and the pressure angle αn. You can then use the Sizing buttons. Click the Sizing buttons to calculate the correct value in each case for the specified base circle. Click the **Cutter...** button to open the **Define cutter** window (see chapter 15.4.1.1, Tool: Hobbing cutters), which displays a list of tools. To define the tolerance field, you can either enter the generating profile shift coefficients directly (**Own input**), or use the pretreatment or final treatment tolerances.

The hobbing cutter data can also be input as coefficients or as absolute lengths (mm or inch). These selection options make your job much easier if the hobbing cutter data are the lengths (in mm or inches) given in a drawing.

When sizing haP0*, the system calculates the value, which is then used to manufacture the involute up to the active root diameter. The proposed value shown here is the exactly calculated value, to which 0.05 is added (to obtain a small distance between the root diameter and the active root diameter).

If you use the Sizing button to define the grinding wheel, the radius ϱaP0 should be small (e.g. 0.1*mn), otherwise the grinding process may reach the root radius.

► **Note**

The cutter information entered here is independent of the data specified in the **Reference profile** input window. In other words, the tooth form calculation is based exclusively on the values defined in the **Tooth form** input window.

15.9.2.3 Generating a cylindrical gear with an imported hobbing cutter

You can import the cutter contour from the CAD system in **dxf** format. To do this, define a half tooth (or a full tooth for an asymmetric tooth) from the predefined layer.

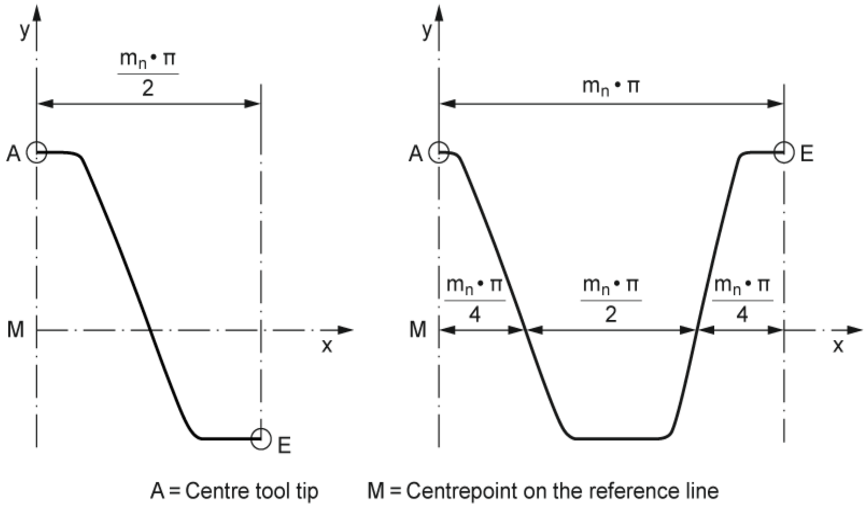


Figure 15.72: Tool profile

You can either specify the layer that includes the contour or select **ALL** for all the data. You can then decide whether to import the tool in transverse section or in normal section, and also change the module. The profile shift coefficients you enter here determine the tooth thickness.

Click on the "Cutter for displaced generation" option to select a normal module for the tool that differs from the cylindrical gear generated by the program.

Click on the "Input data as a reference" option to modify the module in the drawing. The cutter is then scaled to the normal module specified in the basic data.

► **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.4 Generating a cylindrical gear with a pinion type cutter

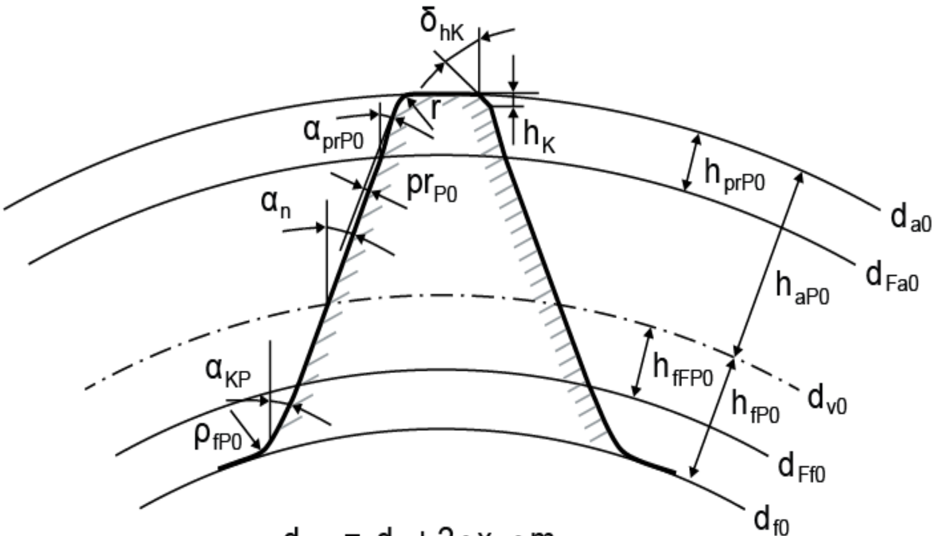
If you want to calculate the tooth form of gears manufactured using a shaping process you must define the geometry of the pinion type cutter.

Required input data:

- Reference profile of the pinion type cutter
  In the reference profile of the pinion type cutter, swap the values of the tip and root

used in the reference profile of the work gear at $x_0 + x_E = 0$. In other cases, you need to input a displacement at x0.

- Z_0 number of teeth on the pinion type cutter
- x_0 profile shift for the pinion type cutter
  (however, if x_0 not known, you can use the cylindrical gear calculation to define the profile shift from the tip diameter or the base tangent length (see chapter 15.1.8, Profile shift coefficient))
- or determine the length of the chamfer on the pinion tooth tip s or the radius of the rounding r on the pinion tooth tip (see Figure 15.73)



$$d_{v0} = d_0 + 2 \cdot x_0 \cdot m_n$$

$$d_{Fa0} = d_{v0} + 2 \cdot (h_{aP0} - h_{prP0})$$

$$d_{Ff0} = d_{v0} - 2 \cdot h_{fFP0}$$

Figure 15.73: Tool profile

15.9.2.5 Generating a cylindrical gear with an imported pinion type cutter

You can import a pinion type cutter as a .dxf file. To do this, define a half tooth (or a full tooth for an asymmetric tooth) from the predefined layer (select ALL for all layers).

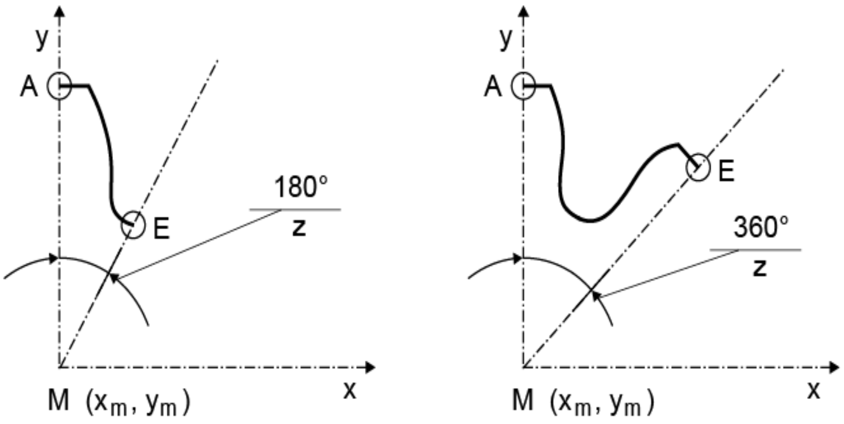


Figure 15.74: Pinion type cutter coordinates

A: Middle tooth tip: Start of contour
E: Middle tooth space (middle of the tooth tip in asymmetric gears): End of contour
M: Center point (x_m, y_m is a required entry)
z: Number of teeth

Click on the "Input data as a reference" option to modify the module in the drawing. The cutter is then scaled to the normal module specified in the basic data.

▶ **Note**

The file (**.dxf**) must only contain contours A to E in the layer you can specify for importing. In this case, you must specify the number of teeth on the pinion type cutter and the manufacturing center distance.

▶ **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.6 Reading (importing) a cylindrical gear

You can import a cylindrical gear directly as a **.dxf** file. To do this, define a half tooth (or a full tooth for an asymmetric tooth) from the predefined layer (select ALL for all layers).

The tooth form can also be imported from a group of points (x, y, N, R). A sample file for the import is stored in the DAT folder (Example_ToothFormAsPoints.DAT).

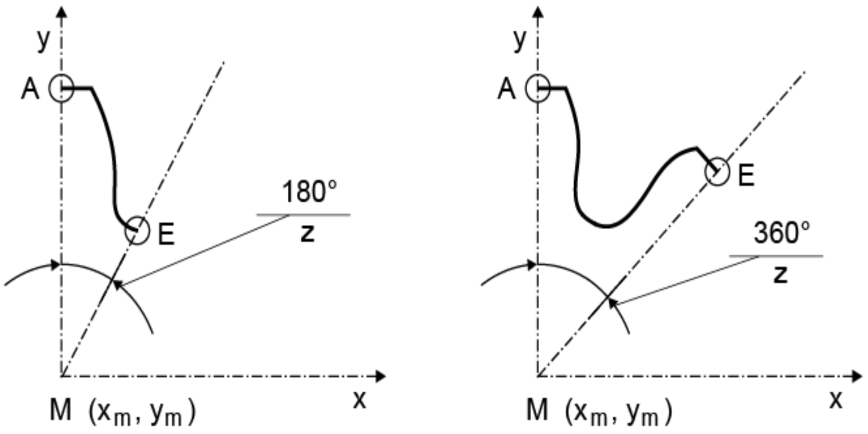


Figure 15.75: Coordinate system for the import

A: Mid tooth tip: Start of contour
E: Middle tooth space (middle of the tooth tip in asymmetric gears): End of contour
M: Center point (x_m, y_m is a required entry)
z: Number of teeth

Click on the **Input data as a reference** option to modify the module in the drawing. The cutter data is then scaled to the normal module specified in the basic data.

However, if the imported tooth form has straight elements (e.g. it is a polyline), the local normals and curvatures must be calculated as approximations so that a contact analysis can be performed. In these cases, click on the **Determine local flank normal and local curvature approximately** checkbox.

► **Note**

The file (**.dxf**) must only contain contours A to E in the layer you can specify for importing.

► **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.7 Adding tip rounding

You can add tip rounding as a tooth form modification. The rounding can be added either in the transverse or axial section.

15.9.2.8 Adding tip chamfer

You can add a tip chamfer as a tooth form modification. The chamfer can be added either in the transverse or axial section and is defined by the starting diameter and an angle.

15.9.2.9 Linear profile modification

In a linear profile modification, the tooth thickness is reduced in a linear progression from the starting diameter to the tip (relief C_a on each flank as a tooth thickness modification).

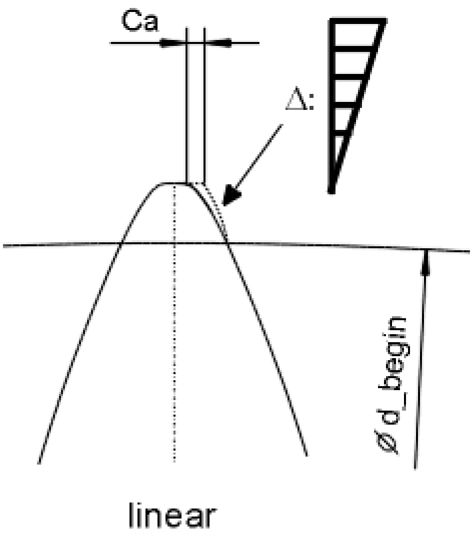


Figure 15.76: Linear profile modification

15.9.2.10 Logarithmic profile modification

In a logarithmic profile modification, the tooth thickness is reduced in a linear progression from the starting diameter to the tip. The profile modification is calculated as described in FVA 609 [30].

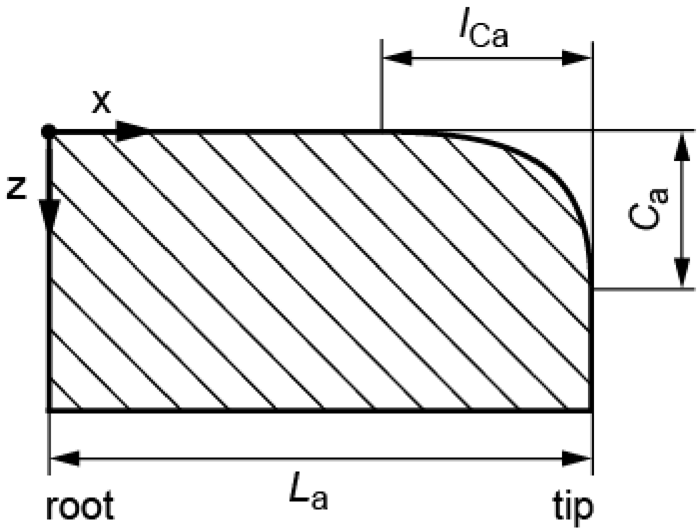


Figure 15.77: Logarithmic profile modification

Equations:

$$z(x) = \frac{C_{\alpha a}}{\log\left(\frac{1}{1-0.995^2}\right)} \log\left(\frac{1}{1-k^2}\right)$$

$$k = \frac{(L_\alpha - x - l_{Ca}) \cdot 0.995}{l_{Ca}}$$

$$x \geq L_\alpha - l_{Ca}$$

15.9.2.11 Progressive profile modification

In a progressive profile modification, the tooth thickness is reduced from a starting diameter to the tip (relief C_a on each flank as a tooth thickness modification) in accordance with

$\Delta A_s = 2 \cdot C_a \left(\frac{d - d_{begin}}{d_a - d_{begin}} \right)^{factor/5}$	(14.21)

. The coefficient controls the course of the relief. A coefficient of 5 represents a linear relief. For more information, see also Figure 14.44. If a coefficient greater than 5 is used, the progressive profile modification moves tangentially into the unmodified tooth flank. This is the preferred option if larger reliefs are to be achieved. We do not recommend you use a coefficient of less than 5 (some of these lower values are simply ignored by the program). Coefficients greater than 20 are also ignored. In this case, a coefficient of 20 is used.

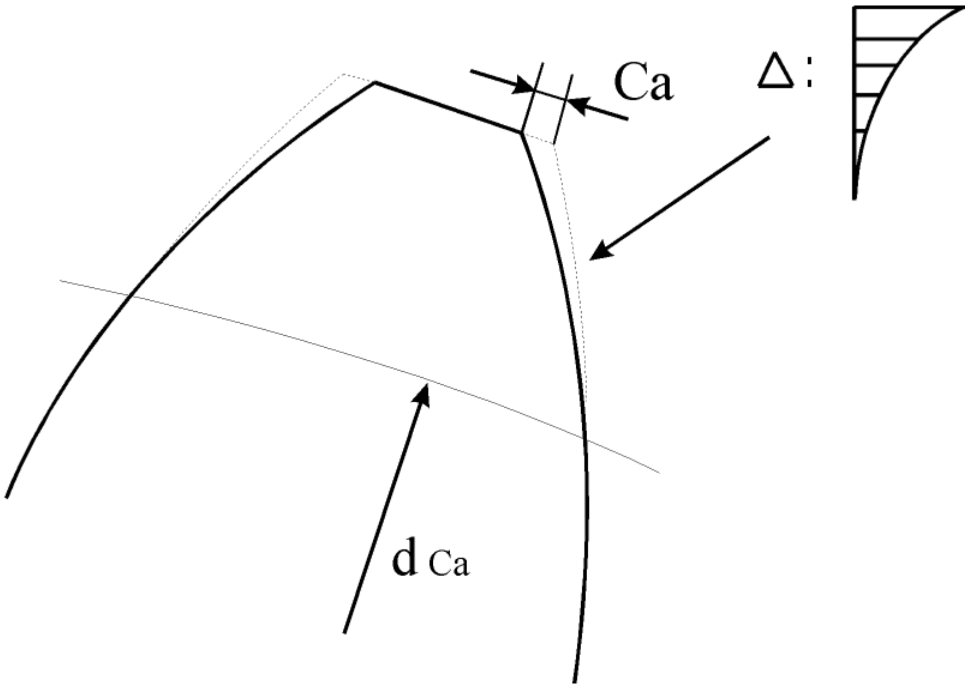


Figure 15.78: Progressive profile modification

15.9.2.12 Profile modification according to Hirn

An entry curve that passes into the involute tangentially is applied to the tooth tip starting from the specific diameter d_{begin}. This entry curve consists of three arcs of circle. The bend in the curve increases from arc of circle to arc of circle, so that the final arc of circle is tangential to the tip circle. This modified tooth form (also called a hybrid tooth) has significant benefits, because it results in extremely quiet running, despite relatively imprecise production methods. For this reason, the modification is applied for plastic products, for preference (see Figure 15.79).

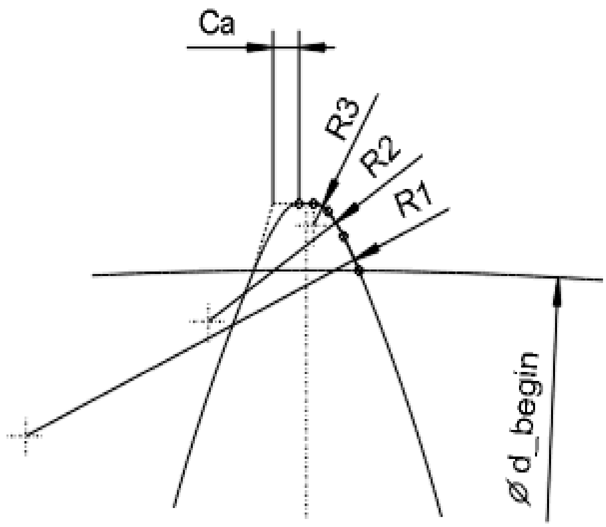


Figure 15.79: Profile modification according to Hirn

An entry curve is usually only applied to deep tooth forms with transverse contact ratios of greater than 2.1. In addition, KISSsoft can use its sizing function to suggest a suitable starting point (diameter) for the entry curve and the tip relief value. To do this, it uses the profile modification calculation (see chapter 15.7, Modifications).

The start of the entry curve is defined as follows:

- For a transverse contact ratio of 2.0: The active involute is reduced until the transverse contact ratio is exactly 2.0.
- For a transverse contact ratio of less than 2.0: The diameter is calculated so that an average tip relief is created, i.e. a transverse contact ratio of above 1.0 is reduced by approximately 50%.
- For example, from 1.8 to 1.8 - 0.5 . 0.8 = 1.4.

The exact definition is:
For a transverse contact ratio greater than > 2.0 : d_{Beginn} = Minimum (d_{PunktD}, $d_{PunktE0.2}$)
For a transverse contact ratio < 2.0 : d_{Beginn} = Minimum ($d_{PunktDE}$, $d_{PunktE0.2}$)

The relief C_a at the tip is defined as shown here:

- For top lands less than 0.21 .m_n: 0.5 . Tooth thickness - 0.01 .m_n
- For top lands greater than 0.21 .m_n: 0.10 .m_n...0.12 .m_n

15.9.2.13 Elliptical root modification

The root fillet is replaced by an ellipse-shaped contour which progresses tangentially in the flank and root circle. The aim is to achieve the greatest possible radius of curvature. The course of the contour can be influenced by the coefficient in the range 1 ÷ 20. If you click on the Sizing button for the diameter at the start of modification, the software suggests a value for the active root diameter (slightly increased, to prevent problems when applying the modification to an undercut). The definable length on the root circle is then set to > 0 if you want an area of the tooth form to run on to the root circle. For example, this is a good idea if the root circle is to be measured with measuring rollers.

The greater tooth thickness in the root area means that the generation process with the other gear in the pair must be checked. For a mathematical description of contours that are similar to ellipses, please contact KISSsoft Support and ask for the separate "kisssoft-anl-123-E-Elliptical root modification" instructions.

15.9.2.14 Taking into account modifications

Modifications, that are defined in the **Modifications** tab, are applied to the tooth form.

15.9.2.15 Root radius

The root contour is replaced by an exact arc of circle with a specifically definable radius. After you make this modification, check the generation process using the other gear in the pair.

15.9.2.16 Cutting the tooth tip

The gear's tip diameter is reduced to the predefined diameter (or increased, in the case of internal toothing).

15.9.2.17 Theoretical involute/form grinding

The tooth form is constructed mathematically. The involute is defined using the module and pressure angle along with the tip and root diameter. The tooth thickness is defined by the profile shift coefficients. You can also define a root radius (in the transverse section). This option is suitable for involute gears that cannot be manufactured by a gear generation process (e.g. internal gears with 4 teeth) or for a processing step involving form grinding.

► **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.18 Cycloid

You can select a cycloid as a special tooth form. The cycloid is defined with two rolling circles and the tip and root diameters. In the main calculation, the tooth thickness is defined by the allowances. Rolling circle 1 rolls on the inside on the reference circle and therefore cuts the dedendum flank. Rolling circle 2 rolls on the outside and generates the tip. Rolling circle 1 of the first gear should correspond to rolling circle 2 of the second gear. Sizing a cycloid toothing is made easier if you calculate the other gear in the pair using the data from the first gear during the optimization process.

Use the Stress curve and Kinematics analyses modules to analyze the strength and geometry properties of cycloid toothings.

► **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.19 Circular arc teeth

The circular arc teeth special toothing type can be defined using the tooth flank radius and the tooth thickness at the reference circle. An arc of circle is created in the root area.

A classic arrangement of circular pitched teeth, for example, as specified in NIHS 20-25 [31] consists of an arc of circle with radius r starting from the reference circle, a straight line that progresses in the direction of the center of the gear below the reference circle, and a full root rounding.

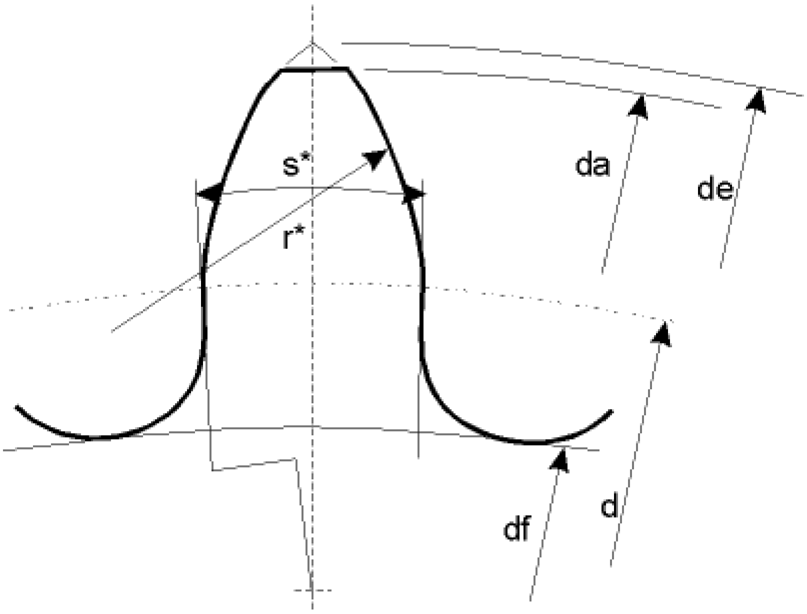


Figure 15.80: Arcs of circle on the tooth

► **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.20 Straight line flank

You can select a straight line flank as a special tooth form. The straight line flank is defined by the tooth thickness at the reference circle (theoretical toothing), the space width angle in transverse section, the tip and root diameter as well as the manufacturing profile shift coefficient (dependent on the tolerance). You can also predefine radii for tip and root rounding.

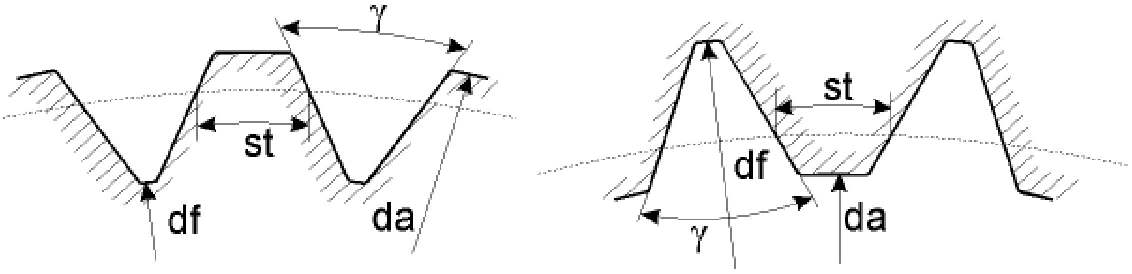


Figure 15.81: Straight line flank

► **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.21 Generation with the other gear in the pair

You can use the other gear in the pair to calculate the tooth form on all the gears, except on gear 1 (gear number - 1). In this case, you can overwrite the manufacturing center distance and the tip circle. The clearance between the gears can be generated either by reducing the manufacturing center distance or by inputting the circumferential backlash. The tip clearance is achieved by increasing the tip circle of the tool.

15.9.2.22 Calculating the reference profile

You can calculate the reference profile of an existing tooth form. A hobbing cutter can then be used to manufacture it. The manufacturing center distance can be changed in this calculation. This has a significant effect on the practicability of creating a tooth form using the generation process. In contrast, the value you input for the profile shift has no effect on the profile. Instead this influences the null point.

The calculated reference profile is then used as a cutter to calculate the cylindrical gear again. By comparing the two tooth forms you can then evaluate the extent to which the tooth form can be manufactured using the generation process. Click **Tool** to display the reference profile in the graphic.

15.9.2.23 Calculating a pinion type cutter

You can calculate a pinion type cutter for an existing tooth form. To do this, enter the number of teeth on the pinion type cutter and the manufacturing center distance. The center distance has a significant effect on the practicability of creating a tooth form using the generation process. Try out a number of different values to find the best one.

The calculated pinion type cutter is then used as a tool for calculating the cylindrical gear again. By comparing the two tooth forms you can then evaluate the extent to which the tooth form can be manufactured using the generation process. Click **Tool** to display the pinion type cutter.

15.9.2.24 Elliptical deformation

Applicable on the external gear (Gear1) of an internal-external cylindrical gear pair. This enables you to display the elliptical deformation of the race in a special gear unit in 2D. Typically z1+z2 = -2 applies here.

The contour of the race is stretched vertically by the lengthening factor and compressed horizontally so that the root circumference of the ellipse matches the root circle circumference of the undeformed gear. In a 2 D display, it is important you check:

- that the gear can be generated without collision over a pitch.

- that opposing sides mesh correctly.

If you need to make a modification, select a different lengthening factor or a different number of teeth (if the total number of teeth is an even number). Values between 0 and 5 % can be used as the lengthening factor.

Note: You cannot create a 3D output for this variant.

15.9.2.25 Generating a face gear with a pinion type cutter

This operation is not yet available. To generate a face gear, select the **Automatically** option. Define the pinion type cutter in the **Reference profile** input window.

15.9.2.26 Generate a rack with a hobbing cutter

Once again, enter the rack's reference profile, as you do when generating a cylindrical gear using a milling cutter. In this case, the addendum is only relevant if you are using a topping tool. The profile shift is measured, starting from a reference line, which is defined by the rack height minus the reference profile addendum in the main screen.

The profile shift coefficients can be either input directly or defined by the pre-machining and final treatment tolerances.

15.9.2.27 Generating a rack with imported hobbing cutter data

You can define a cutter as a **.dxf** file. In this case, the contour must be output as follows so that it can be read correctly by KISSsoft:

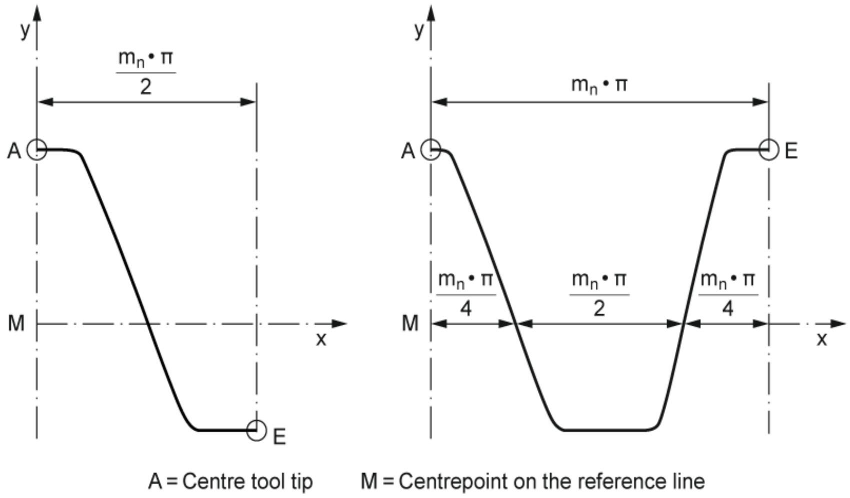


Figure 15.82: Tool profile

▶ **Note**

The file (**.dxf**) must only contain contours A to E in the layer you can specify for importing.

In addition to the contour, you must also define the manufacturing center distance. In this case, the reference line for the center distance is defined using the rack height.

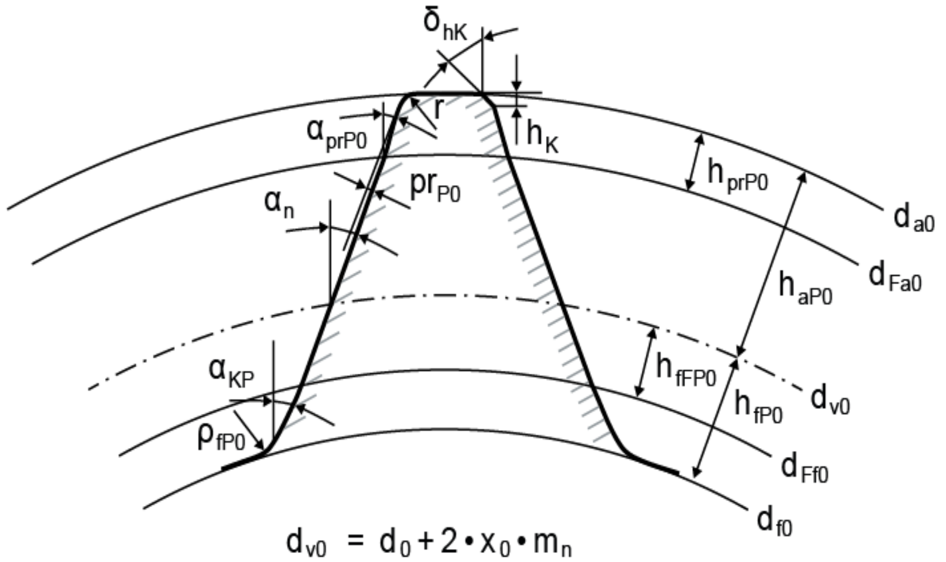
▶ **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.28 Generate rack with a pinion type cutter

Once again, enter the reference profile of the pinion type cutter, as you do when generating a cylindrical gear using a pinion type cutter. The profile shift is measured, starting from a reference line, which is defined by the rack height minus the reference profile addendum in the main screen.

The profile shift coefficients can be either input directly or defined by the pre-machining and final treatment tolerances.



$$d_{v0} = d_0 + 2 \cdot x_0 \cdot m_n$$

$$d_{Fa0} = d_{v0} + 2 \cdot (h_{aP0} - h_{prP0})$$

$$d_{Ff0} = d_{v0} - 2 \cdot h_{fFP0}$$

Figure 15.83: Cutter tooth geometry

15.9.2.29 Generating a rack with imported pinion type cutter

You can generate a rack with an imported pinion type cutter. In this case, you must specify the number of teeth on the pinion type cutter and the manufacturing center distance, in addition to the pinion type cutter contour in **.dxf** format.

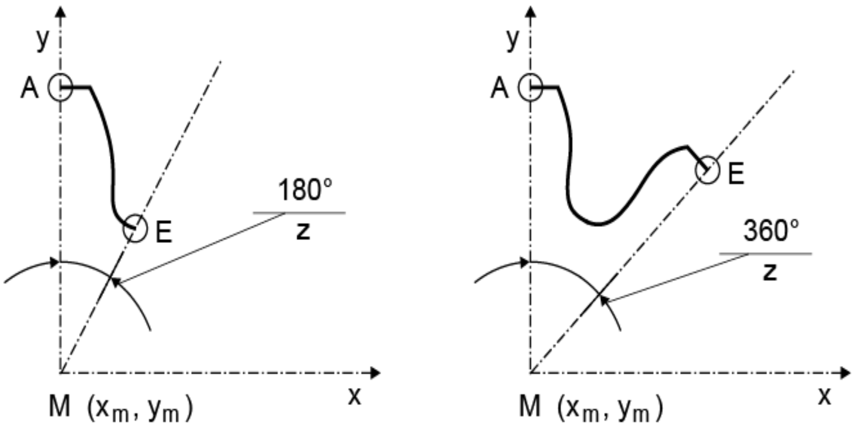


Figure 15.84: Coordinate system for the import

A	:	Mid tooth tip: Start of contour
E	:	Middle tooth space: End of contour
M	:	Center point (x_m, y_m) is a required entry
z	:	Number of teeth

▶ **Note**

The file (**.dxf**) must only contain contours A to E in the layer you can specify for importing.

▶ **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.30 Importing the rack

You can import a rack directly as a **.dxf** file in the following format:

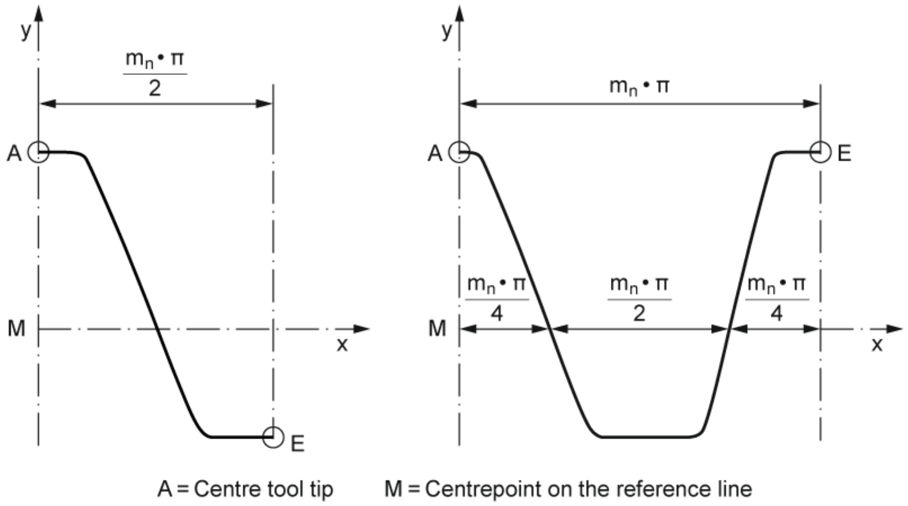


Figure 15.85: Tool profile

▶ **Note**

The file (**.dxf**) must only contain contours A to E in the layer you can specify for importing.

▶ **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.31 Generate a SA worm

This function is currently only available as the **Automatically** option.

15.9.2.32 Importing the data for a worm into the axial section

You can also import a worm in its axial section. In this case the contour is basically the same as the contour of the hobbing cutter, apart from the null point which forms the axis of the worm.

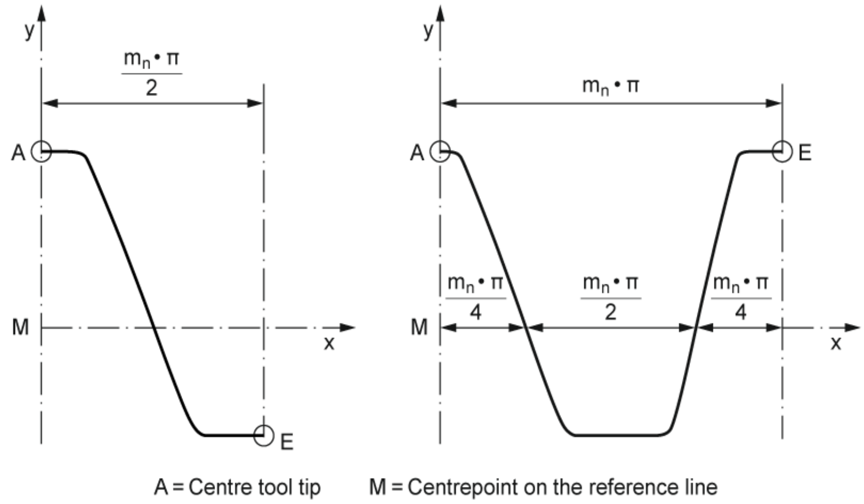


Figure 15.86: Tool profile

▶ **Note**

The file (**.dxf**) must only contain contours A to E in the layer you can specify for importing.

▶ **Note**

This operation should not be combined with the "automatic" operation, if this is not intended. To deactivate "automatic", right-click "Deactivate".

15.9.2.33 Modification for mold making

When plastic gears are manufactured using the injection molding process, the material shrinks as it cools. To counter this effect, and manufacture precise tooth forms, the size of the cutter must be increased by the shrinkage amount. Shrinkage may occur either radially or tangentially depending on what type of material is involved. If you enter the same values in the radial and tangential directions, the strain will be uniform in all directions

If the gear is injection molded around an inlay body, you must also input the external diameter of this body. The radial strains will then calculated using the "external diameter of inlay body".

The modifications only affect the transverse section of the tooth form. No strain in the axial direction is present when a 3D volume model is generated. If you want to create an expanded 3D model of a helical toothed gear (if the strain is to be the same in all three axes), you can achieve this by scaling the module (m_n), the center distance and the facewidth.

▶ **Example**

In the main screen, increase the module, center distance and facewidths by the required strain coefficient.
Coefficient = 1.02

$$m_n' = 1.02 \cdot m_n$$

$$a' = 1.02 \cdot a$$

$$b_1' = 1.02 \cdot b_1$$

$$b_2' = 1.02 \cdot b_2$$

Then, do not input a value for strains in the tooth form calculation.
This modification also increases the lead height p_z by the same coefficient. However, the angle of rotation of the spirals across the facewidth remains the same.

Usual values are:

- Radial shrinkage approx. 2%
- Tangential shrinkage approx. 2%

15.9.2.34 Modification for wire erosion

In the erosion process, the electrodes must maintain a specific distance from the required shape, because additional material is removed due to the spark gap. This is usually taken into account by the machines involved in the wire erosion process.

When sink eroding an injection mold, the eroding wire must therefore be thinner than the required shape by the amount of the spark gap. If a gear shaped electrode is used, the tooth will be correspondingly thinner. To achieve this, enter a negative value for the spark gap. Usual values for the spark gap are 0.03 to 0.07 mm.

After this modification you can also calculate the reference profile in the next step to determine the shape of a hobbing cutter for the electrodes.

► **Note**

You can also use the wire erosion modification to check the practicability of using the wire erosion method. If the aim is to erode external teeth, enter one modification with a positive wire radius and then the second with a negative radius. If the aim is to erode an injection mold for external teeth, first input a negative radius and then run a modification with a positive radius. By comparing the tooth forms you can then see whether the form can be manufactured, or whether a practical form can be created using these two steps.

15.9.2.35 Modification for pinion type cutter

The effective cutting angle and the draft angle of the pinion type cutter cause a tooth form deformation in the projection of the pinion type cutter in the horizontal plane.
The conversion performed here deforms the tooth form in the horizontal plane so that the projection once again shows the exact tooth form once the pinion type cutter has been manufactured.

By grinding with angle ϕ (effective cutting angle) Q moves to P (see Figure 15.87). If the projection P' is to agree (exact contour in the horizontal plane), P must equal Q in the H plane.

$PP' = (r_a - r_p) \cdot \tan(\phi)$	(12.22)
$P'Q = (PP') \cdot \tan(\xi)$	(12.23)
$somit: \ P'Q = (r_a - r_p) \cdot \tan(\phi) \cdot \tan(\xi)$	(12.24)

where

ϕ	Effective cutting angle
ξ	Tip draft angle in axial section
M	Pinion type cutter axis
r_a	Pinion type cutter tip circle radius
r_p	Coordinate of the point P

Conversion of the tooth form:

Given:	Exact tooth form in polar coordinates P = r (Angle)
Searched for:	Tooth form in H-plane P' = r' (Angle)
Solution:	$r' = r + \tan(\phi)$. $\tan(\zeta)(r_{a-r})$

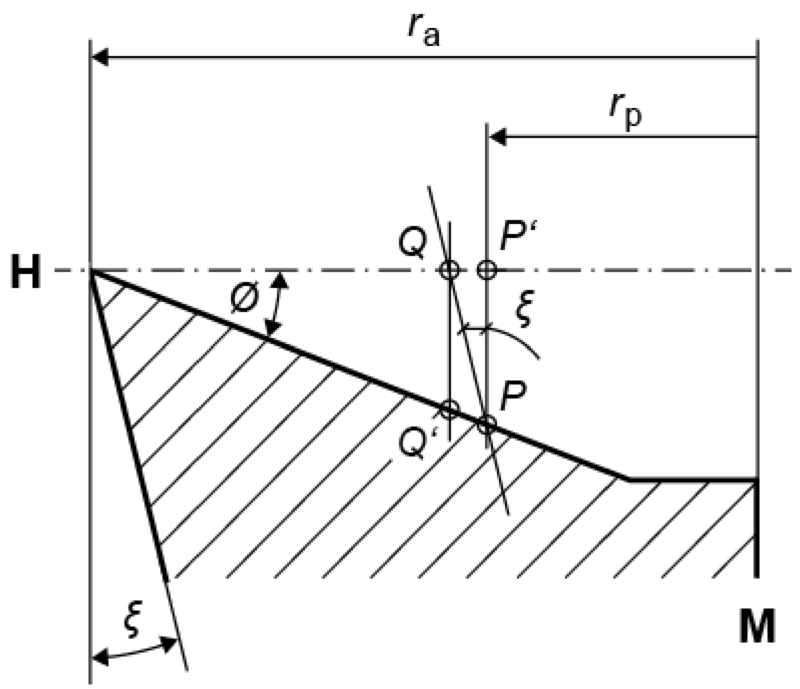


Figure 15.87: Pinion type cutter profile

15.10 Asymmetric gears

Click on "Module specific settings" in the "General" tab to calculate asymmetric gears. The user interface has been changed to make it possible to enter additional parameters for calculating asymmetric gears (pressure angle, reference profile, modifications, etc.).

The strength of asymmetrical gearing can be calculated according to ISO 6336, VDI 2545 and VDI 2736. However, the calculation methods have been modified to handle asymmetrical tooth forms on the basis of the technical literature [32]. The calculation is performed twice– once for the right side, and once for the left (however, in this case, both calculations are based on the special calculation procedure for asymmetrical gearing).

The corresponding flank results are displayed in the graphics, depending on which working flank is selected.

Not all the functions for asymmetrical gearing are currently available (unlike the functions for symmetrical cylindrical gears). For example, pre-machining cannot be performed for asymmetrical gears.

An overview of the advantages and disadvantages of asymmetric gears is given in [33].

15.11 Contact analysis

The load is taken into account when calculating the path of contact. This also calculates the face load factor $K_{H\beta}$ using the more precise method defined in ISO 6336-1, Annex E. In this case, the meshing stiffness can be calculated either according to Weber/Banaschek [21], ISO 6336-1 or **Own input** (see "Contact analysis/Face load factor" section in the Settings chapter).
You can view the calculation results in the report or select Graphic > Contact analysis to display it. The contact analysis can calculate either the transmission error as a length on the path of contact in mm or the angle of rotation error as an angle on the driven gear in °.

Resolution

You can select the levels "Own Input", "low", "medium", "high" or "very high" for the resolution. Resolution defines the termination criterion of the convergence condition, ε (10^-3 to 10^-6).

$$\left|\frac{T_c}{T_n} - 1\right| \leq \varepsilon$$

T_c= the calculated torque and
T_n= nominal torque

of the contact analysis and the number of slices of the discretized model (see the Theory of contact analysis section). The number of slices is automatically set according to the gear geometry and the selected resolution. The higher the selected resolution, the higher the number of slices that are defined automatically. You can also enter the number of steps, slices and pitches manually by setting the accuracy of calculation to "Own input" and clicking the Plus button next to it. The number of steps entered is per pitch.

Partial load and load factors

The "Partial load for calculation Wt" can be input for the load. The partial load is taken into account both when calculating the shaft deformation and when calculating the nominal torque. The partial load can be scaled by selecting **Take into account load factors** and entering ISO coefficients k_A, K_γ, and K_v. To perform the calculation according to ISO, set **Take into account load factors** to ‚$K_A K_\gamma K_v$‘. The "Resulting partial load factor W't" field shows the resulting partial load used for the contact analysis.

If the **Take load spectrum into account** option is selected, contact analysis is calculated using one of the load spectra defined in the **Rating** tab. To take into account individual load bins, you must select the element with the **Consider only one load bin in the load spectra** option in the **Rating** tab. When load spectra are taken into account, the configuration of the driving wheel, the working flank, and the direction of rotation, change according to the load bin's algebraic sign.

Coefficient of friction

If a "coefficient of friction" has been defined, the contact analysis calculates the power loss using the friction force Fr. Click on the Sizing button to the right of the "Coefficient of friction" to size the coefficient of friction according to ISO/TS 6336-22. The partial load and the load factors are then taken into account in the contact analysis when the coefficient of friction is sized. The coefficient of friction between the flanks is assumed to be a constant in the meshing.

Runout error

You can enter the runout error Fr here. This is then included in the contact analysis as a modification to the center distance. You should always perform a calculation with a positive and a negative runout error in the selection list with that name.

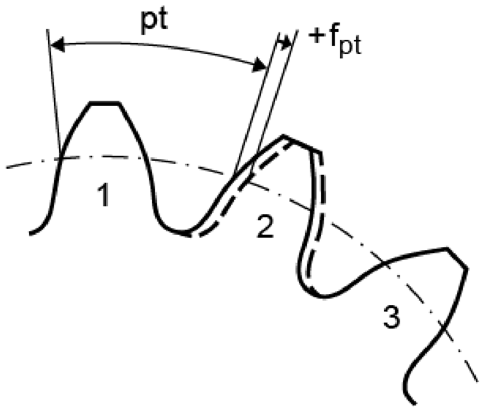
$$F_r = \sqrt{F_{r1}{}^2 + F_{r2}{}^2}$$

Single normal pitch deviation

You can predefine the single normal pitch deviation ƒpt here. The program them calculates a proposed value for the single normal pitch deviation. This can be entered with either a positive or negative prefix operator. The results are then output for the case that the distance is too large or small. The contact analysis is performed over two pitches when single normal pitch deviation is taken into account.

Note:
Numerical problems may arise if the selected single normal pitch deviation is too large relative to the partial load.

$$f_{pt} = \sqrt{f_{pt1}^2 + f_{pt2}^2}$$



Manufacturing deviation

To take the effect of manufacturing errors (fpar, ΣfHβ) into account, select an appropriate value from the "Manufacturing allowances" drop-down list in the **Contact analysis** tab.
The manufacturing error increases the flank gap in the normal flank direction.

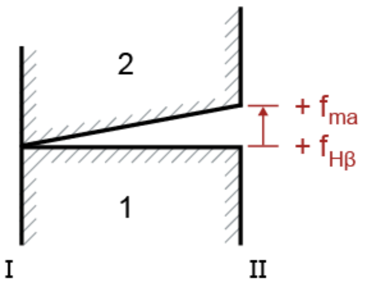


Figure 15.88: Figure: Definition of the positive direction of manufacturing errors fma and fHβ

A linear error distribution is assumed here so that the manufacturing error on side I is 0, is at its maximum on side II, and increases in a linear progression along the facewidth. Manufacturing errors are taken into account in pairs either positively or negatively.

Center distance and center distance tolerance
The "Center distance" field displays the center distance used for the calculation. This corresponds to the center distance in the selection list, either the "Center distance tolerance", the three center distance allowances (lower/middle/top) or the nominal center distance or the center distance defined by the user.

Wear

You can use the wear iteration function to define wear along the tooth flank in more detail because it performs several steps of the contact analysis with the worn tooth flank. However, this does significantly increase the time it takes this calculation to run. Click the **Calculate wear iteratively** checkbox to select this option.

You can input the maximum permitted wear per step. In the contact analysis shown below, the service life after one iteration was reduced by only applying the maximum permissible wear. In the next step in the contact analysis process, the tooth form with wear is taken into account. The process is repeated until the total service life is reached.

In the Module specific settings (**Contact analysis** tab), you have the option of defining the extent to which the results of the iterative wear calculation are to be smoothed.

Conical profile shift coefficients

If this option is selected (in the **Calculation > Settings > Contact analysis** tab), the profile shift coefficients in the **Basic data** can be overwritten, and a conical profile shift can be added to the gear pair with reference to gear 1. When used together with an axial offset, this can reduce the toothing clearance.

15.11.1 Theory of contact analysis

Contact analysis is based on the theory of deformation δ of the meshing of gear pairs as stated in Weber/Banaschek [21]. It can be split into three components:

- Gear body deformation
- Bending
- Hertzian Flattening

Bending:

$$\delta_z = \frac{F_{bti}}{b}\cos^2\alpha_{Fy}\,\frac{1-\nu^2}{E}\left[12\int_0^{y_p}\frac{(y_p-y)^2}{(2x')^3}dy+\left(\frac{2.4}{1-\nu}+\tan^2\alpha_{Fy}\right)\int_0^{y_p}\frac{dy}{2x'}\right]$$

Gear body deformation:

$$\delta_{RK} = \frac{F_{bti}}{b}\cos^2\alpha_{Fy}\,\frac{1-\nu^2}{E}\left[\frac{18}{\pi}\frac{y_p^2}{\bar{s}_{f20}^2}+\frac{2(1-2\nu)}{1-\nu}\frac{y_p}{\bar{s}_{f20}}+\frac{4.8}{\pi}\left(1+\frac{1-\nu}{2.4}\tan^2\alpha_{Fy}\right)\right]$$

Hertzian flattening:

$$\delta_{H1,2} = \frac{F_{bti}}{\pi b_g}\left[\left|\frac{1-\nu_1^2}{E_1}\ln\left(\frac{b_H^2}{4t_1^2}\right)+\frac{\nu_1(1+\nu_1)}{E_1}\right|+\left|\frac{1-\nu_2^2}{E_2}\ln\left(\frac{b_H^2}{4t_2^2}\right)+\frac{\nu_2(1+\nu_2)}{E_2}\right|\right]$$

The total deformation δ has the effect that the contact point is displaced along the path of contact and the theoretical length of path of contact is elongated, in comparison to the actual length of path of contact. The transverse contact ratio under load is therefore greater than in the load-free state.

The spring equation F=δ*C can be applied to calculate the components of the single contact stiffness from the individual deformation components and the normal force. The following applies for the tooth pair spring stiffness in a meshing gear pair:

$$\frac{1}{C} = \frac{1}{C_{B1}} + \frac{1}{C_{R1}} + \frac{1}{C_{H1/2}} + \frac{1}{C_{R2}} + \frac{1}{C_{B2}}$$

15.11.2 Asymmetrical gear teeth in the contact analysis

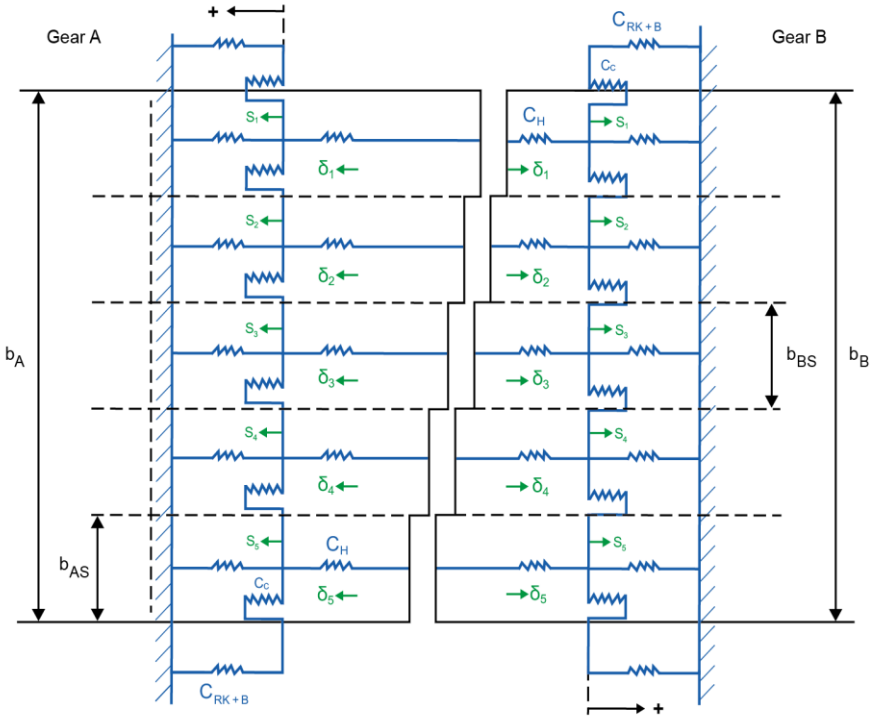
If the contact analysis is performed with asymmetric gear teeth, all the deformation components are calculated with a modified version of the Weber/Banaschek formulae [21] according to Langheinrich [22]. Essentially, this means that the simplified assumption of the symmetrical tooth thickness specified in Weber/Banaschek [21] x' should be replaced by the actual tooth thickness S_{zy} and the clamping point calculation should be modified to suit the new circumstances, as stated in [22].

15.11.3 Discretized model

A discretized toothing model has been generated so that the deformation theory of meshing in gear pairs developed by Weber/Banaschek can be applied to three dimensional cylindrical gears with helical gear teeth. To achieve this, the teeth are distributed in N slices across the width and are coupled together by torsional stiffness Cc.

$$C_c = 0.04N^2\frac{C_{R+B,i} + C_{R+B,i+1}}{2}$$

0.04: Empirical coefficient confirmed by comparative calculations with FEM. The user can change this coefficient (slice coupling factor) in the **Module specific settings**.



15.11.4 Reduced stiffness on the side edges

The reduced normal tooth thickness at the edge of a helical gear tooth then has the effect of reducing the bending stiffness of the tooth to the edges.

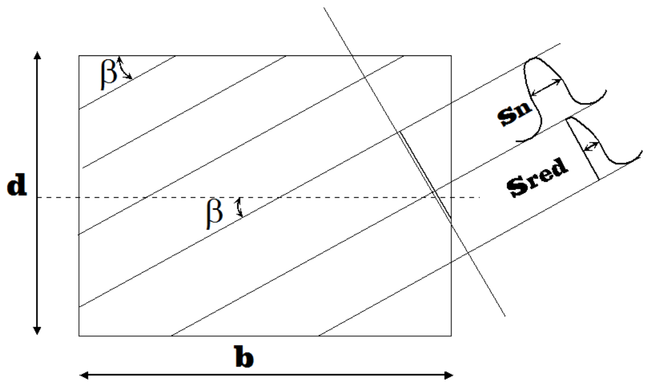


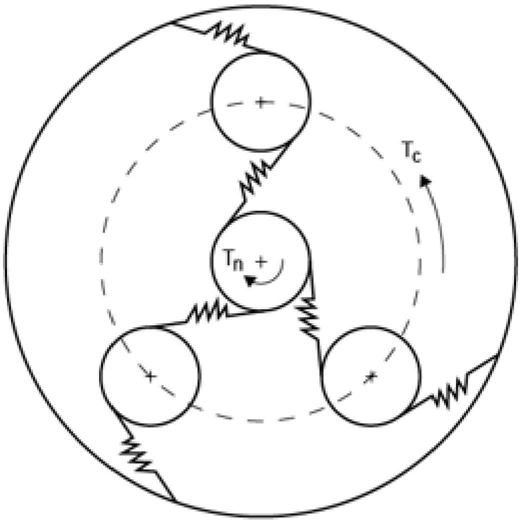
Figure 15.89: Illustration of two cuts for a gear with helical teeth

$$C_{red} = C\left(\frac{S_{red}}{S_n}\right)^{0.5}$$

Exponent 0.5 was evaluated in comparative analyses with FEM and LVR. The reciprocal value of this exponent (border weakening factor (buttressing)) can be changed by the user. It has a decisive effect on the buttressing effect that occurs in helical gear teeth.

15.11.5 Contact analysis model for planetary systems

In the contact analysis for planetary systems, the planet carrier is rotated around a fixed sun and internal gear. Each of the N planets uses the two pair stiffnesses of sun/planet and planet/internal gear to adapt its rotating position and thereby compensate for all torques. This approach also involves an iterative calculation of the system so that the sun's torque corresponds to the nominal torque.



15.11.6 Meshing position for contact analysis

Normally, the contact analysis is performed for a pitch (or for a system period in the case of planetary systems). The meshing position angle of the gears involved here is calculated as follows:

for a **cylindrical gear pair**, for a pitch, the base meshing position *i* for *N* meshing positions is:

$$\varphi_{Ritzel,i} = -\frac{\pi}{z_{Ritzel}} + i\frac{2\pi}{z_{Ritzel} \cdot N}$$

$$\varphi_{Rad,i} = -\pi + \frac{\pi}{z_{Rad}} - \varphi_{Ritzel,i}\frac{z_{Ritzel}}{z_{Rad}} - TE$$

TE = transmission error

For a **planetary system**, for a pitch, the base meshing position (sun and planet carrier) *i*, for *N* meshing positions, is:

$$\varphi_p = \frac{2\pi\left(z_S + |z_H|\right)}{z_S \cdot |z_H|}$$

$$\varphi_{C,i} = -\frac{\pi}{z_S} + i\frac{\varphi_p}{N} - \frac{\pi}{2} - TE_c\frac{z_S + |z_H|}{z_S}$$

$$\varphi_{S,i} = \varphi_{C,i} + \tan^{-1}\frac{c_y}{c_x} - \varphi_{C,i}\frac{z_S + |z_H|}{z_S} - \frac{s_n}{d_w}$$

Here, the following apply:

p = system period

C = planet carrier

S = sun

H = internal gear

*TE*c = transmission error, planet carrier

*s*n = tooth thickness, sun

*d*w = operating pitch diameter

*c*x,y = position of the first planet in the Cartesian coordinate system

15.11.7 Exporting results

All contact analysis results can be called natively, using SKRIPT, or serially, using the COM interface. The results can then be edited and processed in other programs such as Excel or Matlab. To call the data natively, using SKRIPT, `caResults.StepCAData[GearPair][Step].slices[Slice].resultname` can be used, for example. To call the data using the COM interface, the `RetrieveContactAccessResults(GearPair,ResultID)` function must be used.

```
//Example script:

// Calculate contact analysis

CalculateCA()

// Retrieve result via native variable access, e.g. Normalload
number Fn = caResults.StepCAData[0][0].slices[0].tooth[0].Fn
```

```
Write (Fn)

// Or retrieve result by using the RetrieveContactAnalysisResults
function, available in all tools and languages that support the MS COM
interface

number[][] result = RetrieveContactAnalysisResults(0,17)

Write(result[0][0])
```

Additional examples and a detailed list of all results that can be called are available from KISSsoft Support on request.

15.12 Gear pump

If you ignore the return volume, you can calculate the transport volume when you perform the normal calculation. You will find the parameter for this in the Basic data input window (see chapter 15.1, Basic data). In this case, click the **Calculation of the displacement volume of gear wheel pumps** checkbox in the **Calculations** tab in the **Settings** window, which you display by selecting the **Calculation** menu.

Click on **Calculation** to perform a more detailed calculation in the **Gear pump** tab.

The system calculates and displays the changes to the critical parameters of a pump that occur during meshing. These include geometric parameters such as the pinched volume (between two meshed tooth pairs, return volume), the volume with a critical inflow area (if possible, the flow of oil should be kept constant), the narrowest point (minimum distance between the first tooth pair without contact), inflow speed, oil inflow at the entry point (with Fourier analysis to evaluate the noise levels) and volume under pressure at input. Other important information is the progression of torque on the two gears, the progression of the Hertzian pressure σ_H, the sliding velocity v_g and the wear coefficient $\sigma_H \cdot v_g$. Hertzian flattening can be included when calculating forces because this effect has a significant influence. The pinched volume depends on how the pump construction functions under pressure at input or output. This is defined by the appropriate input value and has a considerable effect on the torque curve. When the pinched volume is reduced, you see a significant momentary increase in compression in this volume. This produces strong pulsing forces on the support and therefore generates noise. A pressure release groove must be installed to avoid this increase in pressure. For this reason, it is very useful to calculate and display pressure flow in the pinched volume.

Using this calculation, you can analyze any type of cylindrical gear with involute and non-involute tooth forms. At present, the only fundamental restriction is that this procedure is limited to spur gear teeth.

Optimization strategies for gear pumps

The most important and critical problems regarding gear pumps are

- Noise
- efficiency
- Size
- Wear

A number of tips about the criteria you can use to evaluate pumps are provided below.

- Noise:
  Variations in flow through the pump generate noise in the pipes. For this reason, the flow (Q) should be as continuous as possible. The enclosed volume (V1) should not be reduced during the generation process. A reduction in this volume would create a massive increase in compression in V1 and generate dynamic forces on both the bearing and the shafts. This effect can be reduced by the precise sizing of relief grooves. The inlet speed of the oil through the narrowest point should be kept as low as possible.
- Efficiency:
  The return volume should be kept as low as possible.
- Size:
  The KISSsoft Fine Sizing functions provide a very efficient method for achieving the highest possible displacement volume for a specified size.
- Wear:
  Take into account how the wear coefficient progresses (sliding velocity and Hertzian pressure between the tooth flanks)

► **Note:**

You will find more detailed information about gear pump analyses in KISSsoft-anl-035-E-GearPumpInstructions.doc [34] (available on request).

- The "Gear pump" report shows the input torque on gear 1 [T1] and the torque transferred from gear 1 to gear 2 [T1Contact].
- You should use the torque at the point of contact in the strength calculation and the contact analysis (calculated from P_{out} and P_{in}). Enter this data in the **Basic data** tab.
- The total power [P] and torque [T1] at the pump inlet are only documented in the "Gear pumps" report. Otherwise, they are not used. All the graphics shown under **Graphics > Gear pump** are based on the pressure. The torque curve used in the graphic is the input torque [T1].
- Helical gears are not taken into account. The equivalent spur gear is used.

- $\varepsilon\alpha >= 1 + \varepsilon\beta$ applies for helical pumps. Otherwise, a hydraulic short-circuit would occur. Significant noise reduction is only achieved if $\varepsilon\beta = 1$. Consequently, a deep tooth form with $\varepsilon\alpha > 2$ should be used.

15.13 Operating backlash

In addition to calculating the theoretical backlash as defined in DIN 3967, the backlash after mounting (including toothing deviations, deviation error of axis according to ISO 10064 or DIN 3964 (see Table 15.23), form and mounting deviations) and the operating backlash (including the temperature differences between the gears and the housings) can also be calculated. To calculate the operating backlash, input a temperature range for the gears and the housing, and the maximum and minimum difference in temperature between them. Two cases are calculated simultaneously, one that produces the maximum operating backlash (with the given temperature inputs), and one that produces the minimum operating backlash.

If the module < 1, the statically evaluated circumferential backlash is also calculated according to DIN 58405.

The reduction of the backlash due to single flank deviations is then calculated with tolerances Fb, Ff and fp according to the corresponding quality standard.

The reduction in clearance due to single flank deviations is not taken into account for crossed helical gears.

The effect of the runout error can also be taken into consideration. In this case, the roller runout tolerance (determined using the approximation formula Fr = Fi'' - fi'') is used instead of the runout error Fr for module < 1.

Bearing center distance	**Axis alignment accuracy class**											
L_G (nominal length) in mm	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**
up to 50	5	6	8	10	12	16	20	25	32	40	50	63
more than 50 and up to 125	6	8	10	112	16	20	25	32	40	50	63	80
more than 125 and up to 280	8	10	12	16	20	25	32	40	50	63	80	100
more than 280 and up to 560	10	12	16	20	25	32	40	50	63	80	100	125
more than 560 and up to 1000	12	16	20	25	32	40	50	63	80	100	125	160

more than 1000 and up to 1600	16	20	25	32	40	50	63	80	100	125	160	200
more than 1600 and up to 2500	20	25	32	40	50	63	80	100	125	160	200	250
more than 2500 and up to 3150	25	32	40	50	63	80	100	125	160	200	250	320

Table 15.23: Deviation error of axis according to DIN 3964, values in [mm]

As shown in Table (see Table 15.23), the values in the Axis position accuracy and Distance between bearings input fields are used to calculate the axis deviation error according to DIN 3964. Backlashes are calculated according to DIN 3967.

Circumferential backlash calculation:

The circumferential backlash is calculated on the reference circle with the following formula, according to DIN 3967:

$$j_t = (-A_s / \cos\beta) + 2 \cdot A_a \cdot \tan\alpha_t$$

In KISSsoft, the operating backslash is calculated in the operating pitch circle, using the more precise formula:

$$j_t = (-A_s / \cos\beta \cdot \frac{\cos\alpha_t}{\cos\alpha_{wt}}) + 2 \cdot A_a \cdot \tan\alpha_{wt}$$

Reduced tolerance ranges for the center distance, tooth thickness, tip and root diameter, runout and manufacturing tolerances can also be taken into account.

Planetary gear units are handled differently in the operating backslash calculation. Here, there are 2 operating pitch diameters for the planets (sun/planet and planet/internal gear). The change in operating pitch diameter due to thermal expansion is defined here for the operating pitch circle determined in this process.

In addition, the change in tip clearance due to thermal expansion (and water absorption for plastics) is also calculated.

Any strains that occur in the body of the gear also change its pitch. A single normal pitch deviation occurs as soon as both gears show unequal strain. The increase or decrease in pitch caused by thermal expansion is defined as follows:

$$\Delta pt_1 = pt_1 \cdot \alpha_1 \cdot (\Theta_1 - \Theta_{ref})$$
$$\Delta pt_2 = pt_2 \cdot \alpha_2 \cdot (\Theta_2 - \Theta_{ref})$$
$$fpt = \Delta pt_1 - \Delta pt_2$$

pt: pitch

a: coefficient of thermal expansion

Θ: temperatures

fpt: single pitch deviation

Plastics also expand due to water absorption.

15.13.1 Temperatures

The **Reference temperature**T is the ambient temperature specified for manufacturing. The tooth thicknesses input here apply to this temperature.

The **Temperature range gears** for specific gears defines the thermal expansion coefficient for individual gears. The wheel bulk temperature of the scuffing calculation can be used as here as a starting point.

Taken together with the coefficient of thermal expansion, the **Temperature range housing** then defines the coefficients of thermal expansion that occur for the housing.

The **Permitted temperature difference** defines the maximum permitted difference between the gear temperature and the housing temperature.

15.13.2 Relative water absorption during swelling

Input this value as a percentage of the volume. To calculate clearance, DIN 3967 specifies that: According to DIN 3967, for plastics, the linear expansion due to water absorption is approximately 1/3 of the amount of water absorbed. However, for fiber-reinforced plastics, it is only around 1/12 of the volume of water absorbed. You can also set your own value for the linear expansion due to water absorption.

15.13.3 Coefficient of thermal expansion for housing

If you select a material from the database, this field merely provides information about the coefficient of expansion of the selected housing material. In this case, you cannot change the value. However, if you have set the **Housing material** drop-down list to **Own input**, you can enter your own value.

For plastic materials, it is also possible to define the linear expansion of the housing due to water absorption.

15.13.4 Take into account shaft bending and flank line modifications

In order to use this option, the load distribution calculation ($K_{H\beta}$) according to ISO 6336-1, Annex E, must be activated. (It is used to calculate the shaft bending.) It then determines the position with the lowest backlash change Δjt.i across the facewidth. (This position is documented in the "Face load factor" report). For load spectra, the lowest value found in all load bins is determined.
If Δjt.i is negative, the operating clearance is reduced. This therefore changes the minimum operating clearance. (The maximum operating clearance remains unchanged, as it represents the load-free state.) If Δjt.i is positive, the operating clearance increases. This leads to a change in the maximum operating clearance. (The minimum operating clearance remains unchanged.)
To determine the backlash change caused by bending, only the components in the axial plane, including the component of the flank line modification in the peripheral direction, are taken into account. The bending component normal to the axial plane is not considered, as the flanks lie above the entire facewidth, under load (if $K_{H\beta} < 2$), and therefore do not cause any backlash change.

15.13.5 Take into account tooth bending

Tooth deformation is only taken into account if the line load w>=100 N/mm (otherwise the calculation of the bending according to ISO 6336 is too inaccurate). Tooth deformation is only taken into account in the case of the minimum operating clearance. (The maximum operating clearance remains unchanged, as it represents the load-free state.) It is questionable whether taking the tooth deformation into consideration is sensible. The calculation of the bending is only approximate and can result in the combined result being too conservative.

15.14 Master gear

You can use this KISSsoft calculation module to size and check master gears.

To perform a test for a double flank composite transmission, you require one master gear which is then rotated on a test device together with the gear you want to test. In the test run, the test gear and the master gear are pressed lightly together so that no backlash is generated. The deviation in center distance is then measured carefully. The difference between the minimum and maximum value calculated here is the tooth-to-tooth composite error. To obtain accurate information about how the test gear will run after installation in the gearbox, the test gear's active involute should be generated as completely as possible during the test run. However, it is essential that you prevent the master gear from meshing too deeply in the root area: If the value for the test gear root form circle is not achieved, this will cause meshing interference which will, in turn, generate measurement results that are massively incorrect. You can call the master gear sizing function for each gear in a particular calculation. When you open the sizing window, the default values for a suitable standard master gear taken from DIN 3970 are displayed. The analysis functions check the maximum and minimum tolerance fields of the tooth thickness of the test gear whose involute is being processed. The report then shows which area of the active involute has been tested, or not tested. If the value for the root form circle is not achieved, the program displays a warning to prompt you to reduce the tip circle diameter of the master gear. This calculation is also available for cylindrical gears with a minimum number of teeth greater than 4. Click the **Save** button to save the master gear data and the master gear-test gear pair as KISSsoft files.

Take into account total radial composite deviation (according to AGMA 2002): When calculating the smallest test center distance [aMin], the theoretical center distance stated in AGMA 2002 (equation 8.5) is further reduced by the total radial composite deviation (Vcq specified in AGMA 2000). If the manufacturing tolerances specified in ISO or DIN are being applied, Fi" is used for that purpose.

15.15 AGMA 925

In this input window, you can specify the probability of scuffing and wear and the susceptibility to micropitting (frosting), as specified in AGMA 925.

AGMA 925-A03 *Effect of Lubrication on Gear Surface Distress* calculates the conditions in the lubrication gap across the gear meshing. AGMA 925 defines how to calculate the lubrication gap height while taking into account the flank deformation, lubricant properties, sliding velocity, and local Hertzian stress. The standard then uses this base data to calculate the probability of wear. The wear is caused by the metal surfaces contacting each other if the lubrication gap is too narrow. The probability of wear calculated by the standard is greater than the values that occur in practice.

The standard does not give any indications about safety against micropitting (frosting). However, data provided by the relevant technical literature, and the results of research, reveal that there is a direct correlation between the minimum lubrication gap-to-surface roughness ratio and the occurrence of micropitting (frosting). You can therefore use this calculation method to optimize gear teeth for micropitting (frosting). AGMA 925 also includes a definition of the probability of scuffing. This analysis uses the same base data (Blok's equations) as the calculation of scuffing according to the flash temperature criteria given in DIN 3990, Part 4. However, defining the permitted scuffing temperature according to AGMA 925 presents more of a problem, because of the lack of comprehensive or generally applicable information. In particular, there is no reference to a scuffing load capacity specification as given in the FZG test. There is therefore a tendency to underevaluate oils that have effective EP additives.

The values for the pressure-viscosity coefficient α of typical gear unit oils vary between 0.00725mm^2/N and 0.029 mm^2/N, and are defined as follows in AGMA 925-A03:

$$\alpha = k \cdot \eta_M^5$$	(14.25)

where

α	pressure-viscosity coefficient	mm^2/N
k	see Table 2 in AGMA 925-A03	-
η_M	dynamic viscosity for tooth temperature θM	mPa . s

In practice, calculating wear according to Wellauer results in risk of wear values that are too high. For this reason, the analysis is performed according to Dowson (as in Annex E of AGMA 925). The report shows the results for both methods.

15.16 Root stress FEM

15.16.1 Calculating 2D FEM tooth root stress

This special calculation can be used to calculate the stresses at the root of a gear tooth using the FEM method under the assumption of a plane stress or plane strain state.

The following input fields are available:

1. **Select gear pair/Select gear** for the gear/gear pair to be analyzed. The gear pair selection option is only available in modules with multiple gear pairs (e.g. three gears train).
2. **Boundary condition, gear segment** (fixation condition for the FEM analysis). This can either be the internal or external diameter of the gear (for external or internal gears), the sides of the gear segment selected for the analysis, or the inside or external diameter with sliding on the side. The last variant (diameter fixed, sliding sides) is usually the best.
3. The **mesh density of the FE model** can be defined using seven mesh density levels. The first level represents a coarse mesh and the last level represents a very fine mesh. The tooth form in the FE model is represented more accurately as the mesh density increases. This has significant impact on the resulting stresses. We recommended that levels 1-4 are used for the first stages of the design and levels 5-7 for the final stress evaluation.
4. **Stress criterion** (stress type to be used when searching for maximum stress). This can be either the maximum principal stress or the Von Mises stress. As the calculation is

used to analyze the tooth root bending stress, we recommend you use the maximum principal stress.

5. **Assumption for 2D modeling** for the plane state. Here, the user can select either a plane stress or a plain strain state. We recommend you select a plain strain state for standard gears. Plane stress is suitable for gears with a very small facewidth (facewidth smaller than tooth thickness).
6. **Number of teeth** to be included in the FE model.

Before the FE analysis, the load is calculated and applied to the force application angle at the point of contact of the middle tooth. If the **Input application of force data** option is selected, the user can enter the load, the force application diameter and the force application angle manually. Load is only applied to the middle tooth in the FE model, no matter how many teeth are being modeled. Therefore the mesh of this tooth is finer than that of any modelled teeth around it (although the difference on the mesh density is quite small for high mesh density levels). The software automatically generates a finer mesh around notches and at the root region of the middle tooth, to provide a more accurate calculation of the stresses around these areas.

In the 2D analysis for helical gears, the equivalent spur gear is calculated and used, according to ISO 6336-3. The equivalent gear can be exported from KISSsoft at any time (**File > Export**).

The FEM results are provided at the KISSsoft result window and in the report. Click on **Graphic > FEM > 2D Results (2D Root Stress) > Stress at root area (middle section)** to visualize the stress distribution (principal stress or von Mises equivalent stress, depending on the user selection) at the middle section of the gear from the tooth root to the point of force application. Click on **Graphic > FEM > 3D FEM Results** to view the modeled cross section of the gear. You can also display other results, such as the displacements or principal stresses.

15.16.2 Calculate 3D tooth root stress with FEM

This special calculation can be used to calculate the stresses at the root of a gear tooth using the FEM method. A three dimensional (3D) model is created for this purpose. As this calculation takes the total length of path of contact and the associated load distribution from the contact analysis results, it is essential that the contact analysis has been performed successfully before this special calculation can be performed.

The following input fields are available:

1. **Select gear pair/Select gear** for the gear/gear pair to be analyzed. The gear pair selection option is only available in modules with multiple gear pairs (e.g. three gears train).
2. **Number of teeth** to be included in the FE model.
3. **Mesh density (XY plane)** controls the mesh fineness at the face of the modeled gear. The mesh density can be defined using seven mesh density levels. The first level represents a coarse mesh and the last level represents a very fine mesh. The tooth

form in the FE model is represented more accurately as the mesh density increases. This has significant impact on the resulting stresses. We recommended that levels 1-4 are used for the first stages of design and levels 5-7 for the final evaluation.

4. **Number of slices** defines the number of finite elements along the facewidth and also the number of slices for the contact analysis. The same **Number of slices** setting is used under the **Resolution** option in the contact analysis. The user can select the **Own input** option in order to enter the number of slices directly in an additional field.
5. **Use quadratic finite elements** uses the finite elements of the quadratic approach. These finite elements usually provide more accurate results than linear finite elements when the FE mesh is relatively coarse. However, the calculation time increases exponentially when quadratic finite elements are used. Therefore, we recommend this function is only used with low mesh densities (recommendation: levels 1-3 for number of teeth = 5, levels 3-5 for number of teeth = 3, levels 6-7 for number of teeth = 1).
6. **Load case** for the calculation. This option is only displayed if the contact analysis has been performed successfully. The user can now select the load case for the calculation. The default load case is the one which leads to maximum bending moment at the middle tooth. The user can also select any of the positions calculated in the contact analysis.

The FE mesh used for the calculation is always displayed on the right-hand side. When the contact analysis has been performed successfully, the resulting contact forces are displayed as red lines on the FE mesh. These always correspond to the currently selected load case. The FE mesh of the middle tooth is finer than that of any modeled teeth around it (although the difference in density in high mesh densities is quite small). The software automatically generates a finer mesh around notches and at the root area of the middle tooth, to provide a more accurate calculation of the stresses around these areas.

The tooth's fixation condition (required boundary condition for the FEM analysis) is defined at the inside diameter and the sides of the gear. If the inside diameter is zero, the software automatically defines an inside diameter of 20% of the tip diameter so that the calculation can be performed. Investigations have shown that this has no significant influence on the tooth root stress results.

The FEM results are provided at the KISSsoft result window and in the report. The stress results for every slice along the tooth form of all the modeled teeth are provided in the report. Click on **Graphic > FEM > 3D FEM Results** to display more results, such as displacements and principle stresses, in 3D. All the results in the range between the inside diameter and the active diameter are displayed.

15.17 Gear body

The **Gear body calculation** tab is used in order to include a gear body in the KISSsoft calculations. This determines the deformation of the gear body under load. The deformation has an impact on the load distribution in the loaded tooth contact analysis.

The consideration of the gear body is based on the use of a reduced stiffness matrix, as calculated by the Finite Element Method (FEM), either directly derived in this calculation, or imported through a file. A rim may be included in the calculation, connecting the main gear body with the gear teeth. The tooth rim and the gear body may have different materials.

15.17.1 Definition option

There are three different ways to define the geometry of the gear body:

1. **Manual definition:** You define some points in the gear body cross section, using the option **Manual definition** and the provided input table. Fillet radii may be defined for each point.
2. **Import STEP file:** You import the gear body geometry from a step file. Make sure that the imported data are being correctly positioned and aligned using the provided information sketch.
3. **Import old k016 file (until version 2022):** In order to be able to use gear bodies defined in KISSsoft version 2022 or earlier (K016 module), you should define the path to a K16 file.

15.17.2 Calculation settings

The material of the gear body and of the gear are the same unless a rim has been defined. In this case, the material of the rim and of the gear are the same, but the material of the gear body may be different. The gear rim thickness is either defined directly in the **Module** tab, or it is calculated based on the gear diameters defined in the **Basic data > Details** tab. The force components are taken from the basic calculation and are shown for reference, together with other input data taken from the user interface (rim thickness, width, etc.). You can also select to defeature the geometry defined in the table (i.e. ignoring fillet radii), resulting in a faster FE mesh generation and a structured mesh. The stiffness matrix of the gear body is always calculated, but may also be saved to a file, if the respective checkbox is activated. Finally, different FE mesh densities may be selected. Using the **Preview** button, you can preview the 3D model of the gear body and the generated FE mesh.

15.17.3 Results and graphics

Once the calculation is finished, several important results are displayed in the results window. The deformation of the FEM nodes on the line of load application can be displayed under **Graphics > FEM > 2D Results (Gear body)**. The FEM post-processing graphic is located under **Graphics > FEM > FEM results**. A detailed report showing all results is also provided. The results are displayed per gear body and/or rim based on the current module.

15.18 Rough sizing macrogeometry

KISSsoft has very powerful sizing functions, which are described in this section and those that follow. The process for sizing a gear stage, from start to end, involves rough sizing macrogeometry, fine sizing macrogeometry and, finally, fine sizing modifications (microgeometry).

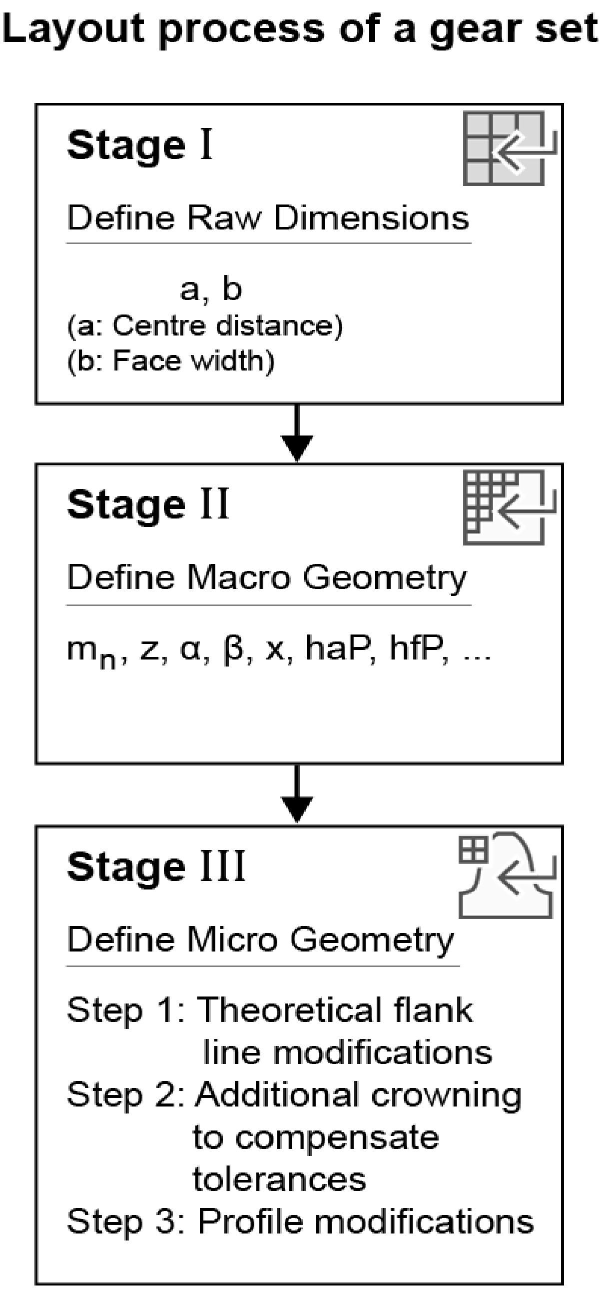


Figure 15.90: Phases involved in sizing gears

Rough sizing proposes possible gear teeth configurations based on the data entered for the ratio and the load. The purpose of rough sizing is to ascertain the possible range of suitable solutions, all sized for the specified torque, according to all the specified required safeties. The total weight is possibly the most important output, because this can be regarded as roughly proportional to the manufacturing cost. The weight of the different solutions usually varies by a factor of up to 3!

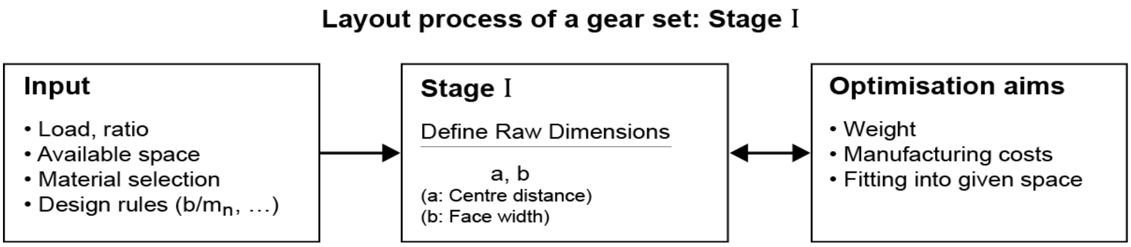


Figure 15.91: Gear sizing, Phase 1

To call the rough sizing function, either go to the **Calculation** menu and select the **Rough sizing** option or click on the icon in the Tool bar.

At present, you can apply this to cylindrical gear pairs with internal or external teeth, and to planetary gears. The nominal ratio is the most important input parameter. For an internal gear pair, the ratio must be entered as a negative value in the **Geometry** area. For planetary stages the nominal ratio must be > 2.0.

The operating data (power, speed, etc.) is taken from the KISSsoft main window (and can be changed there if required). You can also specify
a helix angle or a required overlap ratio (e.g. ε_β= 1.0).

Some important design parameters for gear stages can be set (ratios b/mn, B/d1 and b/a). All three parameters are always taken into account during rough sizing. Since these parameters may restrict each other, you can specify which parameter is to be prioritized by selecting the appropriate button.

The facewidth to normal module ratio is a characteristic value used to achieve reasonable dimensions for gear stages. If the gears involved are too narrow, the axial stiffness of the teeth cannot be guaranteed. For this reason, b/mn should be greater than 6 (see Niemann, Table 22.1/7 [7]).

If the gears are too wide, it is essential that the load is spread evenly across the entire facewidth. Depending on the gear type and accuracy grade, b/mn should be less than 15...40 (see Niemann, Table 22.1/10 [7]).

The facewidth to reference diameter ratio is a characteristic value used to achieve sensible dimensions for gear stages. Depending on what type of heat treatment is involved, this ratio should be less than 0.8 to 1.6 (see Niemann, Table 22.1/5 [7]).

The facewidth to center distance ratio is a characteristic value used to design standard gear units of modular construction. Depending on the stiffness of the housing, this value should be smaller than 0.3 to 0.5 (see Niemann, Table 22.1/6 [7]).

Click on the **Calculate** button to display a list of suggested values that you can use to set the parameters for your gears.

The parameters in the results table are displayed with formula symbols which match the formula symbols used in the rest of the interface, and in the reports. Hover the mouse pointer over a formula symbol in the table to display a description of it in plain text. Right-click on the results table to open a dialog, in which you can either hide or display additional parameters.

Rough sizing automatically finds the most important tooth parameters (center distance, module, number of teeth, width) for the required power and ratio, using the strength calculation according to the selected calculation standard. Dimensioning is performed according to predefined minimum safeties (see chapter 15.22.5, Safety factors).

To specify the intervals for the ratios b/m_n-, b/a, b/d, select the **Calculation** menu option in the **Settings -> Sizings** menu. (see chapter 15.22.2, Sizings)

The program displays a number of different solutions which you can select. You can then use them to perform an optimization in fine sizing. The window remains open, to enable you to select more solutions. You will find more detailed information about fine sizing in section 15.19.

The most important result of this sizing process is that it enables you to define the achievable center distance ranges and module ranges, as well as the facewidth. You can then decide how much space is required for the gear unit itself.

If you select the DIN 3990 calculation method, the standard modules specified in DIN 780 Series I and II are used. If you select a calculation method according to AGMA and enter the module as "Diametral Pitch", the module series according to ISO 54 is converted into diametral pitch and then applied. The module series specified in ISO 54 Series I and II are used for all other calculation methods. As ISO 54 Series I and II only go up to a standard module m =1, this standard module series for m < 1 has been extended by the addition of values from DIN 780.

Solutions with a number between 1 and 5 show solutions with any module. Solutions from 6 onwards show solutions with standardized modules according to DIN 780 (series of modules for gears).

- Number 1: Solution with the most exact ratio
- Number 2: Solution with the greatest center distance
- Number 3: Solution with the smallest center distance
- Number 4: Solution with the largest module
- Number 5: Solution with the smallest module

You can fix the center distance for special cases. However, in these cases, you must remember that the program's sizing options are not exhaustive, and fine sizing represents a better alternative.

Sizing of strength for a planetary gear

When performing rough sizing for planetary stages, it is assumed that the rim is static. If the rim rotates, you must change the speed after sizing.

- **Proposal of number of teeth according to Niemann**

  Table of standard numbers of pinion teeth according to Niemann [7], Table 22.1/8.

Ratio *u*	**1**	**2**	**4**	**8**
Through hardened or hardened				
Counter-through hardened to 230 HB	32..60	29..55	25..50	22..45
Over 300 HB	30..50	27..45	23..40	20..35
Cast iron	26..45	23..40	21..35	18..30
Nitrided	24..40	21..35	19..31	16..26
case-hardened	21..32	19..29	16..25	14..22

Click the Sizing button to transfer these values from the program automatically.

15.19 Fine sizing macrogeometry

The fine sizing function is one of KISSsoft's most powerful tools. It generates and displays all the possible geometry variants (module, number of teeth, etc.) for the specified facewidth and center distance (the gear rim diameter is usually specified for planetary stages and the center distance varied accordingly). The solutions are displayed as graphics, so you can easily see the best possible macrogeometric variant for your purpose.

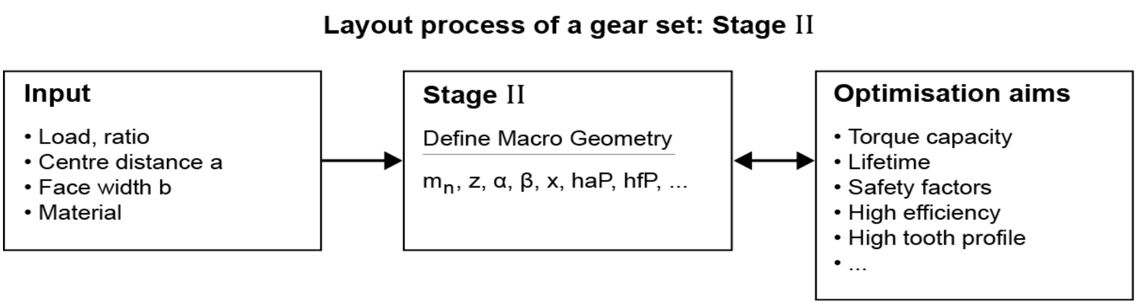


Figure 14: **Gear sizing, Phase II**

To call the **Fine sizing** function, either go to the **Calculation** menu and select the **Fine sizing** option or click the icon in the tool bar.

If you input a nominal ratio, a center distance, and intervals for the module, facewidth and helix angle, as well as the pressure angle, KISSsoft calculates and displays suggestions for the number of teeth, module, helix angle and profile shift. It also shows the deviation from the nominal ratio, the specific sliding and the contact ratios. This module can also be used to size planetary stages, three and four gears trains.

The input of the initial parameters (intervals) can be done through several options available in the drop-down menu. The available options are:

1. Minimum, maximum, step size
2. Minimum, maximum, number of steps
3. Minimum, step size, number of steps
4. Minimum, step size in %, number of steps
5. Nominal, +- step size in %, number of steps

The number of steps can be a rational number.

All the variants found by this process can be evaluated by a wide range of different criteria (accuracy of ratio, weight, strength, tooth contact stiffness deviation etc.)

Depending on your requirements, limits can also be set on the most important parameters (tip circle, root circle, minimum number of teeth, tolerated undercut etc.). In addition to creating text reports detailing the solutions and the summary, the summary can also be displayed as a graphic.

The facewidth appears in the input screen, where you can modify it if required.

Sizing of strength for a planetary gear

When performing rough sizing for planetary stages, it is assumed that the rim is static. If the rim rotates, you must change the speed after sizing.

15.19.1 Necessary entries in the input window

Before you start the fine sizing process, you must enter the following data correctly in the **Basic data** or **Geometry** and **Strength** standard tabs to ensure the calculation returns the results you require.

Geometry:

- Reference profile
- Number of idler gears/planets (to configure planetary gear stage, 3-gear and 4-gear)

Strength:

- Materials
- Power/Speed
- Application factor

- Required service life
- Lubrication

15.19.2 Conditions I

You can predefine the module range for cylindrical gears. If the module flag is set, you can predefine the increments. If the module flag is not set, you can only use modules from the standard module list.

For cylindrical gear pairs, you can either input a fixed center distance (the usual approach) or specify an interval for the center distance. To do this, click the checkbox to the right of the Center distance input fields.

If planetary gear units are involved, you can either perform the calculation with a predefined center distance or with a predefined V-circle diameter (dp = d+2*x*mn) for the internal gear. In practice, it is usually the internal gear diameter that is fixed (gear size remains the same) and the center distance that is varied. In this case, we recommend to first input the required output reduction and the V-circle diameter, then click the Sizing button for the center distance.

Note:
You should check the center distance interval after you change the reference circle or select a variable center distance. You may then need to repeat the sizing process.

15.19.2.1 Limiting the tip diameter

Solutions whose tip circle exceeds the specified value are rejected. Solutions for internal teeth are rejected if $|da| < |da_{limit}|$. If you do not want to limit the tip value, you can input either 0 or 10^{10} .

However, the following problem prevents this option being used sensibly in practice: If a gear is to be installed in an existing housing, it is critical that it does not touch the walls of the housing.

15.19.2.2 Limiting the root diameter

Solutions whose root circle falls below the specified value are rejected. Solutions for internal teeth are rejected if $|df| > |df_{limit}|$. If you do not want to limit the root diameter, you can input 0.

However, the following problem prevents this option being used sensibly in practice: If a gear is mounted on rolling bearings in a speed change gear unit, you must guarantee a minimum thickness of material between the bore and the root circle.

15.19.2.3 Maximum no. of solutions

Proposal: 50 ...250

If the program finds more than the specified number of solutions, you see a warning message and an appropriate note is entered in the report.

► **Note**

You should only perform a final evaluation after all the possible solutions have been displayed. Otherwise, you run the risk that the optimum solution will not be displayed.

15.19.2.4 Limiting the number of teeth

You should not normally use this option and it is therefore inactive by default. However, by clicking the individual checkboxes, you can still fix this parameter. A useful application for this option is when for sizing a planetary gear which has already been modified to fit inside a predefined internal gear. In this case, the module, the number of teeth and the profile shift are predefined for gear 3.

15.19.3 Conditions II

Here, the reference profile h^*_{aP} of the individual gears can be varied step-by-step The dedendum h^*_{fP} is determined via the required tip clearance to the counter gear ($h^*_{fP}2$ - H^*_{aP} 1). If this value is not changed, the tip clearance value in every variant will be the same as the value entered in the Basic tab.
You can also specify that the maximum possible tip rounding radius, ϱ^*_{fP} , is always set automatically.

15.19.4 Conditions III

You can specify other essential functions in the **Conditions III** tab.

1. **Show values of KISSsoft main calculation as additional variant with number 0**
   The toothing data in the KISSsoft Basic tab can also be displayed as a variant with the number 0 (table and graphic). However, the data at the start of the fine sizing process must be consistent before this can happen. This option can either be enabled or disabled. When you enable this option, you must restart the fine sizing process so the variant can also be displayed.
2. **Calculate geometry only**
   If you click this checkbox, no strength calculation is performed.
3. **Strength calculation with load spectrum**
   Before you can perform calculations with a load spectrum, you must specify a load spectrum in the KISSsoft main window before you start the fine sizing process and run the calculation (to ensure the data is consistent). In this case, when you start the fine sizing process, you are prompted to confirm that you want to perform the calculation

with a load spectrum. The flag in the window only shows whether (or not) a load spectrum is being used. You cannot change this.

4. **Permit undercut**
   If this option is selected, solutions with undercut are not rejected.
5. **Reject results with specific sliding higher than 3**
   Specific sliding should not usually be greater than 3.
6. **Reject solutions with non-hunting tooth**
   If this option is selected, solutions without a "non-hunting tooth" are rejected.
7. **Consider minimum tooth thickness**
   If this option is selected, solutions with a tip tooth thickness that is less than the predefined minimum tooth thickness (see **Calculation > Settings General**) are rejected.
8. **Vary gear quality**
   If this option is selected, solutions with varying gear quality are displayed. The varying quality is set up for each gear and it varies in a range from ± nominal value. The standard for gear quality variation is linked to the settings in the **Basic data** tab. Therefore, the quality variation can be done according to ISO 1328, DIN ISO 1328, AGMA 2015, DIN 3961-3963, JIS B 1702, AGMA 2000 and GOST 1643-81. The maximum range for the variation is ± 4.
9. **Allow small geometry errors**
   Minor meshing interference and similar geometry errors will now be tolerated when the system calculates variants! You can make separate settings to take into account the undercut and the minimum tooth thickness at the tip (see points 2 and 4). You must set this option if the program finds solutions where the number of teeth is less than 7, or in other exceptional situations. We do not recommend you set this option in any other situation!

► **Note:**

In these situations, you must also change the minimum number of teeth accordingly (see point 12).

10. **Suppress integer ratios**
    If this option is selected, results with a whole number gear ratio will be rejected.
11. **List of cutters for reference profile**
    Instead of using the predefined reference profile, you can use a list of hobbing cutters for fine sizing. In this case, the calculation is performed for every default cutter in the given module and pressure angle range and the tool is displayed in the results list. The same hobbing cutter is used for each gear. Internal toothings are not affected by this setting.

12. **Sizing of deep tooth forms**
    Special reference profiles with larger addendums and dedendums are used for deep tooth forms. This sizing function, ε_α=$\varepsilon_{\alpha\ target}$, calculates the necessary reference profile on the basis of the required transverse contact ratio $\varepsilon_{\alpha\ target}$. If this function is activated in fine sizing, the reference profile for every solution is calculated so that the exact required transverse contact ratio is achieved. As a result, only those solutions that have exactly the required transverse contact ratio are displayed. However, the ε_α>= $\varepsilon_{\alpha\ target}$ function only changes the reference profile when the transverse contact ratio calculated with the original reference profile results in a transverse contact ratio that is smaller than $\varepsilon_{\alpha\ target}$.

► **Note:**

In both cases (εα= εα $_{target}$ and ε_α>= ε_α target), you must ensure that automatic tip alteration k*m_n is not performed (and is set to zero). Both the reference profile h*$_{aP}$ value and the tip alteration k*m_n have the same effect on the tip circle, which is why only one of these two values should be changed.

13. **Transmission Error**
    If the **Contact analysis** option is selected, contact analysis is performed for every variant. If the **Contact analysis** and **Profile modification sizing** option is selected, the profile modification length and value are determined automatically according to the settings made for the modification method. Click the Plus button to display the profile modification settings window. The modification method takes into account the objective (for high load capacity gears or smooth meshing), tip and/or root relief, length (short or long), and type (linear, arc, progressive or linear with transition radius). It is important to note that the transmission error can be minimized only for one load, and the partial load for sizing should be set correctly according to the applied load level.
    During the contact analysis calculation, default settings are used to prevent the calculation generating an inaccurate result. Click the Plus button to define the calculation accuracy. We recommend you select "medium" or "low" to reduce the calculation time. Set the coefficient of friction in the **Contact analysis** tab. As a consequence, the transmission error in fine sizing might not be exactly the same as the error determined in contact analysis, depending on which settings have been selected.

    The default values are as follows:
    Calculation for: right flank
    Torque for gear A: not considered
    Torque for gear B: not considered
    Partial load range for calculation: 100 %
    Center distance: Average center distance allowance
    Pitch error (pitch variation): 0 mm

Deviation error of axis: 0 mm
Inclination error of axis: 0 mm

The results list shows:
Transmission error (PPTE)
Average wear on the tooth flank (Δw_{n1}, Δw_{n2})
Maximum flash temperature (θ_{flm})
Variation in bearing forces (ΔF)

The calculation time increases significantly if the calculation with transmission option is selected. We therefore recommend you limit the number of results before starting the calculation.

14. **Reject solutions with lower than required safety factors**
Variants which do not meet the predefined minimum safety levels (see **Calculation> Settings> Required safeties**) will be rejected.

► **Note:**

Variants with insufficient safety against scuffing will not be rejected.

15. **Sizing of profile shift coefficient x1**
Fine sizing usually generates 3 or 4 variants in which only the profile shift is different. In this case, the profile shift x1 is changed in increments of 0.1. Here you can specify the criterion used to determine the largest profile shift used, x1.
16. **Minimum number of teeth z_{min}**
Practical values range for the minimum number of teeth:

For helical gear:7 ...9
For spur gear:10 ...12
Click the Sizing button to display a suggested value for the minimum number of teeth.

► **Note:**

If you want to find solutions in which the number of teeth is less than 7, you must first select the **Allow small geometry errors** option.

17. **Minimum distance between root form diameter and active root diameter d_{Nf} - d_{Ff}**
Meshing interferences occur if the active root circle is less than the root form circle. Here, you can specify a minimum value for the distance between the active root circle and the root form circle, i.e. between active and manufactured involutes. The input value is the minimum difference between the two diameters.

18. **Minimum distance between root form circle and base circle d_{Ff} - d_b**
    If the start of the manufactured involute is closer to the base circle, this will cause greater wear on a tool during the manufacturing process. Here, you can specify a minimum value for the distance between the root form circle and the base circle. The input value is the minimum difference between the two diameters.

15.19.5 Results

Click the **Report** button to open the editor and display a list of the best results. A brief description of the criteria used to evaluate the best variants is given here. Please note that these criteria are not relevant to every case, and only need to be queried in particular applications!

1. Evaluation of the variants for the accuracy of gear ratio:
   The difference between the actual gear ratio and the required gear ratio is evaluated here.
2. Weight: this is an indicator for the manufacturing price
3. Specific sliding: maximum value
4. Sliding velocity: maximum value
5. Ratio AC/AE
   AC: length of path of contact from meshing point to pitch point
   AE: total length of path of contact
   "Pushing" sliding occurs in the AC area of contact (the sliding velocity of the driving gear is greater than that of the driven gear). As this area is critical for unlubricated plastic gears, the AC/AE relationship should be as small as possible in this case.
6. Evaluate variants for vibrations:
   The variation in the total stiffness of the meshing is evaluated. The lower the variation, the better.
   The calculation is based on empirical formulae unless the **Calculate mesh stiffness** option is set in "Conditions II".
7. Evaluate variants for strength:
   Evaluate root and flank safety with regard to required safety. Although safeties of less than the required safety are given a very negative evaluation, large safety margins above the required safety have very little influence.
8. Transmission error (PPTE)
   Transmission error is displayed if the corresponding option is set in "Conditions II".
9. Evaluation **Summary**:
   The **Summary** evaluation weights each component to form a total evaluation coefficient. To set the weighting of individual components, select **Calculation > Settings > Evaluation**. This weighting depends to a great extent on which solution you

require, for example, whether you want a solution that is optimized for noise reduction or strength.

► **Note**

The Rough sizing section includes a complete list of all the available parameters (see chapter 15.18, Rough sizing macrogeometry). You will find information about noise optimization in [35].

15.19.6 Graphics

The graphic in the Fine Sizing window gives you a quick overview of the number of solutions. Three parameters can be displayed simultaneously. You can change them in the selection lists. In addition to the two axes, the third parameter is displayed as a color.

15.19.7 Geometry-Fine Sizing for 3 gears

Center distances cannot be changed in the fine sizing process. The center distances entered in the **Basic data** tab are used in this calculation.

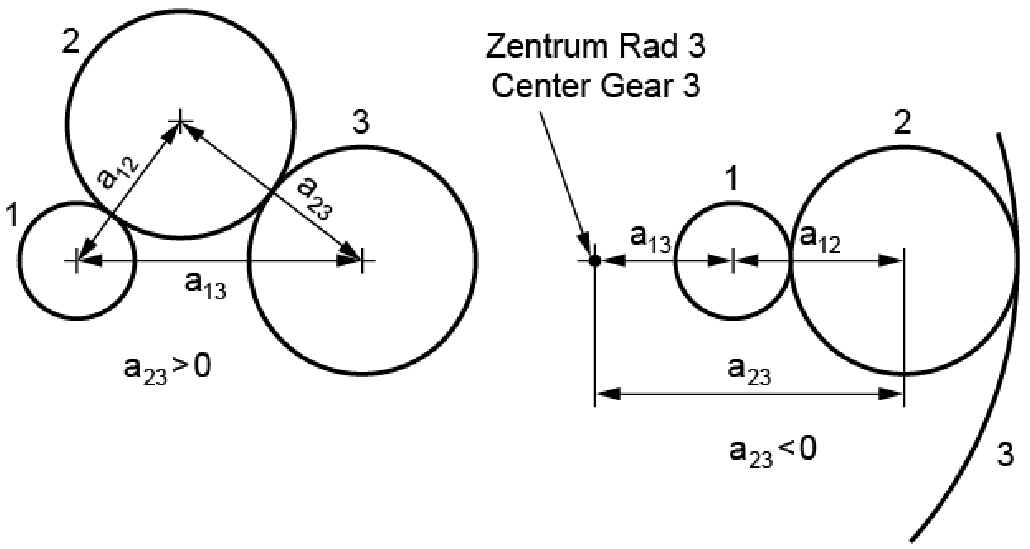


Figure 15.92: Defining center distances

15.19.8 Geometry-Fine Sizing for 4 gears

Center distances cannot be changed in the fine sizing process. The center distances entered in the **Basic data** tab are used in this calculation.

However, if gear 4 is an internal toothing, you can also select the **double planetary stage** option. If you select the **double planetary stage** option, the internal gear's V-circle diameter is also checked and the required output reduction is z3/z2. In this case, all center distances are varied automatically, and all possible solutions are displayed. Values for αM213, clearance13 and clearance24 are displayed in the results.

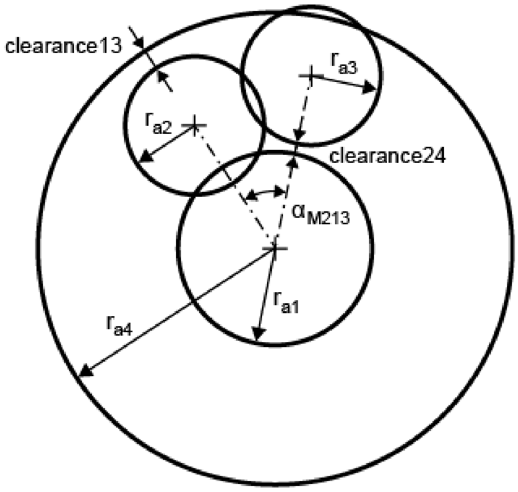


Figure 15.93: Figure: 4-gear configuration

15.19.9 Additional strength calculation of all variants

The KISSsoft system also calculates the strength (tooth root, flank and scuffing) for every variant and displays these values in a list. This option can be used for cylindrical gear pairs, planetary stages and cylindrical gear stages that have an idler gear. If you click on the **Only calculate geometry** checkbox in the **Conditions II** tab, the calculation does not include tooth safeties.

15.20 Fine sizing modifications (microgeometry)

Sizing the profile and flank line modifications is the last and most complex phase in sizing a gear. This modification variant generator can save you time and effort by calculating the optimal modifications quickly and directly.

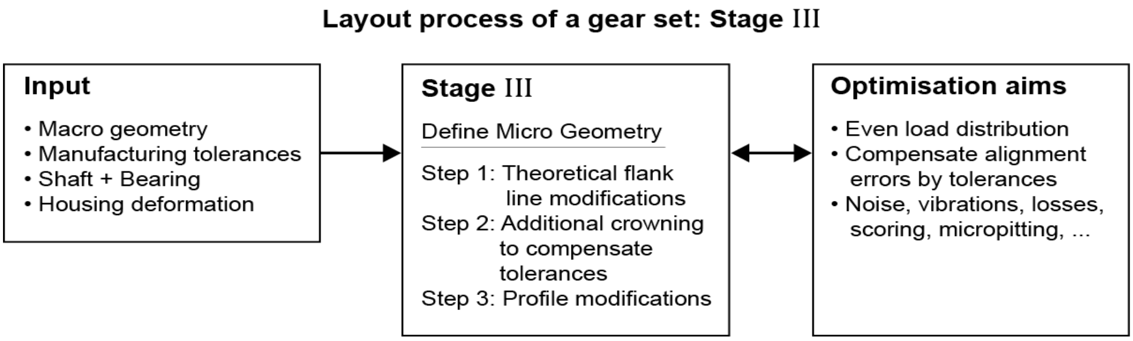


Figure 14.: **Gear sizing, Phase III**

To call the **Fine sizing modifications** function, select the icon in the tool bar, select **Calculation** in the menu and then select **Fine sizing modifications**.

If you call the optimization functions without opening the **Contact analysis** tab, the default settings in the tab will be used in the calculation.

15.20.1 Conditions

Conditions

The **Conditions** tab is where you define general conditions that are valid for every solution.

Select the **Take load spectrum into account** option to take into account the load spectrum previously defined in the strength calculation module.

If the **Excluding contact analysis, only service life calculation using $K_{H\beta}$ according to ISO 6336-1, Annex E** option is enabled, the solution range is only performed using the service life and $K_{H\beta}$ calculation.

Select the **Cross-vary value, Factor 1, Factor 2** option to run an additional variation of the coefficients with the modification value.

Select the **Calculate solution without modifications** option to skip the calculation of the solution without any modifications.

Select the **Calculate shaft deformation just once for each partial load (calculation with basic modification only)** option to calculate the diagrams of bending once for the shafts, for each partial load, and not for every modification configuration. This option makes the calculation slightly less accurate. However, if you are performing calculations with just a few load cases, but a large number of modification configurations, selecting this option can significantly speed up the sizing process.

Every modification can be calculated for a larger partial load range. You can set the necessary settings in the **Partial load range** section. The torque range used for contact analysis and the face load factor calculation result is also output here.

Modifications

The **Modifications** tab is the main tab for fine sizing modifications. In it, you define new **Modification types**, such as tip relief, root relief, etc. for the sizing. (You will find a complete list at (see chapter 15.7, Modifications)). If the **Synchro ID** column contains a different value than for the specific line number (ID), the modification is synchronized with the modification you have selected, and all the variants are performed with the same number of steps. In the **Gear** column, you select a gear to which the modification is to be applied. Use the **Flank** option to specify the flank on which the modification is to be applied. If **active** is selected in the **Status** column, the modification will be varied while it is being sized. If **static** is set, a predefined value will be used for the modification without being changed in any way. If **inactive** is selected, the modification will be ignored and only displayed in the Graphic window. You can enter the **Number of steps** per modification to set the number of steps between the minimum and maximum value, starting from the minimum value. You can enter the minimum and maximum value for the **Value, Factor 1** and **Value, Factor 2**, limiting the inputs for the modification sizing.

15.20.2 Results

All the solutions are displayed in the **Results** tab. You can then select the solution that best suits your requirements. Click on Accept or double-click on the solution to transfer its data to the **Modifications** tab.

The most important results in the result overview:

- No: solution ID. You can use this ID to search for more details about the results in the reports.
- w_t: The partial load of the calculated solution in % (depending on the number of iteration steps specified in the "Number of steps for partial load" field), e.g. 50% partial load relative to the nominal load defined in the **Basic data** tab.
- H: The minimum service life achieved by the gear pair in hours.
- PPTE: Transmission error amplitude of the driven gear along the driven gear's path of contact.
- rel. PPTE: Relative amplitude of the transmission error/angle of rotation error relative to the uncorrected toothing.
- PPFE: Amplitude of the excitation force error of the driven gear along the driven gear's path of contact.
- ε_a: Transverse contact ratio under load
- ε_β: Overlap ratio under load
- ε_v: Total contact ratio under load
- $K_{H\beta}$: Face load factor (If the calculation is performed with load spectra, only the face load factor for the last load bin is ever displayed.)
- σ_{Hmax}: Maximum Hertzian pressure that occurs in the gear teeth
- S_λ: Safety against micropitting as specified in ISO TR 6336-22 Method A
- η: efficiency
- $\Delta W_{nA/B}$: Wear on gear A/B
- ΔT: Torque amplitude of the driven gear
- and others...

15.20.3 Graphic I

All the solutions are displayed as graphics in the **Graphic I** tab. You can display a maximum of up to 10 graphics at the same time. Each graphic can process its own dataset.

Select the required partial load from the partial load selection list (red is the largest partial load, blue is the smallest partial load).

15.20.4 Graphic II

This graphic gives you a quick overview of the number of solutions. Three parameters can be displayed simultaneously. You can change them in the selection lists. In addition to the two axes, the third parameter is displayed as a color.

15.20.5 Report

The results are documented in three different, detailed reports. We suggest you begin by looking at the summary report which gives a broad overview. The other two types of report are considerably longer, and also document intermediate results.

The main calculation performs a series of contact analysis calculations. Each one has a different combination of modifications with all the intermediate steps, and for each wt% level. A contact analysis without modifications is also performed.

A frequently asked question:

How can the length of the modification and the relief Ca be varied independently of each other, to find out which length/value combination gives the best result?

Reply: For example, if the tip relief Ca is to be varied between 100 and 220 µm, and the length factor is to be varied between 0.78 and 1.56, to determine all possible value/value combinations.

15.21 Measurement grid

A measurement grid report is available for cylindrical and bevel gears (select **Calculations > Measurement grid**).

Setting	**Description**
Gear	Setting the gear for calculating the measurement grid.
Measurement grid area	Setting the measurement array for the calculation.

	0: Tooth flank 1: Fillet surface
Measurement machine	Setting the report format for a particular measurement machine 0: Klingelnberg 1: Gleason
Number of columns	Setting the number of columns across the facewidth (>=3) Number of columns (number of sections – 2) for parasolid settings, because the sections should not include both ends of a tooth.
Number of rows	Setting the number of rows across the tooth profile (>=3)
Distance from root form circle	Distance from root form circle. Default value 0.1* normal module (middle).
Distance from tooth tip	Distance from tooth tip. Default value 0.1* normal module (middle).
Distance from side I/toe	Distance from side I for cylindrical gears, distance from toe for bevel gears. Default value is (facewidth)/(number of columns + 1).
Distance from side II/heel	Distance from side II for cylindrical gears, distance from heel for bevel gears. Default value is (facewidth)/(number of columns + 1).

The report includes the coordinates and the normal vector of the grid points in the format [XP YP ZP XN YN ZN]. The reference point and the tooth thickness angle are displayed in the report header.

The reference coordinates of the data may differ according to which type of measuring machine is used. For example, the following convention applies to Klingelnberg machines.

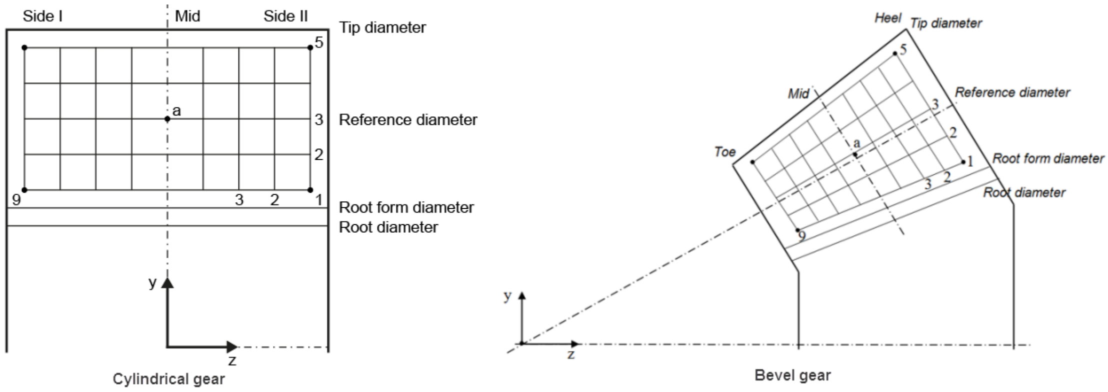


Figure 15.94: Measurement grid for cylindrical gears and bevel gears for Klingelnberg machines

The sequence of index numbers for points and sections is defined according to ISO/TR 10064-6, i.e. the index for lines runs from bottom to top, and the index for columns runs from side II (heel) to side I (toe).

15.22 Settings

To open the **Module specific settings** window, select the **Calculation** menu and then click on the **Settings** menu option. A huge number of these settings are available for cylindrical gear calculations. You can activate the widest variety of possible special functions. Normally there is no need to change the settings.

15.22.1 General

15.22.1.1 Input of normal diametral pitch instead of normal module

If you select this option, the **Normal module** input field in the **Basic data** or **Geometry** input window is replaced by the Diametral Pitch input field.

15.22.1.2 Input of number of teeth with decimal places

In KISSsoft, you can perform a calculation with a fractional number for the number of teeth. We recommend you use this option for arcs of circle or for non-symmetrical teeth.

15.22.1.3 Don’t abort if geometry errors occur

If you select this option, the software will continue the calculation even if severe geometry errors, such as pointed teeth, meshing interference etc. occur. This option enables you to continue the calculation in critical cases, however, you should then use its results with extreme caution!

15.22.1.4 Generate GDE format (VDI 2610) and generate detailed data for profile diagram, flank line diagram and tooth form

Click on **Generate GDE format (VDI 2610)** and **Generate detailed data for profile diagram, flank line diagram and tooth form** to calculate the individual points used to generate the profile diagram and the flank line diagram. This option also calculates the points for the particular tooth form. Click on **Report > Special reports > Detailed data for profile diagram, flank line diagram and tooth form** to display a report documenting the points for the diagrams and the tooth form points. Click on **File > Export > GDE** to generate the GDE output file. This creates a "GEAR DATA EXCHANGE" xml file (in GDE format).

However, if several gears, or even several gear pairs, are to be mapped, first gear 1, then gear 2, and then the <mating_data> section, for pair 1, is written to the file, followed by the same data for pair 2 (if present), etc., until all the gears have been added to the file.

The following modifications, which are present in KISSsoft, are supported in GDE format:

- Tip and root relief, linear
- Profile crowning, roll length-centered
- Flank line crowning
- Twist
- Pressure angle modification
- Helix angle modification (arc)
- End relief, linear

Versions 3.1 and 3.2 are currently supported.

15.22.1.5 Generate GAMA code

Click on "Generate GAMA code" to export gear geometry data in GAMA format. Click on File -> Export… -> GAMA. A GAMA file which can be imported into the Gleason GAMA software is then generated.
The GAMA code currently supports the following parameters, which are present in KISSsoft:

- Normal module
- Number of teeth
- Helix angle
- Facewidth
- Profile shift
- Root diameter
- Start of the involute
- Reference diameter
- Normal tooth thickness, arc

15.22.1.6 Check if mounting of planets is possible

Planets are usually arranged with a constant pitch on the planet carrier (if 3 planets are involved, they will each be at 120°, etc.). In these situations, the number of teeth must fulfill certain conditions to ensure the planets can be mounted correctly. KISSsoft performs the check if this checkbox is selected.

15.22.1.7 Minimum distance between 2 planets

You can specify a minimum distance between the tip circles of two planets in this input field. If this minimum distance is not reached, the system displays a warning message.

15.22.1.8 Factor for minimum tooth thickness at tip

For manufacturing reasons, a specific minimum tooth thickness at the tooth tip must always be achieved. The minimum tooth thickness is: module x coefficient. As defined in DIN 3960, the coefficient is usually 0.2.

15.22.1.9 Factor for minimum tip clearance

The tip clearance is the distance between one gear's tip circle and the root circle of the other gear in the pair. You can specify a minimum tip clearance. If this clearance (taking into account the tip circle and root circle allowances) is not reached, the software displays a warning message.

15.22.1.10 Properties which depend on material combinations

The coefficient of friction, wear coefficients and heat transfer coefficients, which significantly depend on the selected material combination, can now be defined in separate .dat files. The material-dependent properties are defined in the CoefficientOfFriction.dat, WearFactors.dat and HeatTransferCoefficient.dat files, stored in the kiss/dat directory.

To enable these options, select **Module specific settings** > **Plastic** and select the properties for which material-dependent properties should be read from a file. If the options are selected, the values for the coefficients of friction and the wear coefficient (defined in the individual material .dat files) will be overwritten by the values in the material pairing .dat files. The values defined for the heat transfer coefficient in VDI 2736-2 will be overwritten by the values defined in the HeatTransferCoefficient.dat file.

The properties for material pairings are displayed in the figure below.
Material pairings are defined as "material ID number_material ID number_". A .dat file contains information used to describe a TABLE FUNCTION in different ways. You will find the material ID number in the **ID column**, in the Database tool: select Database tool > **Material of gears**. The function can also be used for user-defined materials.

```
 -- E-modulus [N/mm2] for PA66 (data for BASF Ultramid A3K, dry)
:TABLE FUNCTION ElastizitatsModul_dry
      INPUT X ZahnTempFuss TREAT LINEAR
DATA
    -40    -20      0     23     40     60     80    100    120    150
   4480   4170   3970   3350   3160   2820   1210   800    700    640
END

 -- Ultimate tensile strength sigFb [N/mm2] (data for BASF Ultramid A3K, dry)
:TABLE FUNCTION SigFb_dry
      INPUT X ZahnTempFuss TREAT LINEAR
DATA
    -40    -20      0     23     40     60     80    100     120     150
    110    107    103     85     73     50     23     21      18      15
END

-- Tensile strength sigFs [N/mm2] at 0.5% strain (data for BASF Ultramid A3K, dry)
:TABLE FUNCTION SigFs_dry
      INPUT X ZahnTempFuss TREAT LINEAR
DATA
    -40    -20      0     23     40     60     80    100     120     150
     92     90     85     80     68     46     17     15      13      10
END

-- Material data for E-modulus, sigFb and SigFs available also for dry material
-- Should be set only for PA6 and PA66 materials
-- Values: 0-not required, 1-available
:TABLE FUNCTION DryDataGeneralProperties
      INPUT X None TREAT LINEAR
DATA
        0
        1
END

-- Material data for tooth flank/root strength available for dry material (not conditioned)
-- Should be set only for PA6 and PA66 materials
-- Values: 0-not required, 1-required and available for dry material, 2-required and available for conditioned material
:TABLE FUNCTION DryDataGearFatigue
      INPUT X None TREAT LINEAR
DATA
        0
        2
END
```

Figure 15.95: Defining the properties for material combinations in the appropriate DAT file

15.22.1.11 Consider dry material properties

If materials (such as PA6, PA66, etc.) are used, whose mechanical properties (Young's modulus, tensile stresses, tooth root and flank fatigue) change significantly when they absorb water, activate this option to calculate the mechanical properties of dry materials. If data is already available for dry materials, it can be added to the corresponding DAT material file (see Figure 15.95). For example, you can use material file Z014-PA66_VDI2736.DAT.

15.22.1.12 Calculation with operating center distance and profile shift according to manufacture

Cylindrical gear geometry according to DIN 3960 is based on the calculation of the gear teeth (which is theoretically without clearance). This enables the sum of the profile shifts with the specified center distance to be calculated.

Use this option to enter the profile shifts independently of the center distance. This is very useful as it provides a way to check the limits of a toothing (clearance, contact ratio etc.) if there are major variations in the center distance (e.g. in the case of center distance tolerance fields).

15.22.1.13 Allow large profile shift

Use this option to significantly increase the range of profile shifts that can be used ($-1.2 \leq x^* \leq +1.5$). This is very useful for special cases. We recommend you use it for: cylindrical gears, bevel gears, worms, crossed helical gears.

15.22.1.14 Maintain tip circle when changing profile shift

In KISSsoft, the reference profile is usually maintained and the tip and root circle are modified to suit. If you select this option, the tip circle is maintained, and the reference profile is modified to match it when the profile shift is changed. The tip circle value is maintained as long as the number of teeth and the transverse module stay the same. This option is only available if the Center distance checkbox is selected.

15.22.1.15 Maintain root circle when changing profile shift

In KISSsoft, the reference profile is usually maintained and the tip and root circle are modified to suit. If you select this option, the root circle is maintained and the reference profile is modified to match it when the profile shift is changed. The root circle value is maintained as long as the number of teeth and the transverse module stay the same. This option is only available if the Center distance checkbox is selected.

15.22.2 Sizings

15.22.2.1 Required transverse contact ratio

You can specify the required transverse contact ratio for sizing deep tooth forms.

15.22.2.2 Sizing of gear geometry

The Coefficient for active tip - form diameter difference and Coefficient for tip clearance (also known as "form clearance") are settings that are needed for the functions used to size the reference profile, in the "Reference profile" tab and in Fine sizing. For example, you may need to adjust the reference profile to achieve an exact form clearance and to ensure that the minimum tip clearance is also exceeded.

15.22.3 Calculations

15.22.3.1 Calculate form diameters from tooth form

The tooth form calculation simulates the manufacturing process. In doing so it calculates the effective undercut in the tooth root. Select the **Calculate form diameter from tooth form** option to calculate the tooth form in every calculation run, define any undercut that is present and include it in the calculation. This is then used to calculate the transverse contact ratio and the root and tip form circles (generated diameters). As KISSSOFT already automatically calculates the form diameters correctly for protuberance and undercut, we recommend you no longer use this option.

You can select whether the root form circle, the tip form circle, or both these values, are to be included in the tooth form. Up to now, the form diameter for racks has not been taken from the tooth form.

► **Note:**

If this option is selected and profile modifications have been predefined, the calculated form diameter will be at the beginning of the modification. This often results in very small transverse contact ratios εα.i and εα.e. This is correct because, the tooth form at the start of the modification no longer exactly matches the involute. However, the message that is displayed, to inform the user that the transverse contact ratio is too low, is rather confusing. If the profile modification has been sized correctly, so that meshing under load involves a whole tooth height, this message can be ignored. This is because the transverse contact ratio under load corresponds to the theoretical transverse contact ratio εα. Generally speaking, we recommend you do NOT use this option with profile modifications.

15.22.3.2 Calculation with own S-N curve (Woehler line)

The S-N curve (Woehler line) of metallic materials is usually defined by the endurance limit values sigFlim, sigHlim, entered in the database, and the finite life calculation values Y_{NT} (root) and Z_{NT} (flank) in accordance with ISO, AGMA or DIN. If this option has been selected, and you input your own S-N curves (Woehler lines) for material, the strength calculation is performed using your S-N curve (Woehler lines).

If you use your own S-N curves (Woehler lines) to calculate plastics, the **Calculation with own S-N curve (Woehler line)** flag has no effect.

Notes about calculation methods using your own S-N curves (Woehler lines):

- You can use the calculation methods specified in ISO, DIN and AGMA for metallic materials.
- The S-N curves (Woehler lines) are stored in a file (see under: Database). The material's allowable stress (σ_{Fadm} for root and/or σ_{Hadm} for flank) is defined according to the number of cycles N_L .

- The endurance limit values σ_{Flim} and σ_{Hlim}, that are input directly in the database, are also required for documentation purposes and should be detailed together with the S-N curve (Woehler line) data, in a meaningful combination. We recommend to use for σ_{Flim} / σ_{Hlim} the value σ_{Fadm} / σ_{Hadm} if $N_L=10^7$.
- The service life factors, factor Y_{NT} and Z_{NT} are defined and reported as follows: $Y_{NT} = \sigma_{Fadm} / \sigma_{Flim}$, $Z_{NT} = \sigma_{Hadm} / \sigma_{Hlim}$
- The other factors which influence the permitted material value, such as Y_{drel}, Y_{RrelT}, Y_X, Z_L, Z_V, Z_R and Z_W, are calculated and used in accordance with the selected calculation method (ISO, DIN or AGMA). For this reason, the selected permitted material value σ_{FG} or σ_{HG} is not exactly equal to the value σ_{Fadm} / σ_{Hadm} from the S-N curve (Woehler line).

15.22.3.3 Calculation of the displacement volume of gear pumps

This option calculates the transport volume without taking the return volume into consideration. If you select this option, the tooth spaces are integrated numerically to calculate the transport volume and the result is output in the report. In Fine sizing, the transport volume of each variant is also calculated and output. This enables you to identify, for example, the variant with the largest displacement volume.

15.22.3.4 Take into account user specific additions

If KISSsoft AG has added customer-specific upgrades to the software, you can select them and view them here.

15.22.3.5 Take protuberance into account

If the angle difference (protuberance, or buckling root flank) to the pressure angle is greater than the maximum difference defined here, its influence on the tip and root form circles, and also the transverse contact ratio, are taken into account. The contact ratio then reduces accordingly.

15.22.3.6 Permissible maximum wear of tooth thickness

When you calculate wear safety, you must specify a permitted wear threshold value. A usual value for plastic is 15% (tooth thickness wear in the reference circle). If no, or only very little, wear can be tolerated, we recommend you input a value between 5 and 10%.

15.22.4 Tooth form

15.22.4.1 Calculate moment of inertia from tooth form

The toothing moment of inertia is calculated exactly from the tooth form in the tip to root diameter range. To achieve this, the KISSsoft tooth form calculation is run automatically for each calculation and defines the effective tooth form by the numerical integration of the moment of inertia. The result is output in the calculation report. The calculation is also performed in fine sizing and the results are documented.

15.22.4.2 Backlash calculation from tooth form

This option enables you to calculate the load-free backlash from the tooth form, taking into account profile and flank line modifications. The crossed helical gears calculation only covers worm wheels with an axial crossing angle of 90°.

15.22.4.3 Use points (instead of curves) for the tooth form definition

The tooth form calculation uses a highly reliable algorithm to generate the tooth form from curves (lines, involutes, arcs of circle, etc.). In a few, rare cases, this algorithm does not produce a usable solution. In those situations, it may be sensible to use the alternative algorithm, which uses points.

15.22.4.4 Use the alternative algorithm for tooth forms that are defined with points.

The tooth form calculation uses a highly reliable algorithm for determining the points on a tooth form. However, in a few special cases, this algorithm does not provide a good solution. In such situations, using an alternative algorithm may help.

15.22.5 Safety factors

You must set required safeties, not only for every service life calculation, but also for rough sizing and fine sizing.
Calculating reliability using Weibull distribution uses the calculated service life, and so also takes the required safeties into account. To calculate reliability without taking required safeties into account, set the safeties to 1.0.

Safeties are not size-dependent

Experience has shown that much lower minimum safeties can be used for smaller modules. Although the standards do not provide any information about this, this knowledge is based on experience with many different applications. However, if you do not require size-dependent safeties, you can still select the "Safeties are not depending on size" variant.

Minimum safety for calculation according to AGMA

In the tooth strength calculation according to AGMA 2001, the permitted tooth bending stress sat is half the size of the value specified in ISO 6336. Although its meaning is similar, the corresponding s_{at} value in the ISO standard must be multiplied by a factor of 2, the reference gear's stress correction factor Y_{st}. Therefore, if the tooth strength is calculated according to AGMA 2001, the resulting safety is approximately 50% less than that in the calculation using ISO 6336. As a consequence, the safety required for the calculation according to AGMA 2001 is smaller.

Service coefficient

Some applications of the AGMA calculation method require a predefined service coefficient. In actual fact, this is merely a minimum safety. For this reason, if required, you can input Service coefficient C_{SF} for flank strength and K_{SF} for tooth bending strength.

15.22.5.1 Safety factor for the calculation of the shear stress at hardening depth

The safety factor is multiplied by the shear stress, which is then used to calculate the hardness. The hardening depth is then defined using this value.

15.22.6 Contact analysis

15.22.6.1 Calculation

Calculation method contact stiffness: Here you can select either the calculation method defined by Weber/Banaschek [21] (dynamic stiffness analysis: default setting), the method defined in ISO 6336-1 Method B and "Own input".

Fixing position of the tooth: The contact analysis according to Weber/Banaschek [21] is based on a bending beam model with a random form. This setting defines the method for determining the fixing position of the bending beam/tooth. You can select the methods according to Weber/Banaschek [21] or according to Langheinrich [22].

Single contact stiffness: If **Own input** has been selected as the contact stiffness calculation method, you can enter your own value for the single contact stiffness.

Coupling stiffness modification factor: The factor can be defined according to Raabe or Börner [36], see the next figure.

According to Raabe: $C_{Ci} = k_C \cdot 0.04 \cdot N^2 \cdot \frac{C_i + C_{i+1}}{2}$

According to Börner: $C_{Ci} = k_C \cdot 2.75 \cdot \left(\frac{m_n}{b_c}\right)^2 \cdot \frac{C_i + C_{i+1}}{2}$

C_{Ci} – coupling stiffness of slices i and $i + 1$, k_C – coupling stiffness factor
N – number of slices, m_n – normal module, b_c – slice width, C_i – stiffness of slice i

Figure 15.96: Coupling stiffness modification factor according to Raabe or Börner

Border weakening factor: Border weakening factor for a weakening of stiffness on the edge of helical gear teeth.

Correction factor for Hertzian stiffness (according to Winter): Correction factor for Hertzian flattening as described in the experiments performed by Winter/Podlesnik [37].

Number of orders in the amplitude spectrum (transmission error/contact stiffness): This is where you enter the number of orders to be calculated. At least one order must be calculated, and the calculation must be performed with no more than half the number of meshing positions (set this value in the **Contact analysis > Accuracy of calculation** tab).

Flash temperature and micropitting with coefficient of friction according to ISO/TS 6336-22: This overwrites the coefficients of friction defined in the **Contact analysis** tab with a coefficient of friction sized according to ISO/TS 6336-22.

Interpolate stress increase caused by tip rounding: In the case of a tip rounding, the calculation of the tooth form results in a sudden change in the radii of curvature. This in turn results in stress increases at this transition point in the contact analysis calculation. For this reason, you can specify whether the mathematical solution is to be used, to perform the calculation, or whether this stress increase is to be interpolated.

Calculate force excitation: Force excitation (according to FVA Report 487) results from toothing stiffness and the average transmission error. In contrast to the process for calculating transmission error, calculating the excitation force enables a better evaluation of how different toothing variants generate noise. This is because the gear meshing forces, not the equalizing movement (transmission error), of the gears, are the decisive factor in generating noise.

Conical profile shift: Select this option to enable the conical profile shift in the **Contact analysis** tab.

Take into account plastic deformation: Use this setting to specify whether plastic deformation is to be taken into account in the contact analysis. If plasticity is to be taken into account, the maximum contact stress, calculated using the elastic contact theory, is reduced on the basis of the specified "Maximum permitted flank pressure". If the maximum elastic flank pressure is exceeded, the radii of the contact body are changed locally so that the resulting elastic i.e. contact stress matches this maximum value. Only a percentage rate of the new radii is used, on the basis of the specified "Weighting of the plastic deformation".

Smooth iterative wear calculation: If you select this option, the tooth form is smoothed after every iteration of the wear calculation.

15.22.6.2 Display

Unit of the transmission error Here, you can select either the length on the length of path of contact (transmission error) or the angle on the driven gear (angle of rotation error).

Smooth results: This function uses a low-pass filter to smooth the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). By default, this function not selected, but it can be used to smooth the results if they are affected by strong numerical noise.

Area analyzed on tooth height: This defines the maximum area along the tooth height for evaluating the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). This setting generates additional results and does not change the results of the contact analysis.

Area analyzed on facewidth: This defines the maximum area along the facewidth for evaluating the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). This setting generates additional results and does not change the results of the contact analysis.

Draw data for path of contact: If this option is enabled, the results of contact analysis are displayed quadratically in the 3D diagrams. This makes the data suitable for export as a matrix.

Take into account backlash in the transmission error graphic: When this setting is selected, the backlash is taken into account in the transmission error. This causes a displacement of the value of the transmission error even though the amplitude remains the same.

15.22.7 Diagrams

15.22.7.1 Meshing in the diagrams

You can select different values for the X-axis from a drop-down list. Here, you can select the roll angle, the length (length of path of contact), the diameter of gear A and the angle of rotation.

► **Note**

If you select the angle of rotation for the X-axis, the gear axis is 0°.

In every graphic that displays the length of path of contact A-E as defined in ISO 21771, you must remember that these particular points for the non-deformed tooth pair are calculated without modifications, as specified in ISO 21771. Contact analysis takes tooth bending into account. Depending on which tooth modification is used, this creates a contact that occurs earlier and is also longer. This is clearly visible if, for example, normal force is displayed with the angle of rotation (or roll angle, length of path of contact) as the X-axis. In contrast, the force diagram looks different if the diameter is displayed as the X-axis. The premature meshing starts at a slightly higher point on the tooth root and then moves vertically downwards until point A is reached. Only then does it run in the direction of the tooth tip. The force diagram therefore varies, depending on which X-axis is selected.

In special cases, involving large profile modifications, this results in extremely significant differences if the tooth contact no longer runs to the tooth tip and the path of contact varies from the theoretical progression (see Figure 15.99). The profile modifications increase the range of the angle of rotation.

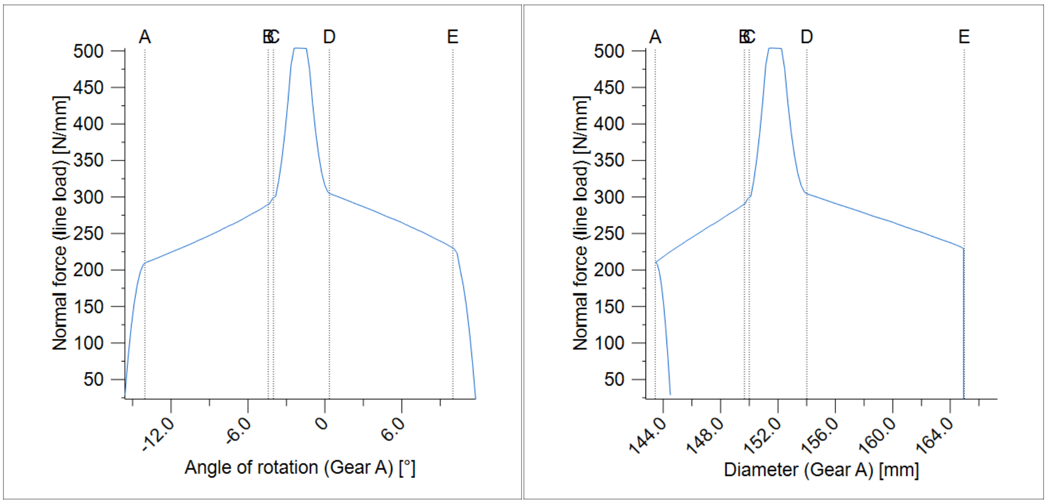


Figure 15.97: Normal force curve for toothing without any modifications; left: display via the angle of rotation; right: via the diameter

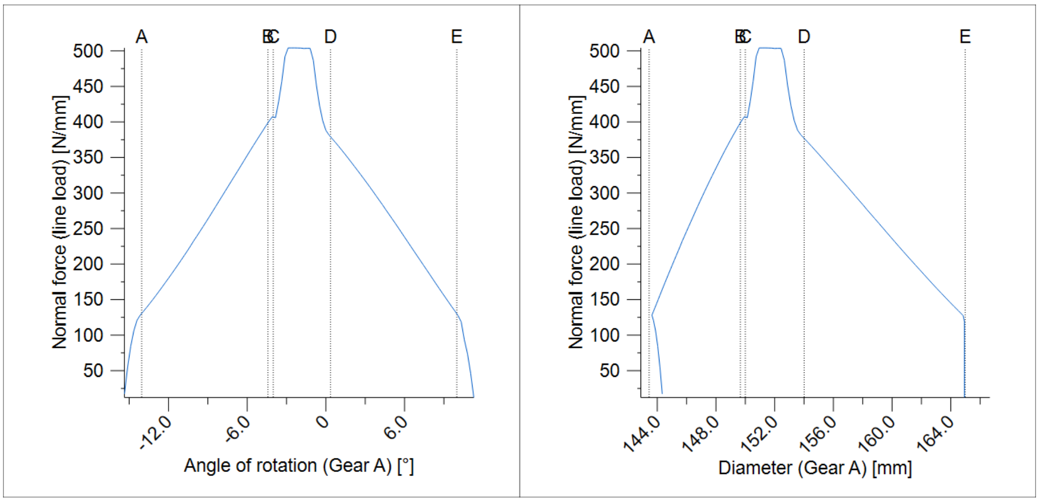


Figure 15.98: Normal force curve for toothing with low profile crowning (meshing occurs up to the tooth tip)

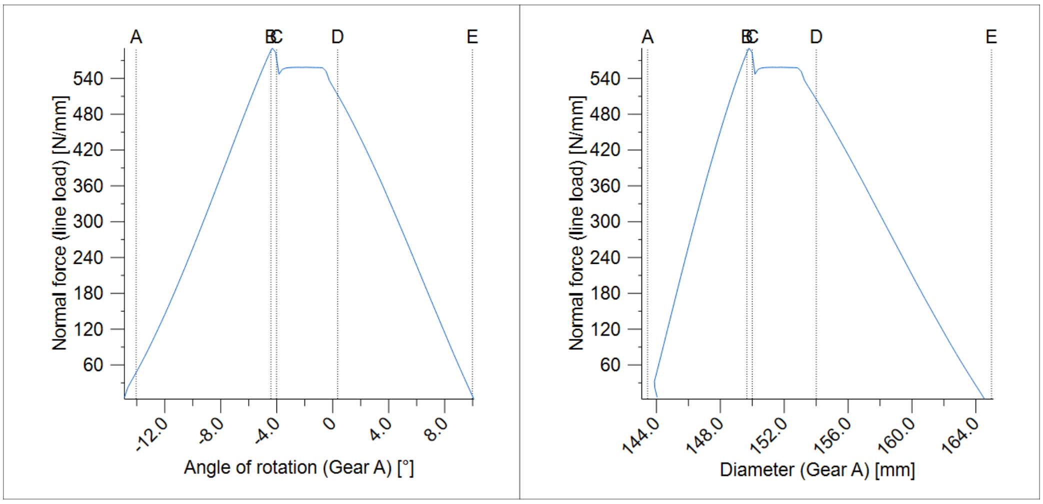
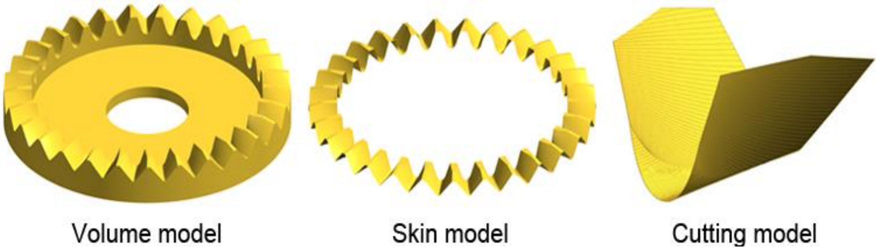


Figure 15.99: Special case of a normal force curve with a higher load and greater profile crowning (meshing does not occur up to the tooth tip)

15.22.8 Generate a 3D model

This is where you modify the parameters used to generate 3D models.

- Under **"Model type"**, specify the type of model to be generated (volume model, skin model, cutting model). The volume model can be used for other applications such as machining by CNC or finite element analysis. The skin model is most suitable for contact analysis. The cutting model is only suitable for the gear models that use cutting simulation, such as face gear and enveloping worm gear, and is used to view the actual cutting simulation.



- The **Number of cutting steps** sets the number of cuts per half pitch for the cutting process. The minimum value is 1, and the default value is 20. The quality of the final model can be increased by increasing the number of cutting steps, but this also increases the probability of manufacturing errors. The "Scale factor" is used for solving the failure problem. If the operation fails, we recommend you use a lower number of generation steps with a larger scale factor.
- The **Number of sections along facewidth** defines the number of sections along the facewidth for approximating the tooth flank form. The minimum value is 2, and the

default value is 11. Normally, the quality of the final model can be improved by increasing this value, but we do not recommend that you set a number that is excessively high, compared with the facewidth. The coefficient is used for the gear models using cutting simulation and gear models using multiple cross sections, such as spiral bevel gears and cylindrical gears with lead modification.

- The **Scaling factor for the cutting model** is used to scale the model during the cutting simulation process. The minimum value is 1 and the default value is 10. Sometimes the cutting simulation can fail due to an internal operation error in the Parasolid kernel, especially when the model has a very small module and/or a large number of generation steps. To prevent this type of operating error, use the model with its size set by the scale factor in the cutting process. Consequently the cutting model can have different dimensions than the actual design. However, the volume and skin models are automatically returned to their original scale (size) after the operation, and therefore have the same dimensions as the entered gear.
- The "**Modeling operation tolerance**" sets the tolerance for the internal operations of the Parasolid kernel, such as the chordal approximation and clash detection in Boolean operations. The default value is 1 µm.
- The **Rendering quality** sets the resolution for the resulting graphics in the 3D geometry view. This is used only to improve the viewer display (usability) and does not affect the quality of the generated model. If the rotation operation in the Viewer is slow, you can increase the quality value to speed up the operation. The default value is 5 µm.
- Click on **Constant root radius along the facewidth** to specify the method used to generate the root fillet radius for a bevel gear. The bevel gear's root fillet radius changes by the factor of the normal module along the facewidth. If you set this checkbox, the constant root fillet radius defined by the normal module is used in the middle section. (Available for bevel gears)
- The **Constant protuberance along the facewidth** value sets the protuberance of the bevel gear's reference profile. The protuberance of the bevel gear's reference profile changes by the factor of the normal module along the facewidth. If you set this checkbox, the constant protuberance defined by the normal module is used in the middle section. (Available for bevel gears)
- If **Display 2D geometry for outside and inside** is selected, the tooth forms on the internal and external sides are represented as a 2D graphic. (Available for bevel gears)
- If **Generate tooth system model in the saved position** is selected, the system model is generated at the position you saved. This position is saved in the calculation file, and you will be able to restore the contact pattern's checking position in the future. (Available for bevel gears)
- Click on **Number of points on the edge of cut for spline approximation** to specify the number of modeling (intermediate) points on each edge that are used to approximate the spline curves for the root area or the tooth flank. The figure shows a diagram in

which the points that are to be used are scanned. The end points (nodes) are removed because they add waviness to the curve. We use only the intermediate points on the cutting edge if it can be assumed that the parametric distance between the points is the same. We usually recommend that more points are used in the root area. However, this model will help the user determine the optimum value. (Available for enveloping worm gear)

- Click on **Oversize factor for worm wheel cutter** to define the coefficient used to increase the worm wheel cutter. There are different methods of implementing the interference tool. Those methods include the axial pitch method, the base pitch method, the extra thread method, and the normal pitch design method. KISSsoft uses the normal pitch method because this is practically regarded as the industry standard. These methods are based on the principle, that the worm wheel cutter uses the same normal pitch and the same normal pressure angle in the normal section as the worm. The cutting distance between the hob and the gear will then be changed accordingly, to ensure a consistent result for the root and tip diameters on the gear. If you are using the oversize factor, the generated surface will not be match the worm surface and will not give the best contact pattern. Therefore we recommend you do not use the oversize factor if you want to use theoretical surface geometry rather than a conventional cutting method. In practice, the tooth thickness of the cutter is increased to take the tooth thickness tolerance of the worm wheel into account. In this case, we recommend you use a small oversize factor to compensate for the tolerance order to get the best contact. (Available for enveloping worm gear)
- Click on **Cutter shaft angle change** to modify the worm wheel cutter shaft angle during the simulated milling run. The angle can be both positive and negative. The positive angle is defined as shown below. (Available for enveloping worm gear)
- Click on **Change in pressure angle of the worm wheel cutter in normal section** to set the worm wheel cutter to a different pressure angle than the worm. (Available for enveloping worm gear)
- Click on **Flank shape of worm wheel cutter** to set a different tooth form for the worm wheel cutter than for the worm. Extensive research has shown how different combinations of tooth forms can be used to get a better contact pattern in worm wheels. This setting is used for this purpose. If this option is not selected, the same tooth form is used for both the worm wheel cutter and the worm. (Available for enveloping worm gear)

- Click on **Axial expansion**, to take the axial length expansion/contraction factor α of the gears into account for injection-molding or sintering processes. The helix angle value for helical gear teeth is based on the new facewidth, calculated again from

$$\beta = tan^{-1}\frac{tan\beta}{\alpha}$$

.

15.23 Tooth thickness

Select the **Calculation > Tooth thickness** menu option to calculate the normal tooth thickness and the normal spacewidth at any diameter.

The tooth thickness is output as an arc length and as a chordal length. To help measure the tooth thickness, the chordal height is output along with the tooth thickness allowances.

15.24 Tooth form export

In the **Calculations** menu, click on "Tooth form export" to export the shape of the gear or tool to a text file, as coordinates.
There are a number of settings available here for the export functions:

- select the cross-section
- select the operation
- select the separator
- text file format
- reduce the number of points
- remove duplicate points

15.25 Gear mesh frequencies

Gear mesh frequencies provide valuable information for damage and defect analysis.

A gear assembly typically shows not only normal low-frequency harmonics in the vibration spectrum but also high activity in the high-frequency region due to the gear teeth. This frequency is called **gear mesh frequency** (GMF, f_z). The GMF is calculated by the product of the number of teeth and its respective running speed according to equation 1.

$$f_z = z \cdot f_n$$
(eq. 1)

The gear mesh frequency will have running speed sidebands (f_s) relative to the shaft speed to which the gear is attached. If the gear pair is in good running condition, all peaks have low amplitudes, and no natural gear frequencies are excited.

$$f_s = f_z \pm f_n$$
(eq. 2)

Tooth damage (wear) and backlash can excite natural gear frequencies along with the gear mesh frequencies and their sidebands. Generally distributed faults such as eccentricity and gear misalignment will produce sidebands and harmonics with high amplitude close to the gear mesh frequency. Localized defects, such as a cracked tooth, produce sidebands that are spread more widely across the spectrum.

Typical **symptoms** for different gear assembly damages:

Eccentric gear or bent shaft

This type of damage causes modulation in the GMF at the rotating speed of the eccentric gear. Modulation may also occur at the shaft rotating frequencies in case of a sufficiently serious problem. In the case of an eccentric gear, its peak at 1× rpm will have a greater amplitude, and the sidebands will appear spaced at that same 1× rpm frequency. Significant sidebands may also be found in the 2× and 3× GMF.

Backlash

An improper backlash of the gear pair usually excites the GMF and sidebands at 1x rpm. If the backlash causes the problem, the GMF amplitude will often decrease with increasing load.

Gear misalignment

A gear misalignment almost always excites second-order or even higher GMF harmonics. Therefore, it will only show smaller amplitudes at 1x GMF, but much larger levels at 2× or 3× GMF.

Another frequency type observed in gear pairs is the gear **hunting-tooth-frequency** (HTF, f_k). In contrast to the gear mesh frequency, which is system dependent, this frequency is pair-dependent. In some literature, it is named “frequency of the same tooth position” and calculated using equation 3. Here k stands for the greatest common divisor of the number of teeth of both gears in contact. It is also referred to as assembly phase factor. and is defined as the smallest common integer multiple of the number of teeth on the pinion and the gear. This frequency is usually very low compared to GMF and is often used to detect faults that occur during the manufacturing process or due to mishandling. It can cause quite high vibrations, and the HTF is usually audible.

$$f_k = k \cdot \frac{f_{n2}}{z_1} = k \cdot \frac{f_{n1}}{z_2}$$
(eq. 3)

The assembly phase factor is also used to calculate the assembly phase frequency (APF, f_a) of a gear pair (equation 4). This frequency is typically used to evaluate the wear distribution between the gear and the pinion. APF is present only when gear pair tooth number combinations don’t share common prime numbers between the gear and the pinion (k > 1).

$$f_a = \frac{f_z}{k}$$
(eq. 4)

The characteristic frequency of a fault gear in a **planetary gearbox**

A planetary gearbox consists of at least two types of gear pairs, sun-planet and planet-internal gear (ring gear). These two pairs have identical meshing frequency. If the internal gear is stationary, the gear mesh frequency can be calculated using equation 5. In all other cases, the velocity of the internal gear is considered when calculating GMF, see equation 6. In this case, f_H is the rotational frequency of planet carrier, n_r is the rotational frequency of the internal gear and z_r is the absolute value of the number of teeth on the internal gear.

$$f_z = f_H \cdot z_r$$
(eq. 5)

$$f_z = abs(f_H - n_r) \cdot z_r$$
(eq. 6)

In addition to the gear mesh frequency, several other (see chapter 13.3.7, Characteristic frequencies) can be found in a planetary gear unit. Those faulty frequencies can typically be divided into **local damage cases** and **distributed damage cases**.

A local damage case is defined by how many times the faulty gear tooth meshes with mating gears per unit of time. This is due to the fact that a local gear damage modulates a gear meshing vibration at the repeating frequency of the sudden changes caused by the contact of the locally damaged gear tooth with mating gears.

A distributed damage case is defined as the relative rotating frequency of the faulty gears with respect to the planet carrier. The reason is the distributed gear damage that modulates gear meshing vibration at a period equal to the damaged gear totaling cycle relative to the planet carrier.

The calculation of the local and distributed frequency for planetary gears is performed according to equation 7-12 where K is the number of planet gears, z_s is the rotational frequency of the sun and z_p is the rotational frequency of the planet gear.

Characteristic frequency of sun gear

$$f_{csd} = \frac{f_z}{z_s}$$
(eq. 7)

$$f_{csl} = K \cdot \frac{f_z}{z_s}$$
(eq. 8)

Characteristic frequency of planet gear

$$f_{cpd} = \frac{f_z}{z_p}$$
(eq. 9)

$f_{cpl} = 2f_{cpd}$
(eq. 10)

Characteristic frequency of internal gear

$f_{crd} = f_H$
(eq. 11)

$f_{crl} = K \cdot f_H$
(eq. 12)

In KISSsoft, the gear mesh frequency calculation can be enabled through the **Strength** tab, and the results are shown in the main report under the **Supplementary data** chapter.

The **Settings** tab allows you to change various options that can be considered during the calculation. These options enable or disable the calculation of the sidebands. You can enter the number of harmonics that are to be considered during the calculation and also specify whether the calculation should consider variations in operating speed range by ± %.

When the gear calculation considers the variation of the input parameters through load spectrum calculation the gear mesh frequency settings can also enable the consideration of them in the calculation. In addition, different selection for plotting the results are available from the **Settings** menu.

The results of the gear mesh frequency can be shown in three different units, namely, Hertz (Hz), cycle per minute (CPM) or nominated by the reference gear speed.

The graphic plot of the results can be shown through the options **Graphic > Evaluation** . There are two different types of diagrams. The first one is gear mesh frequency, where on X axis rotation of the reference gear speed is shown and on Y axis the values of characteristic frequencies are shown. The second diagram is gear mesh frequency – amplitude plot. Currently, the amplitude value is unitless.

Literature according to which the calculation of gear mesh frequencies is performed, see [38], [39], [40], [41], [42], [43] and [44].

15.26 Planetary phasing calculation

Planetary phasing calculation ([45] and [46]) including the angle between the planets, is done automatically for planetary gears. The results can be found in the main calculation report under **Supplementary data**.

The phasing classification for a planetary gear is based on the first harmonic of the excitations of the meshing. Possible phases for equally spaced planets are in-phase, sequential-phase and counter-phase. Higher harmonics may have a different phase by the number of planets which can be used to determine the phase position when the planets are equally spaced:

$k_{\Phi} = mod(|Z_{inner}|, N)$

In particular, phasing is of interest to understand which excitations in the tooth mesh are reinforced or canceled in the system. Excitation amplifications and cancellations are studied for the following components:

1. torques and axial forces, i.e. thrusts (helical only)
2. radial forces

In-phase

In-phase means that each gear mesh has the same phase angle. In-phase planet gears have reinforced torques and thrusts but canceled radial forces. The condition for in-phase is:

$k_{\Phi} = 0$

Sequential-phase

Sequential-phase means that the phase angles are equally separated for subsequent gear meshes. Sequential-phase planet gears have canceled torques and thrusts but increased radial forces. The condition for sequential-phase is:

$k_{\Phi} = 1, N - 1$

Counter-phase

Counter-phase means that opposite phase angles exist for opposite planets. All summed components of the excitation are canceled for counter-phase planetary gear units. The condition for counter-phase is:

$k_{\Phi} = 2, 3 ..., N - 2$

Mixed-phase

Mixed-phase means that there is a combination of phasing types. This can only occur with unequally spaced planets, as found in diametrically opposed constructions. In such cases, the gear mesh phase angles of the tooth meshes must be calculated and used in the sum of the harmonics to determine which reinforcements and cancellations occur.

16 Bevel and Hypoid Gears

Use this module to calculate the geometry and strength of straight, helical and spiral bevel gears (gear axes intersect, offset is 0) and hypoid gears (crossed gear axes, offset not equal to 0). Geometry as specified in ISO 23509 and DIN 3971, tolerances according to ISO 17485 and DIN 3975, strength calculation as specified in ISO 10300 (replacement cylindrical gear toothing method), AGMA 2003, DIN 3991, or Klingelnberg in-house standard KN 3030. The calculation only includes the geometry of bevel gears insofar as is necessary for the strength calculation (see chapter 16.4.1, Methods used for strength calculation), no matter which manufacturing process is used.

16.1 Underlying principles of calculation

16.1.1 General

The geometry of bevel gears is calculated according to ISO 23509 or DIN 3971. The strength calculation is performed in two steps. A virtual cylindrical gear toothing is defined first. This is then used for the strength calculation in a similar way to cylindrical gears. The process is described in [47], [48] and [12].

Bevel gear machine tool manufacturers (such as Klingelnberg in Germany) also have their own methods that differ slightly from the processes mentioned above.

Hypoid bevel gears are primarily used in vehicle axle drives. Strength is calculated by defining a virtual cylindrical gear toothing.

16.1.2 Overview of the bevel gear manufacturing process and the terminology used in it

Various manufacturing processes are used to create bevel gears. Unlike cylindrical gears, the tooth length forms and tooth depth forms differ according to which manufacturing process is used. In particular, the process used to manufacture bevel gears with spiral teeth uses a multitude of terms, the most important of which are described below.

The most important differences are shown in the tooth length form, which can be manufactured as an arc of circle (face milling procedure), epicycloid or involute toothing (face hobbing process). Circular arc teeth were developed by the company Gleason, and are produced using the face milling approach, in which every gap is milled separately, and then the gear is rotated further by the width of that tooth space. Epicycloid gear teeth are used by Oerlikon and Klingelnberg. In this process, the gear rotates constantly during the milling process. Only the palloid manufacturing process is used to create the involute tooth length form. Although, nowadays, Klingelnberg and Gleason, the market leaders in machine manufacturing, can produce toothing using both the face milling and face hobbing processes, these companies are still associated with their traditional processes in the technical literature. For more details, see (see chapter 16.1.3, Calculation according to Klingelnberg, Gleason and Oerlikon) and (see chapter 16.2.1, Type).

Although alternative processes for bevel gears are available, they are not listed here.

16.1.3 Calculation according to Klingelnberg, Gleason and Oerlikon

The strength calculation defined in ISO 10300 or DIN 3991 only includes the relationships (module, helix angle) in the middle of the facewidth in the virtual cylindrical gear toothing method calculation. The shape of the bevel, and the process used to manufacture it, are ignored. As a result, the strength calculation method can be applied no matter which procedure is being used. This also reflects the experience that the load capacity of spiral bevel gears is only slightly affected by the manufacturing process.

The geometry calculation procedure defines the dimensions, such as diameter and tooth thickness, in the middle of the facewidth. It also calculates the diameter at the outside and inside end of the facewidth. These dimensions depend on the type of the bevel. However, the dimensions of the gear blank may differ from the results calculated by machine-specific software because the processes are not described in sufficient detail. This is particularly true for the Gleason process.

- **Klingelnberg process:**
  The **Bevel gear (KN3028 and KN3030) and Hypoid gears (KN3029 and KN3030)** calculation methods enable you to calculate geometry and strength and check the manufacturing process according to the Klingelnberg in-house standard. However, these methods do not calculate the machine settings for the selected Klingelnberg machine. When you input formula data from a Klingelnberg program, you must remember that the toothing data, such as module and helix angle, always applies to the middle of the facewidth (unless otherwise specified).
- **Gleason process:**
  Depending on which calculation program Gleason uses, toothing data such as the module and helix angle, is either predefined for the outside end of the facewidth or for the middle of the facewidth.
  Use the **Conversion from GLEASON data sheets** dialog window to convert Gleason data from the outside end of the facewidth into data for the middle of the facewidth (see chapter 16.2.1, Type). Once this data has been converted, you can perform the strength calculation. Although the bevel dimensions (tip and root diameter) do not always exactly match the actual geometry, they are close enough to enable you to check the assembly conditions (in a gearbox). This procedure does not check to see whether the part can be manufactured on Gleason machines.
- **Oerlikon process:**
  The Oerlikon process is broadly similar to the Klingelnberg process (select the Klingelnberg bevel type).

16.2 Basic data

16.2.1 Type

You will find a drop-down list for the type on the top left of the screen, in the **Geometry** tab.

The following bevel gear shapes are available here 5e83442bef4dd:

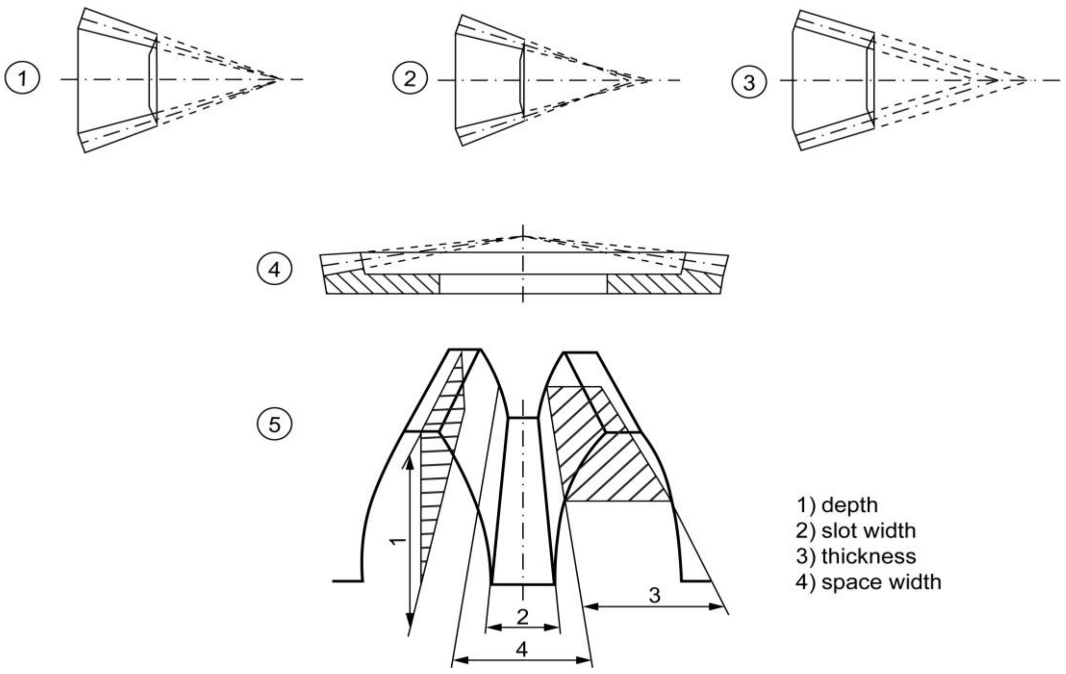


Figure 16.1: Basic types of bevel gears

- **Standard, Figure 1 (tip, pitch and root cone apex in one point)**
  The geometry is calculated according to ISO 23509. No offset possible. If you click the Sizing button, the cone angle is calculated so that the bevels meet each other in the crossing points of the gear axes (similar to the standard specified in ISO 23509, Annex C.5.2). In this case, the tip clearance is not constant. We recommend this type for the simplified sizing of form-forged, injection molded or sintered bevel gears.
- **Standard, Figure 4 (pitch and root apex in one point)**
  Sizing of the tooth tip angle of gear 2 according to ISO 23509, Annex C.5.2, or own input. No offset possible. The tip clearance is constant. A constant tip clearance is taken into account while calculating the cone angle of the counter gear.
- **Standard, Figure 2 (tip, pitch and root cone apex NOT in one point)**
  Sizing of the tooth tip and root apex of gear 2 according to ISO 23509, Annex C.5.2, or own input. No offset possible. A constant tip clearance is taken into account while calculating the cone angle of the counter gear. We recommend this type for bevel gears

with straight or helical teeth with general cone angles, for example differential bevel gears.

- **Constant slot width, Figure 2, (face milling, Gleason-Duplex)**
  The geometry is calculated according to ISO 23509. You can perform this calculation either without offset (Method 0, spiral bevel gears), or with offset (Method 1, hypoid gears). If you click the Sizing button, the cone angle is calculated with a "constant slot width" (ISO 23509, Annex C.5.2). The tip clearance is constant. Gap 2 in Figure 5 does not change. A typical application of this is a ground bevel gear toothing produced in the "completing" process (duplex), where the pinion and the bevel gear are each ground in one work step. This process requires machines that have an additional helical motion.
- **Modified slot width, Figure 2 (face milling, Gleason)**
  The geometry is calculated according to ISO 23509. You can perform this calculation either without offset (Method 0, spiral bevel gears), or with offset (Method 1, hypoid gears). If you click the Sizing button, the cone angle is calculated with a "modified slot width" (ISO 23509, Annex C.5.2). Gap 2 in Figure 5 changes. A typical application is the 5-section process, where the pinion is manufactured with 2 different machine settings, and a modified slot width is consequently created. The bevel shape is often also referred to as a TRL (Tilted Root Line).
- **Uniform tooth depth, Figure 3 (face hobbing, Klingelnberg)**
  The geometry is calculated according to ISO 23509. You can perform this calculation without offset (Method 0, spiral bevel gears), with offset (Method 3, hypoid gears) or according to KN 3028 and KN 3029. The tip and root cone are parallel. Applications are the cyclo-palloid® process and the palloid process. Palloid toothing is characterized by an involute tooth length form with a constant normal module over the facewidth.
- **Uniform tooth depth, Figure 3 (face hobbing, Oerlikon)**
  The geometry is calculated according to ISO 23509. You can perform this calculation either without offset (Method 0, spiral bevel gears), or with offset (Method 2, hypoid gears). The tip and root cone are parallel. Applications are Oerlikon processes such as Spiroflex and Spirac.

16.2.1.1 Converting or inputting Gleason toothing data

The Convert button and the Plus button are enabled for the **Standard, Figure 2, Constant slot width**, **Modified slot width** and **Uniform tooth depth** types. Using these two icons, you can enter data according to the Gleason definition.

Select the Convert button if a Gleason data sheet is present. You can then input the data in the window and then click on Calculate. When the calculation is finished, the Report and Accept buttons will be enabled. Click on the Report button to generate a short report. If you want to generate a more detailed report, click the  button in the main menu. Click the **Accept** button to transfer the data to the main window.

Click the Plus button to open a dialog window in which you can calculate bevel gear data according to different Gleason methods. The results of the geometry calculation will not match the Gleason dimensions sheet exactly, but will be close enough to calculate strength according to ISO 10300 (or AGMA or DIN).

In the "Type of gear" selection list, you can select one of a number of different Gleason methods (the default setting is to use a constant helix angle):

1. **Constant helix angle (straight or angled)**
   A constant helix angle represents a bevel gear with a constant helix angle. You can modify the helix angle to compare the geometry data with the Zerol geometry data if required. If you click the Accept button to close the dialog, the calculation is usually performed with the selection "Standard, Figure 4 (part and root apex in one point)".
2. **Duplex (constant slot width)**
   The term "duplex" refers to bevel and hypoid gears that are manufactured with a constant slot width across the entire tooth length of both gears. These gear types usually have a spiral angle of 35° in the middle of the facewidth with a continuously changing spiral angle in the axial direction. If you selected Duplex (constant slot width) and then clicked the Accept button to close the dialog, the calculation is usually performed with a "Constant slot width".
3. **Spiral toothing, default (modified slot width)**
   These gear types usually have a spiral angle of 35° in the middle of the facewidth with a continuously changing spiral angle in the axial direction. This gear type is described as having a "modified slot width". If you select this gear type, and then click on Accept, the calculation is usually performed with a "modified slot width". In this case, the root gap of the gear pair is constant over the entire tooth length and any gap modifications are performed on the pinion.
4. **Zerol "Duplex taper"**
   This is a Zerol design (see Zerol), but a root angle variation is performed to achieve duplex dimensions. If you select Zerol duplex and then close the dialog by clicking the Accept button, the calculation is usually performed with the "Constant root gap" selection.
5. **Zerol "standard"**
   The Zerol standard is a gear pair with a spiral angle of less than 10° in the middle of the facewidth, with a continuously changing spiral angle in the axial direction. In this case, the internal spiral angle is usually negative. To ensure the program can take into account the change across the tooth length, a value of b=0.001 is assumed for the case b=0. If you close the dialog by clicking the Accept button, the calculation is usually performed with a "modified slot width".

16.2.2 Mean normal module

You can input the normal module in the center of the facewidth. However, if you know the pitch, transverse module or diametral pitch instead of this, click on the Convert button to open a dialog window in which you can perform the conversion. If you would rather input the diametral pitch instead of the normal module, select the **Input normal diametral pitch instead of normal module** option in **Calculation > Settings > General**.

16.2.3 Pitch diameter gear 2

The external reference diameter of gear 2 (de2) is usually entered for bevel and hypoid gears. This is useful for designers because the bevel gear's assembly conditions are often predefined by the housing. The module is then recalculated (not optional).

16.2.4 Pressure angle at normal section

For standard meshings, the pressure angle is α_n = 20°. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Greater pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases.

For hypoid gears, click the Plus button to enter the pressure angle for the driving flank and the driven flank independently from each other. The driving flank is the concave flank of the pinion and the convex flank of the gear. The driven flank is the convex flank of the pinion and the concave flank of the gear.

16.2.5 Pressure angle driving/driven flank: Hypoid gears

Bevel gears are usually able to withstand stress better when driven by the concave pinion flank, i.e. when the sense of rotation and spiral direction of the driving pinion rotate in the same direction.

The concave flank of the pinion is usually called the driving flank (index D for "Drive"), and the convex flank is known as the driven flank (index C for "Coast"). On the bevel gear, the concave flank is the driven flank (index C) and the convex flank is the driving flank (index D). Since the effective pressure angle on the driving flank is greater by the amount of the limit pressure angle, and on the driven flank it is less than the pressure angle in a normal section, by the amount of the limit pressure angle, the pressure angle on the driving flank and driven flank can be entered independently.

As specified in ISO 23509, you should input the nominal design pressure angle for hypoid gears as α_{dD}, α_{dC}. This is used to calculate the generated pressure angle ("effective pressure angle") α_{nD}, α_{nC} for the driving side (index D for "Drive") and the effective pressure angle α_{eD}, α_{eC} for the driven side (index C for "Coast").

The equations specified in ISO 23509 are:

$\alpha_{nD} = \alpha_{dD} + f\alpha_{lim} * \alpha_{lim}$

$\alpha_{eD} = \alpha_{nD} - \alpha_{lim}$

If, as a result, α_{nD} is known, $_{a}$dD can be calculated as follows:

$\alpha_{dD} = \alpha_{nD} - f\alpha_{lim} * \alpha_{lim}$

$\alpha_{dC} = \alpha_{nC} + f\alpha_{lim} * \alpha_{lim}$

or if α_{eD} has been specified, $_{\alpha}$dD can be calculated like this:

$\alpha_{dD} = \alpha_{eD} + \alpha_{lim} * (1- f\alpha_{lim})$

$\alpha_{dC} = \alpha_{eC} - \alpha_{lim} * (1- f\alpha_{lim})$

The limit pressure angle α_{lim} is calculated and output in the report.

The limit pressure angle influence factor $f\alpha_{lim}$ has been introduced so that you do not always need to take the total value of the limit pressure angle into consideration when calculating the flank angle on the tool. For forming tools (Klingelnberg process), $f\alpha_{lim} = 0$. If you use the procedure with a constant slot width (Gleason), $f\alpha_{lim} = 0.5$ is set, otherwise $f\alpha_{lim} = 1.0$ is often used.

However, if more accurate data is not available, you can use the pressure angle in the normal section in the calculation (with $\alpha_{dD} = \alpha_{dC} = \alpha_n$ and $f\alpha_{lim} = 1.0$).

► **Note**

These input fields are only available if you are calculating the strength of hypoid gears (see chapter 16.4.1, Methods used for strength calculation).

16.2.6 Spiral and helix angle

The angle is input in the middle of the facewidth. In the case of helical-toothed bevel gears, the helix angle remains constant across the facewidth. However, in spiral bevel gears the spiral angle changes across the facewidth.

In hypoid gears, the spiral angle is specified in the middle of the facewidth for Gear 2. This value is then used to calculate the value for Gear 1 (pinion).

You can select any value as the spiral angle in the middle of the facewidth. However, we recommend you use a larger angle of between 30° and 45° to ensure optimum running performance. You should only select a value that is less than this guide value if the bearing load has to be reduced.

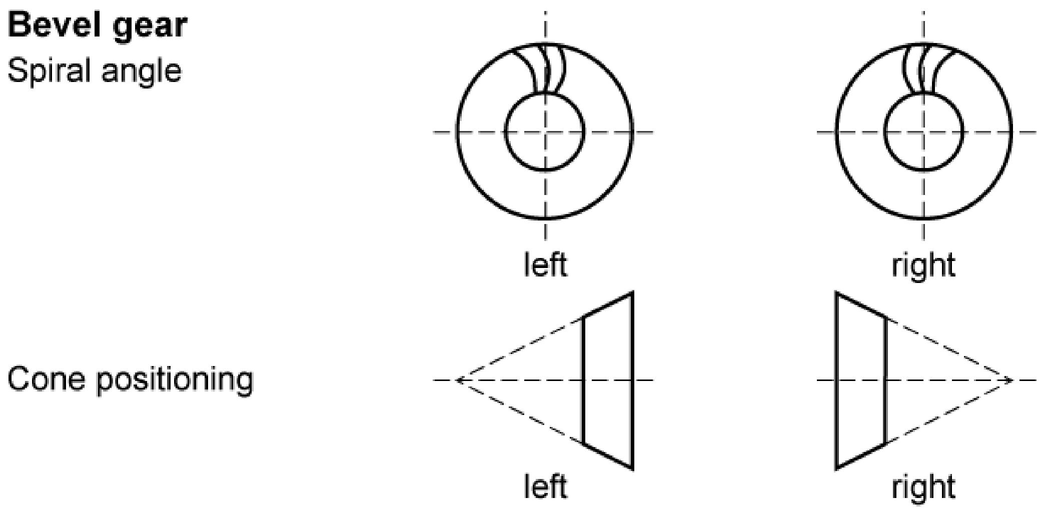


Figure 16.2: Spiral and helix angle

Click the Plus button to the right of the spiral angle input field to display the **Additional data for spiral teeth** window, where you can check the internal and external spiral angle for spiral bevel gears.

▶

16.2.7 Addendum angle and root angle

All the necessary data required to create the bevel gear drawing can be calculated from the addendum angle and dedendum angle. These are the tip and active root diameter on the outside and inside bevel, and the tooth thickness on the external and internal cone diameter (see Figure 16.3) and (see Figure 16.4). In the case of bevel gears with spiral teeth, the addendum angle and dedendum angle are calculated using the selected method [ISO 23509 or DIN 3971].

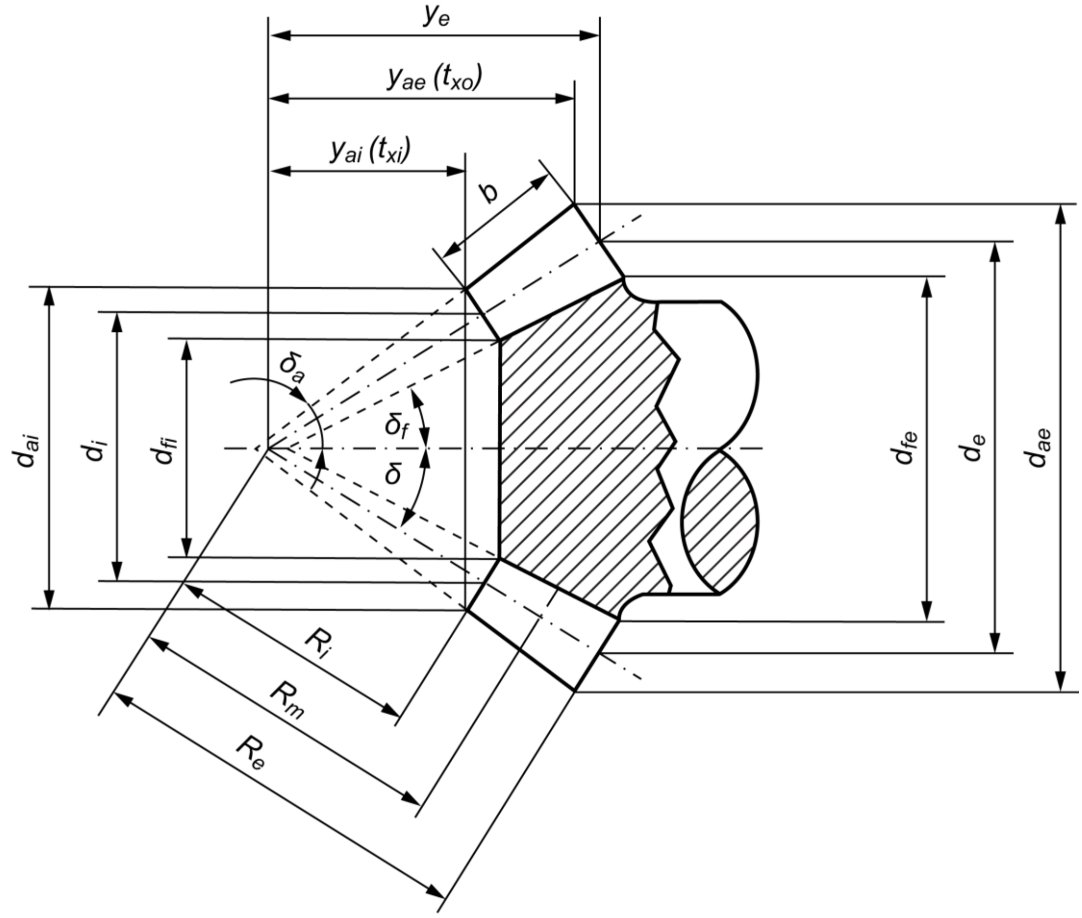


Figure 16.3: Dimensioning a bevel gear

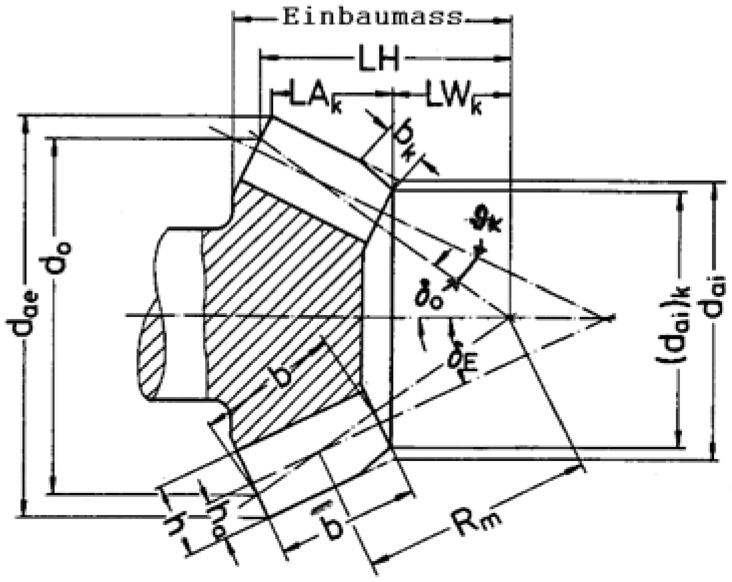


Figure 16.4: Dimensioning a bevel gear according to Klingelnberg

16.2.8 Angle modification

In some less than ideal situations, it may happen that the cutter head cuts into any shaft pins that are located immediately next to the toothing. If this cannot be prevented by modifying either the design or the toothing data, the cutter tip level at the calculation point at d_m of the gear and pinion can be tilted by a slight angle ϑ_k from its intended position $\delta_{o1,2}$ towards the reference cone angle $\delta_{E1,2}$ (see Figure 16.3) and (see Figure 16.4).

16.2.9 Number of teeth

You will find reference values for bevel gears with a shaft angle of 90° in this table.

u	1	1.25	2	2.5	3	4	5	6
z_1	18..40	17..36	15..30	13..26	12..23	10..18	8..14	7..11

Table 16.1: Recommended pairing transmission ratio u - number of teeth, pinion z1 in accordance with Niemann [12].

16.2.10 Facewidth

The facewidth should not usually be larger than the one given in the recommendations (facewidth ratio to outer cone distance (see chapter 16.2.7, Addendum angle and root angle), module ratio (see chapter 16.9.2, Module ratio)). The contact pattern deteriorates if the facewidth is too large.

16.2.11 Profile shift coefficient

You will find reference values for the profile shift coefficient of bevel gears with a shaft angle of 90° in this table:

u	1	1.12	1.25	1.6	2	2.5	3	4	5	6
x*	0.00	0.10	0.19	0.27	0.33	0.38	0.40	0.43	0.44	0.45

Table 16.2: In accordance with Niemann, 24/4 [12] recommended transmission ratio u - profile shift coefficient x*

Click on the Sizing button to the right of the profile shift coefficient input field to display the minimum profile shift coefficient for the pinion required to prevent undercut, and also the recommended value according to Niemann [12].

► **Note**

The ISO 23509 standard defines two different data types that can be used to describe tooth height factors and profile shift. The formulae used to convert data between these two data types are listed in ISO 23509. The Gleason calculation sheets also give partial descriptions of coefficients K and C1.

Although these are very similar to data type II, there are slight differences. Click the Convert button to convert data type II data.

16.2.12 Tooth thickness modification coefficient

Use the tooth thickness modification coefficient to modify the tooth thicknesses of the pinion and bevel gear. You can do this to compensate for tooth root strengths. You will find reference values for bevel gears with a shaft angle of 90° in this table

u	1	1.12	1.25	1.6	2	2.5	3	4	5	6
x_s	0.00	0.010	0.018	0.024	0.030	0.039	0.048	0.065	0.082	0.100

Table 16.3: Recommended pairing transmission ratio u - tooth thickness modification coefficient xs in accordance with Niemann [12].

► **Note**

The tooth thickness modification coefficient is achieved by using different tools. Please contact the manufacturer if you are using universal tools. If individual cutter sizes are used, the backlash occurs when the pinion and bevel gear have different tooth thickness coefficients.

16.2.13 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, click on the Settings button and then select the **Input quality** option. The accuracy grade according to ISO 17485 is very similar to that in DIN 3965.

You will find notes about the achievable toothing quality in the Manufacturing process (see chapter 16.3.1, Manufacturing process).

You can also click the Settings button to set different quality-related options. The following options are available:

Input quality

The manufacturing deviations that are output in the report and used for particular coefficients in the strength calculation are defined either in the ISO 17585 or DIN 3965 standards. You can specify which standard is to be used. If the Calculation method for strength option is selected, the system applies the standard that is best suited to the strength calculation method.

Varying qualities

If you select this option, the Plus button is displayed next to the Quality entry field in the main screen. Click this button to input specific tolerances manually. You will find a more detailed description of this in Qualities (see chapter 15.1.10, Quality).

16.2.14 Shaft angle

The shaft angle for bevel gears is usually 90°. However, you can perform the calculation for any shaft angle.

16.2.15 Offset

The offset is 0 for bevel gears. The offset for hypoid gears is greater than or less than 0. This application enables you to achieve higher contact ratios and greater strength at the tooth root. It is primarily used in automotive engineering (see Figure 16.5).

► **Note**

A positive hypoid offset is almost always applied to hypoid bevel gears, because this is the only way of achieving the improvements to the characteristics described above.

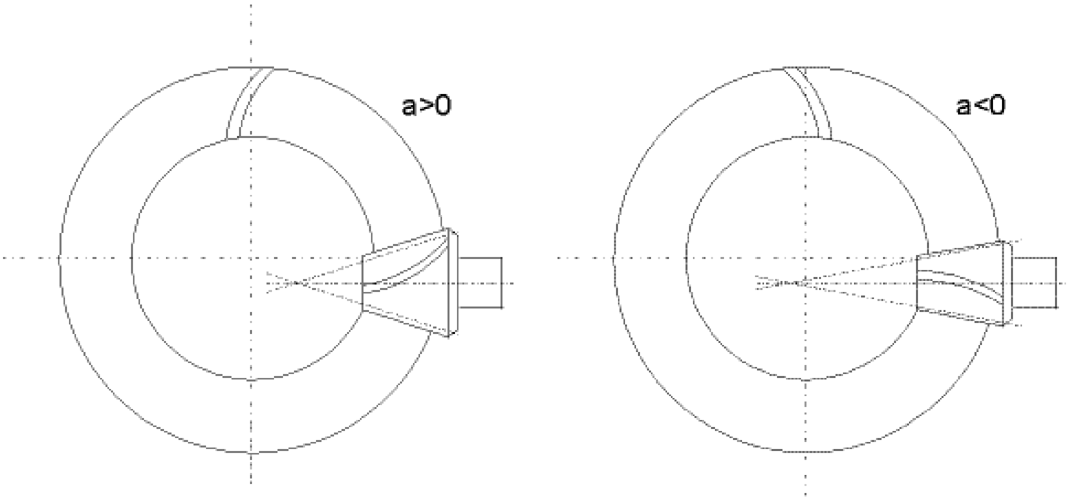
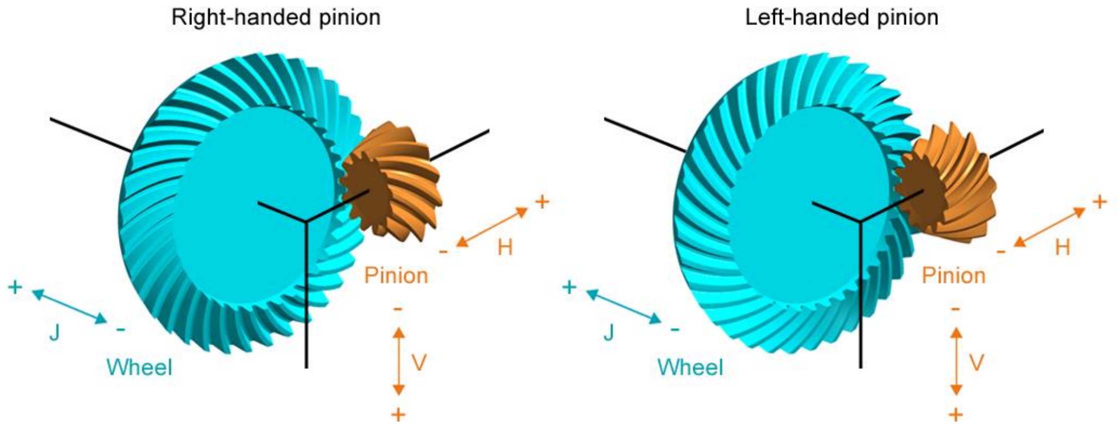


Figure 16.5: Hypoid bevel gear configurations. Positive offset (a > 0): Gear 1 left-hand spiral, Gear 2 right-hand spiral. Negative offset (a < 0): Gear 1 right-hand spiral, Gear 2 left-hand spiral

16.2.16 Geometry details

Click the **Details...** button in the upper right-hand part of the Geometry area to display the **Define details of geometry** dialog window. You can enter these parameters here.

The V-, H- and J misalignments of the bevel gear pinion are system data and are used to calculate the contact pattern.



Conversion of misalignment parameters (right-handed pinion)

Klingelnberg (KISSsoft input)	Gleason
V-	E+
H	P
J	G

Conversion of misalignment parameters (left-handed pinion)

Klingelnberg (KISSsoft input)	Gleason
V+	E+
H	P
J	G

Parameters are defined in the middle of the gear's facewidth.

Figure 16.6: Misalignment of the bevel gear pinion for calculating the contact pattern

You can specify the drawing number and the internal diameter for each gear. The data for dimensions yo, yu and the mounting distance (see chapter 16.2.16.1, Pitch apexes to front and back of gear blank/mounting distance) must be taken into account.

16.2.16.1 Pitch apexes to front and back of gear blank/mounting distance

The Pitch apex to the front of the gear blank is the distance from the pitch apex to the front face of the unworked blank, in the axial direction.

The Pitch apex to back of gear blank is the distance from the pitch apex to the rear face of the unworked blank, in the axial direction.

The Mounting distance can be defined as required. Usually, this means the distance from the pitch apex to the shaft shoulder in integral bevel pinion shafts is specified for the next rolling bearing. In contrast, in the case of bevel gears (without a shaft), the mounting distance usually corresponds to yo. This distance is usually specified on the assembly drawing and checked during mounting.

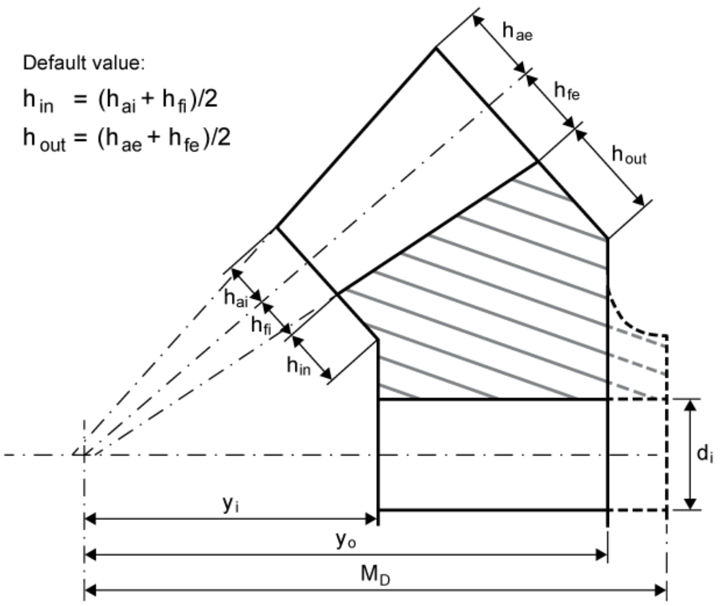


Figure 16.7: Figure: Bevel gear dimensioning

16.3 Process

16.3.1 Manufacturing process

The next table shows the relationship between the manufacturing process and the achievable accuracy grade.

Process	Achievable accuracy grade (ISO 17485, DIN 3965)
Milling only	8
Lapping	7
Skiving	6
Grinding	6

Table 16.4: Interrelationship between manufacturing process and achievable accuracy grade

16.3.2 Manufacture type

Either "generated" or "formate" can be selected as the accuracy grade for the two processes. In gear sets with ratios greater than 2.5, the bevel gear (gear 2) is usually only "formate". The pinion (gear 1) is always generated. In a cyclo-palloid® toothing, the pinion and bevel gear are always generated.

Manufacturer's data for spiral teeth: Face Milling and Face Hobbing

The process used to manufacture bevel gears with spiral teeth is closely linked to this manufacturing process. There are two basic processes used here. The arc of circle toothing process (face milling, traditionally known as the Gleason process) and the continuous face hobbing (face hobbing, traditionally known as the Klingelnberg and Oerlikon process). For more details, see Calculation process (see chapter 16.1.3, Calculation according to Klingelnberg, Gleason and Oerlikon).

16.3.3 Cutter radius

In the case of spiral teeth bevel gears, the size of the cutter radius r_{c0} influences the curvature of the flanks and therefore also the properties of the bevel gear pair. This effect applies both to the position of the contact pattern and the strength, and must be taken into account when calculating the transverse coefficient K_{Fa} according to ISO 10300.

► **Note**

This parameter is not present if you use the Klingelnberg method to calculate strength. In that case, you select the cutter radius together with the machine type.

16.3.4 Number of blade groups

The number of blade groups describes the number of cutter blade groups on the cutter head used to manufacture bevel gears with spiral teeth and, when face hobbing is in use, this number, together with the cutter radius, influences the bevel of the tooth length. You must enter the number of blade groups as defined in ISO 23509, Annex E or as specified in the manufacturers' instructions.

16.4 Load

16.4.1 Methods used for strength calculation

You can select the following methods:

1. Bevel gears, static calculation
Implements the strength calculation for cylindrical gears (see chapter 15.2.1, Calculation methods).

2. Differential, static calculation
The static calculation can be used for differential gears. In a typical construction of rear axles with one bevel or hypoid gear on the differential housing, the torque on gear 2 (side shaft gear) is specified to calculate differential bevel gears. The torque on gear 2 is half the torque on the differential housing. You must also input the number of strands (click on "Details – Number of

strands"). For "2-pinion designs", input 2 strands. For "4-pinion designs", input 4, etc. The calculation is performed with the highest circumferential force F1 or F2 (see Figure 16.8)

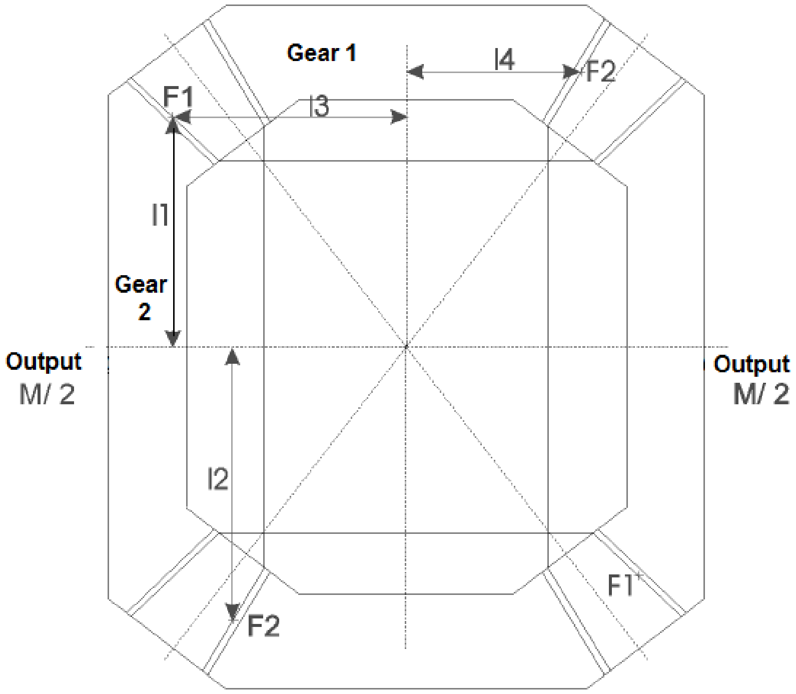


Figure 16.8: Bevel gears in differential gears

$$M = 2 \cdot F_1 \cdot l_1 + 2 \cdot F_2 \cdot l_2 \qquad (13.1)$$

$$F_1 = F_2 \cdot \frac{l_4}{l_3} \qquad (13.2)$$

$$l_1 = \sqrt{R_{m1}^2 - \left(\frac{d_{m1}}{2}\right)^2} - 0.5 \cdot \tan(\delta_1) \cdot (d_{am1} - d_{m1}) \qquad (13.3)$$

$$l_2 = \frac{d_{am2}}{2} \qquad (13.4)$$

$$l_3 = \frac{d_{am1}}{2} \qquad (13.5)$$

$$l_4 = \sqrt{R_{m2}^2 - \left(\frac{d_{m2}}{2}\right)^2} - 0.5 \cdot \tan(\delta_2) \cdot (d_{am2} - d_{m2}) \qquad (13.6)$$

3. Bevel gears according to ISO 10300, Method B (C)

ISO 10300, Parts 1, 2 ,3: Load capacity calculation for bevel gears.

4. Bevel gears according to ISO 10300 (2022)
ISO 10300, Parts 1, 2, 3, 4, 20: Load capacity calculation for bevel and hypoid gears.

5. Bevel gears according to AGMA 2003-B97 or AGMA 2003-C10
ANSI/AGMA 2003-B97 or AGMA 2003-C10: Rating the Pitting Resistance and Bending Strength of Generated Straight Bevel, Zerol Bevel and Spiral Bevel Gear Teeth

6. Bevel gears according to DIN 3991
DIN 3991, Parts 1, 2, 3, 4: Load capacity calculation for bevel gears.
This calculation is usually performed as defined in method B. The tooth form factor is calculated as defined in method C.

7. Bevel gears Klingelnberg KN 3028/KN 3030
This calculation is the same as the Klingelnberg in-house KN 3028 and KN 3030 standards. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

8. Bevel gears Klingelnberg palloid KN 3025/KN 3030
This calculation is the same as the Klingelnberg in-house KN 3025 and KN 3030 standards. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

9. Bevel gears Plastic
The equivalent cylindrical gear pair is calculated here (see also DIN 3991). Here the calculation is performed according to Niemann/VDI/VDI-mod. in the same way as the cylindrical gear calculation (see chapter 15, Cylindrical gears).

10. DNV41.2 Calculation guideline for ships' engines
The Det Norske Veritas calculation guideline [11] for ships' engines corresponds in principle to ISO 10300 (root, flank) and ISO 13989 (scuffing). However, it does have some significant differences, especially where S-N curves (Woehler lines) are concerned. These differences are detailed in our kisssoft-anl-076-DE-Application_of_DNV42_1.pdf information sheet, which is available on request.

11. Hypoid gears according to ISO 10300

12. Hypoid gears, according to Klingelnberg KN 3029/KN 3030
This calculation is the same as the Klingelnberg in-house KN 3029 and KN 3030 standards. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

14. Hypoid gears, according to Klingelnberg KN 3026/KN 3030
This calculation is the same as the Klingelnberg in-house KN 3026 and KN 3030 standards. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

► **Note**

For additional notes about the strength calculation as specified in Klingelnberg, see (see chapter 16.13, Notes about calculations according to the Klingelnberg standard).

16.4.2 Driving gear and working flank gear 1

To define flank alignment, you require the data for the driving gear (pinion or bevel gear) and the working flank (left flank or right flank) on gear 1. This data is then used together with the definition of the spiral direction (see figure below) to determine the flank alignment for the driving and driven gear.

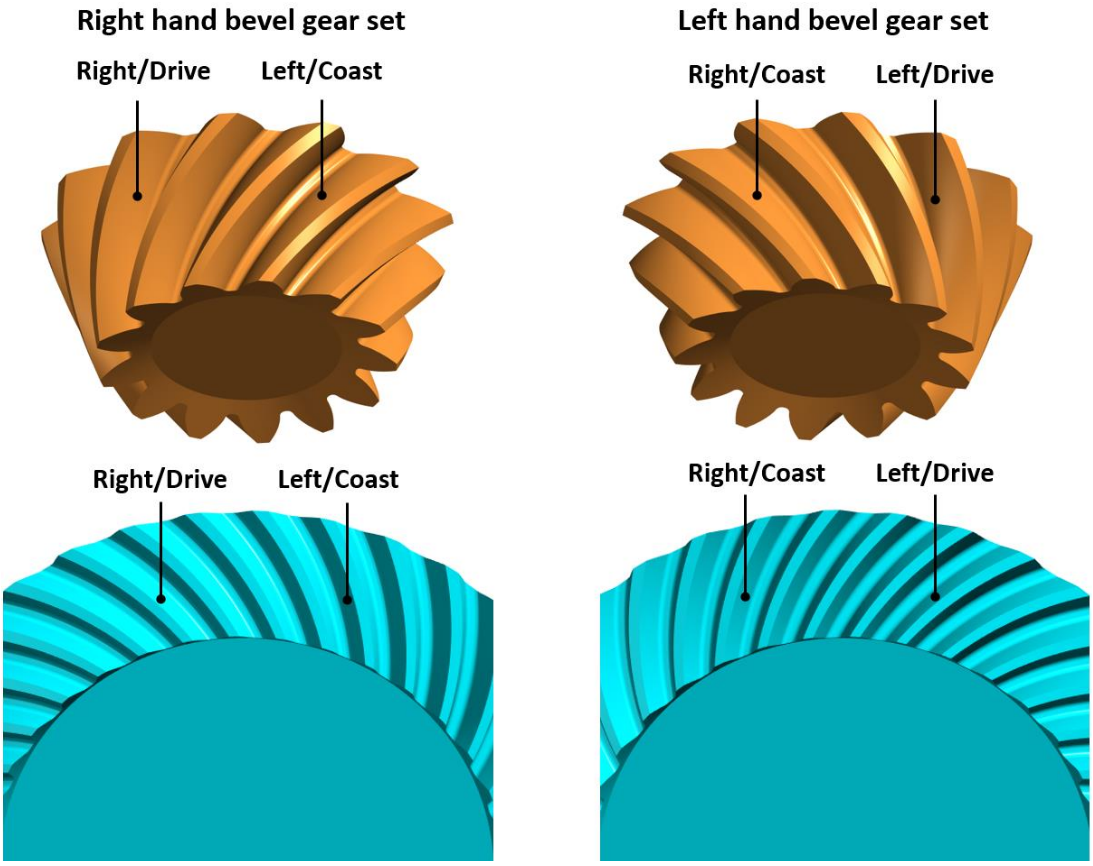


Figure 16.9: Definition of driving and driven flank for right- and left-hand helix bevel gears

16.4.3 Power, torque and speed

Click the Sizing button next to the power input field (for torque) to calculate the power (torque) that is needed to maintain a predefined minimum level of safety (see chapter 15.22.5, Safety factors).

16.4.4 Required service life

You enter the required service life directly in this input field.

Click the Sizing button to size the service life based on the minimum safeties for tooth root and flank strength. The service life for all the gears in the configuration is displayed. You can also click the Sizing button to size the service life with or without defining a load spectrum (see chapter 15.2.8, Define load spectrum). More detailed information about defining load spectra is provided here (see chapter 15.2.8, Define load spectrum).

16.4.5 Peak overload factor

The peak overload factor K_{AP} according to DNV 41.2 is used when calculating scuffing, when a short-period high overload, which is not covered by the application factor K_A, occurs. In this case, the root and flank are calculated with K_A, but the scuffing is calculated with K_{AP}. This procedure is especially useful in the case of special applications. It enables the methodology to also be used for calculations according to the ISO standard.

16.4.6 Strength details

Click on the **Details** button for the root and flank strength calculation to display a dialog in which you can make additional settings for the strength calculation.

The parameters described in other places are:

- Limited life (see chapter 15.2.6.6, Life factors as defined in ISO 6336)
- Modification of S-N curve (Woehler lines) in the range of endurance limit
- Tooth flank with load spectrum
- Tooth root with load spectrum
- Minor pitting (see chapter 15.2.6.10, Small amount of pitting permissible)
- Tooth mass temperature
- Lubricant factor XL
- Toothing is well run in
- Relative structural factor

16.4.6.1 Profile modification

Profile modification (in the sense of tip relief) is not usual for bevel gears. The run-in amount specified in ISO 10300 is the most commonly used.

16.4.6.2 Profile crowning (depth crowning)

ISO 10300 states that two values for profile crowning can be entered: "strong" or "low". This value changes the load distribution within the path of contact and therefore affects all the relevant safeties such as flank, root or scuffing.

16.4.6.3 Effective facewidth calculated with

Flank and root safety as defined in ISO 10300 is calculated with the length of the contact line on the middle of the tooth height l_{bm}. Select this checkbox to perform this calculation with a modified width instead of using the one defined in ISO 10300.

$$b' = (faktor) \cdot b$$

.

The usual contact pattern width is 0.85*facewidth (for example, as specified by ISO 10300). If you have sufficient experience, or are performing the calculation with contact analysis, you can modify this value.

► **Note**

You can only see this value if you are using the ISO 10300 calculation method.

16.4.6.4 Oil level

The oil level value is used to calculate scuffing according to ISO 10300-20. The depth of immersion influences power loss and therefore the bulk temperature.

16.5 Reference profile

16.5.1 Default values for tip clearance

The tip clearance for spiral bevel gears is usually 0.2 to 0.3 times the mean normal module. However, a greater amount of clearance is used for toothing that is manufactured with tilt. This prevents the tooth tip intersecting with the root of the opposing gear.

Default values are (as stated in the "Kegelräder" book produced by Klingelnberg [49]):

"Gleason, modified slot width" process: 0.3

"Gleason, constant slot width" process: 0.35

"Klingelnberg, palloid" process: 0.3

"Klingelnberg, cyclo-palloid" process: 0.25

"Oerlikon" process: 0.25

16.5.2 Default values for addendum coefficients

The addendum coefficient is usually 1.0.

16.6 Contact analysis

In the Bevel and Hypoid gears module, the contact analysis calculates the path of contact under load for bevel gears with straight, helical, and spiral teeth. Hypoid gears are not supported.

A pair of bevel gears with virtual cylindrical gear toothings are approximated for the analysis. Each one of the gears in this cylindrical gear pair has a number of teeth that varies across the facewidth, an operating pitch diameter, and a helix angle (spiral toothing). For a more detailed description of the theory of contact analysis, refer to the Cylindrical gear contact analysis section (see chapter 15.11, Contact analysis) and [21].

Axis alignment
The contact analysis takes into account the specified H, G and V misalignment, and also the direction of the torsion. As for contact analysis for cylindrical gear pairs, the deformation of the shafts can also be taken into account when calculating bevel gears (see chapter 15.3.7, Taking into account shaft bending (face load factor and contact analysis)). When the deformation of the shafts is taken into account, the equivalent H, G and V misalignment is also documented in the contact analysis report.

Other settings for the contact analysis can be made in the Module specific settings (see chapter 15.22.6.1, Calculation).

16.7 Modifications

The **Modifications** tab is where you define the profile and flank line modifications, a tip chamfer or a tip rounding.

You will find data about the tip chamfer and also the profile and flank line modifications here (see chapter 15.7, Modifications).
For technical manufacturing-related reasons, not all modifications that could be used for cylindrical gears are used for bevel gears.

Tip alterations are used for bevel gears, in special cases, to ensure sufficient tip clearance is achieved. The definition of the data to be input here is shown in the figure (see Figure 16.10).

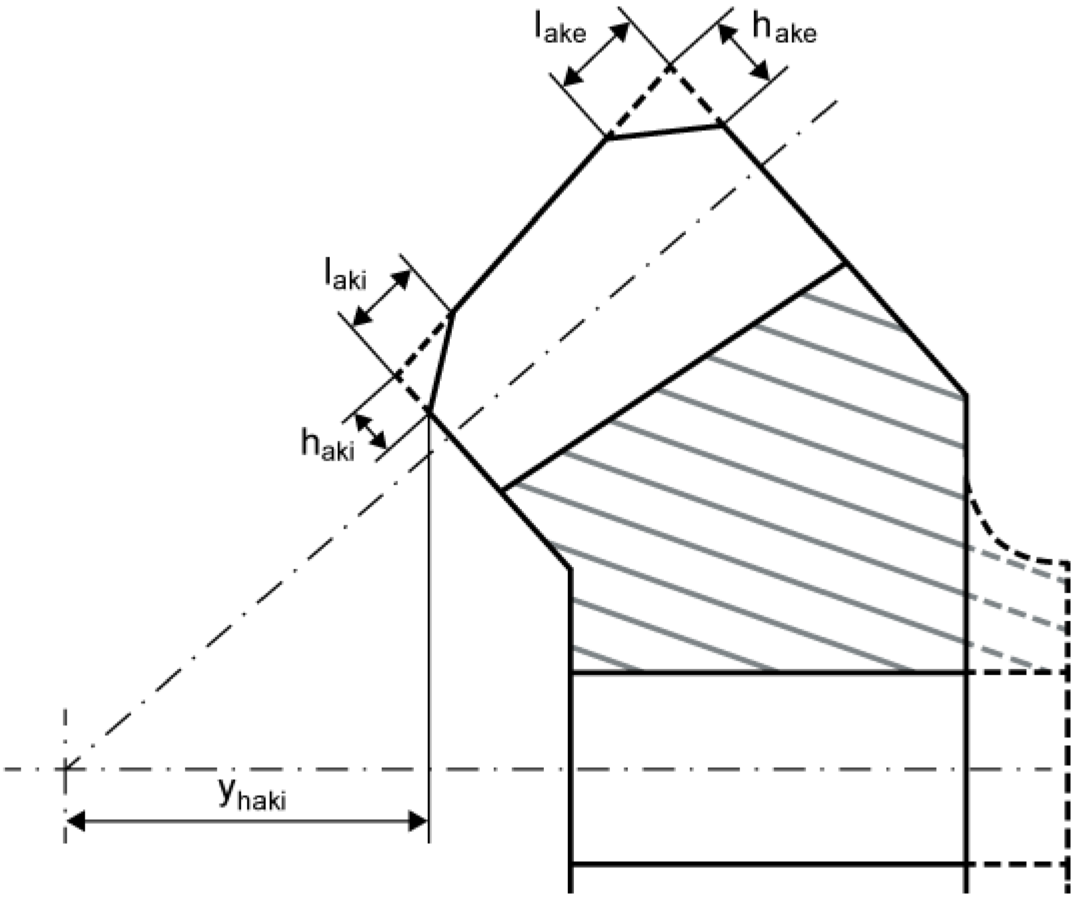


Figure 16.10: Tip alterations for bevel gears

Use the tip alteration Sizing button for the internal face to generate a suggested value for a constant tip width with 0.2*normal module, corresponding to length bk (tip relief width), as specified in the Klingelnberg in-house standard. To do so, the calculation according to Klingelnberg KN 3028 is required.

The length and width values for the gear body can be modified on the inside and outside (for the 3D view). Then, click the conversion button to generate modifications with a parallel axis bearing. This then opens a window, in which you can use the sizing function for the external face to generate a proposed value for the maximum possible height of the modification, h_ake, up to the external cone length diameter. The maximum possible length of modification l_ake is limited to half the facewidth so that the unchanged tooth height remains in the tooth middle, in the 3D model of the bevel gear.

Select the "**Modified blank**" option to generate special forms of the gear body(see Figure 16.11)

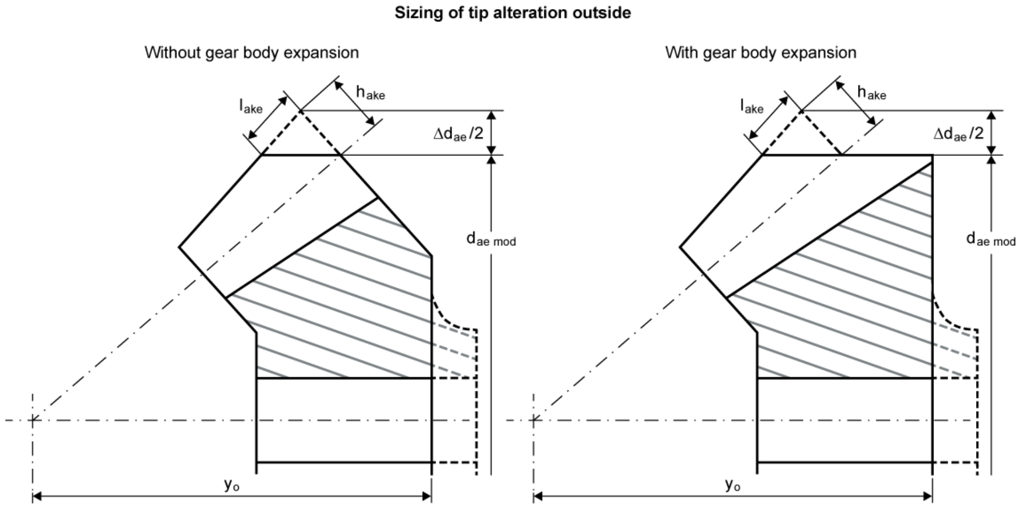


Figure 16.11: Gear body modification on the external face of the bevel gear

You can also click the conversion button to perform a conversion for the internal face. This then opens a window, in which you can use the sizing function for the internal face to generate a proposed value for the maximum possible height of the modification, h_aki, up to the internal cone length diameter.

Select the "**Modified blank**" option to generate special forms of the gear body(see Figure 16.12). The Sizing button in this window for "**Distance in axial direction to the pitch apex yaimod**" calculates yai in such a way that the bevel gear body is given a shape according to Δyai=0, (see Figure 16.12), left or right.

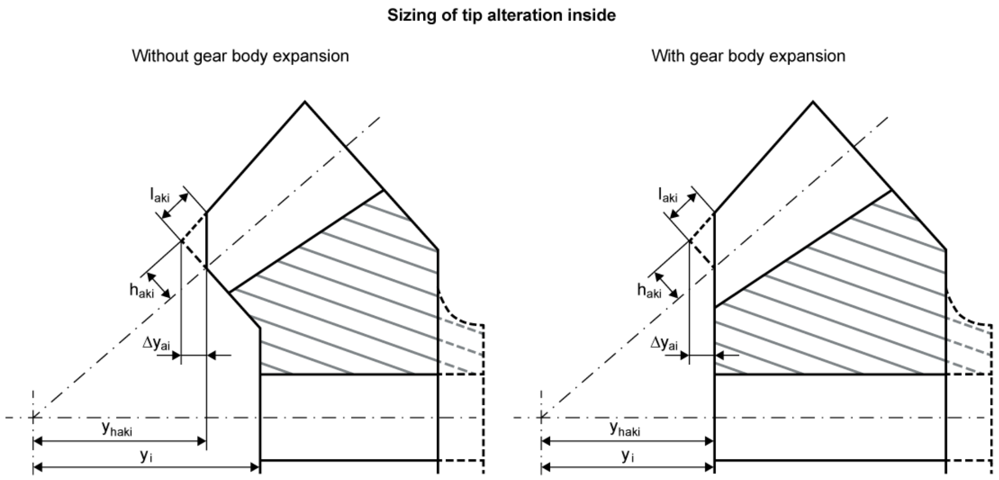


Figure 16.12: Gear body modification on the inside face of the bevel gear

16.8 Factors

16.8.1 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. The next table provides information about the coefficient values. You will find more detailed comments in ISO 10300, ISO 6336, DIN 3990 and DIN 3991.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 16.5: Assignment of operational behavior to application factor

16.8.2 Dynamic factor

To calculate the dynamic factor K_v, as defined by Klingelnberg, use the coefficient K_1 either for preliminary calculations based on the planned manufacturing method (lapped, HPG) or on the basis of the derived accuracy grade (see also Klingelnberg standard KN 3030, Table 5.2-1 or 5.2-2).

16.8.3 Face load factor

The face load factors $K_{H\beta}$, $K_{F\beta}$ and $K_{B\beta}$ take into account the influence of an uneven load distribution over the facewidth on the contact stress, tooth root stress and scuffing stress. You can specify that the face load factor is either to be set as a constant value or calculated from other values. If you already know the face load factor $K_{H\beta}$, select the **Own input** method and input this value.

The usual setting here is **Calculation according to calculation method**. The face load factor is then calculated according to the formulae used in the strength calculation standard (ISO, AGMA or DIN). You will need to input a number of values (mounting factor). Other values can be entered in the **Settings for face load factor** window.

The **calculation according to calculation method and Shaft misalignment** method calculates the face load factor according to the formulae in the strength calculation standard (ISO, AGMA, DIN). The misalignments V(E), H(P), J(G) and axis angle error are also calculated. The associated shafts that were stored in the shaft calculation are also used in the **Axis alignment …** window. Misalignments are calculated without iterating the load distribution in the tooth contact and therefore do not change the shaft bending line due to load distribution in the tooth contact.

Torsion of the gear body can be taken into account. Here, the calculation assumes a solid cylinder or hollow cylinder (external diameter = root circle + 0.4*normal module or operating pitch circle, depending on what has been predefined under **Settings**, bore = inside diameter) is involved. In other words, the inside diameter is taken into account and the torque on one side is zero. The torque is distributed in a linear fashion along the facewidth (parabolic course of deformation by torsion). You can select which side is to be subjected to torsional moment. In this case, I and II refer to the same side, as is also the case when you enter the toothing modifications.

The **Calculation according to ISO 6336 Annex E and misalignment of shafts** method calculates the face load factor on the basis of the tooth contact. As described in X, it calculates any gaping during meshing, and therefore defines the load distribution over the entire facewidth. The face load factor is calculated with iterating the load distribution in the tooth contact and therefore includes the change in the shaft bending line due to load distribution in the tooth contact. Shaft deflections and flank modifications are take into account in the load distribution. However, the face load distribution due to bevel and hypoid gear flank geometry (without modifications) is not taken into account and is assumed to be homogenous. As bevel gears and hypoid gears usually have higher length correction, you should specify the usual modifications in the **Modifications** tab to achieve a realistic calculation of the face load factor.

The facewidth is divided into slices to help you calculate the face load factor as defined in ISO 6336, Annex E: You can set the accuracy of the face load factor calculation according to Annex E in the **Define number of slices** dialog. Click the plus button next to the calculation method to open this dialog.

In **Settings for face load factor**, you can define the how load factors K_A and K_γ are used. They can be taken into account when calculating load distribution and axis alignment according to ISO 6336-1, Annex E.

The misalignments V(E), H(P), J(G) and axis angle error are also calculated. The associated shafts that were stored in the shaft calculation are also used in the **Axis alignment ...** window. Misalignments are calculated without iterating the load distribution in the tooth contact and therefore do not change the shaft bending line due to load distribution in the tooth contact.

16.8.3.1 Bearing application factor

The tables below show the **bearing type** → **mounting factor** for different standards.

Support for pinion and bevel gear	**Bearing application factor**		
	a	**b**	**c**
both on both sides	1.00	1.05	1.20
one on both sides, one floating	1.00	1.10	1.32
both floating	1.00	1.25	1.50

Table 16.6: Mounting factor according to ISO 10300

a: Contact pattern in the gearbox tested under full load
b: Contact pattern in the gearbox tested under partial load
c: Contact pattern only tested in specific tests

Support for pinion and bevel gear	**Bearing application factor**
both on both sides	1.10
one on both sides, one floating	1.25
both floating	1.50

Table 16.7: Mounting factor according to DIN 3991

Support for pinion and bevel gear	**Bearing application factor**
both on both sides	1.10
one on both sides, one floating	1.10
both floating	1.25

Table 16.8: Mounting factor according to AGMA 2003

The face load factors $K_{H\beta}$,$K_{F\beta}$ and $K_{B\beta}$ are calculated as follows from the mounting factor $K_{H\beta be}$ as defined in the standard:

$$K_{H\beta} = K_{F\beta} = K_{B\beta} = 1.5 \cdot K_{H\beta be} \quad (15.7)$$

16.9 Rough sizing

The method used to size bevel and hypoid gears, according to suggestions from technical literature [Kegelräder, pub. Klingelnberg], provides geometrically satisfactory sizing recommendations for gear pairs. This proposal does not provide sufficiently precise solutions to the problems of achieving the required safeties against tooth fracture and pitting, because it is based on values gathered through years of experience. If you verify gear teeth that have been dimensioned according to this method, you may discover certain deviations from the required safety values.

16.9.1 Facewidth ratio

Depending on how and where a gear unit is to be used, the facewidth should be in a specific ratio to the cone distance and correspond to the following values:

Light and medium-heavy load gear units for machines and vehicles	$3.5 \le (Re/b) \le 5.0$

Heavy load gear units for machines and vehicles	3.0≤ (Re/b) ≤ 3.5

16.9.2 Module ratio

The normal module mn should be in a ratio to the facewidth b within specific limits which can only be exceeded (or not reached) for exceptional reasons:

surface hardened bevel gears at risk of tooth fracture	7 ≤ (b/mn) ≤ 12
bevel gears at risk of pitting or through hardened or not hardened	10≤ (b/mn) ≤ 14

16.10 Fine sizing

To start the **Fine Sizing** process, click the **Calculation** menu and select the **Fine Sizing** option or click the  icon in the Tool bar.

If you input a nominal ratio, a center distance, intervals for the module and helix angle, and the pressure angle, the system calculates and displays all the possible suggestions for the number of teeth, module, helix angle and profile shift. It also shows the deviation from the nominal ratio, the specific sliding and the contact ratios. This module can also be used to size planetary stages and three gear chains.

All the variants found by this process can be evaluated by a wide range of different criteria (accuracy of ratio, weight, strength, etc.).

Depending on your requirements, limits can also be set on the most important parameters (minimum number of teeth, tolerated undercut, etc.). In addition to creating text reports detailing the solutions and the summary, the summary can also be displayed as a graphic.

16.10.1 Required entries in the standard tabs

Before you start the fine sizing process, you must enter the following data correctly in the **Basic data** or **Geometry** and **Strength** standard tabs to ensure the calculation returns the results you require.

Geometry:

- Reference profile
- Type: Standard, Gleason, Klingelnberg

Strength:

- Materials
- Power/Speed
- Application factor
- Required service life
- Lubrication

16.10.2 Conditions I

16.10.2.1 Maximum no. of solutions

If the program finds more than the specified number of solutions, you see a warning message and an appropriate note is entered in the report.

► **Note**

You should only perform a final evaluation after all the possible solutions have been displayed. Otherwise, you run the risk that the optimum solution will not be displayed.

16.10.2.2 Normal module (middle), reference diameter, length of reference cone

Use the three available options to vary and restrict the gear size.

16.10.3 Conditions II

You can define more parameters in the **Conditions II** tab.

16.10.3.1 Addendum coefficient gear 1 (middle), addendum coefficient gear 2 (middle)

You can vary the reference profile of the bevel gears by changing the addendum coefficients of gear 1 and gear 2. You can then calculate the dedendum coefficients of the counter gear (gear 2 and gear 1) by specifying the addendum coefficient and the "Required tip clearance".

16.10.3.2 Addendum angle gear 2, dedendum angle gear 2

By varying the addendum and dedendum angle on gear 2 you can then vary the tooth height along the facewidth. To calculate the addendum and dedendum angle on the mating gear (gear 1), input a constant tip clearance (parallel tip cone and root cone for the mating gear).

Restrictions due to gear type: You cannot vary the cone angle for gear types for which the angle cannot be changed. You cannot vary the addendum angle and root angle of the "Standard, Figure 1" type. Although the face angle can be varied for the "Standard, Figure 4" type, you cannot vary the root angle. The addendum angle and root angle cannot be varied at all for the "Uniform tooth depth, Figure 3" types.

► **Note**

These options for varying the parameters in **Conditions II** are useful for differential bevel gears, which are characterized by major geometric variations during manufacture. However, do remember that the usual conditions must be met when using conventional manufacturing methods for spiral teeth.

16.10.4 Conditions III

You can define more parameters in the **Conditions III** tab.

1. Show values of KISSsoft main calculation as additional variant with number 0

The toothing data displayed in the KISSsoft Basic tab can also be displayed as a variant with number 0 (table and graphic). However, **the data at the start of the fine sizing process must be consistent before this can happen.**

2. Only calculate geometry

If you select this setting, no strength calculation is performed.

3. Strength calculation with load spectrum

Before you can perform calculations with a load spectrum, you must specify a load spectrum in the KISSsoft main window before you start the fine sizing process and run the calculation (to ensure the data is consistent). In this case, when you start the fine sizing process, you are prompted to confirm that you want to perform the calculation with a load spectrum. Click the **Strength calculation with load spectrum** option to perform the calculation with a load spectrum, otherwise the calculation is performed without a load spectrum.

4. Suspend results which do not meet the required safeties

Variants which do not meet the predefined minimum safety levels (see **Calculation > Settings > Required safeties**) will be rejected.

5. Transmission error

If the **Calculation of the transmission error** option is selected, contact analysis is performed for every variant. During the transmission error contact analysis, most of the default settings are used to prevent the calculation generating an inaccurate result. However, the coefficient of friction and accuracy of calculation are not used. Input the settings in the main program, in the **Contact analysis** tab. You can also specify the accuracy of the calculation. We strongly recommend you use "medium" or "low" to reduce the processing time. As a consequence, the transmission error in fine sizing may not be exactly the same as you get in the contact analysis, depending on which settings have been selected.

- The default values are as follows:
  - Calculation for: right flank
  - Torque for gear A: not considered
  - Torque for gear B: not considered
  - Partial load range for calculation: 100 %
  - Center distance: Average center distance allowance
  - Single normal pitch deviation: 0 mm
- Then, the results list shows
  - Transmission error (PPTE)
  - Medium wear on the tooth flank (delwn1, delwn2)
  - Maximum flash temperature (theflamax)
  - Variation in bearing forces (VarL)

The calculation time increases significantly with the transmission error calculation option. For this reason, we recommend you limit the number of variants to be calculated before you start the calculation.

16.10.4.1 Ratio of cone distance to facewidth

A standard sizing characteristic value for bevel and hypoid gears is the "Ratio of cone distance to facewidth". If this flag is set, solutions which lie outside this range are rejected.

► **Note**

Make this range relatively small when calculating bevel gears with spiral teeth. Select a larger range for differential gear bevel gears.

16.10.4.2 Ratio of facewidth to normal module

A standard sizing characteristic value for bevel and hypoid gears is "Ratio of facewidth to normal module". Small values result in modules that tend to be large and sizings that are optimized for root strength. If this flag is set, solutions which lie outside this range are rejected.

► **Note**

Make this range relatively small when calculating bevel gears with spiral teeth. Select a larger range for differential bevel gears

16.10.4.3 Only take solutions into account if the following conditions are fulfilled:

The user can also define other criteria to ensure unsatisfactory solutions are rejected. These values are calculated and checked on the virtual cylindrical gear toothing:

1. Minimum distance of active diameter to form diameter d_{Nf} - d_{Ff}

Meshing interferences occur if the active root circle is less than the root form circle. Here you can specify a minimum value for the distance between the active root diameter and the root form circle, i.e. between active and manufactured involutes. The input value is the minimum difference between the two diameters. Only solutions greater than, or equal to, the input value are taken into account in the results view.

2. Minimum transverse pressure angle at a point on root form circle alphafF

For differential bevel gears, a minimum profile angle in the transverse section is required to ensure the axial demoldability. Only solutions greater than, or equal to, the input value are taken into account in the results view.

3. Minimum root rounding radius in the reference profile rhofp

A minimum root rounding radius may be required for reasons of manufacturability (absolute value in mm). Only solutions greater than, or equal to, the input value are taken into account in the results view.

4. Minimum tip clearance c

A minimum tip clearance may be required for reasons of manufacturability (absolute value in mm). This is compared with tip clearance c. Only solutions with a tip clearance greater than, or equal to, the input value are displayed in the results view.

5. Minimum tooth thickness on tip form circle sFvan

The minimum tooth thickness on the tip form circle, sFvan, is critical for achieving the required tip rounding radius. This calculation takes into account the tip alterations from the **Modifications** tab. Only solutions with a tooth thickness at the tip form circle that is greater than, or equal to, the input value are displayed in the results view. The tooth thickness sFvan is checked in the middle of the facewidth. Select **Additions for differential gears** in the Module specific settings if you also want the tooth thickness to be checked outside and inside, in sections I and II.

6. Manufacturing must be possible with tip rounding rK

Only solutions in which the tip rounding rK as defined by the entries in the **Modifications** tab can be executed are displayed in the results view.

► **Note**

If **Module specific settings –> Differential gears** has been selected, these criteria are also checked in an "inside" and "outside" section. Only solutions which meet the predefined criteria are then taken into account.

16.10.5 Results

Click the **Report** button to open the editor and display a list of the best results. A brief description of the criteria used to evaluate the best variants is given here. Note that these criteria are not relevant to every case, and only need to be queried in particular applications!

16.10.6 Graphics

The graphic in the Fine Sizing window gives you a quick overview of the number of solutions. Three parameters can be displayed simultaneously. You can change them in the selection lists. In addition to the two axes, the third parameter is displayed as a color.

16.11 Measurement grid

A measurement grid is required so that topological measurements can be performed on the flank surface. KISSsoft calculates the measurement grid in Gleason and Klingelnberg formats. For more precise instructions about these entries, please contact KISSsoft Support and request the document KISSsoft-anl-068-E-3D Geometry of Spiral Bevel Gear.pdf.

16.12 Topological modifications

When re-engineering existing bevel gears, simply import the measurement grid from an existing bevel gear into KISSsoft and then calculate a topological modification. For more precise instructions about these entries, please contact KISSsoft Support and request the document KISSsoft-anl-068-E-3D Geometry of Spiral Bevel Gear.pdf.

16.13 Notes about calculations according to the Klingelnberg standard

16.13.1 Bevel gears with cyclo-palloid® gear teeth

Geometry, manufacturability and strength calculation of bevel gears as defined in the Klingelnberg cyclo-palloid® process.

As stated in the Klingelnberg in-house standard KN 3028 (geometry and manufacturing) and KN 3030 (strength calculation) a complete calculation is performed for cyclo-palloid®toothing:

- Calculate machine distance for machine types FK41B, AMK400, AMK635, AMK855, AMK1602 with all relevant cutters, cutter radii and number of times the machines have been started. A warning is displayed if you select an incorrect machine type or cutter tip.
- You can specify any shaft angle, or angle modification here.
- Overall geometry, modules (inside, middle, outside), spiral angle (inside, outside), checks on cut back, undercut space, calculation of profile shift for balanced sliding, checks on backwards cut, checking and calculating the necessary tip shortening on the internal diameter, transverse contact ratio and overlap ratio, tooth form factor and stress correction factor.
- Calculation of all toothing dimensions.
- Calculates pitting, tooth root and scuffing load capacity (as defined by the integral temperature criterion) with all modifications in the KN 3030 in-house standard.

16.13.2 Hypoid gears with cyclo-palloid gear teeth

Geometry, manufacturability and strength calculation of hypoid gears (bevel gears with offset) according to the Klingelnberg process.

As stated in the Klingelnberg in-house standard KN 3029 (geometry and manufacturing) and KN 3030 (strength calculation) a complete calculation is performed for cyclo-palloid toothing:

- Calculate machine distance for machine types FK41B, KNC40, KNC60, AMK855, AMK1602 with all relevant cutters, cutter radii and number of times the machines have been started. A warning is displayed if you select an incorrect machine type or cutter tip.
- You can use any value as the shaft angle, angle modification, pressure angle for the driving and driven flank.

- Overall geometry with calculation of the facewidths, modules (inside, middle, outside), spiral angle (inside, outside), undercut boundary, calculation of gap widths, checks on backwards cut, checking and calculating the necessary tip shortening on the internal diameter, transverse contact ratio and overlap ratio, tooth form factor and stress correction factor either for the driving or driven flank.
- Calculation of all toothing dimensions.
- Calculation of pitting, tooth root and scuffing load capacity (as defined by the integral temperature criterion for the replacement crossed helical gear) with all modifications in the KN 3030 in-house standard.

16.13.3 Bevel gears with palloid gear teeth

Geometry and strength calculation of bevel gears according to the Klingelnberg process.

A complete calculation for palloid gear teeth is performed according to the Klingelnberg KN 3025 in-house standard (Geometry, Edition No. 10) and KN 3030 (strength calculation).

- Take into account palloid milling cutter dimensions by specifying a small diameter dK and milling cutter cut length SF. You can also input special milling cutters here.
- A warning is displayed if the cutter does not cover the crown wheel at either the inner or outer end of the tooth
- You can select any shaft angle, or angle modifications
- Overall geometry, modules (inside, middle, outside), spiral angle (inside, middle, outside), checks on profile shift for balanced sliding and undercut space, checking and calculating the necessary tip shortening on the internal diameter, profile and overlap ratio, tooth form factor and stress correction factor
- Calculation of all toothing dimensions
- Calculate forces for contact pattern core for reference cone length Rpr and Rm
- Calculate pitting, tooth root and scuffing load capacity (as defined by the integral temperature criterion) for all modifications in the Klingelnberg standard KN 3030 (taking into account the forces at cone distance Rpr)

► **Note**

The forces at cone distance Rm are used for the transfer to KISSsys, to ensure that forces can be calculated independently of the toothing procedure. However, including the theoretical contact pattern core in the Klingelnberg in-house standard is very difficult to implement in the manufacturing process.

16.13.4 Minimum safeties

We recommend you use the following minimum safeties:

Application	**Minimum safeties**
Flank	1.1 ... 1.2
Root	1.5 ... 1.6
Scuffing	1.8 ... 2.0

Table 16.9: Recommended minimum safeties

16.13.5 Surface roughness at tooth root

Treatment	**Surface roughness [mm]**
through hardened	0.016
lapped	0.016
hard-cut	0.008

Table 16.10: Surface roughness values

16.13.6 Manufacturing quality for bevel gears

Treatment	**Quality number**
through hardened	7
lapped	7
hard-cut	6

Table 16.11: Manufacturing quality for bevel gears

16.13.7 Characteristic number

The product of the lubrication, speed and roughness factor $Z_L Z_V Z_R$ for different surface treatments is shown in the next table:

Treatment	**Characteristic number $Z_L Z_V Z_R$**
Through hardened	0.85
Lapped	0.92
Hard-cut	1.0

Table 16.12: Characteristic number ZLZV ZR depending on surface finish

► **Note**

You will find a similar definition in ISO 10300-2:2001, section 14.4. Here the characteristic number is also dependent on the defined level of roughness R_z.

16.13.7.1 Single normal pitch deviation

This is calculated according to DIN 3965.

16.13.7.2 Meshing stiffness

The meshing stiffness is assumed to be constant.

$$c_g = 20 \frac{N}{mm \cdot \mu m}$$

16.14 Settings

In the **Calculation** menu, you will find the **Settings** option. Select this sub-menu option to display the **Module specific Settings** window. From here, you can access the tabs listed below to input other calculation parameters (see chapter 15.22, Settings)

16.14.1 General

During the mounting process, you can modify the mounting distance to achieve additional backlash. You can also specify how much additional backlash you require with Δj (enter this as a coefficient in the module). The required axial displacement for the integral pinion shaft Δα1 and the gear shaft Δα2 is then calculated according to ISO/TR 22849. The additional backlash that would be achieved by entering a predefined modification to the mounting distance is also calculated.

16.14.2 Calculations

16.14.2.1 Coefficient of friction for hypoid gears

Due to longitudinal sliding, hypoid gears have more power loss than spiral bevel gears. For this reason, the calculation of gear meshing forces in KN 3030 takes the coefficient of friction into account. If necessary, you can enter the size of the coefficient of friction in the **Module specific settings**.

16.14.3 Differential gears

16.14.3.1 Additional geometry calculations, external and internal

If the extensions for differential gears are selected, the geometry parameters are calculated at positions Li and Le. The data for the virtual cylindrical gear toothing at these two positions is then also documented in the report. The tip alteration can then also be applied up to underneath the cone length.

16.14.3.2 Entries for the webbing

As an alternative to entering these values manually, click on the **Modifications** tab to use an algorithm to enter the webbing values. The permissible pressure for the thrust washer is used to determine the external diameter of the thrust washer. The distance from bore to thrust washer (Delta05Bf) includes the radial distance of the bore to the internal diameter of the thrust washer and is used, together with the bevel gear bore, to determine the internal diameter. The value for the required webbing thickness at the thrust washer (SBfAS) includes the axial distance of the external diameter of the thrust washer to the webbing on the outside of the root. This distance is used to size the webbing on the outside of the root (webbing length and height, see **Modifications** tab).

The required inside wall thickness (for dFi) includes the radial distance from the bore to the webbing at the inside root.

Click the appropriate **Perform appropriate tip and root shortening automatically** option and the software will define the webbing in the **Modifications** tab. The root reductions are determined using the values input as described above. Tip reductions are defined using the value input as the necessary tip clearance for the root webbing (of the counter gear).

16.14.4 Helpful information about the Generation of 3D model tab

► **Note:**

For more precise instructions about these entries, please contact KISSsoft Support and request the document KISSsoft-anl-068-E-3D Geometry of Spiral Bevel Gear.pdf.

16.14.5 Calculation

Calculation method contact stiffness: Here you can select either the calculation method defined by Weber/Banaschek [21] (dynamic stiffness analysis: default setting), the method defined in ISO 6336-1 Method B and "Own input".

Fixing position of the tooth: The contact analysis according to Weber/Banaschek [21] is based on a bending beam model with a random form. This setting defines the method for determining the fixing position of the bending beam/tooth. You can select the methods according to Weber/Banaschek [21] or according to Langheinrich [22].

Single contact stiffness: If "Own input" has been selected as the contact stiffness calculation method, you can enter your own value for the single contact stiffness.

Coupling stiffness modification factor: The factor can be defined according to Raabe or Börner [36], see the next figure.

According to Raabe: $$C_{Ci} = k_C \cdot 0.04 \cdot N^2 \cdot \frac{C_i + C_{i+1}}{2}$$

According to Börner: $$C_{Ci} = k_C \cdot 2.75 \cdot \left(\frac{m_n}{b_c}\right)^2 \cdot \frac{C_i + C_{i+1}}{2}$$

C_{Ci} – coupling stiffness of slices i and $i + 1$, k_C – coupling stiffness factor
N – number of slices, m_n – normal module, b_c – slice width, C_i – stiffness of slice i

Figure 16.13: Coupling stiffness modification factor according to Raabe or Börner

Border weakening factor: Border weakening factor for a weakening of stiffness on the edge of helical gear teeth.

Correction factor for Hertzian stiffness (according to Winter): Correction factor for Hertzian flattening as described in the experiments performed by Winter/Podlesnik [37].

Number of orders in the amplitude spectrum (transmission error/contact stiffness): This is where you enter the number of orders to be calculated. At least one order must be calculated, and the calculation must be performed with no more than half the number of meshing positions (set this value in the **Contact analysis > Accuracy of calculation** tab).

Interpolate stress increase caused by tip rounding: In the case of a tip rounding, the calculation of the tooth form results in a sudden change in the radii of curvature. This in turn results in stress increases at this transition point in the contact analysis calculation. For this reason, you can specify whether the mathematical solution is to be used, to perform the calculation, or whether this stress increase is to be interpolated.

Calculate force excitation: Force excitation (according to FVA Report 487) results from toothing stiffness and the average transmission error. In contrast to the process for calculating transmission error, calculating the excitation force enables a better evaluation of how different toothing variants generate noise. This is because the gear meshing forces, not the equalizing movement (transmission error), of the gears, are the decisive factor in generating noise.

Take into account plastic deformation: Use this setting to specify whether plastic deformation is to be taken into account in the contact analysis. If plasticity is to be taken into account, the maximum contact stress, calculated using the elastic contact theory, is reduced on the basis of the specified

"Maximum permitted flank pressure". If the maximum elastic flank pressure is exceeded, the radii of the contact body are changed locally so that the resulting elastic i.e. contact stress matches this maximum value. Only a percentage rate of the new radii is used, on the basis of the specified "Weighting of the plastic deformation".

16.15 GEMS interface

16.15.1 Import

Click on **Import** to import an XML file from GEMS. The macrogeometry data for the bevel or hypoid gear (spiral teeth only) and the load data are then imported to KISSsoft.

16.15.2 Export

Click on **Export** to generate an XML file for GEMS. This contains the macrogeometry for the bevel or hypoid gear (spiral teeth only), the load data and the EPG data from KISSsoft.

17 Face gears

Face gears are a special type of bevel gear. Although a face gear pinion is a normal cylindrical gear, it has a complex 3D-tooth form. Unlike a bevel gear, a face gear is absolutely unaffected by axial displacement. For this reason, face gears are much easier to assemble.

The KISSsoft **Face gears** calculation module calculates the geometry of pairs of straight or helical cylindrical gear pinions with face gears with offset and with any shaft angle Σ. In this case, the strength and 2D geometry are calculated for an offset of 0 mm and a shaft angle Σ=90°. In every other case, you can perform the pre-sizing with these restrictions and then add the required hypoid offset and shaft angle to the 3D volume model. In the **Geometry** docking window, you can display the tooth form of a face gear for its inside, middle and outside diameter or for any number of sections all at the same time. You use this tool to check for undercut and pointed teeth on the internal or external diameter of the face gear. In the **Modifications** tab, you will find the value/length of tip alteration at outside or inside input fields, $h_{ake(i)}$ and $l_{ake(i)}$. Here, you can input additional parameters that will help prevent pointed teeth occurring in the gear. The system calculates the tooth form on the face gear by simulating manufacturing using a pinion type cutter. The strength calculation is based on the use of established standards for cylindrical or bevel gears.

17.1 Underlying principles of calculation

A face gear has common features with a curved rack. However, unlike this simple gearbox, when sizing a face gear, engineers are always confronted with the constraints imposed by that very bending. As the tooth flank in a straight-toothed face gear must run parallel to one radius of the face gear - the contacting pinion has flanks parallel to its own axis - the immediate result of the theorem of intersecting lines is that the pressure angle must reduce from outside to inside. This equation [50] is the central formula for sizing the geometry of face gears. Here it is applied for spur gear teeth. See equation (16.1).

$d_2 = \dfrac{m_n \cdot z_2 \cdot \cos\alpha_n}{\cos\alpha_2}$	(16.1)

with

d_2	diameter of face gear
m_n	normal module pinion
z_2	number of teeth on face gear
α_n	pinion pressure angle on the reference circle
α_2	pressure angle on face gear for diameter d_2

From this, you can, for example, define the pressure angle from the external diameter to the internal diameter. If the inside tooth flanks are steep, the involute will be short and only bear a small part of the tooth height. The risk of an undercut in the direction of the crown gear center grows. Any undercut here would further reduce the usable area. The result is a minimum internal diameter and a maximum external diameter, which limit the total facewidth of the face gear. This is a fundamental difference to the bevel gear set. A pair of bevel gears can transmit higher torques because of its increased facewidth. Face gears are limited in this respect. However, if you select the right axial offset b_v, i.e. by moving the facewidth middle b/2 relative to the reference circle d_{Pm}, you can optimize the maximum permitted facewidth.

When sizing a face gear, it is a good idea to define a minimum and a maximum pressure angle and then the achievable internal and external diameter. If external conditions limit this diameter (this usually affects the external diameter), you can use the conversion in equation (16.1) to change the range available for the module.

$$m_{\min/\max} = \frac{d_{2,\min/\max} \cdot \cos(\alpha_{2,\min/\max})}{z_2 \cdot \cos(\alpha_n)} \quad (16.2)$$

In addition to the bare numbers, you may find it helpful to look at a graphical presentation of the teeth.

The vast majority of applications use face gears with spur gears. However, face gears with helical teeth, when sized correctly, do offer a number of benefits related to noise reduction and strength. Unfortunately, these benefits are offset by the problem that the tooth flanks are not symmetrical, i.e. the left flank no longer matches the right flank. In practice, this means that any undercut that occurs will happen earlier on one flank than on the other. These flank differences also have a significant influence on strength, which results in a difference in the transmittable power between the directions of rotation. However, if only one sense of rotation is used, (as is the case for power tools), you can optimize the flank involved without taking into account the effect on the non-working flank.

Experience has shown that theoretical geometry considerations that describe the tooth form with involute functions, lines and arcs of circle, will sooner or later reach their limit. Tried and tested, and much more reliable, are tooth form calculations which are based upon the simulation of the meshing process, or, even better of the manufacturing process. Here, the trajectory of a point on the active surface of the tool is followed until its velocity normal to the tool surface reaches a zero crossing (see Figure 17.1).

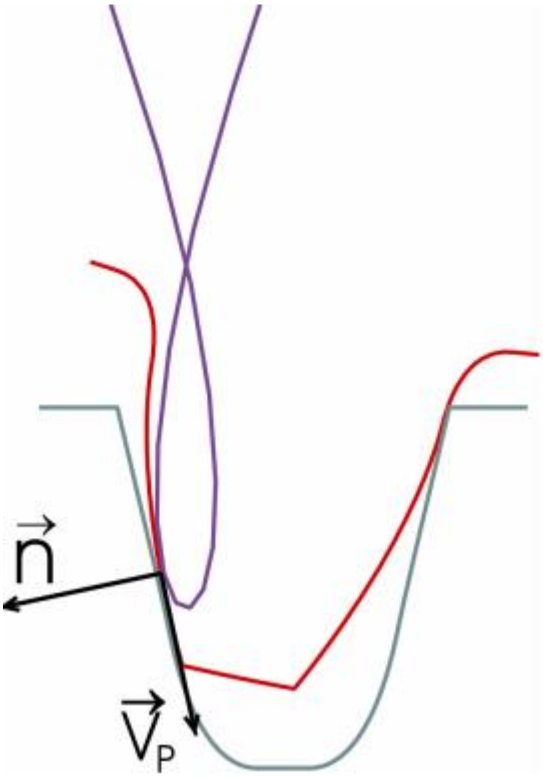


Figure 17.1: Spur curve (blue) of the pinion type cutter tool (red) on the face gear (green)

These places are potential points on the tooth form surface. The actual points on the surface must then be identified separately from the “imaginary” points at which, although the normal speed also disappears, the remaining points are also marked as being outside of the material. One of the most difficult aspects of the procedure described here is how to separate the real points from the imaginary points. In addition to referring to the usual standard algorithms for classifying points in a level, you must also use empirical approaches that use the known properties of the tooth form to be sure of achieving a well-defined tooth form with sufficient safety. This enables you to match the data derived from calculating a 3D tooth form for a face gear with the data derived from generating with a pinion type cutter, using a classic manufacturing method. By outputting the 3D body in IGES, STEP or SAT format, you can design the form in any CAD system. The face gears can be manufactured in an injection molding, sintered or precision forging process. However, 2D cross section view is much more suitable if you want to check a face gear for undercut or pointed tooth tip. This displays the inside, middle and outside of the face gear tooth form all at the same time. If you rotate the gears step by step, you can check every aspect of gear generation very accurately. If a tooth is pointed, or if the meshing ratios are not good enough, you must reduce the tooth height in the same way as you do for hypoid gears. To reduce the gear’s sensitivity to errors in the axis alignment or the center distance, you can permit flank line crowning on the tooth flank (flank line). You can generate this quite easily for face gears by using a pinion type cutter that has one or more teeth more than the pinion in the manufacturing process [3]. When you compare the tooth forms, you can see the effect that the increased number of teeth on the pinion type cutter had on the generated tooth form. However, if the face gear has a large axial offset b_v, the crowning can shift to one side! In every axial section through the cylindrical gear, the face gear gear unit corresponds to a pinion-rack gear unit.

Using the rack theory as a basis, you can therefore define the pressure angle, contact lines and contact ratio in each section.

The examples in this section are based on the publication in [51].

17.2 Basic data

17.2.1 Normal module

Enter the normal module. However, if you know the pitch, transverse module or diametral pitch instead of this, click on the Convert button to display a dialog window in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select **Input normal diametral pitch instead of normal module** by selecting **Calculation > Settings > General**.

If the geometry of a face gear has been completely defined, you will receive the following message after clicking the Convert button:

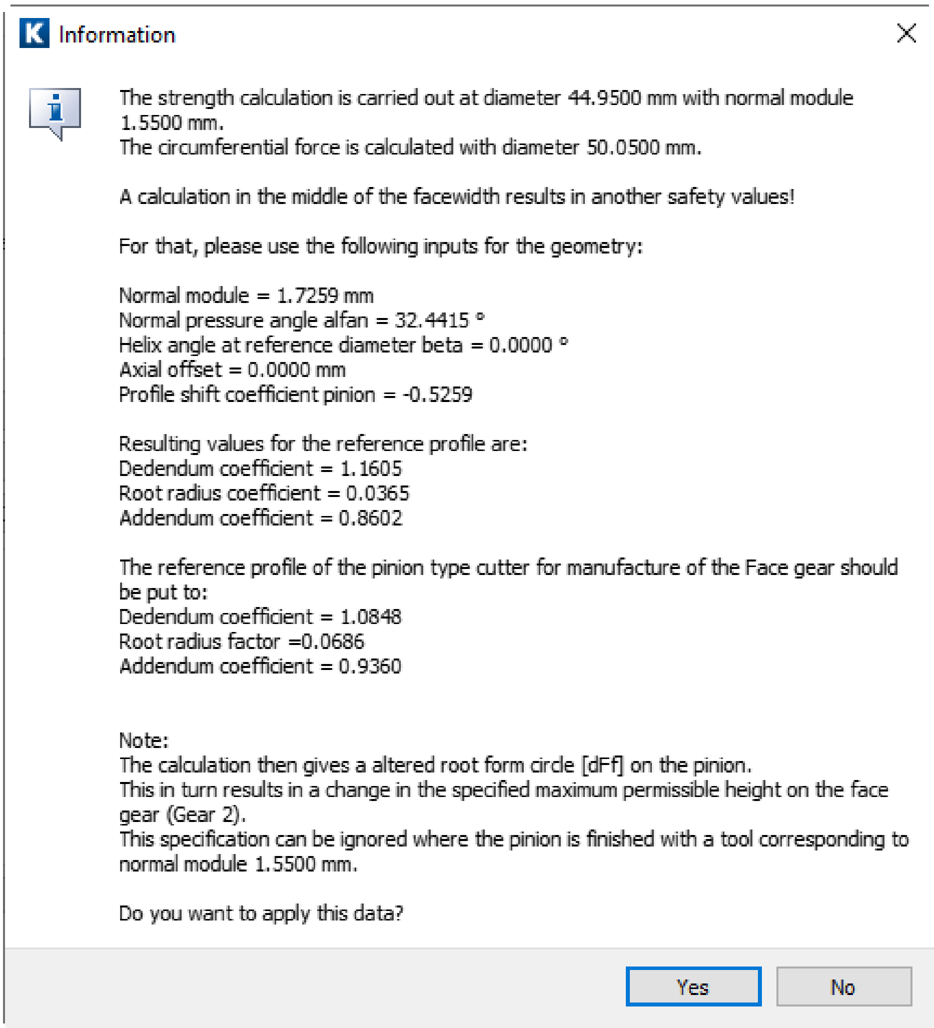


Figure 17.2: Information window for sizing the normal module

The strength calculation is performed for the mean diameter of the face gear as part of the bevel gear calculation performed according to ISO 10300 or DIN 3991. If the axial offset b_v <> 0, the conditions for this type of calculation have not been met. For this reason, the functionality triggered with the Sizing button supports the conversion of normal module m_n and pressure angle α_n, to ensure that b_v = 0. Although this changes the root fillet radius of the pinion, the shape of flank remains the same.

▶ **Note**

We recommend you only use this conversion method when you perform the strength calculation. The conversion changes the module and you can no longer use the tool. For this reason, you must save your geometry data before you perform the conversion.

17.2.2 Pressure angle at normal section

The normal pressure angle at the reference circle is also the reference profile flank angle. For standard meshings, the pressure angle is α_n = 20°. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Greater pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

▶ **Note**

The working transverse pressure angle α_{wt} changes across the width of the gear teeth.

17.2.3 Helix angle at reference circle

Enter the helix angle in [°]. Click the Convert button in the **Convert helix angle** window to calculate this angle from the helix angle at base circle β_b or from the helix angle at tip circle β_a. Helical gear teeth usually generate less noise than spur gear teeth. However, they also have the disadvantage that they involve additional axial force components.

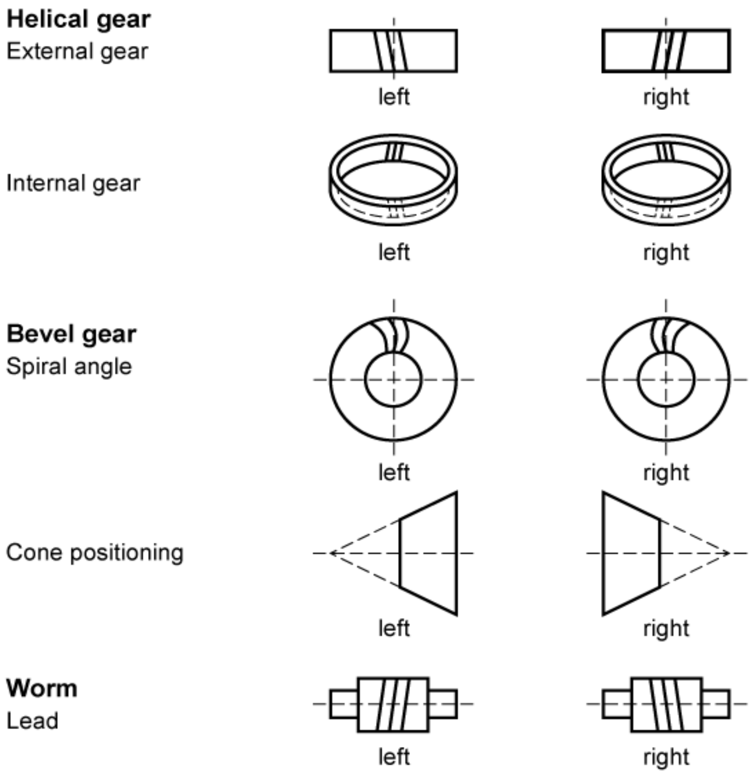


Figure 17.3: Helix angle

17.2.4 Axial offset

The axial offset is the distance from the pinion center to the mean diameter of the face gear.

Click the Sizing button to the right of the **Axial offset** input field to calculate greatest possible width of the face gear b_2 and the corresponding axial offset b_v, so that the pressure angle lies within the predefined limits.

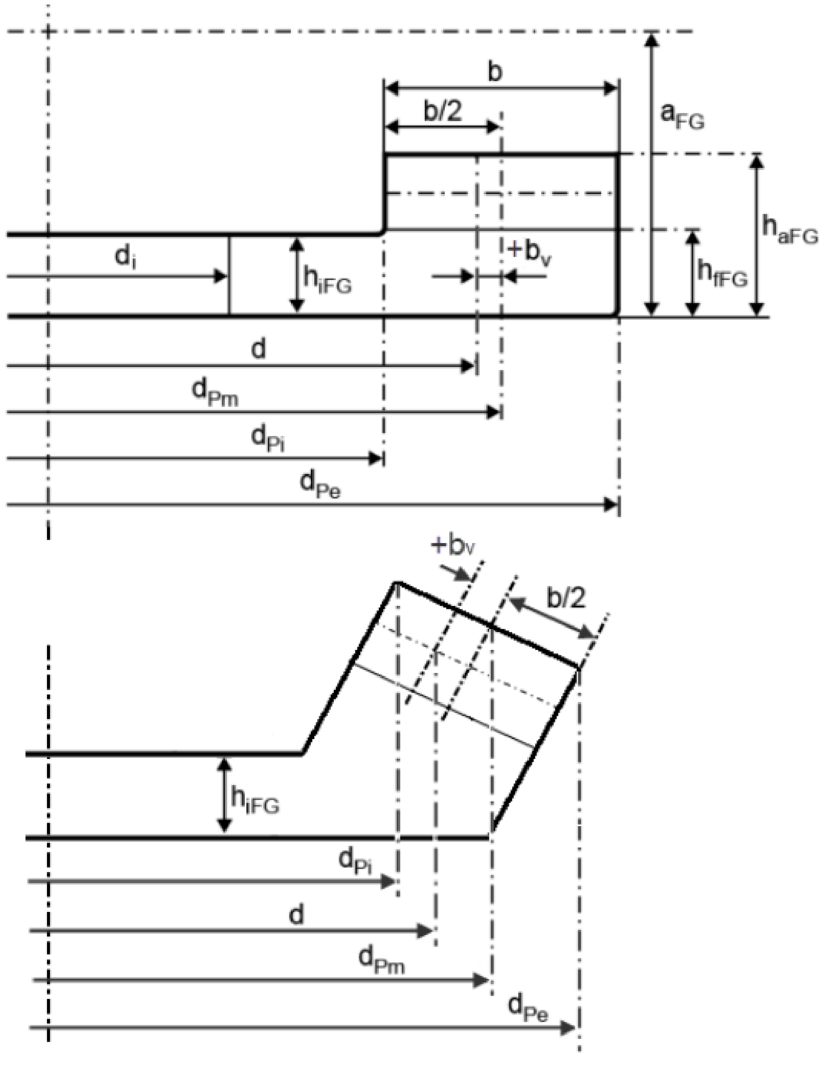


Figure 17.4: Axial offset of the face gear

17.2.5 Profile shift coefficient

The tool can be adjusted during the manufacturing process. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift, the tool is pushed deeper into the material, with the result that the tooth thickness is smaller and there is more danger of undercut. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

You can modify the profile shift according to different criteria. To achieve this, use the various sizing options provided by clicking the Sizing button in the **Sizing of profile shift coefficient** window:

- **For undercut boundary**
- **For minimum topland per gear.**
  You can specify the minimum thickness of the topland under **Calculation > Settings > General > Coefficient for minimum tooth thickness at the tip**.

► **Note**

The pinion should have a reasonable high value for the tooth thickness at the tip because the pinion type cutter used to manufacture a face gear has a somewhat higher tip and still must not be permitted to become pointed.

Click the Convert button and KISSsoft will determine whether the profile shift coefficients (see chapter 15.1.8, Profile shift coefficient) are to be taken from measured data or from values given in drawings.

17.2.6 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, click on the Settings button and then select the **Input quality** option. The accuracy grade according to ISO 1328 (DIN ISO 1328) is very similar to the same quality in AGMA 2015.

The manufacturing qualities that can be achieved are displayed in the next table.

Manufacturing process	Quality according to ISO		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 17.1: Accuracy grades for different manufacturing processes

► **Note**

The values in brackets can only achieved in exceptional situations.

You can also click the Settings button to set different quality-related options. The following options are available:

Input quality

The manufacturing deviations that are output in the report and used for particular coefficients in the strength calculation procedure are defined either in the ISO 1328 (DIN ISO 1328), DIN 3961:1978 or AGMA 2015 standards. You can specify which standard is to be used. If you click the **Calculation method for strength** option, the system applies the standard that is best suited to the strength calculation method (for example, ISO 1328 is used if you are using the ISO 6336 calculation method).

Varying qualities

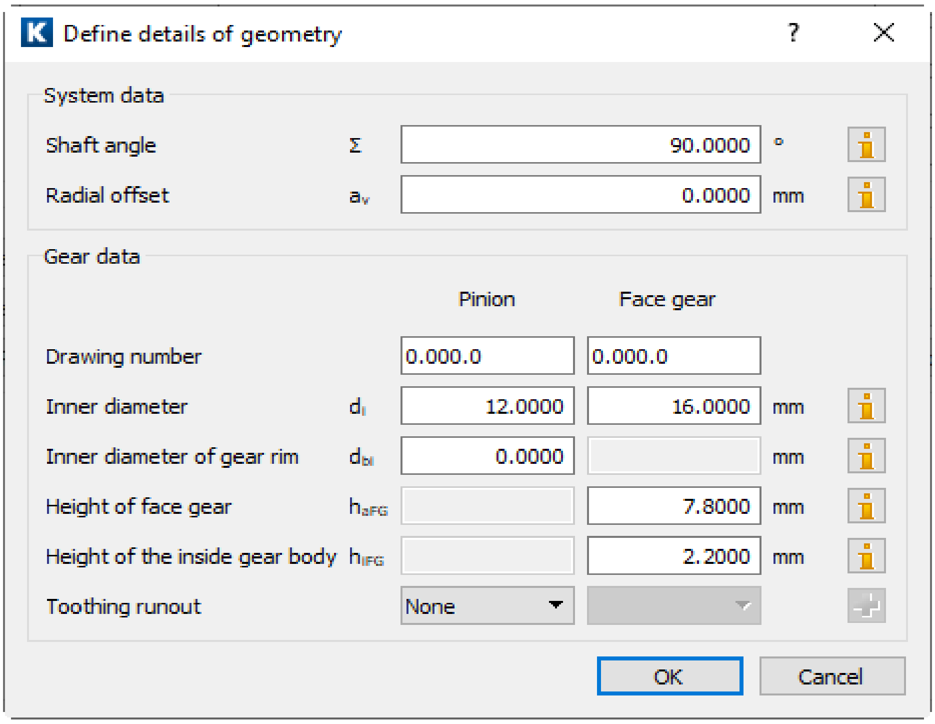
If you select this option, the Plus button is displayed next to the Quality entry field in the main screen. Click this button to input specific tolerances manually. You will find a more detailed description of this in Qualities (see chapter 15.1.10, Quality).

Fp tolerance as specified in tables in DIN 3962
The total cumulative pitch deviation Fp given in the tables in DIN 3962 can be very different from the Fp calculated using the formulae in DIN 3961.

Extrapolate tolerance values
The tolerances detailed in ISO 1328:2013, DIN ISO 1328:2018, AGMA 2000 and AGMA 2015 are calculated using the formulae in each particular standard and with the effective geometric data (mn, d, b…). The range of validity must be defined in each case. For example, the tolerances specified in ISO 1328 apply for a module range 0.5 mm <= mn <= 70 mm. However, these formulae cannot be applied to gear teeth that lie outside of the range of validity. Despite this, these formulae are still used in such cases, due to the lack of any other information. In KISSsoft, the relevant limiting value is usually used to determine tolerances (for example, in the ISO standard, the tolerance is defined as 70 mm, and for a module it equals 80 mm). Alternatively, you can select **Extrapolate tolerance values** to calculate tolerances using the effective value (i.e. with 80 mm). In this case, in ISO 1328 (2013 edition), tolerances are also output when the helix angle is greater than 45°.

In DIN 3961:1978 and ISO 1328 (1996 edition), tolerances are calculated with the geometric mean values and therefore no extrapolations can be performed here.

17.2.7 Geometry details



Click the **Details...** button in the upper right-hand part of the Geometry area to display the **Define details of geometry** dialog window. You can enter these parameters here.

17.2.7.1 shaft angle

You can select your own shaft angle here. However, to perform a strength calculation you should set it to $\Sigma = 90°$.

17.2.7.2 Internal diameter

The internal diameter is needed to calculate the mass moment of inertia. As defined in ISO or AGMA, the gear rim thickness does affect the strength. For solid wheels, enter 0. For external wheels with webs, enter the relevant diameter d_i.
The internal gear rim diameter is required for calculations according to ISO or AGMA. Where thin gear rims are used, this factor can greatly influence the calculation results, as shown in the figure.

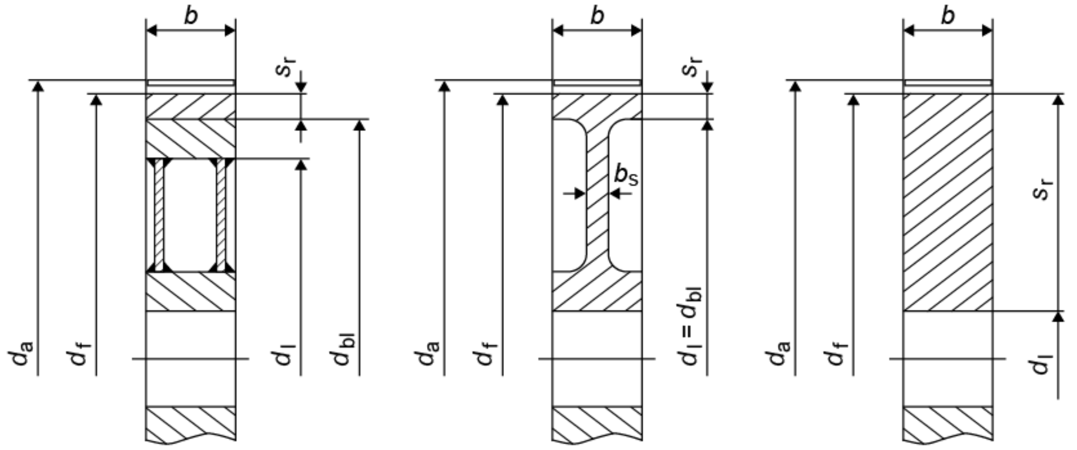


Figure 17.5: Dimensioning the diameter

17.2.7.3 Height of face gear

To define the height of face gear h_{aFG} (see Figure 17.8).

17.2.8 Material and lubrication

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select **Own Input** from the list or enter the material in the database (see chapter 9.4, External tables) first. Click the Plus button to display the **Material pinion (Face gear)** window, in which you can select your material from a list of materials that are available in the database. Select the **Own Input** option to enter specific material characteristics. This option corresponds to the **Create a new entry** window in the database tool.

17.3 Load

17.3.1 Methods used for strength calculation

To enable developers to use the calculation method they require, KISSsoft can perform the strength calculation either according to ISO 6336, DIN 3990, DIN 3991, ISO 10300 or DIN 3991.

17.3.1.1 Only geometry calculation

If you select this method, no strength calculation is performed. Therefore, you no longer need to enter the data that is only required for the strength, such as power, application factor, etc.

17.3.1.2 Static strength

- The strength calculation for cylindrical gears is implemented here (see chapter 15.2.1, Calculation methods).

17.3.1.3 Method ISO 6336-B/Literature

We recommend you use the method described here.

The method used to calculate the strength of face gears as originally proposed by Crown Gear [50], is based on the cylindrical gear calculation according to DIN 3990. The inclined lines of contact in a face gear increase the total contact ratio due to pitch overlap. This can be compared with the overlap ratio in helical gear cylindrical gears (an overlap ratio is also present in helical face gears due to the helix angle β_n). You can therefore derive the virtual helix angle β_v from the inclination of the lines of contact. In the strength calculation, this effect is taken into account by helix load factors Y_β and Z_β. The value at the middle of the facewidth is then used as the transverse contact ratio ε_a. It is clear that the face load factor $K_{H\beta}$ and transverse coefficient K_{Ha} according to DIN 3990 cannot be used for face gears. In crown gear calculations, these values are usually set to $K_{H\beta}$ = 1.5 and K_{Ha} = 1.1, and therefore enable the same procedure to be used as the one for calculating bevel gears (DIN 3991, ISO 10300). However, the international acceptance of the strength calculation method specified in ISO 6336 makes it a logical alternative to DIN 3990. As ISO 6336 is very similar to DIN 3990, the same restrictions also apply.

In contrast to the Crown Gear program, the following data is used in the calculation:

- The arithmetical facewidth (pitting) corresponds to the minimum contact line length (Lcont)

- The circumferential force F_t is determined from d_{Pm} (middle of facewidth)

17.3.1.4 Crown Gear Method (DIN 3990)

This calculation method produces results that correspond to those produced by the Crown Gear program. The underlying principle of calculation is described earlier in the "ISO 6336/Literature" (see chapter 17.3.1.3, Method ISO 6336-B/Literature) method.

The main differences between it and the "ISO 6336/Literature" method are:

- The calculation is based on the method defined in DIN 3990.
- The arithmetical facewidth (pitting) corresponds to the facewidth (even if the minimum contact line length is shorter than the facewidth).
- The circumferential force F_t is determined from d_{Pd} (reference circle = module * number of teeth), even if d_{Pd} is not the middle of the facewidth.

17.3.1.5 Similar to ISO 10300, Method B

As already mentioned, you can use ISO 10300 as a good alternative method for calculating the strength of bevel gears. Face gears are classified as bevel gears and can therefore be regarded as bevel gears where the cone angle is 0° (pinion) and 90° (face gear). The strength of bevel gears is calculated on the basis of the virtual cylindrical gear (cylindrical gear with the same tooth form as the bevel gear). However, for a face gear the virtual gear number of teeth for the pinion is $z_{1v} = z_1$ and for the gear z_{2v} it is infinite. If you verify the examples, using the Crown Gear program (similar method to the one defined in DIN 3990) and the ISO 10300 method in KISSsoft, you will get a good match of values. The deviation in root and flank safeties is less than 10% and usually less than 5%. This shows that both calculation methods in DIN 3990 and ISO 10300 (DIN 3991) are reliable and effective.

17.3.1.6 Analogous to DIN 3991, Method B

The same notes as for the "Analog to ISO 10300" (see chapter 17.3.1.5, Similar to ISO 10300, Method B) method also apply here.

17.3.2 Service life

The value in the **Service life** input field is used together with the speed to calculate the number of load cycles.

17.3.2.1 Number of load cycles

KISSsoft calculates the number of load cycles from the speed and the required service life. If you want to influence the value, you can define it in the **Number of load cycles for gear** n **window**. Click the Plus button to access this. Here, you can select one of five different calculations for calculating the number of load cycles.

1. **Automatically** The number of load cycles is calculated automatically from the rating life, speed, and number of idler gears.
2. **Number of load cycles** Here, you enter the number of load cycles in millions. You must select this option for all the gears involved in the calculation, to ensure this value is taken into account.
3. **Load cycles per revolution** Here you enter the number of load cycles per revolution. For a planetary gear unit with three planets, enter **3** for the sun and **1** for the planets in the input field.

   Note:
   If the **Automatically** selection button in the calculation module is selected, KISSsoft will

determine the number of load cycles, taking into account the number of planets, in the **Planetary stage** calculation module.

4. **Load cycles per minute** Here you enter the number of load cycles per minute. This may be useful, for example, for racks or gear stages where the direction of rotation changes frequently, but for which no permanent speed has been defined.
5. **Effective length of rack** The rack length entered here is used to calculate the number of load cycles for the rack. The rack length must be greater than the gear's perimeter. Otherwise, the calculation must take into account the fact that not every gear tooth will mesh with another. You must enter a value here for rack and pinion pairs. Otherwise the values N_L(rack) = N_L(pinion)/10 are set.

► **Note**

This calculation method is used for transmissions that only travel over one oscillation angle.

Assume a scenario in which a reduction is present,

$$i = \frac{z_2}{z_1}$$

and an oscillation angle w in [°] from gear 2, where gear 2 constantly performs forwards and backwards movements with the angle value w_2. The effective endurance is given as the service life. The two coefficients f_{NL1} and f_{NL2}, which modify the absolute number of load cycles, N_L, are now calculated. To do this:

- a) Set the alternating bending factor of the pinion and gear to 0.7, or calculate it as defined in ISO 6336-3:2006. In this case, one complete forwards/backwards movement is counted as one load cycle.
- b) Coefficients f_{NL1} and F_{NL2} for pinion and gear are defined as follows:

$$f_{NL1,2} = \frac{ROUNDU\,P(\frac{W_{1,2}}{360})}{2 * \frac{W_{1,2}}{360}}$$

- w_2 = oscillation angle gear 2
- w_1 = W2*i
- ROUNDUP = round up to a whole number

The value in the counter displays the actual number of loads that occur during a complete cycle (forward and backward oscillation) on the flanks (not teeth) that are most frequently subjected to load. By rounding up this number to the next whole number, every rotation recorded is counted as a load.

Then, to determine the required $f_{NL1,2}$ factor, the actual number of loads that occur per flank is divided by the number of loads that would occur per cycle, if rotation were to continue without a backward rotation at the angle of rotation (1 load for each 360°).

Example calculation for $f_{NL1.2}$:

Gear 1 rotates through a half cycle at **540°** while gear 2 oscillates by **90°** (i = 6).

In a complete cycle, the oscillation angle moves forwards once an backwards once.

The actual number of load cycles that occur in a complete cycle on the flanks that are most frequently subjected to load (only one side of the tooth is taken into consideration) is then:

For gear 1:

$$ROUNDUP(\frac{540}{360}) = 2$$

For gear 2:

$$ROUNDUP(\frac{90}{360}) = 1$$

Without adjusting the coefficients, the number of counted load cycles in a complete cycle would then be:

For gear 1:

$$2 * (\frac{540}{360}) = 3$$

For gear 2:

$$2 * (\frac{90}{360}) = 0.5$$

The coefficients are therefore f_{NL1} and f_{NL2}:

$$f_{NL1} = \frac{2}{3} = 0.667$$

$$f_{NL2} = \frac{1}{0.5} = 2$$

- c) Then, input coefficients fNL1 and fNL2 in the **Load cycles per revolution** input field.

The strength calculation can now be performed for the correct number of load cycles, on the basis of the data entered in steps a through d.

17.3.3 Power, torque and speed

Click the Sizing button next to the power input field (for torque) to calculate the power (torque) appropriate to maintain a predefined minimum level of safety (see chapter 15.22.5, Safety factors).

Click the Plus button next to the Speed input field to enter the direction of rotation of the face gear as specified in the **Define sense of rotation** window (see Figure 17.6).

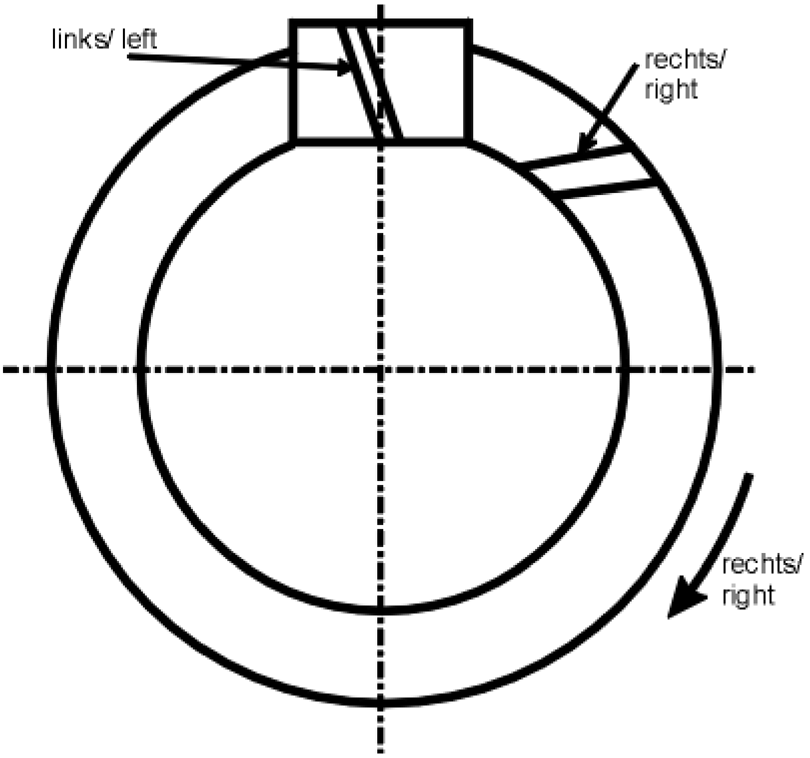


Figure 17.6: Helix angle on a face gear: right; helix angle on the pinion: left; direction of rotation: to the right

17.3.4 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. The next table provides information about the coefficient values. You will find more detailed comments in the ISO 6336 standard.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 17.2: Assignment of operational behavior to application factor

17.3.5 Strength details

Click on the **Details** button for the root and flank strength calculation to display a dialog in which you can make additional settings for the strength calculation.

17.3.5.1 Profile modification

You can modify the theoretical involute in high load capacity gears by grinding the toothing. You will find suggestions for sensible modifications (for cylindrical gears) in KISSsoft module Z15 (see chapter 15.7, Modifications). The type of profile modification has an effect on how the safety against scuffing is calculated. The load sharing factor Xγ is calculated differently depending on the profile modification. The main difference is whether the profile has been modified or not. However, the differences between the versions for **high load capacity gears** and for **smooth meshing** are relatively small. The strength calculation standard presumes that the tip relief C_a is properly sized, but does not provide any concrete guidelines. The load sharing factor Xγ is calculated as follows, depending on the type of profile modification according to DIN 3990:

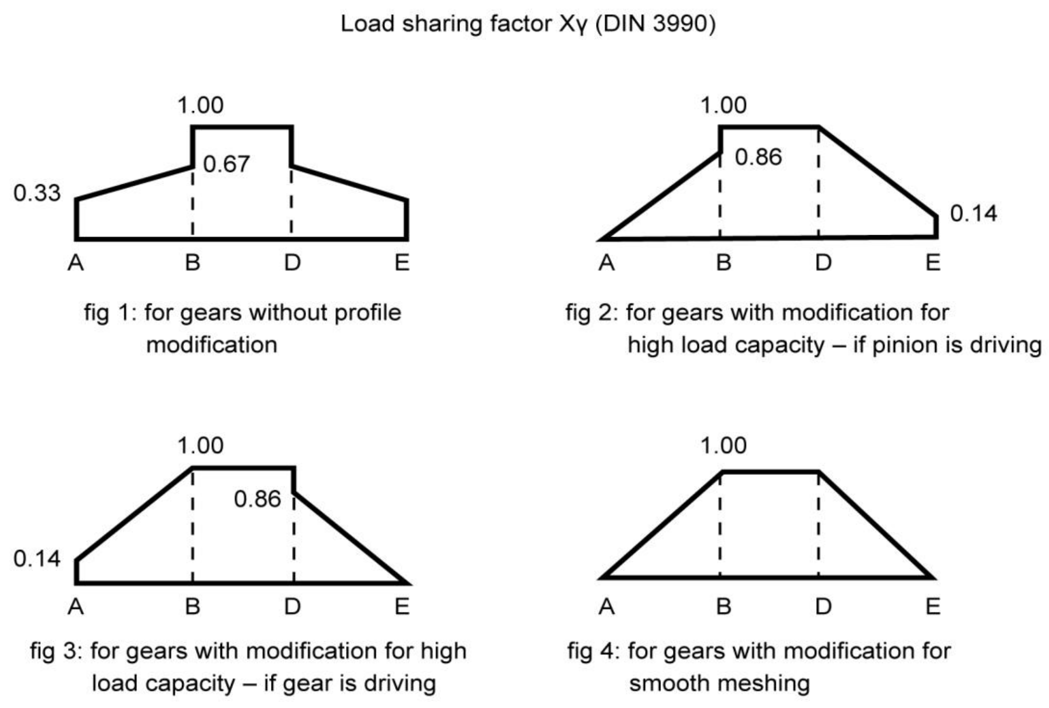


Figure 17.7: Force distribution factor Xγ for different profile modifications

17.3.5.2 Limited life coefficients as defined in ISO 6336

Set the limited life coefficient Z_{NT} to reduce the permitted material stress according to ISO 6336-2:2006:

$\sigma_{H\lim red} = Z_{NT} \cdot \sigma_{H\lim}$	(12.14)
$\sigma_{F\lim red} = Y_{NT} \cdot \sigma_{F\lim}$	(12.15)

As stated in ISO 6336, this value is important for cylindrical gear calculations and is the reason for the lower safeties for the range of endurance limit, compared with DIN 3990.

1. **normal (reduction to 0.85 for 10^{10} cycles):** The permitted material stress in the range of endurance limit (root and flank) is reduced again. The limited life coefficients Y_{NT} and Z_{NT} are set to 0.85 for $\geq 10^{10}$ load cycles.
2. **increased if the quality is better (reduced to 0.92):** Y_{NT} and Z_{NT} are set to 10 for $\geq^{10}$ load cycles (in accordance with ISO 9085).
3. **with optimum quality and experience (always 1.0):** This removes the reduction and therefore corresponds to DIN 3990. However, this assumes the optimum treatment and monitoring of the materials.

17.3.5.3 Optimal tip relief

To calculate safety against micropitting as specified in Method B in ISO/TS 6336-22, you must specify whether or not the profile modification is to be assumed to be optimal. The same applies to calculating the safety against scuffing. The software checks whether the effective tip relief (Ca) roughly corresponds to the optimum tip relief (Ceff). If this check reveals large differences, i.e. Ca < 0.333*Ceff or Ca > 2.5*Ceff, a warning is displayed. In this case, the value you input is ignored and is documented accordingly in the report.

17.3.5.4 Hardening depth, known by its abbreviation "EHT"

You can input the intended hardening depth (for hardness HV400, for nitrided steels, or HV550 for all other steels). The input applies to the depth measured during final machining (after grinding).

When you input this data, the safety of the hardened surface layer is calculated automatically according to DNV 41.2 [11]. The calculation is performed as described in the "Subsurface fatigue" section in [11]. The calculation is performed using different solutions than the calculation of the proposal for the recommended hardening depth, but still returns similar results (see chapter 22.6, Proposals for hardening depth).

17.3.5.5 Load spectra with negative bins

Load spectra with negative load bins (T < 0 and/or n < 0) can be calculated as follows.

IMPORTANT:

- A load bin is considered to be negative if the non-working flank is placed under load.
- This does not apply for the calculation of the pitting safety for idler gears (in the case of planetary gear stages, it only applies to the sun and internal gear; in the case of planets, it is assumed that both flanks are always under load).
- When calculating the root safety, this is only applied to bins where the alternating bending factor is Y_M=1.0.

Torque factor	Speed factor	Flank under load	Load bin is
+	+	Working flank (*)	evaluated as positive
+	-	Working flank (*)	evaluated as positive
-	+	Non-working flank	evaluated as negative
-	-	Non-working flank	evaluated as negative

Table 17.3: Evaluation of a load bin, depending on the prefix operator

(*) Working flank as entered in the **Rating** tab

You can select the following in "**Details**" in the **Strength** group, in the **Rating** tab:

- For calculating pitting safety
  - Evaluate all negative load bins as positive (as up to now)
  - Only consider positive load bins
  - Only consider negative load bins
  - Calculate both cases and document the less favorable case
- For calculating root safety
  - Evaluate all negative load bins as positive (as up to now)
  - Increase tooth root stress for negative load bins by 1/0.7
  - Increase tooth root stress for positive load bins by 1/0.7
  - Calculate both cases and document the more realistic case

17.4 Factors

17.4.1 Face load factor

Face load factors $K_{H\beta}$ take into consideration the influence of an uneven load distribution over the facewidth on the contact stress, tooth root stress and scuffing stress. For face gears, we recommend you use approximately the same coefficients (see chapter 16.8.3.1, Bearing application factor) as for bevel gears.

17.5 Modifications

The **Modifications** (see chapter 15.7, Modifications) tab in the **Face gears** calculation module basically includes the same functionality as for cylindrical gears. Its special features are listed below:

17.5.1 Tip alteration

You specify the tip alteration $h_{ake(i)}$ and the tip alteration length $l_{hake(l)}$ (see Figure 17.8) in the **Modifications** tab. The tip is then altered to prevent the tooth becoming pointed. When you specify the tip alteration, we recommend you select **Calculation > Settings > General** (see chapter 17.6.1, General) and increase the number of calculated sections, to fully display the modification for the 3D export.

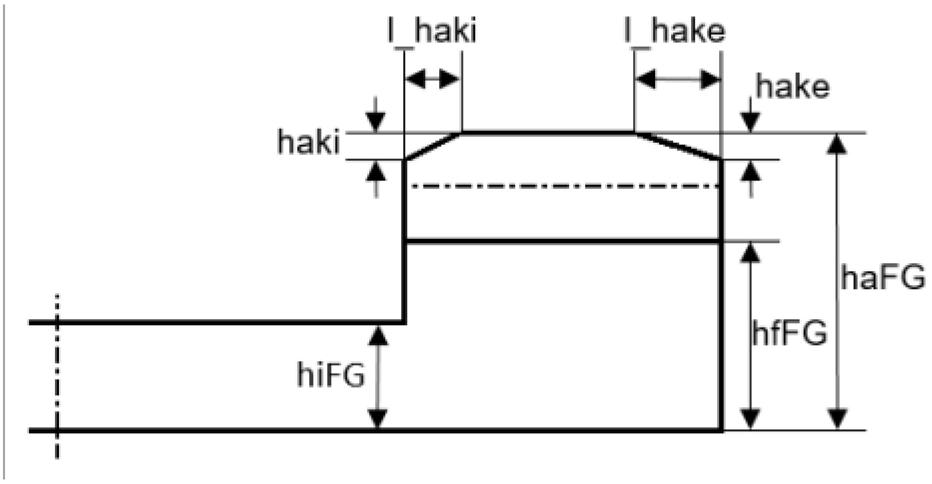


Figure 17.8: Characteristic values of a face gear

17.5.2 Modification type

In the list of modifications (see chapter 15.7.1, Modification type), you can only modify the pinion.

17.6 Settings

Click on **Calculation > Settings** or select the icon to display the window for the **Module specific settings** sub-menu. From here, you can access the tabs listed below and input other calculation parameters in them.

17.6.1 General

The **Number steps for tooth form calculation** input field defines how many equidistant section levels $N \geq 3$ are to be distributed between the outside and internal diameter of the face gear. The default value here is $N = 3$ which defines section levels $r_2 = d_{2i}/2$, $r_2 = d_{2e}/2$ and $r_2 = (d_{2i} + d_{2e})/4$.

► **Note**

You should select N > 10 to ensure an adequate spatial resolution for your 3D export.

17.6.2 Sizings

The values entered in the **Minimum and Maximum pressure angle in transverse section** $\alpha_{t,min/max}$ input fields define the range that contains the values for the face gear tooth flank pressure angle across the width. These values are used, for example, when sizing the facewidth of face gear b_2 and axial offset b_v.

17.7 Notes on face gear calculation

17.7.1 Dimensioning

In KISSsoft, a wide variety of procedures that differ greatly from other commonly-used procedures, e.g. for cylindrical gears, for dimensioning the complex tooth forms in face gears. For a face gear, you must select a geometry that prevents the creation of pointed teeth on the outside face of the gear and ensures that no (or very little) undercut occurs on the inside face. You must perform these checks when you calculate the tooth form. The actual geometry calculation procedure converts the data into the equivalent bevel gear and the virtual cylindrical gear. In the tooth form calculation process, a face gear is calculated in a number of sections set along its facewidth. To specify the number of required sections, select the **Calculation** menu. Then, select **Settings > Module specific settings > General > Number of sections for tooth form**. In the dialog that is then displayed, define the number of sections. In the **Geometry** graphics window, you can display the tooth form simultaneously on the internal diameter, external diameter and in the middle of the tooth. You can see here whether the top land (normal crest width) and undercut are acceptable.

You can take these measures to prevent pointed teeth or undercuts occurring in the gear:

- change facewidth offset b_v
- reduce the facewidth
- change the pressure angle
- alter the tip in the outside part of the facewidth.

► **Notes**

- To generate a crowned tooth form: You can generate flankline crowning on the tooth trace of face gears by using a pinion type cutter that has one or two more teeth than the meshing pinion. Use the data buffer function in the 2D display (select **Graphics > Geometry > Meshing**) to check the difference between the generated tooth forms. To do this, define a pinion type cutter with the same number of teeth as the pinion used to calculate the tooth form. Save the face gear tooth form by clicking the **Save curve** button  and then increasing the number of teeth on the pinion type cutter. If the face gear has a large axial offset b_v, you can displace the crowning to one side.

17.7.2 Pinion - Face gear with Z1 > Z2

No provision has been made for calculating a pinion – face gear pairing when the number of teeth on the face gear (Z2) is less than the number of teeth on the pinion (Z1), because this situation does not happen very often. However, under certain conditions, you can still determine the geometry of this type of pairing.

To do this, select **Module specific settings** and click the **Do not cancel if geometry errors occur** checkbox. Then, we recommend you follow these steps:

- Reduce the facewidth of the face gear (for example, by half)
- Starting with Z2 = Z1, zoom Z2 out step by step, performing a calculation after every step and correcting the inside, middle, and outside aspect of the cuts and, if necessary the tooth height, in the 2D display.
- Once you achieve the required number of teeth Z2, try to increase the facewidth of the face gear again, and modify b_v if necessary.

18 Worms with enveloping worm wheels

The worm geometry is calculated according to ISO 14521 or DIN 3975. Control measurement values for worms and worm wheels are determined, and manufacturing tolerances are according to DIN 3974.

Sizing of facewidth, center distance , lead angle etc. as well as strength calculation is performed according to ISO 14521 or DIN 3996. This includes the definition of: efficiency, temperature safety, pitting safety, safety against wear, root and deflection safety of the worm. Data for various different worm wheel materials are supplied.

The user can also calculate the starting torque under load, which is a critical value when sizing drives.

Calculation of the flank forms can be performed: ZA, ZC, ZI, ZK, ZN (equivalent to A, C, I, K, N according to ISO TR 10828:2015), ZH (equals ZC). The displayed tooth form (2D, 3D) is generated by mirroring half a tooth flank.

The drawings below show the dimensioning of a worm wheel.

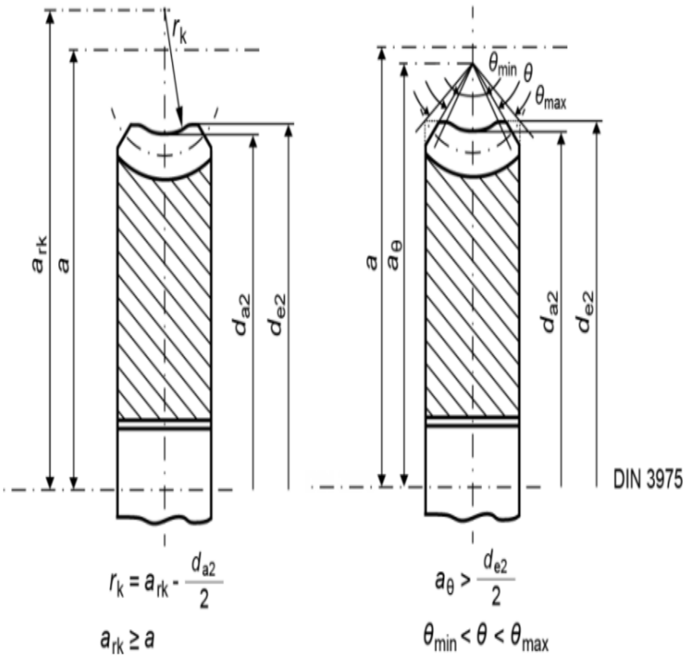


Figure 18.1: Dimensioning a worm wheel

18.1 Underlying principles of calculation

The underlying geometric relationships are defined in ISO 14521 or DIN 3975. You will find additional information, and other important definitions, such as the various worm flank forms (ZA, ZC or ZI, ZH, ZK, ZN), in [12]. Strength (tooth fracture, pitting, wear and temperature safety) is calculated according to ISO 14521 or DIN 3996. These calculations take much less time and effort to perform than those required for cylindrical gears. Worms can be checked throughout the manufacturing process by using what are known as "three wire measurements". This corresponds to the principle of the measurement over two balls that is used for worm gears (and also for cylindrical gears). However, the calculations involved in ascertaining the three wire measurement are very complex. A very useful method for standard flank forms has been developed by G. Bock [52] at the "Physikalisch-Technisches Bundesanstalt" (German national metrology institute) in Berlin. This method takes into account the shape of the worm's flank, which is why it is used in KISSsoft.

► **Note**

When you use the term "module" you must differentiate clearly between the axial and the normal module.

Note about how to use the application factor

In cylindrical gear and bevel gear calculations, the application factor K_A is usually multiplied by the power, for example, so that K_A=1 with P= 5 kW gives exactly the same safeties as K_A=2 and P=2.5 kW. However, this is different for worm calculations performed according to the ISO or DIN standard and may lead to confusion.

The forces and torques are multiplied by the application factor. In contrast, the power is not multiplied by the application factor when determining the bearing power loss P_{VLP} and when calculating the total efficiency η_{Ges}. Therefore, if K_A=2 and P=2.5 kW instead of K_A=1 with P= 5 kW, the power loss [PV] is lower, but the total efficiency η_{Ges} is massively too low.

Results for the example "WormGear 1 (DIN 3996, Example 1).Z80":

	K_A=1; P= 5 kW	K_A=2; P=2.5 kW	
P_{VLP}	0.140	0.070	<< (* 1/K_A)
P_{VD}+P_{V0}	0.199	0.199	=
P_{VZ}	0.530	0.530	=
P_V	0.869	0.799	<
η_z	90.00	90.00	=
η_{Ges}	85.19	75.77	<<
Θ_S	76.6	76.6	=
Θ_M	80.9	80.9	=

S_W	1.386	1.386	=
S_H	1.143	1.143	=
S_δ	2.369	2.369	=
S_F	2.251	2.251	=
S_T	1.306	1.306	=

This difference in the results is not logical so therefore, to determine P_{VLP} and η_{Ges}, the power is also multiplied by K_A to achieve the same results.

18.2 Basic data

18.2.1 Axial/transverse module

Here, you can click on **Module specific settings** in the **Calculations > Calculation with normal module instead of with axial module** tab, to work with the normal module m_n instead of the axial module in future calculations.

► **Note**

This changes the way the tip and root circles(see chapter 18.5.4.2, Calculation with normal module instead of axial module) are calculated.

18.2.2 Pressure angle at normal section

The normal pressure angle at the reference circle is also the reference profile flank angle. For standard meshings, the pressure angle is $\alpha_n = 20°$. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Larger pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

18.2.3 Lead angle at reference diameter

The lead angle on the reference circle in a worm (gear 1) is the complement of the helix angle and is calculated according to equation 17.1. Lead direction (see Figure 17.3).

$\gamma = 90° - \beta$	(17.1)

Click the Convert button to open the **Convert lead angle** dialog window, in which you can calculate the lead angle from other gear values. These options are available here: **from center distance**, **from reference diameter** and **from the reference circle and the center distance (x_2*** is modified). A larger lead angle produces greater efficiency, whereas you can create self-locking toothing if you use a smaller lead angle.

18.2.4 Center distance

Click the Sizing button to calculate the center distance from the values of profile shift coefficient x*, number of teeth z and lead angle γ. In this case, you do not receive a message telling you that the calculation has been performed correctly.

18.2.5 Number of teeth

The number of teeth on a worm usually is in the range 1 ≤ z1 ≤ 4.

18.2.6 Facewidth

For more information about the dimensions of gear teeth and wheel flange widths, please refer to (see Figure 18.2). Enter the width of the worm wheel in the facewidth b_{2R} input field. The facewidths b_{2H} and b_2 of the worm wheel are then calculated using this value.

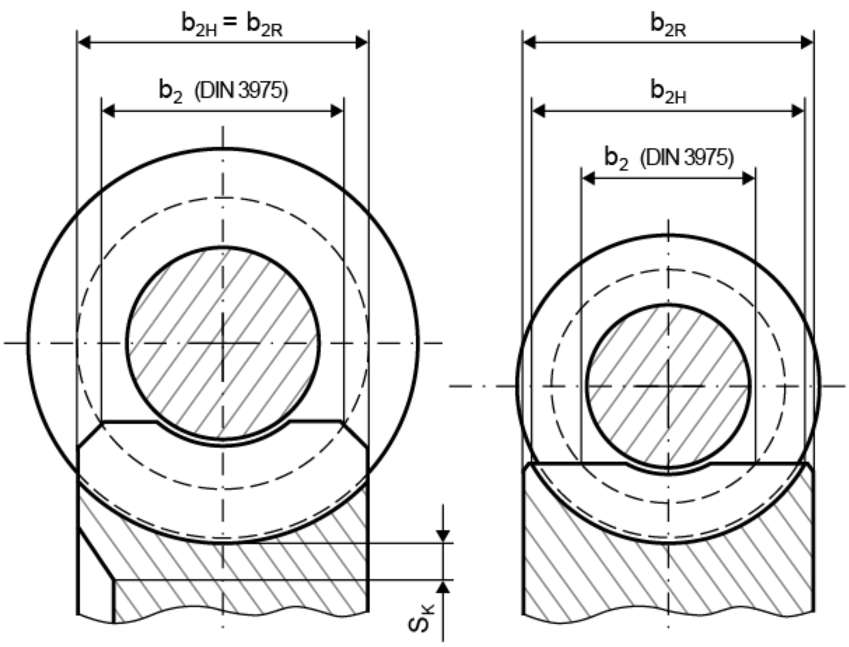


Figure 18.2: Dimensioning the gear tooth and gear rim width

18.2.7 Profile shift coefficient

In the **Worms with enveloping worm wheels** calculation module, the profile shift for worm/gear 1 is set to zero (as defined in the ISO 14521 standard). You can only modify the tooth thickness of the worm in the input window for the **Tooth thickness modification coefficient** x_s.

► **Note**

You should use the **Crossed helical gears** calculation module if you require a worm whose profile shift coefficient is $x_1^* \neq 0$.

18.2.8 Tooth thickness modification coefficient

This factor should only be used in special cases. The factor x_s modifies the tooth thickness with ΔAs = 2 * x_s * m_n, as for bevel gears. $x_{s2} = -x_{s1}$ always applies, so that the clearance remains unchanged if you input x_{s1}. For normal applications, the values for x_{s1} are entered in intervals of $-0.1 \le x_{s1} \le +0.1$.

This factor can be used, for example, if the worm is significantly harder than the gear, which causes the gear to wear during operation. The gear unit fails when the gear tooth breaks because it has become increasingly thinner, due to wear. Modifying the worm's tooth thickness with e.g. $x_{s1} = -0.1$ will make the worm wheel tooth thicker. This will result in a much longer service life.

18.2.9 Quality for worm gear units

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, click on the Settings button and then select the **Quality input** option.

The manufacturing qualities that can be achieved are displayed in the next table.

Manufacturing process	**Quality according to DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 18.1: Accuracy grades for different manufacturing processes

18.2.10 Geometry details

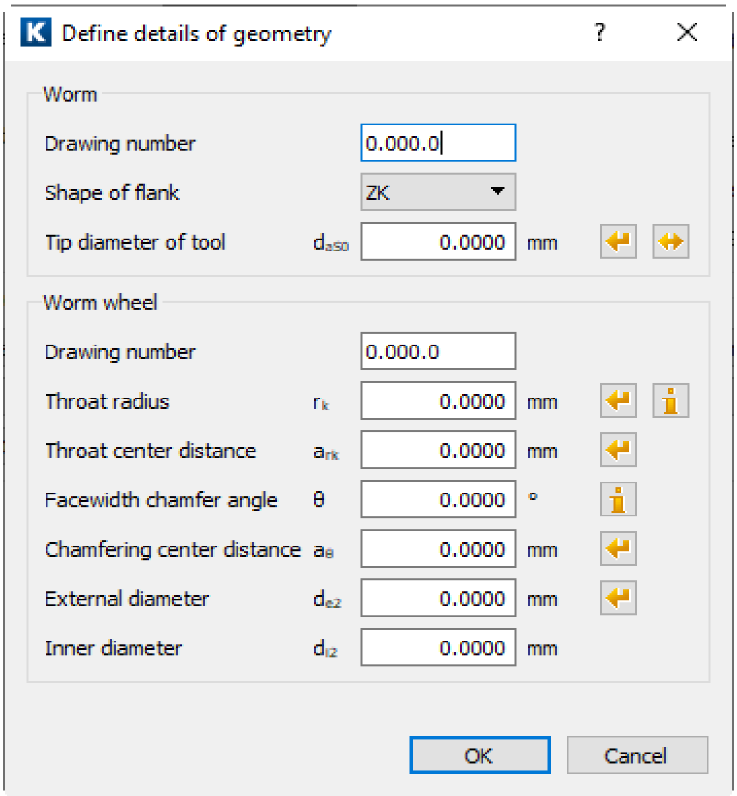


Figure 18.3: Define details of geometry window

Click the **Details...** button in the **Geometry** area to display the **Define geometry details** window, in which you can change the parameters listed below.

18.2.10.1 Shape of flank

The flank shape is a result of the manufacturing process. ZA, ZN, ZK and ZI worms have very similar levels of efficiency and flank load capacity. Although ZC and ZH worms (hollow flanks) have better load capacity in some situations, they do have other major disadvantages.

ZA form:	manufactured on lathe with tool (straight flanks), mounted in axial section
ZN-form:	manufactured on lathe with tool (straight flanks), mounted in normal section
ZI form:	manufactured with hobbing cutter (worm flank is involute)
ZK form:	manufactured with grinding wheel (straight flanks), mounted in normal section
ZC, or ZH form:	Manufacturing with special tools to generate a hollow flank

For more information, please refer to: Dubbel [26], with figures on pages G136 and S79.

18.2.10.2 External diameter and throat radius

You specify values for the external diameter d_{e2} and throat radius r_k as specified in DIN 3975-1:2017-09. According to equation (59) and (67) the following values are suggested for these two dimensions:

$$d_{e2} \approx d_{a2} + m_x$$

$$r_k \geq a - \frac{d_{a2}}{2}$$

with:

d_{a2}: Tip diameter

m_x: Axial module

a: Center distance

18.2.11 Material and lubrication

Materials

The strength calculation method used for worms according to ISO 14521 is based on empirical values determined using these materials:

Worm:

- Case-hardened steels (especially 16MnCr5), HRC = 58 to 62
- Heat treatable steels (especially 42CrMo4), heat- or induction-hardened, HRC = 50 to 56
- Nitriding steels (especially 31CrMoV9), gas-nitrided

Worm wheel:

- Bronze (GZ-CuSn12, GZ-CuSn12Ni, GZ-CuAl10Ni)
- Grey cast iron (GJS40, GC25)
- Polyamide (PA-12, cast)

To calculate strength, you require very special materials data, in particular the wear values. The standard only specifies these values for the most commonly used worm wheel materials (mostly bronze). This is why the selection of materials in KISSsoft is limited. As defining data for materials that are not already documented takes a great deal of time and effort, we strongly recommend you select a material from the list that is closest to the material you actually want to use.

Lubricants

Selecting the right lubricant for a worm is extremely important. Synthetic lubricants (polyglycols or polyalfaolephine) can reduce loss and wear enormously.

18.3 Load

18.3.1 Methods used for strength calculation

The calculations defined in ISO 14521 and DIN 3996:2012 are identical.

However, strength calculation as defined in ISO 14521 includes a number of different methods (A,B,C,D;). KISSsoft uses the most precise documented method, which is usually Method B. This calculation method is not suitable for every material (see chapter 18.2.11, Material and lubrication), because some of the empirical values are missing.

The ISO 14521 standard provides a calculation method for determining:

- efficiency
- Wear and safety against wear
- Pitting safety
- Root safety
- Deflection safety
- Temperature safety

The default value of the shaft angle is set to 90° because this is the default value specified in the strength calculation method defined in DIN 3996. However, you can calculate the geometry with a shaft angle that is not 90° by using the **Crossed Helical Gears** calculation module (see chapter 19, Crossed helical gears and crossed helical gear with rack).

► **Notes:**

- To calculate strength, you require very special materials data, in particular the wear values. The standard only specifies these values for the most commonly used worm wheel materials (mostly bronze). This is why the selection of materials in KISSsoft is limited.

- Grease lubrication: Grease lubrication is not mentioned in DIN 3996. In this situation, KISSsoft performs the calculation as for oil bath lubrication. This assumption is permissible, because the lubrication type has very little influence on the calculation.
- Endurance limit values for tooth root load capacity: The standard provides two different values. If you enter the smaller value in the database, no decrease in quality due to plastic deformation of the teeth will be accepted.

18.3.2 Service life

The value in the **Required service life** input field is used together with the speed to calculate the number of load cycles.

Use the Plus button to set the **Power-on time**. To calculate the service life, multiply the power-on time with the number of load cycles. The temperature calculation also takes into account the power-on time when it determines the amount of heat generated.

Note about calculating temperature:
It is assumed that heat is constantly dissipated, but that, in contrast, heat is only generated during the specified power-on time. The precondition for this is that the gear unit is only run for a short period of time (maximum 15 minutes), and is then stopped again. If this is not the case, the power-on time must be set to 100%.

18.3.3 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. The next table 5ba0bcd9e87 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 18.2: Assignment of operational behavior to application factor

18.3.4 Permissible decrease in quality

Depending on the construction type of the worm wheel, it may experience a decrease in quality over time due to wear. This value must not sink below the value specified in this input field. A decrease in quality is linked to the plastic deformation of the material, and therefore a higher material property. This, in turn, results in a higher safety against plastic deformation in the root.

18.3.5 Power, torque and speed

Click the Sizing button next to the power input field (for torque) to calculate the power (torque) appropriate to maintain a predefined minimum level of safety (see chapter 15.22.5, Safety factors).

18.3.6 Strength details

Click on the **Details** button when sizing root and flank strength to display a dialog in which you can make additional settings for the strength calculation.

18.3.6.1 Support of gearing

The calculation method used to ascertain bearing power loss of the integral worm shaft identifies two different types of bearing.

18.3.6.2 Bearing power loss

If roller bearings are used, the power loss is calculated using the empirical formulae defined in ISO 14521. If plain bearings are used, you must specify the power loss manually.

The empirical formulae defined in ISO 14521 can only be used in a specific range of validity.

KISSsoft checks whether the total power losses are too high.

$(P_{V0} + P_{VLP} + P_{VD}) > 0.2*P$

If the power losses are greater than 0.2*P, the individual power losses P_{V0}, P_{VLP} and P_{VD} are each set to 0.

18.3.6.3 Number of radial sealing rings, worm shaft

To calculate the power loss in sealing, you must enter the number of radial sealing rings on the integral worm shaft. The sealing gaskets on the worm shaft are not taken into account because their slow rotation speed means they lose very little power (the calculation formulae are defined in ISO 14521).

18.3.6.4 Permissible tooth thickness decrease

The permissible tooth thickness decrease (on the gear) is needed to calculate the wear safety and taken into account when calculating the root safety. If this input field contains 0, the permissible tooth thickness decrease is not checked.

18.3.6.5 Permissible mass decrease

You can limit the permissible mass decrease in kg on the worm wheel (for example, by specifying oil change intervals). This limiting value is also used to define wear safety. If this input field contains value 0, the mass decrease is not checked.

► **Note**

The decrease in mass experienced on the worm is not calculated, because the standard assumes that the worm is harder than the worm wheel and therefore will not be subject to wear.

18.3.6.6 Distances on the integral worm shaft

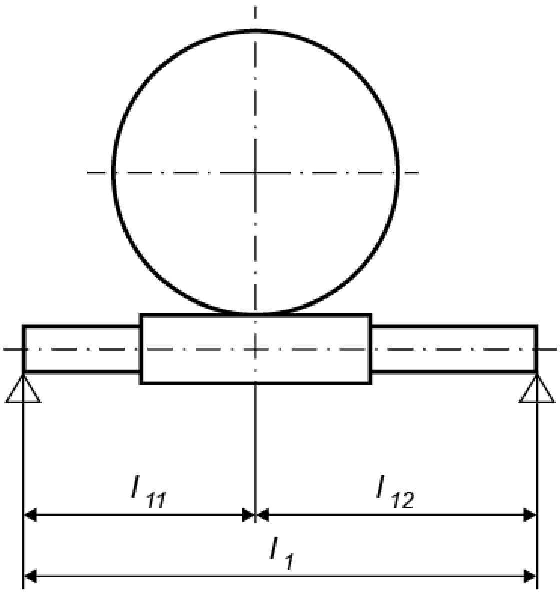


Figure 18.4: Dimensioning the worm/worm wheel

l_1	Distance between the bearings on the integral worm shaft
l_{11}	Distance from bearing 1 to the middle of the worm

You need these values to calculate the deflection safety. The position of the drive has no effect on the calculation.

18.3.6.7 Load spectra with negative bins

Load spectra with negative load bins (T < 0 and/or n < 0) can be calculated as follows.

IMPORTANT:

- A load bin is considered to be negative if the non-working flank is placed under load.
- This does not apply for the calculation of the pitting safety for idler gears (in the case of planetary gear stages, it only applies to the sun and internal gear; in the case of planets, it is assumed that both flanks are always under load).
- When calculating the root safety, this is only applied to bins where the alternating bending factor is Y_M=1.0.

Torque factor	Speed factor	Flank under load	Load bin is
+	+	Working flank (*)	evaluated as positive
+	-	Working flank (*)	evaluated as positive
-	+	Non-working flank	evaluated as negative
-	-	Non-working flank	evaluated as negative

Table 18.3: Evaluation of a load bin, depending on the prefix operator

(*) Working flank as entered in the **Rating** tab

You can select the following in "**Details**" in the **Strength** group, in the **Rating** tab:

- For calculating pitting safety
  - Evaluate all negative load bins as positive (as up to now)
  - Only consider positive load bins
  - Only consider negative load bins
  - Calculate both cases and document the less favorable case
- For calculating root safety
  - Evaluate all negative load bins as positive (as up to now)
  - Increase tooth root stress for negative load bins by 1/0.7
  - Increase tooth root stress for positive load bins by 1/0.7
  - Calculate both cases and document the more realistic case

18.4 Tolerances

The structure and functionality of the Tolerances input window (see chapter 15.6, Tolerances) in the **Worms with enveloping worm wheels** calculation module is the same as the Tolerances input window for cylindrical gears. When you enter allowances for worm calculations, we recommend you click on the **Tooth thickness tolerance** drop-down list and select either the **Worm as defined in Niemann** or **Worm wheel as defined in Niemann** option. The corresponding data is based on recommendations in Niemann [12].

18.5 Settings

Click on **Calculation >Settings** or select the icon to display the window for the **Module specific settings** sub-menu. From here, you can access the tabs listed below and input other calculation parameters in them.

18.5.1 General

You can set general parameters for the calculation in the **General** tab. You will find detailed descriptions of the individual parameters in the (see chapter 15.22.1, General) section.

18.5.2 Reference gearing

This calculation is based on a standard reference gearing, on which tests have been performed. The default data corresponds to the reference gearing in ISO 14521. However, if you have the results of your own tests or empirical values, you can modify this calculation to take advantage of this expertise. For a more detailed description, please refer to ISO 14521.

18.5.3 Sizings

To dimension the gear unit stage, you select or define different sizing criteria in the **Sizings** tab.

18.5.4 Calculations

18.5.4.1 Always calculate transmittable torque (utilization)

Click on this checkbox to define the value so that exactly the required safety (input in the **Sizing** tab) is achieved.
The process is also documented in the main report.

18.5.4.2 Calculation with normal module instead of axial module

The geometry of worm gear pairs is usually calculated with the axial module (or transverse module of the worm wheel). If you click on this checkbox, all the values used for the reference profile are calculated with the normal module (tool module). This particularly affects the tip and root circle. In contrast, the profile shift x* m_x (m_x for an axial module) remains unchanged.

The formula for the tip circle (m_n for the normal module) is then:

$d_{a1} = d_{m1} + 2\, m_n\, h_{aP}$
$d_{a2} = d_2 + 2\, m_x\, x_2 + 2\, m_n\, h_{aP}$

For the root circle, the following apply:

$d_{f1} = d_{m1} - 2\, m_n\, h_{fP}$
$d_{f2} = d_2 + 2\, m_x\, x_2 - 2\, m_n\, h_{fP}$

18.5.4.3 Calculating with alternative formulae (differs from standard)

Alternative calculation methods are used at these points if you select this checkbox:

- Effective tooth thickness at tip (instead of formula (81) in accordance with DIN 3996:2019 or formula (128) in accordance with ISO 14521:2010)
- Mesh power loss PVZ with coefficient 1/9.550 instead of 0.1
- Radial force and meshing efficiency according to Schlecht [10], where μ_{zm} is the tooth's mean coefficient of friction and ϱ_z is the angle of friction of the mean coefficient of friction

$$F_r = F_t \frac{\tan\alpha_{0n} \cdot \cos\rho'}{\sin(\gamma_m + \rho')}$$

$$\eta_z = \frac{\tan\gamma_m}{\tan(\gamma_m + \rho')}$$

$$\tan\rho' = \frac{\tan\rho_z}{\cos\alpha_{0n}}$$

$$\mu_{zm} = \tan\rho_z$$

18.5.5 Safety factors

KISSsoft displays an error message to tell you, if the specified required safeties have not been reached after you completed the calculation. Sizing is always calculated on the basis of the required safeties for tooth fracture, pitting and wear. If you do not wish to use one or more of these criteria, set the appropriate required safety to zero. According to ISO 14521, you must ensure the following safeties:

Root safety	1.1
Pitting safety	1.0
Wear safety	1.1
Deflection safety	1.0
Temperature safety	1.1

You can also modify these values if you are familiar with the process.

19 Crossed helical gears and crossed helical gear with rack

Crossed helical gears are involute cylindrical gears with helical teeth whose shafts have crossed axes. Usually, the axial crossing angle is $\Sigma = 90°$. In contrast to the line contact shown in enveloping worms, crossed helical gears only contact at one point during generation. As a result, they can only transmit very small forces, and are primarily used for control purposes.

In precision engineering, a worm wheel is often manufactured in the same way as a helical cylindrical gear with a cylindrical gear body. This makes it easier to produce and assemble than the "classic" worm wheel globoid gear. In this situation, the crossed helical gear module is used to calculate the geometry of a precision mechanics worm wheel in the same way as a helical-toothed cylindrical gear.

In a crossed helical gear where the profile shift coefficient is not zero, the helix angle on Gear 1 (referred to as the "worm" if a small number of teeth is involved)and the helix angle on Gear 2 differ from each other. Both gears usually have the same hand of gear. In contrast, the total of both helix angles at the operating pitch circle/spiral is exactly the same as the shaft angle.

The axial crossing angle is usually positive, but can also be negative or positive in some special cases. However, it must always be smaller than the helix angle on Gear 1. In this situation, gear 2 has the opposite hand of gear to gear 1. In KISSsoft, you must always input values for the helix angle on Gear 1 and axial crossing angle. The helix angle on Gear 2 is calculated. KISSsoft always uses a positive value for the helix angle on Gear 1 in its internal calculations. However, the angle on Gear 2 can be either positive (as is usually the case) or negative.

Special cases:

In some special cases, the user can consider Gear 1 as a worm. For this the ZR[0].isWorm variable must be set to true directly in the calculation file.

Gear 2 can have internal toothing. In this case, enter a negative number of teeth on Gear 2.

Gear pair with spur gear teeth: Gear 1 cannot be a spur gear. An entry of 0 for the helix angle is not accepted. In contrast, Gear 2 can have spur gear teeth. In this case, select a negative shaft angle.

For the shaft angle, the following conditions apply:

- for external teeth: $\beta1+\beta2 = \Sigma$ (*)
- for internal teeth: $\beta1-\beta2 = \Sigma$ (*)

*): Applies if gear 1 and gear 2 have the same hand of gear (either both right or both left). Otherwise, a negative value for β2 must be entered.

19.1 Underlying principles of calculation

The method used to calculate crossed helical gears (cylindrical gears with crossed axes) is defined in Niemann [12]. The current version describes the methods used to calculate and check the geometry of crossed helical gears for any axial crossing angle. The measures used for checking and fabrication are determined arithmetically.

Although the method detailed in Niemann [12] is used as an approach for calculating the root and flank strength and the scuffing safety, the individual equations used are those specified in ISO 6336. (Niemann uses the equations from an obsolete edition of DIN 3990.)

19.2 Basic data

19.2.1 Normal module

Enter the normal module. However, if you know the pitch, transverse module or diametral pitch instead of this, click the Convert button to display a dialog window in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select **Input normal diametral pitch instead of normal module** by first selecting **Calculation > Settings > Module specific settings** in the **General** tab.

19.2.2 Pressure angle at normal section

The normal pressure angle at the reference circle is also the reference profile flank angle. For standard meshings, the pressure angle is $\alpha_n = 20°$. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Greater pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

19.2.3 Helix angle reference circle gear 1

The center distance, number of teeth, profile shift (x^*_1, x^*_2) and shaft angle can be used to calculate the helix angle of gear 1. It often happens that several helix angles meet the requirements of the toothing geometry. In this situation, when you click the Sizing button, you see an **Information** window that lists the possible values. Here, the **solution that is closest to the current value** is selected automatically. However, if only one value is suitable for the sizing, it is transferred into the input field without any messages being displayed. If the sizing function is unable to find any solutions, it displays a warning message and you must then change either the center distance or the module.

The helix angle value of gear 1 must be entered as a positive value. The hand of gear is set as a right- or left-hand helix. Gear 2's axial crossing angle and helix angle can be negative.

19.2.4 Center distance

The center distance is calculated on the basis of the helix angle of gear 1, the axial crossing angle, the profile shift (x^*_1, x^*_2) and the number of teeth.

19.2.5 Facewidth

Because the facewidth must have a minimum value, the input field has a Sizing button which you can use to define the minimum width based on the parameters you have already defined.

19.2.6 Profile shift coefficient

The tool can be adjusted during the manufacturing process. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift, the tool is pushed further into the material, with the result that the tooth is narrower and undercutting may occur sooner. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

Click the Convert button and KISSsoft will determine whether the profile shift coefficients (see chapter 15.1.8, Profile shift coefficient) are to be taken from measured data or from values given in drawings.

► **Note**

If one of the two profile shift values appears in gray, this means it will be calculated by KISSsoft. This is what happens when you select the checkbox for entering the center distance. If you overwrite a gray field, it will become active and KISSsoft will calculate the value for one of the other gears.

19.2.7 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, click on the Settings button and then select the **Input quality** option. The accuracy grade according to ISO 1328 (DIN ISO 1328) is very similar to the same quality in AGMA 2015.

The manufacturing qualities that can be achieved are displayed in the next table.

Manufacturing process	Quality according to ISO		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9

Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 19.1: Accuracy grades for different manufacturing processes

► **Note**

The values in brackets can only achieved in exceptional situations.

You can also click the Settings button to set different quality-related options. The following options are available:

Input quality

The manufacturing deviations that are output in the report and used for particular coefficients in the strength calculation procedure are defined either in the ISO 1328 (DIN ISO 1328), DIN 3961:1978 or AGMA 2015 standards. You can specify which standard is to be used. If you click the **Calculation method for strength** option, the system applies the standard that is best suited to the strength calculation method (for example, ISO 1328 is used if you are using the ISO 6336 calculation method).

Varying qualities

If you select this option, the Plus button is displayed next to the Quality entry field in the main screen. Click this button to input specific tolerances manually. You will find a more detailed description of this in Qualities (see chapter 15.1.10, Quality).

Fp tolerance as specified in tables in DIN 3962
The total cumulative pitch deviation Fp given in the tables in DIN 3962 can be very different from the Fp calculated using the formulae in DIN 3961.

Extrapolate tolerance values
The tolerances detailed in ISO 1328:2013, DIN ISO 1328:2018, AGMA 2000 and AGMA 2015 are calculated using the formulae in each particular standard and with the effective geometric data (mn, d, b…). The range of validity must be defined in each case. For example, the tolerances specified in ISO 1328 apply for a module range 0.5 mm <= mn <= 70 mm. However, these formulae cannot be applied to gear teeth that lie outside of the range of validity. Despite this, these formulae are still used in such cases, due to the lack of any other information. In KISSsoft, the relevant limiting value is usually used to determine tolerances (for example, in the ISO standard, the tolerance is defined as 70 mm, and for a module it equals 80 mm). Alternatively, you can select **Extrapolate tolerance values** to calculate tolerances using the effective value (i.e. with 80 mm). In this case, in ISO 1328 (2013 edition), tolerances are also output when the helix angle is greater than 45°.

In DIN 3961:1978 and ISO 1328 (1996 edition), tolerances are calculated with the geometric mean values and therefore no extrapolations can be performed here.

19.2.8 Geometry details

Click the **Details...** button in the **Geometry** area to display the **Define geometry details** window, in which you can change the parameters listed below.

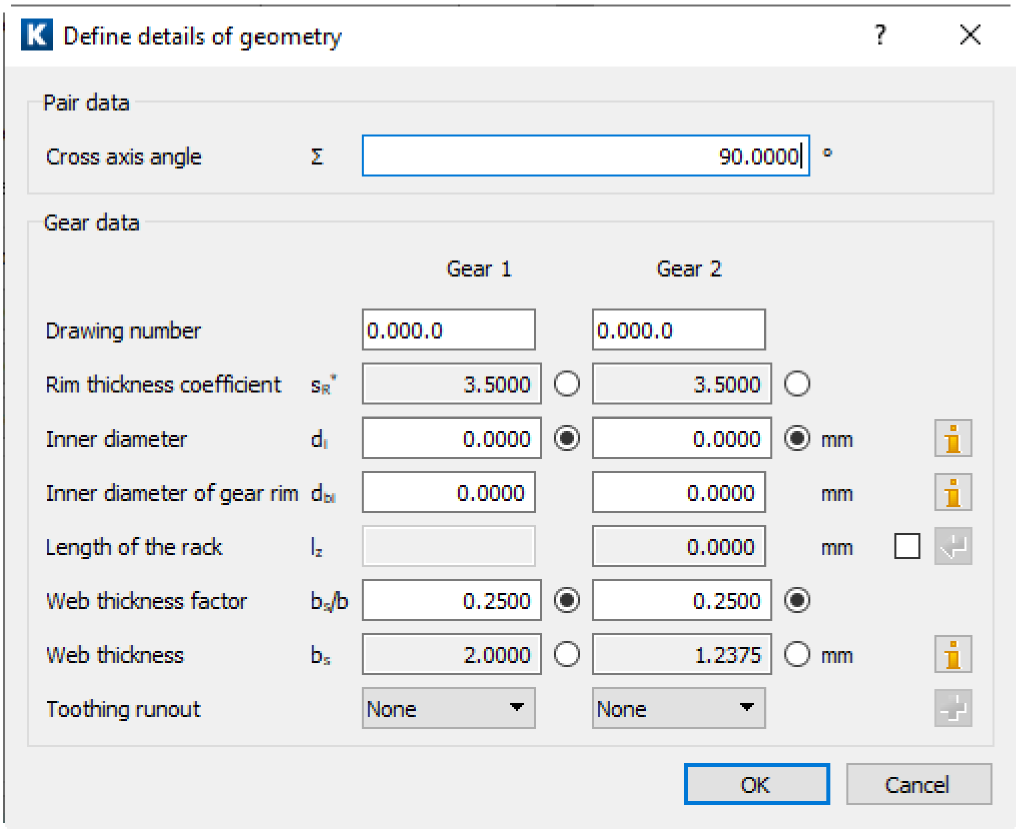


Figure 19.1: Geometry details input window

19.2.8.1 Axial crossing angle

The axial crossing angle is usually Σ = 90°, but you can select your own value here.

19.2.8.2 Internal diameter

The internal diameter is needed to calculate the mass moment of inertia. As defined in ISO or AGMA, the gear rim thickness does affect the strength. For solid wheels, enter 0. For external wheels with webs, enter their diameter d_i. For internal wheels, enter the external diameter of the gear rim.

The internal diameter of the gear rim is required for calculations according to ISO or AGMA. Where thin gear rims are used, this factor can greatly influence the calculation result (see Figure 19.2).

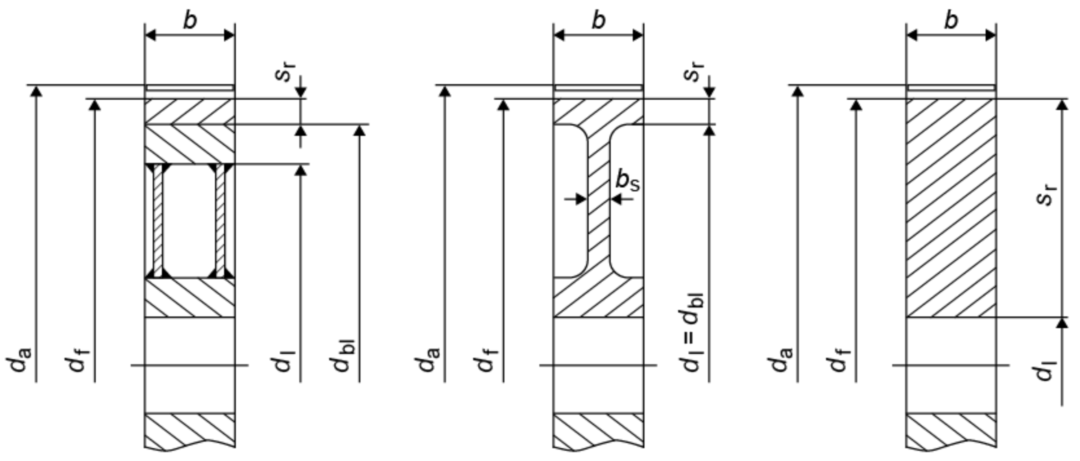


Figure 19.2: Dimensioning the diameter

19.2.9 Material and lubrication

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select **Own Input** from the list or enter the material in the database (see chapter 9, Database Tool and External Tables) first. Click the Plus button to display the **Material gear 1(2)** window, in which you can select a material from the list of materials available in the database. Select the **Own Input** option to enter specific material characteristics. This option corresponds to the **Create a new entry** window in the database tool.

19.2.10 Load

19.2.10.1 Methods used for strength calculation

As yet, no binding standard has been drawn up for the calculation of crossed helical gears. For this reason, KISSsoft recommends using ISO 6336 (see chapter 19.2.10.1.3, Strength calculation according to ISO 6336/Niemann).

You can use one of three different methods to calculate the strength of worms:

19.2.10.1.1 Strength calculation according to Hirn

The method used to calculate worms as defined by H. Hirn is based on an obsolete edition of Niemann's machine elements. It calculates the temperature safety, the flank safety, root safety and deflection safety. Although the material values cannot be compared with the values for worm calculation as defined in DIN 3996, the safeties are, however, similar.

We do not recommend you to use this obsolete method.

► **Note**

The calculation method defined in Hirn also selects a material pairing. This material pairing must lie in the permitted **Material and lubrication** range. Axial crossing angle Σ = 90° and $z_1 < 5$.

19.2.10.1.2 Strength calculation according to Hoechst

You can use the strength calculation in acc. with Hoechst for worm wheels made from Hostaform® (POM), paired with steel worm gears [53]. The permitted load coefficient is c [N/mm²] See equation (18.1) ÷ (18.3), is a value that defines the temperature resistance. This method also checks the worm's permissible contact stress and blocking safety. The decisive value for blocking safety is maximum load, not continuous load.

$c = \dfrac{F_2}{f_z \cdot b \cdot m \cdot \pi}$	(18.1)
$b = \sqrt{d_{a1}^2 - d_{m1}^2}$	(18.2)
$m = \dfrac{m_n}{\cos(\gamma_m)}$	(18.3)

where

F_2	Circumferential force on the worm wheel
f_z	Coefficient for number of teeth
b	Usable width
m_n	Normal module
γ_m	Mean lead angle
d_{a1}	Tip diameter of worm
d_{m1}	Reference diameter of worm

► **Note:**

Axial crossing angle Σ = 90° and $z_1 < 5$. The calculation method involves a worm made of steel and a crossed helical gear made of plastic.

19.2.10.1.3 Strength calculation according to ISO 6336/Niemann

You can perform the strength calculation for crossed helical gears with $z_1 \geq 5$ as defined in Niemann [12]/ISO 6336. As stated in Niemann, the contact ellipse is calculated using a for the width and b for the height of the half axes. An effective facewidth of 2a is assumed for flank safety (pitting). The same value plus twice the module value is used to calculate the strength of the tooth root. This corresponds to the specifications given in ISO 6336, if the facewidth is greater than the contact width. Scuffing safety is calculated as defined in Niemann [12]. This method differs from the DIN 3990-4 guideline because of the high sliding velocities of the crossed helical gears. It is more similar to the method applied to hypoid bevel gears. It supplies a proof of tooth root strength, the flank load capacity and the scuffing load capacity.

► **Note:**

If the number of teeth $z_1 < 5$, this calculation supplies tooth root and contact stress safeties that are too high.

19.2.10.1.4 Strength calculation as defined in VDI 2736

Part 3 of this VDI guideline describes the calculation for a cylindrical worm paired with a thermoplastic helical gear, i.e. a precision mechanics worm gear unit.

19.2.10.1.5 Static calculation

The static calculation performs a static estimate of the safety against fracture and yield point. The calculation is performed according to the documented formulae (see chapter 15.2.1.1, Static calculation).

The calculation in this approach for helical gears returns safeties that tend to be too low, because gear 2 in a worm gear that is to be mated is more likely to be subjected to shearing.

19.2.10.1.6 Static calculation on shearing

Determining how the worm wheel is subjected to shearing as a helical gear:

$\tau_F = F_t 2 * K_A * Y_E / A_T$

$A_T = b_{max}/5*(4*s_{tda2}-s_{tdx2})$

$d_{x2} = 2* a-d_{a1}$

This calculation is performed automatically and is documented in the report under **Tooth root load capacity** or **Static shearing in tooth root of the gear**.

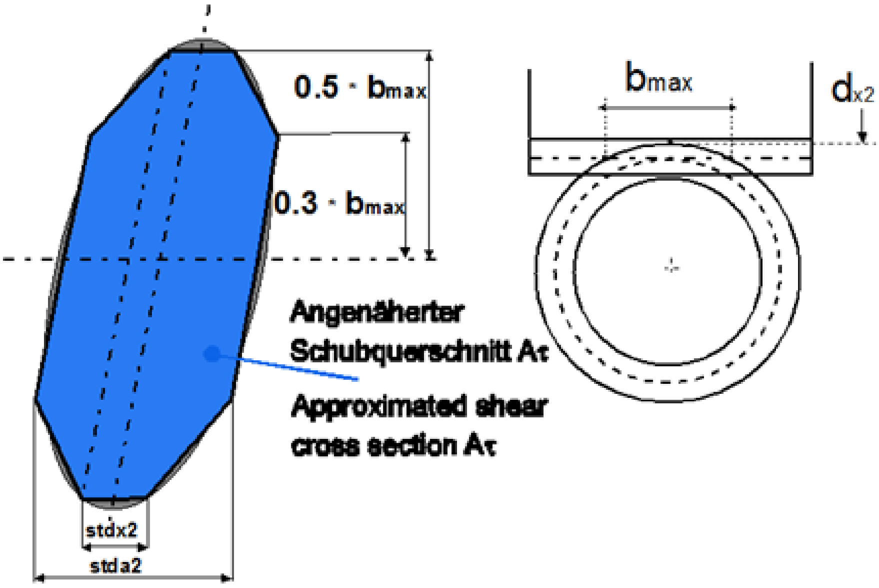


Figure 19.3: Dimensions of the shear cross section.

19.2.10.1.7 Calculating wear on worm gears according to Pech

A calculation for determining the wear on crossed helical gears according to Pech [54] is now available. This process calculates the plastic deformation, the degree of wear and the overall wear (in the normal section on the operating pitch diameter) of plastic worm wheels. The following restrictions apply to this calculation:

- Cylindrical worm wheel pair with an axial crossing angle of 90°
- Grease lubrication
- Calculation without load spectrum
- Material of worm: Steel
- Material of worm wheel: POM, PEEK, PEEK+30% CF or PA46
- Driving gear: Worm

Using KISSsoft, you can also perform calculations for plastic/plastic combinations, but these are subject to special assumptions and limitations (see below).

The coefficient of friction (COF) taken from the material DAT file has no effect on the calculation (the COF is calculated according to Pech). A (user-defined) COF is used to calculate plastic/plastic combinations (select **Rating > Details**).

The entries for the root temperature and flank temperature have no effect on the calculation of steel/plastic combinations (temperatures are calculated according to Pech). User-defined temperatures are used for plastic/plastic combinations. The grease temperature for plastic/plastic combinations is calculated as the mean value of the root temperatures of the two gears.

The flank roughness of the worm wheel has an effect on the calculated coefficient of friction. A greater level of roughness causes a greater amount of wear.

Click on **Module specific settings** to input a coefficient for the permitted level of plastic deformation (**Calculation> Settings> Plastic**).

If you input your own material in the KISSsoft material database, you must enter additional data in the material .dat file (for example for PEEK).

-- Type of plastic material
-- Values: 0-not on the list, 1-POM, 2-PEEK, 3-PEEK+30%CF, 4-PA46, 5-PA66,
-- 6-PA6, 7-PA66+GF, 8-PPS, 9-PPS+GF, 10-PA12, 11-PBT, 12-PET
:TABLE FUNCTION MaterialType
INPUT X None TREAT LINEAR
DATA
0
2
END

The table below shows the parameter limits for calculating wear according to Pech.

Number of teeth: Worm wheel	$16 \le Z_2 \le 80$
Center distance	10 mm ≤ a ≤ 80 mm
Axial module: Worm wheel	0.5 mm ≤ m_x ≤ 3 mm
Gear ratio	$10 \le u \le 80$
Pressure angle	$10° \le a_n \le 22°$
Profile shift coefficient: Worm wheel	$-0.2 \le x_2 \le 1.5$

Table 19.2: Geometry limit values for calculating wear according to Pech

The progression of the tooth trace deviation over time on the loaded and unloaded flank, according to Pech, can be seen in the next figure.

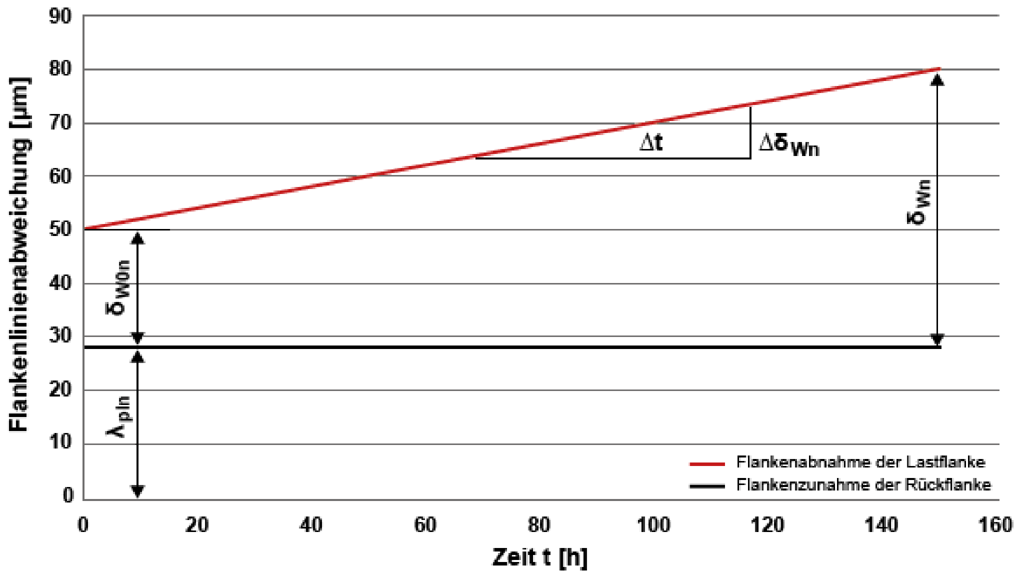


Figure 19.4: Figure: Development of tooth trace deviation on the loaded flank (decrease) and the unloaded flank (increase) according to Pech.

19.2.10.2 Service life

The system displays the required service life in the input field.
To enter it directly, and perform sizing, click the Sizing button. This process uses the minimum safety value for the tooth root and flank strength to calculate the rating life (in hours) for every gear and for every load you specify. The rating life is calculated according to ISO 6336-6:2006 using the Palmgren-Miner Rule. In the range of endurance limit, you can also select a modified form of the S-N curve (Woehler line) instead of ISO 6336 or DIN 3990. The program displays the system rating life and the minimum rating life of all the gears used in the configuration. You can also click the Sizing button to size the service life with or without defining a load spectrum (see chapter 15.2.8, Define load spectrum). For more detailed information about load spectra, see (see chapter 15.2.8, Define load spectrum).

► **Note**

The ISO 6336/Niemann method is primarily used to support the service life calculation.

19.2.10.3 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. The next table 5ba0bcd9e43 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336.

	Operational behavior of the driven machine

Operational behavior of the driving machine	uniform	moderate shocks	average shocks	heavy shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 19.3: Assignment of operational behavior to application factor

19.2.10.4 Power, torque and speed

Click the Sizing button next to the power input field (for torque) to calculate the power (torque) appropriate to maintain a predefined minimum level of safety (see chapter 15.22.5, Safety factors). Click the Plus button next to the power input field to apply a load spectra for power, torque and speed in the Define load spectrum (see chapter 15.2.8, Define load spectrum)window.

19.2.10.5 Strength details

Click on the **Details** button for the root and flank strength calculation to display a dialog in which you can make additional settings for the strength calculation.

19.2.10.5.1 Profile modification

You can modify the theoretical involute in high load capacity gears by grinding the toothing. You will find suggestions for sensible modifications (for cylindrical gears) in KISSsoft module Z15 (see chapter 15.7, Modifications). The type of profile modification has an effect on how the safety against scuffing is calculated. The load sharing factor X_γ is calculated differently depending on the profile modification. The main difference is whether the profile has been modified or not. However, the differences between the versions for **high load capacity gears** and for **smooth meshing** are relatively small. The strength calculation standard presumes that the tip relief C_a is properly sized, but does not provide any concrete guidelines. The load sharing factor X_γ is calculated as follows, depending on the type of profile modification according to DIN 3990:

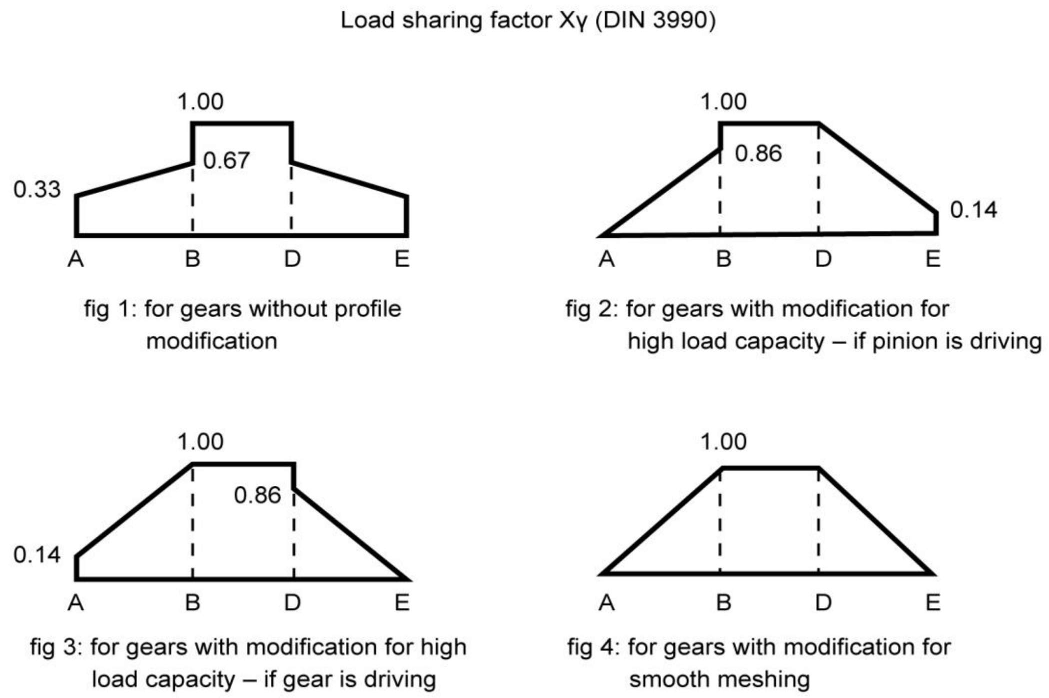


Figure 19.5: Force distribution factor Xγ for different profile modifications

19.2.10.5.2 Limited life coefficients as defined in ISO 6336

Set the limited life coefficient Z_{NT} to reduce the permitted material stress in accordance with ISO 6336- 2:2006:

$\sigma_{H\lim red} = Z_{NT} \cdot \sigma_{H\lim}$	(12.14)
$\sigma_{F\lim red} = Y_{NT} \cdot \sigma_{F\lim}$	(12.15)

As stated in ISO 6336, this value is important for cylindrical gear calculations and is the reason for the lower safeties for the range of endurance limit, compared with DIN 3990.

1. **normal (reduction to 0.85 for 10^{10} cycles):** The permitted material stress in the range of endurance limit (root and flank) is reduced again. The limited life coefficients Y_{NT} and Z_{NT} are set to 0.85 for ≥10^{10} load cycles.
2. **are increased if the quality is better (reduced to 0.92):** Y_{NT} and Z_{NT} are set to 0.92 for ≥10^{10} load cycles (in accordance with ISO 9085).
3. **with optimum quality and experience (always 1.0):** This removes the reduction and therefore corresponds to DIN 3990. However, this assumes the optimum treatment and monitoring of the materials.

19.2.10.5.3 Structural factor XwrelT or structural factor Xw (scuffing)

The structural factor takes into account differences in materials and heat treatment at scuffing temperature. The factor can be set using the appropriate Settings button. Structural factor X_{wrelT} (in DIN 3990 and in ISO/TS 6336-21:2022) or structural factor X_w (in ISO/TS 6336-20:2022) is used, depending on which standard is selected. However, in this case, $X_{wrelT} = X_w/X_{wT}$ and X_{wT}= 1 apply. This results in $X_{wrelT} = X_w$. The two factors are identical.

However, the standards do not provide any details about how to proceed when different types of material have been combined in pairs. You must input this factor yourself, because it is not set automatically by KISSsoft.

Through hardened steels	1.00
Phosphated steels	1.25
Coppered steels	1.50
Nitrided steels	1.50
Case-hardened steels	1.15 (with low austenite content)
Case-hardened steels	1.00 (with normal austenite content)
Case-hardened steels	0.85 (with high austenite content)
Stainless steels	0.45

Table 19.4: Structural factor according to DIN 3990-4

The standard does not provide any details about how this factor is to be applied when the pinion and gear are made of different types of material. In this case it is safer to take the lower value for the pair.

19.2.10.5.4 Number of load cycles

KISSsoft calculates the number of load cycles from the speed and the required service life. If you want to influence the value, you can define it in the **Number of load cycles for gear** n **window**. Click the Plus button to access this. Here, you can select one of five different calculations for calculating the number of load cycles.

1. **Automatically** The number of load cycles is calculated automatically from the rating life, speed, and number of idler gears.
2. **Number of load cycles** Here, you enter the number of load cycles in millions. You must select this option for all the gears involved in the calculation, to ensure this value is taken into account.
3. **Load cycles per revolution** Here you enter the number of load cycles per revolution. For a planetary gear unit with three planets, enter **3** for the sun and **1** for the planets in the input field.

Note:
If the **Automatically** selection button in the calculation module is selected, KISSsoft will determine the number of load cycles, taking into account the number of planets, in the **Planetary stage** calculation module.

4. **Load cycles per minute** Here you enter the number of load cycles per minute. This may be useful, for example, for racks or gear stages where the direction of rotation changes frequently, but for which no permanent speed has been defined.
5. **Effective length of rack** The rack length entered here is used to calculate the number of load cycles for the rack. The rack length must be greater than the gear's perimeter. Otherwise, the calculation must take into account the fact that not every gear tooth will mesh with another. You must enter a value here for rack and pinion pairs. Otherwise the values N_L(rack) = N_L(pinion)/10 are set.

► **Note**

This calculation method is used for transmissions that only travel over one oscillation angle.

Assume a scenario in which a reduction is present,

$$i = \frac{z_2}{z_1}$$

and an oscillation angle w in [°] from gear 2, where gear 2 constantly performs forwards and backwards movements with the angle value w_2. The effective endurance is given as the service life. The two coefficients f_{NL1} and f_{NL2}, which modify the absolute number of load cycles, N_L, are now calculated. To do this:

- a) Set the alternating bending factor of the pinion and gear to 0.7, or calculate it as defined in ISO 6336-3:2006. In this case, one complete forwards/backwards movement is counted as one load cycle.
- b) Coefficients f_{NL1} and F_{NL2} for pinion and gear are defined as follows:

$$f_{NL1,2} = \frac{ROUNDU\ P(\frac{W_{1,2}}{360})}{2 * \frac{W_{1,2}}{360}}$$

- w_2 = oscillation angle gear 2
- w_1 = W2*i
- ROUNDUP = round up to a whole number

The value in the counter displays the actual number of loads that occur during a complete cycle (forward and backward oscillation) on the flanks (not teeth) that are most frequently subjected to load. By rounding up this number to the next whole number, every rotation recorded is counted as a load.

Then, to determine the required $f_{NL1,2}$ factor, the actual number of loads that occur per flank is divided by the number of loads that would occur per cycle, if rotation were to continue without a backward rotation at the angle of rotation (1 load for each 360°).

Example calculation for $f_{NL1.2}$:

Gear 1 rotates through a half cycle at **540°** while gear 2 oscillates by **90°** (i = 6).

In a complete cycle, the oscillation angle moves forwards once an backwards once.

The actual number of load cycles that occur in a complete cycle on the flanks that are most frequently subjected to load (only one side of the tooth is taken into consideration) is then:

For gear 1:

$$ROUNDUP\left(\frac{540}{360}\right) = 2$$

For gear 2:

$$ROUNDUP\left(\frac{90}{360}\right) = 1$$

Without adjusting the coefficients, the number of counted load cycles in a complete cycle would then be:

For gear 1:

$$2 * \left(\frac{540}{360}\right) = 3$$

For gear 2:

$$2 * \left(\frac{90}{360}\right) = 0.5$$

The coefficients are therefore f_{NL1} and f_{NL2}:

$$f_{NL1} = \frac{2}{3} = 0.667$$

$$f_{NL2} = \frac{1}{0.5} = 2$$

- c) Then, input coefficients fNL1 and fNL2 in the **Load cycles per revolution** input field.

The strength calculation can now be performed for the correct number of load cycles, on the basis of the data entered in steps a through d.

19.2.10.5.5 Optimal tip relief

To calculate safety against micropitting as specified in Method B in ISO/TS 6336-22, you must specify whether or not the profile modification is to be assumed to be optimal. The same applies to calculating the safety against scuffing. The software checks whether the effective tip relief (Ca) roughly corresponds to the optimum tip relief (Ceff). If this check reveals large differences, i.e. Ca < 0.333*Ceff or Ca > 2.5*Ceff, a warning is displayed. In this case, the value you input is ignored and is documented accordingly in the report.

19.2.10.5.6 Hardening depth, known by its abbreviation "EHT"

You can input the intended hardening depth (for hardness HV400, for nitrided steels, or HV550 for all other steels). You can also input the hardness HV300. This value is then used to display the hardening curve as a graphic. The input applies to the depth measured during final machining (after grinding).

When you input this data, the safety of the hardened surface layer is calculated automatically according to DNV 41.2 [11]. A minimum value of t400 (nitrided steel) or t550 (all other steels) is used here. If only the value for HV300 is known, this value is then used. However, the calculation should then only be seen as an indication. The calculation is performed as described in the "Subsurface fatigue" section in [11]. The values required to define the CHD hardening depth coefficient Y_C, as specified in DNV 41.2, are also needed. The calculation does not use the same approach as the calculation for the proposal for the recommended hardening depth, but still returns similar results. To obtain a proposal for a sensible hardening depth, we recommend you call the relevant calculation by selecting **Report -> Proposals for hardening depth**. A maximum value for the hardening depth is only used to check the hardening depth at the tooth tip. It is mainly used for documentation purposes.

19.2.10.5.7 Load spectra with negative bins

Load spectra with negative load bins (T < 0 and/or n < 0) can be calculated as follows.

IMPORTANT:

- A load bin is considered to be negative if the non-working flank is placed under load.
- This does not apply for the calculation of the pitting safety for idler gears (in the case of planetary gear stages, it only applies to the sun and internal gear; in the case of planets, it is assumed that both flanks are always under load).
- When calculating the root safety, this is only applied to bins where the alternating bending factor is Y_M=1.0.

Torque factor	Speed factor	Flank under load	Load bin is
+	+	Working flank (*)	evaluated as positive

+	-	Working flank (*)	evaluated as positive
-	+	Non-working flank	evaluated as negative
-	-	Non-working flank	evaluated as negative

Table 19.5: Evaluation of a load bin, depending on the prefix operator

(*) Working flank as entered in the **Rating** tab

You can select the following in "**Details**" in the **Strength** group, in the **Rating** tab:

- For calculating pitting safety
  - Evaluate all negative load bins as positive (as up to now)
  - Only consider positive load bins
  - Only consider negative load bins
  - Calculate both cases and document the less favorable case
- For calculating root safety
  - Evaluate all negative load bins as positive (as up to now)
  - Increase tooth root stress for negative load bins by 1/0.7
  - Increase tooth root stress for positive load bins by 1/0.7
  - Calculate both cases and document the more realistic case

19.3 Settings

Click on **Calculation >Settings** or select the  icon to display the window for the **Module specific settings** sub-menu. From here, you can access the tabs listed below to input other calculation parameters (the following parameters are not described here: (see chapter 15.22, Settings)).

19.4 Notes

19.4.1 Checking the contact pattern

The collision check shown in the 2D graphic (see chapter 23.2.4, Meshing) can only be used to a limited extent for crossed helical gears, because it only works for a shaft angle of 90° and does not supply informative enough results. Some of the flank line modifications are not taken into account, and meshing is only displayed in the axial section, middle for Gear 1 and in the transverse section for Gear 2.

A better method for checking meshing is to use a 3D model. A 3D model includes all modifications and can be displayed for any axial crossing angle. You can use the "skin model" 3D model type to simulate a contact situation and then check it carefully using the meshing animation. In this case, click the appropriate function button to gently engage one gear with the other until the contact between the gear surfaces forms a contact pattern. Then, run the meshing animation. To ensure that the gears do not overlap too much, we recommend to set (in Properties) the number of rotation steps to between 100 and 500 or higher.

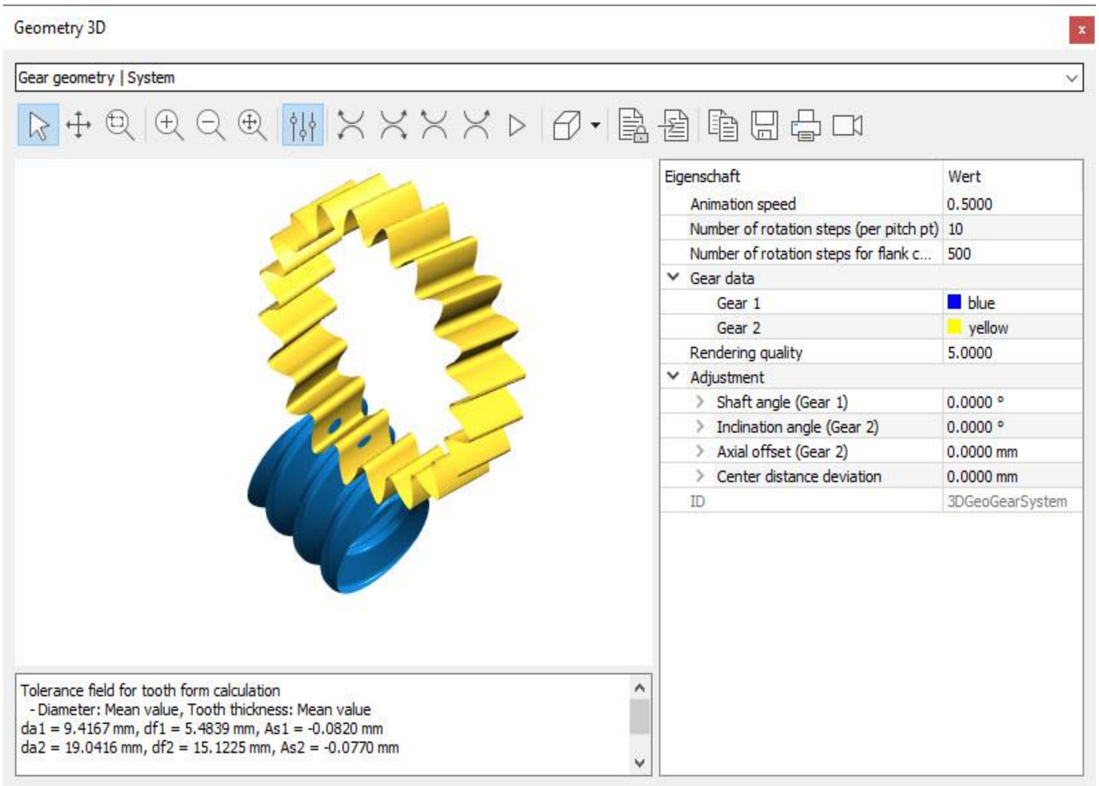


Figure 19.6: Contact pattern of a worm toothing

19.5 Crossed helical gear with rack

Click the Plus button next to the field in which you enter the number of teeth to select the crossed helical gear with rack configuration. Click the Plus button next to the field in which you enter the center distance to specify the height of the rack.

The number of load cycles on each tooth in the rack can either be input directly or calculated from the service life, pinion speed and rack length. Otherwise, the operation is identical to that used for a gear pair.

20 Beveloid Gears

Beveloid gears, also known as conical gears, are generated by a rack-like tool which is tilted by a predefined angle (see K. Roth, Zahnradtechnik – Evolventen-Sonderverzahnung [3])

Beveloid gears are primarily used in two particular areas: to generate a shaft angle between two meshing gears. Alternatively, two beveloid gears with opposing cone angles can be used to generate backlash-free toothing.
Beveloid gears with a shaft angle can be used to achieve a compact type of gear unit.

Unfortunately, no standards or guidelines have yet been drawn up for the calculation of the complex geometry, or for strength.
For this reason, the geometry calculation method used in KISSsoft is based on standard technical literature and publications. The main data used is taken from the publications mentioned in the next section.

For simplicity's sake, the strength in the mid section is calculated as if for a cylindrical gear pair.

20.1 Underlying principles of calculation

The basic calculation of the geometry and tooth form for a single beveloid gear is based on K.Roth [3], and on well known standards for cylindrical gears (e.g. DIN 3960, DIN 867, etc.).

Therefore, a beveloid gear is generated using the same process as a cylindrical gear, except that the profile shift changes along the facewidth. And this therefore changes all the parameters which are affected by the profile shift.

For helical toothed beveloids, the cutter is not only tilted by a cone angle θ, but also by an additional helix angle β. In the transverse section, this creates a trapezoidal reference profile with different pressure angles α on the left and the right side. This has a significant effect on the tooth form, because it changes the base circles.

The changes to the profile shift across the facewidth mean that beveloid gears often run the risk of undercut at the root or having teeth with a pointed tip. The profile shift at the toe and heel is calculated by

$$x^*_{\min,\max} = x^*_m \ \mu \frac{b \bullet \tan\theta}{2 \bullet m_n}$$

The undercut limit and minimum topland are only output in the error message if the values are exceeded by the data that has just been entered. As the two sizes on the left and right may be different (in the case of helical gear teeth), the system displays the more unfavorable value in each case.

The beveloid pair's meshing conditions are calculated on the basis of the publications by S. J. Tsai: see [55] and [56]. In this case it is important to note that the parameters are sub-divided into manufacturing and working parameters ("Manufacturing data and working data" section).

20.2 Basic data

20.2.1 Normal module

You can enter the normal module here.
However, if you know the "Pitch", "Transverse Module" or "Diametral Pitch" instead of this, click on the conversion button to display a dialog window in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select **Input normal diametral pitch instead of normal module** by selecting **Calculation > Settings > General**.

20.2.2 Pressure angle at normal section

This entry relates to the reference profile's flank angle. The normal pressure angle on the beveloid gear's reference circle is dependent on the cone angle and helix angle. [3]

20.2.3 Helix angle

Here you can enter the helix angle, or else select a spur gear toothing. The helix angle entry only applies to gear 1. Gear 2 may have a different helix angle value from gear 1, and is calculated. For gears with total profile shift 0, the following equation applies for determining the second helix angle from the entered parameters:

$$\cos \sum = \cos\theta_1 \cos\theta_2 \cos(\beta_1 + \beta_2) - \sin\theta_1 \sin\theta_2$$

20.2.4 Shaft angle

You can specify the shaft angle between the two axes of rotation here.

The shaft angle between any two straight pitches can be determined from the scalar product of the direction vectors of the two straight pitches. This corresponds to the angle between the two straight pitches in the plan view along the distance vector between the two straight pitches.

20.2.5 Number of teeth

The number of teeth defines the transmission ratio of the gears. Only even numbered, positive values are permitted.

20.2.6 Width

Facewidth of the gears. Please note that, when the width and cone angle are very large, the profile shifts between the toe and heel may be very different. For this reason, you cannot input just any value for the width, because this might, for example, create a pointed tooth.

At present, you cannot specify an axial offset. This means the gear pair contact is always in the middle of the gear.

20.2.7 Cone angle

The specified cone angle corresponds to the manufacturing parameter used to set the misalignment of the milling cutter to the gear. Both positive and negative cone angles are permitted, however, the total cone angle must be at least 0.

20.2.8 Profile shift coefficient (center)

The profile shift coefficient is defined in the same way as for a standard cylindrical gear, but the value relates to the value at the middle of the beveloid gear. When this calculation is performed, the Results window displays the size of the profile shifts at the toe and heel of the gear.

20.2.9 Quality

The quality achieved when generating the beveloid gear.

20.2.10 Material and lubrication

The entry is the same as the normal entry, as for cylindrical gears.

20.3 Reference profile

In the "Reference profile" tab, you can either input the reference profile for the manufacturing process in the same way as for a cylindrical gear calculation, or define the tools directly.

In this case, you must modify the height in the reference profile as follows to calculate the tooth form in transverse section (see K. Roth [3], section 5.2.6):

$$h_{aC} = \frac{h_{aP}}{\cos\theta}, \quad h_{fC} = \frac{h_{fP}}{\cos\theta}$$

Here, the subscript C represents the heights in the transverse section of the beveloid gear (calculated values) and P represents the heights of the reference profile (input values). You can check these values in the main report by selecting "Summary / Reference profile / Gears".

20.4 Modifications

The selection options for modifications in the beveloid gear module are limited.

In general, the contact pattern for beveloid gears with a shaft angle that is not 0 improves if negative crowning is used. To do this, you can input the "Crowning" modification and define a negative value.

20.5 Factors

The face load factor $K_{h\beta}$ cannot be calculated automatically for beveloid gears, and must therefore be set by the user. A value of 1.5 is used by default.

20.6 Dimensioning

As far as we know, no standards or research projects have yet been completed which involve calculating the load on beveloid gear pairs. For this reason, the calculation of strength is performed using replacement cylindrical gear toothing in the mid section.

In this case, note that $K_{h\beta}$, in particular, can differ a great deal from the values in the common gear standards. For this reason, the factor must be entered manually.
Minor differences may occur in the calculated safeties produced during cylindrical gear calculation and beveloid gear calculation, which are caused by a slight difference in the way the contact ratio is calculated.

20.7 Manufacturing Data and Working Data

As we are performing the calculation of the beveloid pair according to J. Tsai [55], it is important to know the difference between "manufacturing data" and "working data".

Manufacturing data is the data that is decisive for manufacturing. This category includes the values you input in the 'Basic data' tab. In contrast to this is the working data, which relates to the generation geometry of the beveloid gears that are in use. An example is the cone angle θ of the angle at which the tool is tilted during manufacturing. In contrast, working cone angle θw is the angle of the pitch cone of the beveloid gears in the meshing.

The working data is required to calculate a correct pairing, at which the contact point of the gears is in the middle of both beveloid gears. For example, if all the other parameters result in the helix angle value β_w from gear 2 at the operating point, this is then converted into a helix angle β for the manufacturing process.

The working data is also needed to position the two gears relative to each other. To position a gear pair in a 3D CAD environment, gear 2 is positioned relative to gear 1 as follows:

1. displacement along the Y-axis at r_{w1}
2. rotation around the X-axis with θ_{w1}
3. rotation around the negative y-axis with $\beta_{w1}+ \beta_{w2}$
4. rotation around the X-axis with θ_{w2}
5. displacement along the Y-axis at r_{w2}

21 Non-Circular Gears

You can use KISSsoft's non-circular gear module to calculate gears with non-circular gear bodies.

21.1 Input data

Input the geometry, generation and tolerance values in the **Basic data** tab.

Then, enter the details for generating non-circular intermeshing in the **Reference profile** tab.

21.1.1 Geometry

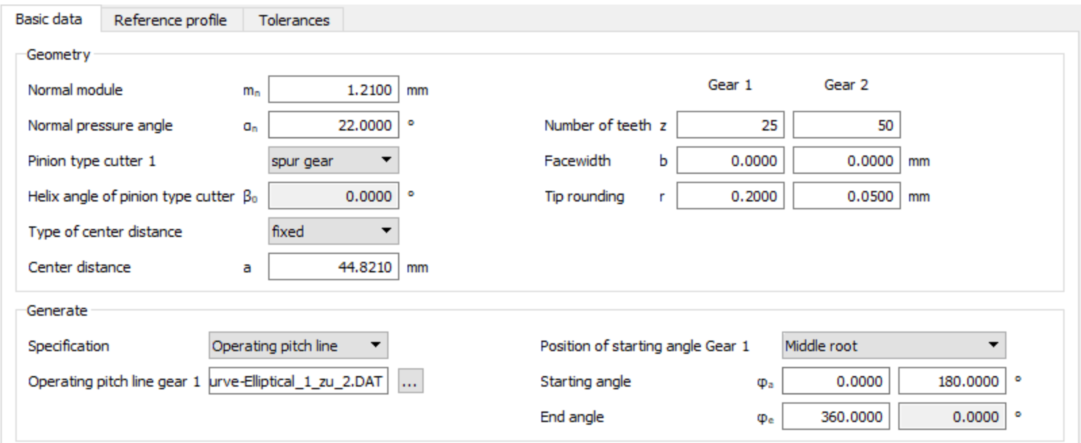


Figure 21.1: Basic data tab: entries for a non-circular gear pair

The module is defined from the "**Results window**" (total length of operating pitch line/[number of teeth* π]=module).

Results (basic calculation)

		Gear 1	Gear 2
Tooth thickness allowance (normal section) (mm)	[As]	-0.030	-0.030
Total length of operating pitch line (mm)	[L_{pitch}]	223.36	223.30
Starting angle (°)	[φ_a]	0.00	180.00
Middle (°)	[φ_m]	180.00	90.00
End angle (°)	[φ_e]	360.00	0.00
Corresponding length on the operating pitch line (mm)	[L_a]	16.71	
(mm)	[L_m]	64.23	
(mm)	[L_e]	111.75	
Exact number of teeth	[zeff]	25.001	zeff = (L_e-L_a)/(m_n*π)

Figure 21.2: Results window

To save time in the first phase of the sizing process, we recommend you do not enter the total number of teeth z. We suggest you perform the calculation with a lower number of teeth (e.g. 2). In this case, although all the operating pitch lines are calculated completely, only the specified number of teeth (2) are calculated and displayed.

Initially, start the calculation with a pressure angle in the normal section α_n of 20°. Later on you can change this angle instead of the profile shift or to optimize the tooth form.

21.1.1.1 Generation

The start and end angles φ_a and $\varphi_e$are important values because they determine the operating pitch line of gear 1, i.e. the area that will be generated In closed curves the angle φ_a is 0° and φ_e is 360°.

The operating pitch lines or the ratio progression are then defined in files. The files must be in either "**dat**" or "**dxf**" format. These files can be stored in any directory. It is important to register these files correctly, using the ... button.

Operating pitch lines are also stored in the .Z40 file. As a consequence, when you load a new calculation, you do not need to access the .dat file. In this case you see a message to tell you the file cannot be found, and existing data will be used instead.

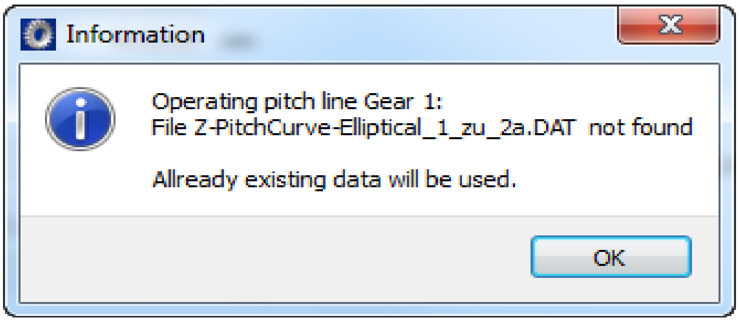


Figure 21.3: Message

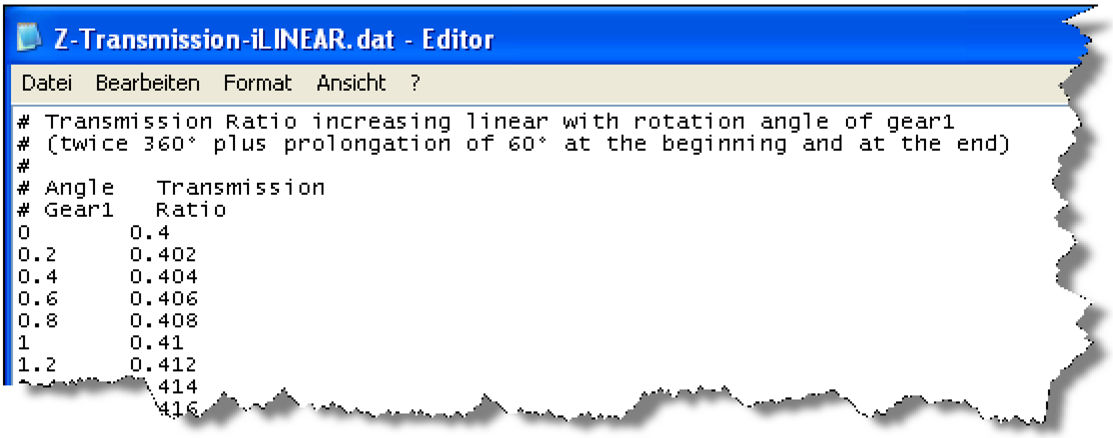
► **Note**

The progression (ratio or operating pitch line) must be defined from at least the starting angle to the end angle. To achieve uniform meshing, the curve must have approximately 30° forward motion and follow-up movement. If the curve has no forward motion and/or follow-up movement, the software extends it automatically.

21.1.1.1.1 Input format for data in imported files

You can predefine one or two operating pitch lines or the ratio progression. The imported files must have ".dat" as their file extension.

A maximum of 7,800 lines can be processed during non-circular gear analysis. Lines that start with # are comments and are ignored. To predefine the ratio progression, input the angle on gear 1 and the ratio.



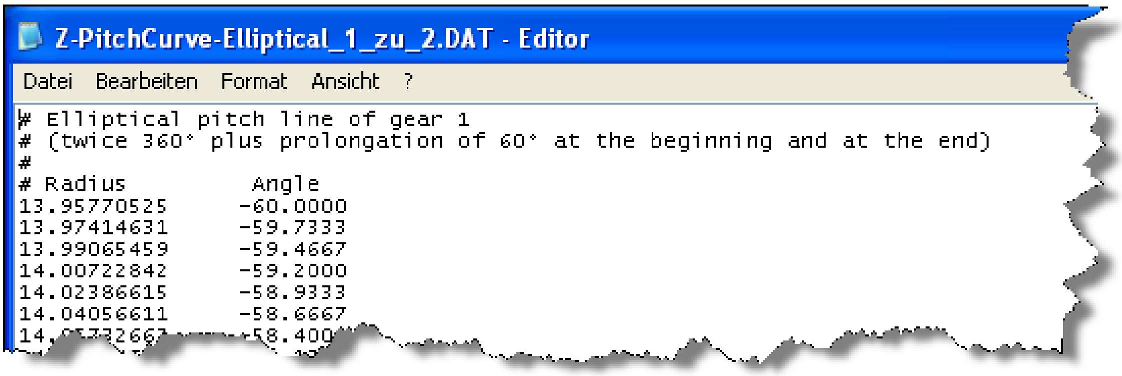
Z-Transmission-iLINEAR.dat - Editor

Datei Bearbeiten Format Ansicht ?

```
# Transmission Ratio increasing linear with rotation angle of gear1
# (twice 360° plus prolongation of 60° at the beginning and at the end)
#
# Angle   Transmission
# Gear1   Ratio
0       0.4
0.2     0.402
0.4     0.404
0.6     0.406
0.8     0.408
1       0.41
1.2     0.412
          414
          416
```

Figure 21.4: Example of ratio progression

To predefine the operating pitch line progression, input the radius and the angle.



Z-PitchCurve-Elliptical_1_zu_2.DAT - Editor

Datei Bearbeiten Format Ansicht ?

```
# Elliptical pitch line of gear 1
# (twice 360° plus prolongation of 60° at the beginning and at the end)
#
# Radius          Angle
13.95770525      -60.0000
13.97414631      -59.7333
13.99065459      -59.4667
14.00722842      -59.2000
14.02386615      -58.9333
14.04056611      -58.6667
14.               -58.400
```

Figure 21.5: Example of an operating pitch line

21.1.2 Tolerances

We recommend you enter sufficiently large tooth thickness allowances A_{sn} (e.g. -0.10/-0.12 for module 2).

21.1.3 Reference profile

You must specify a topping pinion type cutter. The same pinion type cutter is usually defined for both gear 1 and gear 2.

Problems may arise unless the profile shift coefficient of the pinion type cutter is set to 0. You must then carefully check exactly how the gears are generated.

21.2 Notes on how to operate KISSsoft

21.2.1 Angle error

When you use an operating pitch line or gear reduction progression to input a closed curve (Gear 1), it must start at 0° and finish at 360°. For this reason, the rotation of gear 2 must also be 360° (or a multiple of this). If not, this will result in an error.

Figure 21.6: Minor error for Gear 2. φe is 183.256 instead of 180°

However, this error has no effect because the predefined gear backlash is large enough.

21.2.2 Checking the meshing

A useful way of checking the meshing is to change the number of rotation steps (per 360°) to rotate the gear in larger or smaller steps. You change the step sizes, as usual, in the Graphics window.

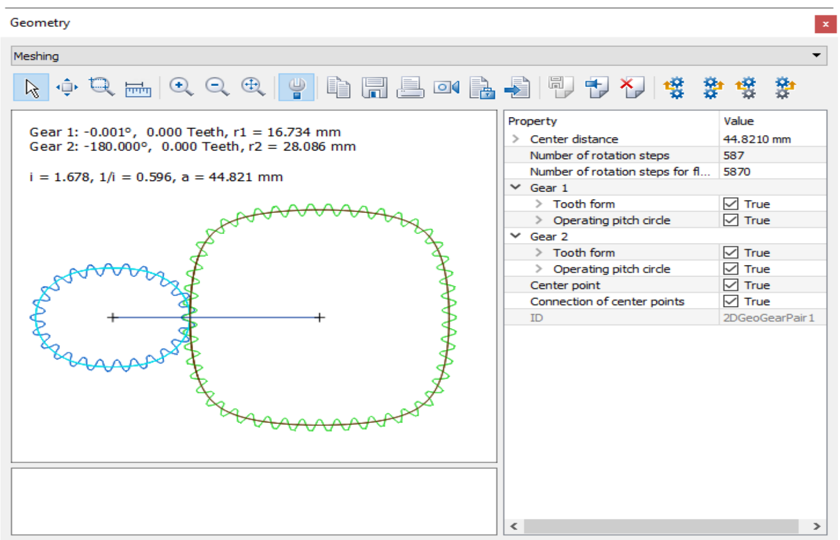


Figure 21.7: Changing the rotation steps

When you generate gears with allowances, we recommend you click the  button to bring the gears into flank contact with each other.

► **Note**

If, when you click the "**Rotate independently to the right**" button, one gear rotates too far against the other (or not far enough), you must adjust the number of "rotation steps" accordingly!

21.2.3 Improving the tooth form

You can change the tooth form of circular gears quite significantly by changing the profile shift. In the current version of the program for non-circular gears, we recommend you set the profile shift coefficient of the pinion type cutter $x^*_0=0$. Despite this, you can still modify the tooth form by changing the pressure angle α_n .

21.2.4 Accuracy of the tooth form

Select **"Calculations" > "Settings"** to predefine the accuracy (and therefore also the size of the file) for an IGES or DXF export.

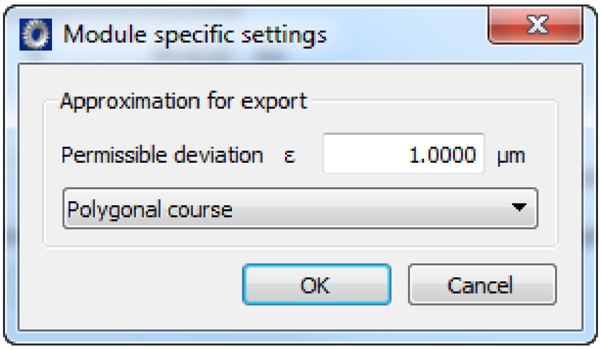


Figure 21.8: Module specific settings

This entry only influences IGES or DXF files.

In the software, the tooth form (each flank) is calculated with 100 points. You will find these results in the TMP files (and in the report). If you want to modify the number of internally calculated points, simply change the appropriate entry in the .Z40 file:

Go to a saved .Z40 file and search for the line that contains

ZSnc.AnzPunkteProFlanke=100;

and, for example, replace 100 with 40. If you do so, only 40 points per flank will be calculated.

21.2.5 Exporting individual teeth

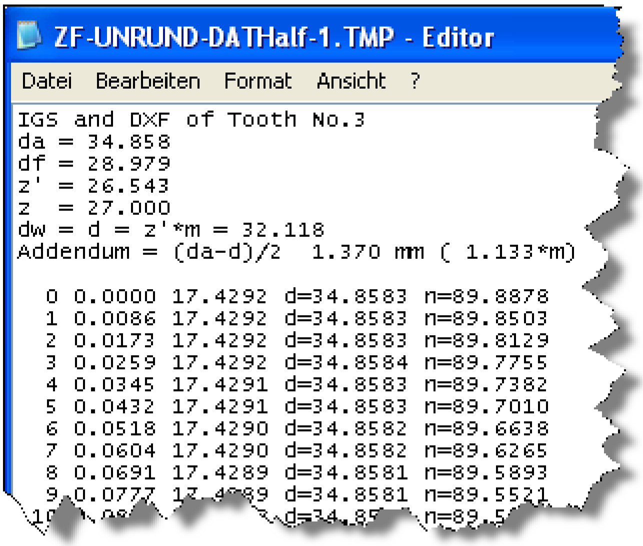
Go to a saved .Z40 file and search for the line that contains

ZRnc[0].AusgabeKontur=0, for Gear 1 or

ZRnc[1].AusgabeKontur=0, for Gear 2.

There, change the variable to the required value, e.g. ZRnc[0].AusgabeKontur=3.

The LEFT flank of the x-th tooth space (therefore the 3rd gap in Gear 1, in the example) is always exported.



```
ZF-UNRUND-DATHalf-1.TMP - Editor
Datei  Bearbeiten  Format  Ansicht  ?
IGS and DXF of Tooth No.3
da = 34.858
df = 28.979
z' = 26.543
z  = 27.000
dw = d = z'*m = 32.118
Addendum = (da-d)/2  1.370 mm ( 1.133*m)

  0 0.0000 17.4292 d=34.8583 n=89.8878
  1 0.0086 17.4292 d=34.8583 n=89.8503
  2 0.0173 17.4292 d=34.8583 n=89.8129
  3 0.0259 17.4292 d=34.8584 n=89.7755
  4 0.0345 17.4291 d=34.8583 n=89.7382
  5 0.0432 17.4291 d=34.8583 n=89.7010
  6 0.0518 17.4290 d=34.8582 n=89.6638
  7 0.0604 17.4290 d=34.8582 n=89.6265
  8 0.0691 17.4289 d=34.8581 n=89.5893
  9 0.0777 17.4289 d=34.8581 n=89.5521
```

Figure 21.9: Temporary file for exporting teeth (ZRnc[0].AusgabeKontur=3, for Gear 1)

21.2.6 Report

If you have select Detailed in **Report settings**, this report will also contain a lot of information. If you want a shorter version, set "Extent of data" to 5 (standard).

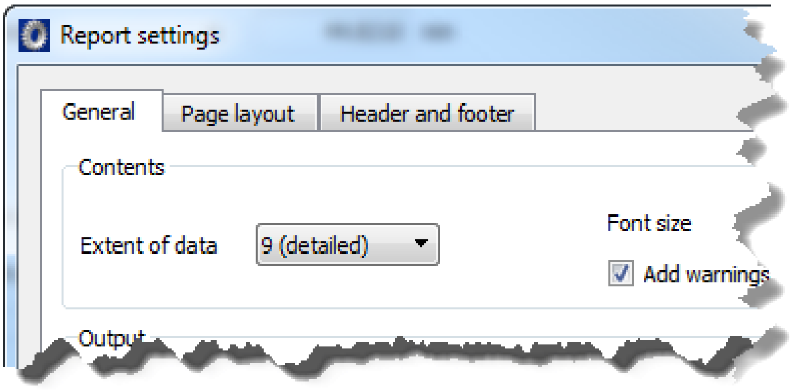


Figure 21.10: Report settings with a changed data scope for output to a report

21.2.7 Temporary files

When a calculation is performed, KISSsoft automatically generates temporary files. The directory in which these files are generated by KISSsoft must be specified in KISS.ini in the "PATH" section. You will find KISS.ini in the KISSsoft main directory. Before changing the default setting, you must ensure that you have read and write permissions for the changed directory. You will find more detailed information in Chapter 2 of the manual, "Setting Up KISSsoft".

ZF-H1_Rad 1 (Schritt 2).TMP: ZF-H1_Rad 2 (Schritt 2).TMP:	Unimportant information: contains details about generating the noncircular gear, flank by flank.
ZF-UNRUND-1.TMP:	Contains interesting information about operating pitch line 1: defining the contact points on operating pitch line 1, calculating operating pitch line 2 from operating pitch line 1, operating pitch line lengths, documentation about the intermeshing (individual points) of non-circular gear 1 with X, Y, normal, diameter and angle
ZF-UNRUND-2.TMP:	Contains interesting information: documentation about the intermeshing (individual points) of non-circular gear 2 with X, Y, normal, diameter and angle
ZF-UNRUND-DAT-1.TMP: ZF-UNRUND-DAT-2.TMP:	Possible further use of the gear teeth (individual points) X,Y coordinates
ZF-UNRUND-OPLINE-1.TMP: ZF-UNRUND-OPLINE-2.TMP:	Possible further uses of the operating pitch line (individual points) x- and y-coordinates
Z-WalzKurve-1.TMP: Z-WalzKurve-2.TMP:	Possible further uses of the operating pitch line r (individual points), ϕ coordinates (*). Has exactly the same format as the .dat file (see "Import format" section).

Z-OpPitchPoints-1.TMP: Z-OpPitchPoints-2.TMP:	Possible further use of meshing points on each tooth in r, ϕ coordinates

22 Report Menu

22.1 Drawing data

Select **Drawing data** to display the toothing data you want to add to a drawing. Use the **Z10GEAR1?.RPT** file (for Gear 1), and the **Z10GEAR2?.RPT** file (for Gear 2), etc. (? = d/e/f/i/s for the required language) to modify the template to your own requirements.

All the angle data for the user-specific Z10GEAR1?.rpt ... Z10GEAR4?.rpt reports is given in degrees-minutes-seconds, and displayed in brackets after the decimal point.

For example the number 20.3529° is displayed as:

20° 21' 10" (20.3529)

22.2 Manufacturing tolerances

Select the **Manufacturing tolerances** menu option to generate a report with all the manufacturing tolerances specified in ISO 1328 (DIN ISO 1328), DIN 3961:1978, AGMA 2000, AGMA 2015 and BS 436 standards.

22.3 Summary

Use the summary function to compare the current toothing with the results of **fine sizing**.

22.4 Service life calculation

This report shows the most important data that is used to calculate service life either with or without a load spectrum (see chapter 15.2.8, Define load spectrum). You can also call the service life calculation by clicking the Sizing button next to the Service life input field. This then displays the service life that should be achieved if required safeties are used.

22.5 Sizing of torque

The sizing of torque displays the most important data required to calculate the transmittable torque (or the maximum transmittable power) with or without load spectrum. You can also call the sizing of torque function directly by clicking the checkbox next to the Torque or Power input fields. You then see a value for the torque that should be achieved if required safeties are used.

22.6 Proposals for hardening depth

A wide range of different proposals for the hardening depth EHT as specified in the standards have been documented. The data specified in the ISO, AGMA and Niemann standards are often very different, because of the very rough approximations involved. The most accurate calculation, which uses the shearing stress criterion from the Hertzian law to define the required hardening depth, is documented in the upper part of the report. You can also specify the safety factor which is to be used for the calculation (see chapter 15.22.5.1, Safety factor for the calculation of the shear stress at hardening depth). For the graphical display (see chapter 23.4.4, Surface layer shear stress).

23 Graphics Menu

In the **Graphics** menu, you can select various menu options to help you display gear teeth and functional processes.

► **Note**

Right-click to display a context menu that contains other operating functions.

The table (see Table 23.1) shows which of the options in the **Graphics** menu are supported by particular gear calculation modules, and where you can find the relevant documentation in this section.

Menu option	**Options**	**Section**											
AGMA 925	Lubricant film thickness/ specific oil film thickness/ temperature in contact/ Hertzian Pressure	23.1.1		✔	✔	✔	✔	✔					
Geometry 2D	Tooth form	23.2.1	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Tool	23.2.2	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Manufacturing	23.2.3	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Meshing	23.2.4		✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Meshing (slices)	23.2.5										✔	
	Profile diagram/flank line diagram	23.2.6	✔	✔	✔	✔	✔	✔		✔		✔	✔
	Drawing	23.2.7	✔	✔	✔	✔	✔	✔	✔				
	Manufacturing drawing	23.2.8	✔	✔	✔	✔	✔	✔					✔
Geometry 3D	Gear geometry	23.3.1	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔

	Modifications	23.3.2	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Axis alignment	23.3.3		✔		✔							
Evaluation	Specific sliding	23.5.2.4		✔	✔	✔	✔	✔	✔				
	Contact temperature	23.4.2		✔	✔	✔	✔	✔	✔				
	Flash temperature	23.4.3		✔	✔	✔	✔	✔	✔				
	Surface layer shear stress	23.4.4		✔	✔	✔	✔	✔	✔	✔			
	Proposed hardening depth	23.4.5		✔	✔	✔	✔	✔					
	Theoretical contact stiffness	23.4.6		✔		✔	✔	✔					
	Woehler lines (S-N curves)	23.4.7		✔	✔	✔	✔	✔	✔				
	Safety factor curves	23.4.8		✔	✔	✔	✔	✔	✔				
	Oil viscosity	23.4.14	✔	✔	✔	✔	✔	✔	✔	✔			
	Reliability	23.4.10		✔	✔	✔	✔	✔	✔	✔	✔		
	Gap analysis	23.4.14		✔	✔	✔	✔	✔					
	Face load distribution	23.4.14		✔	✔	✔	✔	✔					
	Backlash with actual tooth form	23.4.13		✔		✔	✔	✔					
	Tooth flank fracture	23.4.14		✔	✔	✔	✔	✔	✔	✔			
	Sliding velocity	23.4.15								✔			
	Contact line	23.4.16								✔			
	Stress curve	23.4.17								✔			
	Scuffing and sliding velocity	23.4.18								✔			

Contact analysis > Excitation	Meshing	23.5.1.1	✔	✔	✔	✔	✔	✔					
	Transmission error	23.4.17	✔	✔	✔	✔	✔	✔	✔				
	Transmission error acceleration	23.5.1.3		✔	✔	✔	✔	✔					
	Transmission error acceleration amplitude	23.5.1.4		✔	✔	✔	✔	✔	✔				
	Speed curve	23.5.1.5		✔	✔	✔	✔	✔					
	Excitation force	23.5.1.6		✔	✔	✔	✔	✔	✔				
	Excitation force amplitude spectrum	23.5.1.7		✔	✔	✔	✔	✔	✔				
	Torque curve	23.5.1.8		✔	✔	✔	✔	✔	✔				
	Single contact stiffness	23.5.1.9		✔	✔	✔	✔	✔	✔				
	Stiffness curve	23.5.1.10		✔	✔	✔	✔	✔	✔				
	Amplitude spectrum of the contact stiffness	23.5.1.11		✔	✔	✔	✔	✔					
	Kinematics	23.5.1.12		✔	✔	✔	✔	✔	✔				
Contact analysis > Efficiency	Total power loss	23.5.2.1		✔	✔	✔	✔	✔					
	Specific power loss	23.5.2.2		✔	✔	✔	✔	✔					
	Efficiency progression	23.5.2.3		✔	✔	✔	✔	✔					
	Specific sliding	23.5.2.4		✔	✔	✔	✔	✔					
	Heat development	23.5.2.5		✔	✔	✔	✔	✔					
	Heat development	23.5.2.6		✔	✔	✔	✔	✔					

	along the tooth flank												
	Contact temperature	23.5.2.7		✔	✔	✔	✔	✔	✔				
	Lubricant film	23.5.2.8		✔	✔	✔	✔	✔					
	Specific film thickness	23.5.2.9		✔	✔	✔	✔	✔					
Contact analysis > Forces and stresses	Contact lines on tooth flank	23.5.3.1		✔	✔	✔	✔	✔	✔				
	Contact pattern on tooth flank	23.5.3.2		✔	✔		✔	✔	✔				
	Load distribution on operating pitch circle	23.5.3.3		✔	✔		✔	✔					
	Normal force curve (line load, length of path of contact)	23.5.3.4		✔	✔	✔	✔	✔	✔				
	Normal force distribution on tooth (line load, length of path of contact)	23.5.3.5		✔	✔	✔	✔	✔					
	Bearing force curve	23.5.3.6		✔	✔	✔	✔	✔					
	Bearing force curve in %	23.5.3.7		✔	✔	✔	✔	✔					
	Direction of the bearing forces	23.5.3.8		✔	✔	✔	✔	✔					
	Stress curve (Hertzian pressure)	23.5.3.9		✔	✔	✔	✔	✔	✔				
	Tooth root stress	23.5.3.10		✔	✔	✔	✔	✔	✔				

	Tooth root stress across facewidth	23.5.3.11		✔	✔	✔	✔	✔	✔				
	Bending stress in root area	23.5.3.12		✔	✔	✔	✔	✔	✔				
	Stress distribution on tooth	23.5.3.13		✔	✔	✔	✔	✔					
Contact analysis > System	Load distribution of planets	23.5.4.1				✔							
	Position of sun	23.5.4.2				✔							
	Safety against scuffing	23.5.4.3		✔	✔	✔	✔	✔	✔				
	Safety against micropitting	23.5.4.4		✔	✔	✔	✔	✔					
	Safety against micropitting at the tooth	23.5.4.5		✔									
	Wear along the tooth flank	23.5.4.6		✔	✔	✔	✔	✔	✔				
	Wear progress along the tooth flank	23.5.4.7		✔	✔		✔	✔					
	Safety against tooth flank fracture	23.5.4.8		✔	✔	✔	✔	✔	✔				
3D FEM	Maximum tooth root stress	23.7.1		✔	✔	✔	✔	✔					
	Load distribution	23.7.2		✔	✔	✔	✔	✔					

Table 23.1: Graphics menu in the KISSsoft interface menu bar.

- Single gear, - Cylindrical gear pair, - Pinion with rack, - Planetary gear stage, - Three gears, - Four gears, - Bevel and hypoid gears, - Face gears, - Worms with double enveloping worm wheels, - Crossed helical gears, - Splines (Geometry and Strength)

23.1 AGMA 925

23.1.1 Lubricant film thickness and specific oil film thickness

The lubricant film thickness h_e according to AGMA 925 is shown over the meshing cycle. Another figure shows the specific film thickness λ, which is a critical value for evaluating the risk of micropitting. Expressed in simple terms, λ is the ratio of the lubricant film thickness to the surface roughness.

23.2 2D Geometry

You can select a number of different output options from the drop-down list in the Tool bar in the **Geometry** graphics window:

23.2.1 Tooth form

Displaying a gear tooth form.

► **Note:**

Click the Property button above the graphic to specify the number of teeth that are to be displayed. You can display the gear in transverse section, normal section or axial section. Selecting the "Half tooth for export" option is also very useful if you want to export the tooth form and then reimport it into KISSsoft later on.

23.2.2 Tool

This displays the tool associated with the gear, if one is present.

23.2.3 Manufacture

Display the pairing: gear with cutter. Here, the gear is shown in blue and the cutter in green.

23.2.4 Meshing

Displays the meshing of two gears.

▶ **Note about face gears:**

In KISSsoft, the face gear is calculated by simulating the manufacturing process in different sections. You can display different sections at the same time. To do this, open the **Property browser** in the Graphic window, and set the property in the **section** you require to **True** (see Figure 23.1).

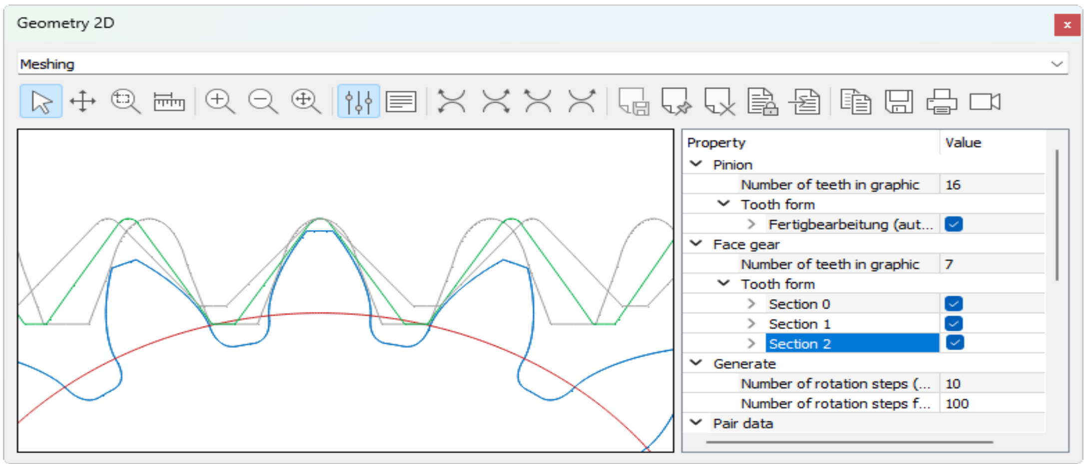


Figure 23.1: Meshing graphics window with Property Browser

The difference between the theory and the effective tooth form means that the tooth has an undercut! You can see this more clearly in the 2D view.

Collision check:

You can select the collision display option when generating two gears (in the graphical display). In the graphic, this shows (with squares) the points at which the gears touch or where collisions may occur.

shown in black: Contact (between 0.005 * module distance and 0.001 * module penetration)

shown in red: Collision (greater than 0.001 * module penetration)

The system identifies and marks collisions in all the meshing teeth. This option is particularly useful for analyzing the generation of non-involute tooth forms or measured tooth forms (using a 3D measurement machine) with a theoretical single flank generation check.

This function is also available for cylindrical gears and worm gears (but with restrictions for worm gears (see chapter 19.4.1, Checking the contact pattern)).

▶ **Note:**

If the **Make flank contact automatically** option is selected, you can only check the tooth meshing on contact. In this case, collisions are no longer displayed.

23.2.5 Meshing (slices)

In the meshing graphic, the tooth form is shown in several parallel sections in the axial direction of the worm (see Figure 23.2). Select **Modul specific settings > Diagrams** to set the number of slices to be displayed and the distance between the parallel sections. The middle meshing slice is always in the middle of the worm.

This graphic is only available for crossed worm/crossed helical gear pairs.

The number of slices is limited to 21. Flank line modifications are not taken into account in the tooth form calculation.

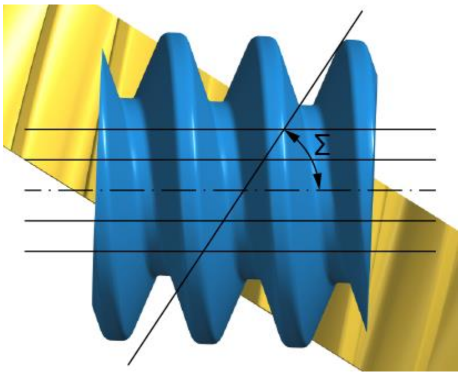


Figure 23.2: Position of slices in the meshing graphic

23.2.6 Profile and flank line diagram

These diagrams are generated by placing two lines diagonally over the tolerance band, as described in ANSI/AGMA: 2000-A88 (Figures 1 and 2).

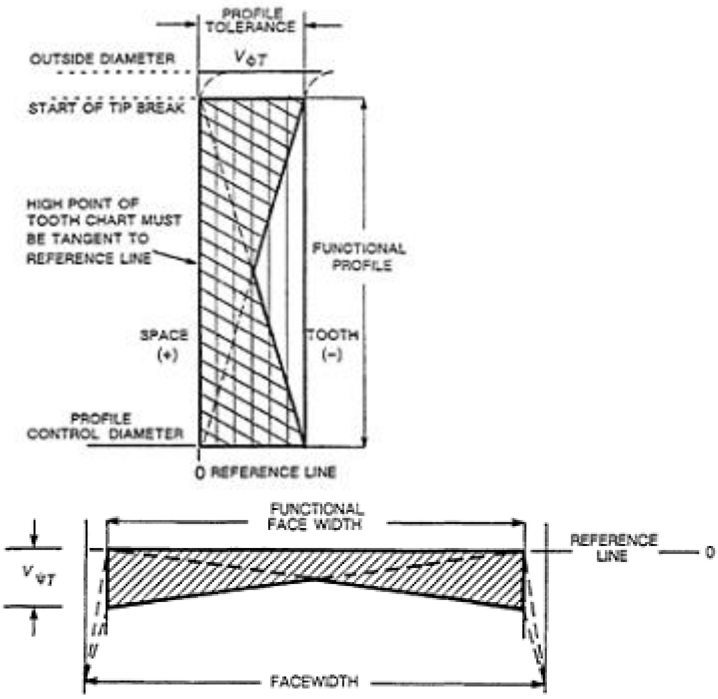


Figure 23.3: Profile diagram (left), flank line diagram (right)

In the figures shown above, $V_{\varphi T}$ is the profile tolerance and $V_{\psi T}$ is the tooth alignment tolerance which correspond to the total profile deviation (F_{α}) and the tooth helix deviation (F_{β}) as detailed in ISO 1328-1.

Although every company has its own method of creating profile and flank line diagrams, the AGMA standard is recognized as the default standard in the industry. ISO TR 10064-1 (and ISO FDIS 21771) also include a general description of profile and flank line diagrams, however without any explanations about the construction method.

In KISSsoft, the profile and flank line modifications are defined in the **Modifications** tab. The relevant diagrams are then generated using this data.

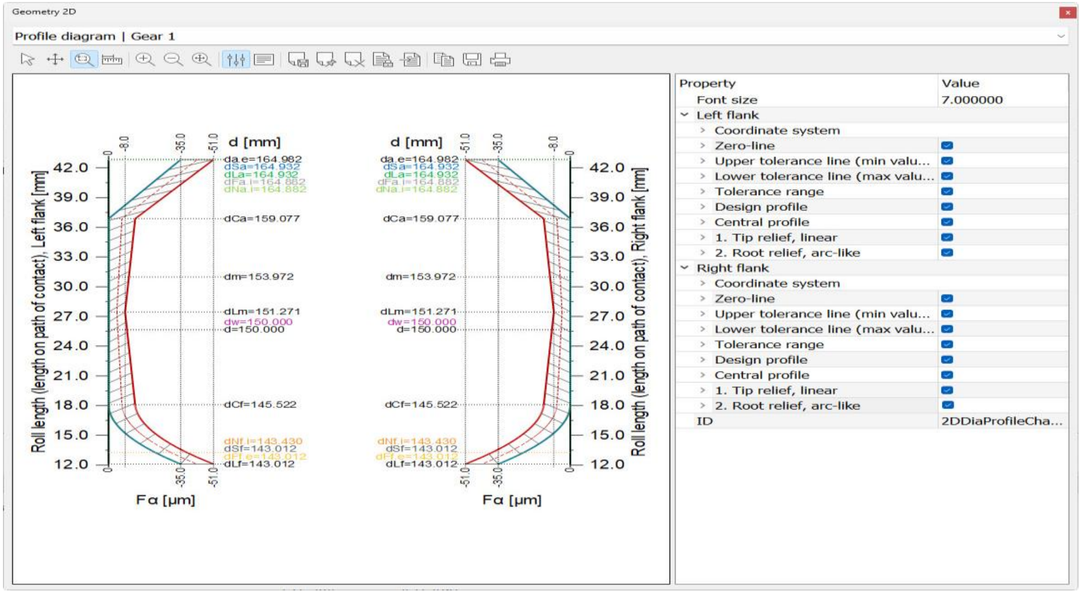


Figure 23.4: Profile diagram for gear 1 according to the predefined modifications

The horizontal axis of the profile diagram shows the profile deviation values and the vertical axis shows the coordinates along the profile. You can select different values for the left-hand vertical axis (roll angle or path of contact length) (**Calculation** > **Settings** > **Diagram**). The values for the right-hand flank are always given as the diameter. You can also specify the tolerance type by clicking on **Calculation** > **Settings** > **Diagram**. If you select the tolerance band type as specified in AGMA 20000-A88, the diagrams are generated according to the method mentioned above. If you set the tolerance band type to constant, the tolerance remains constant along the length or the width of the tooth flank. Click on the **Display central profile in the middle of the tolerance band** checkbox to specify whether the central profile (see below) should usually be displayed.

Description of the specific diameter of the right-hand vertical flank:

- d_{Sa}: Diameter at the end of the test diagram
- d_{Sf}: Diameter at the start of the test diagram
- d_{Ca}: Start of modification at tip
- d_{Cf}: Start of modification at root
- d_{Sm}: $d((L(d_{Sa}) + L(d_{Sf}))/2)$: Middle of the rolling path range (path of contact)

► **Note:**

The profile diagram is in the middle of the facewidth. The **Twist** profile modification is not possible.

Show curves in the diagram:

- Green line: Modifications of 1. **Tip relief, linear** and 2. **Tip relief, arc-like**.

- Blue line: Reference profile (current function profile used for checking and generated from the total of the modified curves).
- Red line: Tolerance curve generated by subtracting the total profile deviation from the reference profile. The profile deviation values are listed in the main report.
- Red line, dashed: Central profile, which can be entered as the target value for processing because it lies in the middle between the reference profile and the tolerance curve.
- Gray line: Tolerance range, which shows the range (as a crosshatched area) in which the actual manufacturing profile can lie.

The manufacturing profile (with tolerance) should lie between the tolerance curve and the reference profile.

You can use the properties to display or hide the individual lines or change their colors or line type.

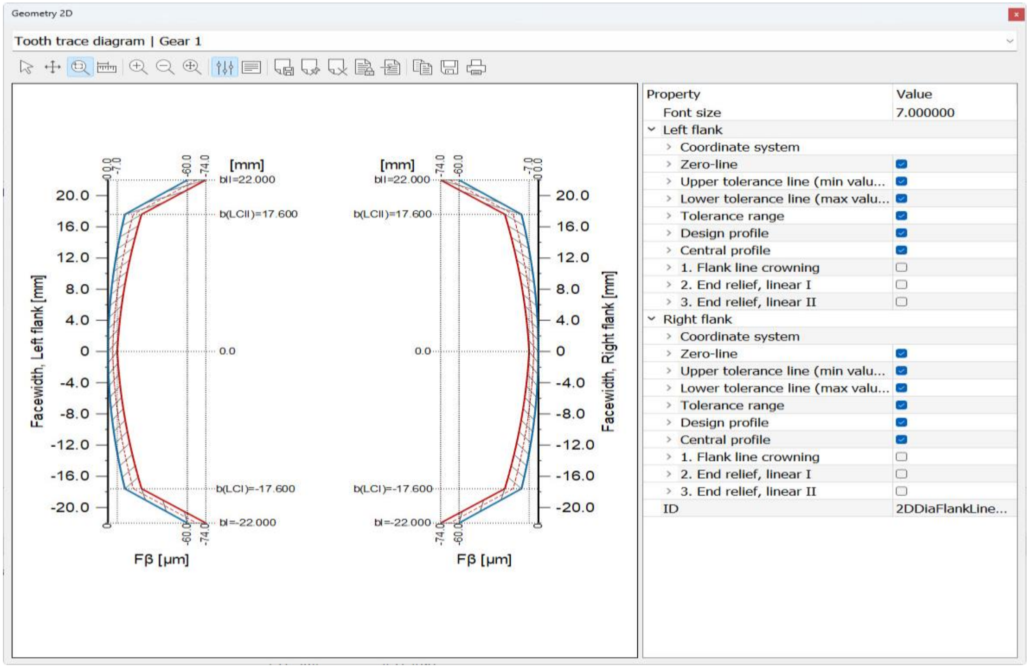


Figure 23.5: Flank line diagram for gear 1 with the predefined modifications

In the figure, the reference profile is shown in blue and the tolerance line is shown in red. The horizontal axis shows the coordinates along the flank line (facewidth) and the vertical axis shows the flank allowance as specified in the usual industrial conventions. The value of the total tooth trace deviation F_β is given in the main report.

The manufacturing flank line (with tolerances) should lie between the tolerance curve and the reference flank line.

Flank curvature radii

In this graphic you see the flank curvature radii along the flank line. Along with the normal force, these are critical values for Hertzian pressure.

Angle of flank normal

The normal angle to the flank is shown in this graphic. Every point on the tooth form has a normal.

23.2.7 Drawing

Use this menu to display gears schematically. The gears are shown in transverse and axial section.

This option is primarily used for bevel gears and worms.

Select **System** to generate a diagram showing how the gears are assembled. The gear (pair) assembly is shown in transverse and axial section.

Two views, section and overview, are given for bevel gears with a shaft angle of 90°. For shaft angles <> 90° only the section of the bevel gear pair is displayed.

23.2.8 Manufacturing drawing

23.2.8.1 General

Manufacturing drawings are designed to display a number of graphics on the same surface, and therefore create a print-ready image that can be used to manufacture a gear. You can also display the drawing data report at the same time. Use a control file to tailor the display to suit specific requirements. The control file is stored in the template directory (usually under KISSDIR\template). It has the module name and the file extension .grc (e.g. Z012gear1.grc).

You can also save the graphic generated here as a .dxf file in the usual way.

23.2.8.2 Editing the control file

You can modify the manufacturing drawing to suit your own requirements by making changes to the control file. The commands used to control the manufacturing drawing are described in the following table.

Papersize: A4 Papersize: A4 portrait Papersize: 297, 210	Specifies the required paper format. This refers to the standard terms used to describe commonly used paper sizes (A3, A4, A5, B4, B5, Letter, Legal and Ledger), and also enables you to input your own dimensions for width and height.

	The default setting is for landscape format. However, you can switch to portrait format by entering the key word "portrait".
Fontsize: 5	Specifies the required font size. The font size influences the size of the report and the diagram titles.
Units: inch	The default setting is that input values are assumed to be in mm. The system can handle these units: inch, mm and cm.

You can now add graphics that have specific properties. The table below gives an overview of the correct inputs.

Draw 2DDiaProfileChart1	“Draw" is the key word used to specify that a graphic is to be added. It is followed by the ID of the graphic you want to insert. The number at the end is part of the ID, and identifies the gear.
Window: 160, 285, 0, 85	“Window" identifies the window in which the graphic is displayed. The values show the limits on the left, right, bottom and top.
Scaletofit	This optional command forces the graphic to distort so that it fills the window in every direction. We recommend this for diagrams, but not for geometric presentations. If this term is not used, the original size and shape of the graphic is retained.
Fixedscale 2	This optional command generates a scaled output of the graphic. The number corresponds to the scale (in this case, 2 for M 2:1, 0.1 for M 1:10, etc.).

You can insert these graphics:

Tooth form	2DGeoToothDrawing
Drawing	2DGeoGearDrawing
Assembly	2DGeoAssemblyDrawing
Tool	2DGeoToolDrawing
Profile diagram	2DDiaProfileChart
Flank line diagram	2DDiaFlankLineChart
Angle of flank normal	2DDiaNormal

Finally, you can now display the report in the required location:

Write report1	“Write" is the key word used to create a gear data report. Enter report1 to select the gear data of gear 1, report2 to select the gear data from gear 2, etc.
Topright: 297, 218	Unlike graphics, you must specify an alignment here, which is defined with the first word. The correct commands are topright:, topleft:, bottomright: and bottomleft:. They represent the alignment (top right, top left, bottom right and bottom left). The next two values represent the particular reference point.

23.3 3D Geometry

The gears are displayed in the 3D Parasolid viewer.

You can select a number of different output options from the drop-down list in the Tool bar in the **Geometry 3D** graphics window. You can store the Parasolid viewer graphics in different file formats such as:

- Windows Bitmap (.bmp)
- Joint Photographic Experts Group (.jpg, .jpeg)
- Portable Network Graphics (.png)
- Standard for the Exchange of Product Model Data (.stp, .step)
- Parasolid Text File Format (.x_t)
- Parasolid Binary File Format (.x_b)

23.3.1 Tooth geometry

You can display the individual gear in 3D in the Parasolid Viewer here.

Select **System** to display the assembled system of gears in 3D.

23.3.2 Modifications

This graphic shows all the defined modifications along the width and the diameter. In the graphic settings you can switch between the flanks.

23.3.3 Axis alignment

Display the axis alignment of gear B relative to the axis of gear A. This display is a very useful way of checking the deviation error of axis and inclination error of axis.

23.4 Evaluation

23.4.1 Specific sliding

The graphic shows the specific sliding of the gears (ratio between sliding and tangential speed) over the angle of rotation. Different values can be represented for each gear: Gears without backlash, gears with an upper center distance allowance (for lower tooth thickness tolerance) and gears with a lower center distance allowance (for upper tooth thickness tolerance).
The graphic is also available for crossed helical gears, in which case the specific sliding is calculated with the replacement cylindrical gear geometry.

When you specify the profile shift (see chapter 15.1.8, Profile shift coefficient), click the Sizing button to see a suggested value for balanced specific sliding.

23.4.2 Contact temperature

The contact temperature is the local temperature on the tooth flank at the moment of contact. It is displayed over the meshing. Based on the contact temperature values and its locations on the flank, appropriate action (e.g. profile modification) can be taken to reduce the temperature if necessary.

23.4.3 Flash temperature

The flash temperature is the increase in local temperature on the tooth flank at the moment of contact. It is displayed over the meshing. Depending on the values used for the flash temperature, and its position on the flank, a number of measures (e.g. profile modification) can be implemented to reduce the temperature.

23.4.4 Surface layer shear stress

This calculates the optimum hardening depth (for case hardened or nitrided gears). It shows the vertical shear stress progression in the depth, relative to the flank surface. This value is displayed directly in the HV values, because HV or HRC values are always used when specifying hardening depth and hardening measurements. If the materials database already contains values for a measured hardness curve, the hardening progression is displayed, accompanied by a warning message if the hardening properties are insufficient.

Proposed values for the recommended hardening depth are displayed in a special report, classified by calculation method, selected material and heat treatment process.

The various different methods are:

- The shearing stress progression in the depth of the gear pair is calculated according to Hertzian law. The shear stress is multiplied by a safety factor. (Enter this under "Settings". The default setting is 1.63). This defines the depth of the maximum shear stress (hmax). The program suggests the value 2*hmax as the hardening depth (EHT).
- For each individual gear in accordance with the proposals given in Niemann/Winter, Vol.II [7] (page 188).
- For each individual gear in accordance with the proposals given in AGMA 2101-D04 [57] (pages 32-34).
- For each individual gear according to the proposals given in ISO 6336 Part 5 [19] (pages 21-23) (to avoid pitting and breaking up of the hard surface layer).

23.4.5 Proposals for hardening depth

Suggestions for hardening depths according to ISO 6336, Niemann, AGMA 2001 and Linke [24] are displayed for different hardening processes.

23.4.6 Theoretical contact stiffness

The graphical display shows the contact stiffness depending on the angle of rotation. Contact stiffness is calculated on the basis of the real tooth forms. The calculation takes into account tooth deformation, gear body deformation, and flattening due to Hertzian pressure. The calculation is performed according to Weber/Banaschek [21].

For helical toothed gears, the overall stiffness is calculated with the section model (the facewidth is split into 100 sections and stiffness is added over all sections). See also [24], page 203. The transmission error is determined according to [7]. The transmission variation in the peripheral direction Δs equals:

$\Delta s = \left(\frac{F_{bt}}{b}\right) \cdot \left(\frac{q}{c'}\right)$	(22.5)
$F_{bt} = \frac{F_t}{\cos(\alpha_t)}$	(22.6)

where (q/c') is replaced by c_{gam}.

► **Note:**

The theoretical contact stiffness and the contact stiffness of the effective toothing under load can be quite different.

23.4.7 S-N curves (Woehler lines)

The graphic displays the S-N curves (Woehler lines) for the tooth root and flank. The S-N curves (Woehler lines) are calculated using the selected calculation method for gears. The individual S-N curves are divided by an appropriate safety factor. The individual load spectra are also displayed in the same graphic.

If a load spectrum is taken into account when calculating the gears, the graphic also shows the curve for damage accumulation (not available for plastics).

23.4.8 Safety factor curves

The graphic shows the progression of the safety factors over the service life. The safety factors are displayed for nominal operating conditions (i.e. without a load spectrum).

23.4.9 Oil viscosity

The graphic shows kinematic viscosity at different oil temperatures.

23.4.10 Reliability

The graphic shows the reliability curves for the gears and the configuration.

23.4.11 Gaping

The graphic shows a gap between the tooth flanks (in the direction of the path of contact) across the width of the meshing gears.

23.4.12 Face load distribution

The graphic shows a line load across the width of the meshing gears.

23.4.13 Backlash with effective tooth form

The evaluation graphic includes a circumferential backlash (in °) across the angle of rotation. The display shows the minimum, average and maximum backlash progressions. The actual tooth form, including flank line and profile modifications, is used in this calculation. The crossed helical gears calculation only covers worm wheels with an axial crossing angle of 90°.

23.4.14 Tooth flank fracture

The graphic shows the material utilization, the material's shearing strength, the equivalent shear stress and the hardness curve for the selected gear.

23.4.15 Sliding velocity (face gear)

The face gear sliding velocity graphic shows the sliding velocity for the tip and root of the face gear.

23.4.16 Contact line (face gear)

The "Contact line (face gear)" graphic shows the progression of the contact lines on the pinion and on the face gear.

23.4.17 Stress curve (face gear)

The graphic shows the stress curve (tooth root and flank) over the face gear's facewidth. The calculation splits the facewidth into individual segments, which can then be sized as rack pairs either according to ISO 6336, DIN 3990 or AGMA 2001. The calculation assumes a constant line load (which results in a slightly different torque for each segment due to the different pitch circle).

When you calculate data to represent the contact line and the stress curve, the most important values are calculated in separate sections and saved to two separate tables. This data is stored in the Z60-H1.TMP and Z60-H2.TMP files.

23.4.18 Scuffing and sliding velocity (face gear)

The graphic displays the safety against scuffing for face gears. However, due to the very different sliding velocities and the changing contact stress across the tooth flank, calculating the safety against scuffing is actually very difficult. Akahori [58] reports massive problems with scuffing at a higher sliding velocity. For this reason, it is appropriate to think about how to calculate the risk of scuffing. One sensible option, as described above for stress distribution, is to calculate the safety against scuffing in separate sections.

The graphic shows the progression of scuffing safety as defined by the flash and integral temperature criterion along the tooth flank. To achieve realistic results from this calculation, it must be ensured that every section is calculated with the same mass temperature. However, when you work through the calculation, you will see there are significant changes in safety when the calculation is performed on the basis of the integral temperature. In particular, this happens as point E on the path of contact gets closer to the pitch point. If you then use the formulae in DIN 3990 to convert the flank temperature at point E to the average flank temperature, the results you get will not be particularly precise. For this reason, we recommend you use the flash temperature as the criterion when you perform this calculation for face gears.

23.5 Contact analysis

► **Notes:**

The usual strength and speed calculations performed on gears assume that an involute tooth form is being used. However, if you use this program module, you can calculate and evaluate any type of gear teeth, such as cycloid toothing, just as accurately as involute tooth forms.

All the graphics can be exported:

- 2D diagrams as:
  - BMP
  - JPG
  - PNG
  - DXF
  - IGES
  - TXT
- 2D curves as:
  - TXT
- 3D diagrams as:
  - BMP
  - JPG
  - PNG
  - DAT (the y-axis is only output for the contact analysis if the "Draw data for path of contact" option is selected in the module-specific settings)

23.5.1 Excitation

23.5.1.1 Meshing

Displays the meshing of two gears, with length of path of contact under load. It also shows the efficiency progression, the standardized speed and the standardized torque along the meshing.

23.5.1.2 Transmission error

The path of contact under load is used to calculate the transmission error. The diagram shows the displacement of the contact point (μ) of the second gear on the length of the path of contact or the angle of rotation (°) of the driven gear.

The amplitude of the transmission error plays a role in how much noise is generated but, despite this, you should not ignore the pitch (how steep the slopes are), because high acceleration also generates high additional loads.

23.5.1.3 Transmission error acceleration

The Transmission error acceleration (second derivative with reference to time) is available as a graphic.

23.5.1.4 Amplitude of transmission error

This graphic displays the spectroscopic analysis results for the transmission error by Fourier transformation.

You can select the number of orders in the amplitude spectrum (transmission error/contact stiffness) in the module-specific settings in the **Face load factor/Contact analysis** tab.

23.5.1.5 Speed curve

This graphic shows the speed progression.

23.5.1.6 Excitation force

Force excitation (according to FVA Report 487) results from toothing stiffness and the average transmission error. In contrast to the process for calculating transmission error, calculating force excitation enables a better evaluation of how different toothing variants generate noise. This is because toothing forces, not the equalizing movement (transmission error) of the gears, are the decisive factor in generating noise.

23.5.1.7 Excitation force amplitude spectrum

The **Amplitude spectrum of the transmission error** graphic displays the spectroscopic analysis results for the excitation force after Fourier transformation.

23.5.1.8 Torque curve

The default value for torque defined in the main screen is kept constant during the calculation. The graphic then shows the torque for gear 1 and the torque for gear 2, divided by the ratio. The torque fluctuation results from the fluctuation in contact stiffness. The offset between the two moments forms the loss due to friction. The loss is dependent on the friction in the tooth contact.

Variations in the input torque depend on the level of accuracy you have specified, and are caused by the accuracy of the iteration.

23.5.1.9 Single contact stiffness

This graphic shows the individual elements of single tooth contact stiffness. These are the bending and tilting stiffness of the two gears and the Hertzian stiffness.

As this is a series-connected spring system, the following applies:

$$\frac{1}{C_{Pair}} = \frac{1}{C_{Gear1}} + \frac{1}{C_{Gear2}} + \frac{1}{C_{H1,2}}$$

23.5.1.10 Stiffness curve

The stiffness curve represents the system's torsional stiffness. It contains secant and torsional stiffness. The stiffness value for gears is usually specified per mm facewidth. To determine the stiffness of the meshing, the entered value must be multiplied by the bearing tooth facewidth.

23.5.1.11 Amplitude spectrum of the contact stiffness

This graphic displays the spectroscopic analysis results for the contact stiffness after Fourier transformation.

You can select the number of orders in the amplitude spectrum (transmission error/contact stiffness) in the module-specific settings in the **Face load factor/Contact analysis** tab.

23.5.1.12 Kinematics

The effective tooth form and the effective path of contact are used to calculate a wide range of kinematic values, which are then displayed along the length of the path of contact:

- specific sliding
- the sliding coefficients
- the standardized sliding velocity

23.5.2 efficiency

23.5.2.1 Total power loss

The power loss of the meshing is output. Power loss is usually greatest at the start and at the end of the mesh because this is where the highest sliding velocities are generated. However, profile modifications can be used to reduce the load at these points so that the maximum value is shifted to the area between the start or end of contact and the operating pitch point.

23.5.2.2 Specific power loss

This graphic shows the specific power loss for a tooth pair along the length of the path of contact. The power loss is output for one width unit. The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.2.3 Efficiency progression

This graphic shows the meshing efficiency progression in %.

23.5.2.4 Specific sliding

You can display specific sliding either alongside the meshing cycle, in **Kinematics**, or alongside the tooth height.

23.5.2.5 Heat development

Heat development links power loss with specific sliding. If the contact point on a gear moves slowly, it creates a higher heat value per length than if the contact point moves more quickly.

Greater heat development on the tooth flank should also correlate with the tendency to scuffing. However, this is not directly attributable to temperature.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.2.6 Heat development along the tooth flank

You can display specific sliding either alongside the meshing cycle, in **Heat development**, or alongside the tooth height.

23.5.2.7 Contact temperature

The effective local temperature shown in the graphic at each point in the path of contact is defined by the gear base temperature (the tooth bulk temperature) plus additional local warming (the flash temperature).

At each point on the path of contact, the calculation uses the following data from the contact analysis calculation to calculate the flash temperature on the tooth flank:

- Sliding velocity
- Speed in tangential direction to the pinion and gear
- Curvature radii on the tooth flanks
- Hertzian Pressure

The coefficient of friction μ is taken from the value input for calculating the path of contact. The bulk temperature is calculated as specified in ISO/TS 6336-22.

The flash temperature is calculated for:

- ISO according to ISO/TS 6336-22

$$\theta_{fl,Y} = \frac{\sqrt{\pi}}{2} \cdot \frac{\mu_m \cdot p_{dyn,Y} \cdot 10^6 \cdot |v_{g,Y}|}{\sqrt{B_{M1} \cdot w_{Y1}} + \sqrt{B_{M2} \cdot w_{Y2}}} \cdot \sqrt{8 \cdot \rho_{rel,Y} \cdot \frac{p_{H,Y}}{1000 \cdot E_r}}$$

- AGMA according to AGMA 925 with Equation 84

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.2.8 Lubricant film

The ISO/TS 6336-22 proposal contains a precise definition of the calculation used to determine the progression of the effective lubrication gap thickness h over the meshing. The lubrication gap can vary significantly depending on local sliding velocity, load and thermal conditions.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.2.9 Specific thickness of film

The ISO/TS 6336-22 proposal contains a precise definition of the calculation used to determine the progression of the effective specific lubrication gap thickness over the meshing. The lubrication gap can vary significantly depending on local sliding velocity, load and thermal conditions.

The location with the smallest specific lubrication gap thickness is the decisive factor in evaluating the risk of micropitting.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.3 Forces and stresses

23.5.3.1 Contact lines on tooth flank

You can examine the contact line along the facewidth in this graphic. All contact lines during one meshing position are shown simultaneously on a flank.

23.5.3.2 Contact pattern on tooth flank

In this 2D graphic, the contact pattern on the tooth flank is shown in color.

23.5.3.3 Load distribution in the operating pitch circle

This load distribution shows the normal force along the facewidth in the operating pitch circle.

23.5.3.4 Normal force curve (line load, length of path of contact)

The normal force curve shows the force per width for a tooth on the cylindrical gear for meshing under load. In a well arranged profile modification, the normal force should increase steadily from zero. Without profile modification, an introduction of the force before contact point A or after contact point E shows the input and exit impact.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.3.5 Normal force distribution on tooth (line load, length of path of contact)

This graphic shows the normal force curve along the tooth flank and facewidth on a 3D gear. You can set the color scaling thresholds in the graphic's properties.

23.5.3.6 Bearing force curve

Use the bearing configuration options in the **Define face load factor** dialog window to calculate the bearing force curve. In this case, both the bearing distance L and the distance s are used in the calculation. The value given for the face load factor calculation is used as the distance between the bearings. The purpose of this graphic is not to display the correct bearing forces, but to represent the variations in these forces.

Variations in the bearing forces cause vibrations in the shafts and changes in housing deformations.

23.5.3.7 Bearing force curve in %

This graphic shows the fluctuations in the bearing force curve in %. Friction and profile modifications lead to slight fluctuations.

23.5.3.8 Direction of the bearing forces

This graphic shows the direction of the bearing forces in degrees. Primarily, the force is in the direction of the pressure angle. Slight fluctuations can however occur due to variations in stiffness. Fluctuations also result if there are profile modifications, as the pressure angle changes in the profile modification area.

23.5.3.9 Stress curve (Hertzian pressure)

This graphic shows the progression of Hertzian pressure, which results from the bends at the particular contact points.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.3.10 Tooth root stress

This graphic shows two curves. Tooth root stress using the graphical method and tooth root stress at the 30° tangents.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.3.11 Tooth root stress across facewidth

The following tooth root stresses can be represented across the facewidth:

- Tooth root stress (graphical method)
- Tooth root stress (at 30° tangents)
- Tooth root stress (graphical method), analysis area
- Tooth root stress (at 30° tangents), analysis area

23.5.3.12 Bending stress in root area

This graphic shows the progression of this bending stress in the root area, for the maximum occurring tooth root stress during meshing.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.3.13 Stress distribution on tooth

This graphic shows the progression of the Hertzian pressure over the tooth flank and facewidth on a 3D gear. You can set the color scaling thresholds in the graphic's properties.

23.5.4 System contact analysis

23.5.4.1 Load distribution of planets

The deformations of the system mean that the torque is not distributed uniformly across all the planets. This graphic shows the load distribution on the individual planets.

23.5.4.2 Position of sun

In the contact analysis, the **Axis alignment** for the sun includes the **Floating** option. If this is active, the graphic shows the position of the sun's center point for the selected position of the first planet, during generation across a pitch.

23.5.4.3 Safety against scuffing

The calculation is performed as specified in ISO/TS 6336-20/21. The contact temperature determined in the contact analysis is used for the calculation.

23.5.4.4 Safety against micropitting

Calculation method

The calculation is performed according to ISO/TS 6336-22, Method A. All the required data is taken from the contact analysis.

Lubrication gap thickness h and specific lubricant film thickness λ_{GFP}

The ISO/TS 6336-22 proposal contains a detailed definition of the calculation used to determine the progression of the effective lubrication gap thickness h and the effective specific lubrication gap thickness λ_{GF} across the meshing. The lubrication gap can vary significantly depending on local sliding velocity, load and thermal conditions. The location with the smallest specific lubrication gap thickness is the decisive factor in evaluating the risk of micropitting.

Permitted specific lubricant film thickness λ_{GFP}

To evaluate the risk of micropitting (frosting), it is vital that you know how large the required smallest specific lubrication gap thickness λ_{GFmin} is to be. The calculation rule states that:

λ_{GFmin} >= λ_{GFP} must be set to prevent micropitting (frosting), or to ensure safety against frosting Sl = λ_{GFminP}/ λ_{GFP}.

If the lubricant's micropitting (frosting) load stage is known, the permitted specific lubricant film thickness is calculated from test rig data, according to ISO/TS 6336-22.

Otherwise, reference values for λ_{GFP} can be derived from the appropriate technical literature.

In [59] you will see a diagram that shows the permitted specific lubrication gap thickness λ_{GFP} for mineral oils, depending on oil viscosity and the micropitting (frosting) damage level SKS.

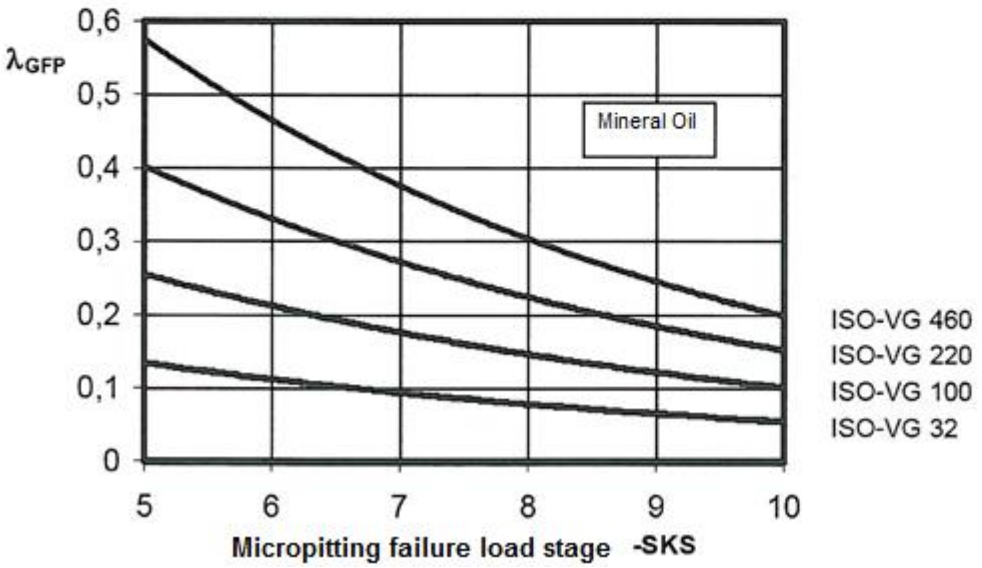


Figure 23.6: Minimum necessary specific thickness of lubricant film λGFP

The micropitting damage level SKS, determined in accordance with the FVA information sheet [60], is nowadays also stated in data sheets produced by various lubricant manufacturers. The data in the diagram applies to mineral oils. Synthetic oils with the same viscosity and frosting damage level show a lower permitted specific lubrication gap thickness λ_{GFP} [59]. Unfortunately, as no systematic research has been carried out on their effects, no properly qualified values are available.

Furthermore, note that the predefined λ_{GFP} values only apply to case-hardened materials. As specified in ISO/TS 6336-22, for other materials, the permitted specific lubrication gap thickness λ_{GFP} can be multiplied by the coefficient Ww.

	Ww
Case hardening steel, with high austenite content <= 25%	1.00
Case hardening steel, with high austenite content >= 25%	0.95
Gas-nitrided (HV > 850)	1.50
Induction or flame-hardened	0.65
Through hardening steel	0.50

Table 23.2: Material coefficient

It is interesting to note that, at least according to the table shown above, materials with a nitrite content are more prone to micropitting than case-hardened materials when the same lubrication gap is used. In contrast, through hardened materials that are not surface hardened are much more resistant.

You should be aware that the data shown here must be used with caution, because information about the micropitting process is still incomplete, and even technical publications will sometimes contain contradictory data.

Safety against micropitting

If the load stage against micropitting as defined in FVA C-GF/8.3/90 [60] is specified for the lubricant, the minimum required lubricant film thickness λ_{GFP} is calculated. This then makes it possible to define the safety against micropitting $S\lambda = \lambda_{GFmin} / \lambda_{GFP}$.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.4.5 Safety against micropitting at the tooth

This graphic shows the progression of the safety against micropitting over the tooth flank and facewidth on a 3D gear. You can set the color scaling thresholds in the graphic's properties.

The graphic can be displayed in 2D or 3D view. You can select which section is to be displayed, in the 2D graphic's properties.

23.5.4.6 Wear along the tooth flank

Before you can calculate local wear on the tooth flank, you must first define the material's wear coefficient k_w. This coefficient can be measured using gear testing equipment or by implementing a simple test procedure (for example pin and disk test rig) to determine the approximate value. Investigations are currently being carried out to see how accurately the coefficient k_w, determined using a simplified measurement method, can be applied to gears. For exact forecasts, you will also need to determine the coefficient $k_w$for the material pairing. For example, POM paired with POM does not supply the same results as POM paired with steel.

Plastics

You can input the wear factor k_w in the polymer data file, for plastics, depending on the temperature (for example, Z014-100.DAT for POM). The data is input in 10^6 mm^3/Nm.

For example:

```
-- Data for wear (PROVISORY! From Dissertation R.Feulner, 2008) Dependency from Temperature unknown
-- in 10^-6 mm3/Nm
:TABLE FUNCTION KwearDry
      INPUT X ZahnTempFlanke TREAT LINEAR
DATA
    -20      23      40      90     120
   1.03    1.03    1.03    1.03    1.03
END
```

Steel

Plewe's investigations have revealed that a rough approximation of the wear coefficients for steel materials can be defined. See also the calculation of wear coefficients for steel (see chapter 15.1.12.2, Calculation of the wear coefficient kw for steel).

Calculation

Wear is calculated according to the following basic equation:

$$\delta_w = K_w \cdot P \cdot V \cdot T$$

(δw [mm], k_w [mm3/Nm], P: Pressure [N/mm2], V:Velocity [m/s], T:Time[s])

As modified to suit gear conditions, local wear results from:

$$\delta_{w\_i} = NL \cdot k_w \cdot w \cdot \zeta_{\_i}$$

(i = 1.2)

(δw_i [mm], k_w [mm3/Nm], NL: Number of load cycles, w: Line load [N/mm], ζ_i: specific sliding)

This equation also corresponds to the data in [61], Equation 6.1.

The calculation used to determine wear on the tooth flank uses the following data at each point of contact taken from the calculation of the path of contact:

- Specific sliding
- Line load

POM against steel (at 23°C), [61] gives a k_w value of 1.03 * 10-6 mm3/Nm. Against steel, it gives a k_w value of 3.7*10-6 mm3/Nm.

When you interpret the results, note that the increasing wear on the tooth flank changes local conditions (line load, sliding velocity) to some extent, and therefore also changes the increase in wear itself.

23.5.4.7 Wear progress along the tooth flank

When the iterative wear calculation is run, this graphic shows the flank consumed by wear for every iterative step.

23.5.4.8 Safety against tooth flank fracture

The calculation is performed as specified in ISO/TS 6336-4, Method A. The Hertzian pressure determined in the contact analysis is used for the calculation.

23.6 Gear pump

Different diagrams provide detailed documentation about the progressions of the characteristic values in a gear pump when it is generated. You will find detailed information about calculating the gear pump in (see chapter 15.12, Gear pump).

23.7 3D FEM

23.7.1 Maximum tooth root stress

The following tooth root stresses can be represented across the facewidth:

- Tooth root stress at the 30 and 60° tangent point (maximum principal stress)
- Tooth root stress at the 30 and 60° tangent point (Von Mises equivalent stress)
- Maximum tooth root stress (maximum principal stress)
- Maximum tooth root stress (Von Mises equivalent stress)

23.7.2 Load distribution

This graphic shows the load distribution across the width. It also shows the point at which application of force occurs. Here, the value 0 corresponds to the root diameter and the value 1 corresponds to the tip diameter.

23.8 3D export

Select **Graphics > 3D Export** to export the geometry of the gears you have just designed to a specified CAD system. The next section (see chapter 23.9, Settings) provides more detailed information about which CAD system you should use, and its interface.

► **Note:**

Before you call this function for the first time, make sure you are using a suitable CAD system. If you have specified a CAD program that has not yet been installed, you may cause a problem when you call this function.

23.9 Settings

Click on the **Graphic > Settings** menu option to configure the graphics and input the settings for video recording.

2D graphics

Option: Clip curves to diagram range

If you do not select this setting, the entire curve progression is displayed, even if it exceeds the current axis range.

3D graphics

Options: Background, CAD system (you can select the interfaces for which licenses have been purchased) and Projection.

Video recordings

Video codec:

- H.264 - a widely used format which is supported almost everywhere.
- H.265 - has better optimization and supports a resolution of up to 8k (UHD 8192×4320). However, this can lead to problems of compatibility.

Mode:

- Constant Bit Rate (CBR) - the constant bit rate is not adjusted dynamically. This can result in a loss of quality if there is too much movement in the image.
- Unconstrained Bit Rate - the video quality depends on the average minimum bit rate. Image sequences involving a lot of movement are recorded with a correspondingly higher bit rate.
- Constrained Bit Rate - the quality of the video depends on the average bit rate and the maximum bit rate. The average bit rate of the entire video must not fall below the specified value. However, it may happen that a particular sequence maxes out the maximum bit. For that reason, the encoder must encode other sequences at less than the average rate.
- Constant Rate Factor (CRF): if this is set, the constant bit rate is based on the value input as the **Constant Rate Factor**.

Frame rate: the number of images per second.

Fixed width: if you input a fixed width, the video is recorded with this width (in pixels). If you do not input a value here, the video is generated at the current size.

Fixed height: if you input a fixed height, the video is recorded at this height (in pixels). If you do not input a value here, the video is generated at the current size.

Constant Rate Factor: a value between 0-100 which defines the file's band width and size. The default value is 24.

Average bit rate: approximate average data rate in bits per second.

Maximum bit rate: maximum data rate in bits per second.

24 Answers to Frequently Asked Questions

24.1 Answers concerning geometry calculation

24.1.1 Precision engineering

KISSsoft is an ideal tool for calculating the gears for precision engineering.

The reference profile and the geometry are calculated as defined in DIN 54800 etc. The strength calculation is performed according to ISO 6336, DIN 3990, VDI 2545 or VDI 2736, since no special strength calculation is available for precision gears. For this reason, "defining required safeties for gear calculation" (see chapter 24.2.4, Required safeties for cylindrical gear units) is important when you are interpreting the results.

If gears are manufactured using topping tools, the tip circle can be used to measure the tooth thickness. In this situation, it is critical that you specify precise value of the addendum in the reference profile to match the relevant cutter or tool. This is because this value is used to calculate the tip circle. The tip alteration $k*m_n$ is not taken into account in the calculation of the manufactured tip circle. The following formula is used:

$d_{aeff} = d + 2 \cdot m_n \cdot x_{eff} + 2 \cdot m_n \cdot h_{aP}$	(23.1)

24.1.2 Deep tooth forms or cylindrical gears with a high transverse contact ratio

Using deep tooth forms is recommended for some specific applications (for example, for spur gears that should not generate a lot of noise).

In KISSsoft, you can easily calculate all aspects of deep toothed gears. To calculate the geometry, you must select a profile of a suitable height when you select the reference profile:

Normal profile height: for example m_n * (1.25 + 1.0)
For a deep tooth form: for example m_n * (1.45 + 1.25)

You must be aware that this type of gear is more prone to errors such as undercut or pointed teeth. Experience has shown that you must select a value of 20 or higher as the number of pinion teeth to ensure that you can create a functionally reliable pair of gears. KISSsoft also has very effective and easy to use strength calculation functionality; as specified in DIN 3990, Part 3, calculation of gears with transverse contact ratios greater than 2.0 tends to be on the conservative side.

The Geometry Variants calculation (Modules Z04 and Z04a) is very good at sizing optimum arrangements of deep toothed gear pairs!

See also section 14.16.

24.1.3 Pairing an external gear to an inside gear that has a slightly different number of teeth

When you pair a pinion (for example, with 39 teeth) with an internal gear (for example, with 40 teeth) that has a slightly different number of teeth, the teeth may collide ("topping") outside the meshing area. This effect is checked and an error message is displayed if it occurs.

To size a functioning pairing of this type, select this strategy:

- Reference profile: Short cut toothing
- Pressure angle: the bigger the better
- Total profile shift: select a negative value
- Pinion profile shift coefficient: between 0.4 and 0.7

24.1.4 Undercut or insufficient effective involute

An insufficiently effective involute occurs if the tip of the other gear in the pair meshes so deeply with the root of the first gear that it reaches a point at which the involute has already passed into the root rounding. These areas are subject to greater wear and tear. Some gear calculation programs do not check this effect and suffer recurrent problems as a consequence.

To keep a close eye on the undercut and effective involute, you should always work with the Calculate form circle from tooth form (see chapter 15.22.3.1, Calculate form diameters from tooth form) option. This function checks the tooth form every time a calculation is performed. It determines any undercut it discovers and takes it into account in the calculation.
(The tooth form calculation takes into account all aspects of the manufacturing process. In contrast, the geometry calculation according to DIN 3960 uses simplified assumptions.)

24.1.5 Tooth thickness at tip

The tooth thickness at the tip circle is calculated for a zero clearance status. In addition, the maximum and minimum value is calculated using all tolerances.

When you check the tooth geometry, the tooth thickness at the tip must usually be at least 0.2 * module (according to DIN 3960). If this limit is not reached, KISSsoft displays the appropriate warning message. Select **Calculation> Settings > General** to change this factor if required.

24.1.6 Special toothing

The term "special toothing" is used to describe toothing with non-involute flanks. The reference profile (or the normal section through the hobbing cutter or rack-shaped cutter) of special toothing is not straight (unlike involute toothing). However, the same generating process is used to manufacture both toothing types. As part of the tooth form calculation, special toothing (cycloid, circular arc teeth) can either be imported from CAD or defined directly. In addition, you can then generate a suitable counter gear by clicking **Generate with the other gear in the pair** in the **Tooth form** tab.

By simulating the generation process, the tooth form and, from this, the geometry can then be defined for special toothing. There are no standards or technical documentation for strength calculations. The calculation for such tooth forms can be derived from the cylindrical gear method. (see chapter 23.5, Contact analysis)

24.1.7 Calculating cylindrical gears manufactured using tools specified in DIN 3972

Profile I and II are profiles for final machining. Simply select the tool you require from the selection list (Reference profiles).

Profiles III and IV are used for tools used in pre-machining. You should always use a finished contour to calculate a gear's strength. These profiles should therefore only be used as a pre-machining cutter.

The reference profiles are dependent on the module, as defined in the following formulae

Profile III	h_{fP} = 1.25 + 0.25 m_n-2/3	h_{aP} = 1.0	ϱ_{fP} = 0.2
Profile IV	h_{fP} = 1.25 + 0.60 m_n-2/3	h_{aP} = 1.0	ϱ_{fP} = 0.2

In the **Reference profile** tab, if the configuration is set to Tool: **Hobbing cutter**, you can select profiles III or IV according to DIN 3972 as the data source. Note that the selection of the cutter depends on the module and pressure angle. If the module or the pressure angle is changed, the cutter must be selected again.

Use the recommendations in the standard to select the correct allowances for pre-machining:

Profile III	Allowance = +0.5 m_n1/3 tan(α_n)
Profile IV	Allowance = +1.2 m_n1/3 tan(α_n)

If Pre-machining has been selected (in the Reference profile tab), you can set the appropriate Grinding allowance for Profile III or IV in the list in the Grinding allowance field.

Click on the Plus button next to Final machining q to input a tolerance interval qTol (=qmax-qmin). The grinding allowance for pre-machining then lies in the range qmin ... qmax, where qmin = q - qTol/2. qmax = q + qTol/2 applies.

The control measurements (base tangent length etc.) for pre-machining are then calculated with the following allowances:

Maximum size with As.e + qmin*2/cos(α_n)

Minimum size with As.i + qmax*2/cos(α_n)

Note:

If you want customer-specific tolerances to be processed automatically, you can define them in a file called "GrindingTolerance.dat". The \dat directory has an example of this type of file, which is called "GrindingToleranceExemple.DAT". When this file is renamed to "GrindingTolerance.dat", its tolerance values are used in the calculation.

24.1.8 Composites as defined in DIN 58405

DIN 58405 specifies the base tangent length allowances and permitted composite errors for gear teeth used in precision engineering. In this case, the reference profile specified in DIN 58400 assumes a pressure angle of α_n=20°. If you use a working transverse pressure angle that is not 20°, DIN 58405 Sheet 3, sections 1.2.10 and 1.2.11, state that the permitted composite error and the permitted rolling deviations must be multiplied with a coefficient L = tan(20°)/tan(abs). This is because the base tangent length allowances are standardized and the center distance error increases as the pressure angle is reduced. KISSsoft takes coefficient L into account when calculating tolerances to comply with DIN 58405, because it is specified in the standard.

However, the tolerances specified in ISO 1328 and DIN 3961 do not include this coefficient because it is not listed in the standard.

24.1.9 Automatic change of reference profiles

A problem has been identified in several calculations. In the **Reference profiles** tab, the factors for the tool addendums and dedendums change automatically if the center distance is changed. Why?

This is because the **Maintain tip circle when changing profile shift** or **Maintain root circle when changing profile shift** checkbox has been selected in the **Module-specific settings**, in the **General** tab.
If you change the center distance, the profile shift coefficient or coefficients also change in the reference profile.

24.1.10 Non-identical (mirrored symmetry) tooth flanks

If the tooth flanks (left, right) are not identical, will this cause an error when the tooth contour is exported?

The tooth flanks used in the calculation (sizing) are identical.

The export function used in the system not only exports the involute but also the entire tooth form. This is an approximated curve.

You can use the export precision (permitted deviation ε) to define how closely you want to approximate the calculated tooth form.

In each case, an approximate curve with the specified level of accuracy is supplied for either half of the tooth or the whole tooth. You can only use mirror symmetry with approximation accuracy.

The smaller the selected deviation, the more detailed the curve.

24.1.11 Internal teeth - differences in the reference profile if you select different configurations

A gear pair with internal teeth has been calculated in the KISSsoft system. A pinion type cutter is then to be used to manufacture this internal gear. The tool is manufactured to suit particular customer requirements and is influenced by the particular tooth form which is used. This must reflect the reference profile geometry of the internal gear. How can you then determine the pinion type cutter geometry?

A gear's reference profile is the relevant rack profile. A regular hobbing cutter for an outside gear has this rack geometry, which makes it easy to define the hobbing cutter profile. However, you must reverse the gear profile to obtain the hobbing cutter profile (the gear reference profile tooth addendum becomes the hobbing cutter tooth dedendum, and so on).

If the manufacturing tool is a pinion type cutter, the situation changes due to the restricted number of teeth on the pinion type cutter. You can begin with the basic approach that the inverse gear reference profile matches that of the pinion type cutter. However, after that, you must modify the pinion type cutter addendum to achieve the necessary root diameter on the internal gear.

First of all, you must define the number of teeth on the pinion type cutter. The reference diameter of the pinion type cutter will already be predefined to some extent, depending on the type of machine tool used to manufacture the gear. This reference diameter must be greater than the diameter of the main shaft of the machine tool that is to be inserted in the pinion type cutter tool. However, if this diameter is too large, when compared with the size of the pinion type cutter, the shank diameter will be too small. This will cause powerful vibrations during the production process and result in a poor accuracy grade. The reference diameter is divided by the module to determine the number of teeth on the pinion cutter.

If you want to use the KISSsoft system to design the pinion type cutter geometry, you must first input the number of teeth on the pinion type cutter. You can start with 0.0 for the profile shift coefficient of the pinion type cutter. A pinion type cutter's profile shift changes as it is used. Every time the pinion type cutter is resharpened, the profile shift is reduced slightly. A new pinion type cutter usually has a positive profile shift (for example +0.2). A worn tool has a negative profile shift.

Be extremely careful when entering the data for a pinion type cutter. Is the required root form diameter dFf achieved? If not, you must reduce the tip fillet radius of the pinion type cutter. If that does not help, you must increase the addendum of the tool reference profile. However, this also changes the active root diameter.

The same problem can also happen with the tip form circle diameter dFa. It often happens that it is not possible to generate the entire involute part up to the tooth tip. In this situation, you must either increase the number of teeth on the pinion cutter tool or reduce the tip diameter of the gear.

If you develop a gear that is manufactured by a pinion type cutter, it is always critically important that you investigate the production process early on in the development process. This is because not every gear geometry can be created with this production process.

24.1.12 Effect of profile modifications

Profile modifications are a popular topic of discussion. Where should these modifications start, and what values should be used to make these modifications?

Linear tip relief is a type of profile modification. It has the following properties: Starting from a particular point, ever increasing amounts of material are removed from the involute toothing part up to the tip diameter.

When there is no load, there is no contact between the tooth flanks in this modified area. Contact only occurs when suitable load is present. This entire area is taken into account when calculating the length of contact to determine the transverse contact ratio ε_a. Shouldn't this be different?

If you use profile modifications, you "delete" the real involute. Why is this a good idea?

This is a complex problem that must be taken into consideration when you design profile modifications. The amount of material removed (tip relief C_a = the reduction in tooth thickness at the tip due to the profile modification) must be applied according to the tooth bending.

For example, if the tooth had infinite stiffness, and you ignored any of the possible effects of compensating for manufacturing errors, the profile modification would simply reduce the transverse contact ratio. This is basically correct for a gear that is subject to a lower load. However, you will usually need to size gears for optimum performance at the operating torque and the tooth deformation that then occurs.

If the tip relief C_a is properly sized, the profile modification then compensates for the tooth bending, so that the tooth contact across the entire tooth height is not compromised. In this case, the transverse contact ratio is not reduced. Here, when compared to a gear without profile modification, you have a modified normal force curve over the meshing.

However, the maximum force (in the operating pitch circle), where only one gear pair is meshing, is not changed. For this reason, the maximum root and flank strains, which determine the service life of the gear unit, remain unchanged. This profile modification reduces the normal force at the start and end of the meshing. This also leads to a significant reduction in the risk of scuffing. The risk of scuffing is due to contact stress and sliding velocity. Sliding is greatest at the start and end of the tooth contact, so, by reducing the contact stress in this area, you can also reduce the risk of scuffing. A profile modification can reduce the influence of tooth strain on stiffness fluctuations across the meshing, and therefore limit the number of transmission errors. This also lowers the levels of vibration and noise.

Therefore, if a profile modification has been properly sized for the gearbox's operating torque, the transverse contact ratio is not reduced. However, when lower loads are involved, gears whose profile has been modified do not mesh as well as gears without profile modification. This is because the transverse contact ratio has been significantly reduced.

24.1.13 Number of teeth with common multiples

A gearing with 15:55 teeth has been sized. Different documents state that you should avoid gear reductions (such as 11:22) that are whole numbers. You will also discover that you should also avoid using numbers of teeth that are common multiples (in this case the 5 in 3*5 to 11*5). Is that true, and is it displayed in KISSsoft?

Let's assume we have a gear which has a fault on one of its teeth. In a whole number reduction, this tooth will always come into contact with the same tooth in the counter gear. The error is then transmitted to the counter tooth. However, if the tooth with the fault comes into contact with a different counter tooth in every rotation, this error will be reduced as the gears wear in.

Nowadays, most gears are surface-hardened. Unlike weak gears, they hardly ever wear in. As a result, this problem is now less critical than it used to be, where it was important that whole number gear reductions (such as 11:22) were avoided even when hardened gears were used. In contrast, whole number toothing combinations with common multiples (such as 15:55) do not cause any issues for surface hardened gears.

In KISSsoft, you will find notes about whole number combinations with hunting multiples in both fine sizing and rough sizing under the keyword "hunting". If you see YES for "hunting" in the table, this means: no common multiple is present.

24.1.14 Allowances for racks

From Release 10/2003 onwards, allowances for racks are defined in conjunction with the paired gear.

This conforms to DIN 3961.

"The tolerances for the gear teeth on a rack should not be greater than the tolerances for its counter gear. If the manufacturer does not know the counter gear value, they can set the rack length to the same value as the counter gear circumference."

24.2 Answers to questions about strength calculation

24.2.1 Differences between different gear calculation programs

You will always discover differences in the results when you compare calculations performed with different gear calculation programs. Many of these differences are due to the different data entered. However, even if all the data entered is the same, you will still get different results.

One of the questions our users often ask is whether the results calculated by KISSsoft are correct.

The main calculation process used in the KISSsoft cylindrical gear calculation functions is based on DIN 3990, ISO 6336, and AGMA. It faithfully follows the procedure described in Method B. However, as DIN 3990 or ISO 6336 offer various different methods (B, C, D) and sub-methods, it is no surprise that different calculation programs produce slightly different results. Most programs do not perform calculations that consistently use Method B. Instead, they partially use Method C or even D, which are easier to program.

To give our users additional reassurance, we have therefore integrated the **FVA program** calculation variant in KISSsoft. This variant supplies exactly the same results as the ST+ FVA program that was developed by the Technical University in Munich, and which can be used as a reference program. The minor differences between KISSsoft's calculations according to DIN 3990 and the FVA programs are due to the slight (permissible) deviations of the FVA program from the standard default process defined in DIN 3990.

24.2.2 Difference between cylindrical gear calculation according to ISO 6336 or DIN 3990

The strength calculation method used in ISO 6336 is virtually the same as that defined in DIN 3990. The majority of the differences only affect minor details which have very little effect on the safeties calculated for tooth root, flank and scuffing.

The only significant difference occurs when calculating the service life factors (Z_{NT} and Y_{NT}). In the endurance limit range (according to DIN, depending on material type and calculation method 107 to 109 load cycles) this coefficient in ISO 6336 decreases from 1.0 to 0.85 at 1010 load cycles. Only with "optimum material treatment and experience" does the coefficient remain 1.0.

As a result, gears in the range of endurance limit supply much smaller safeties (15% lower) when calculated according to ISO 6336 for root and flank! In the case of optimum material treatment, or for the number of load cycles in the limited life range, the safeties are practically identical.

24.2.3 Calculation using Methods B or C (DIN 3990, 3991)

Cylindrical Gears:

Calculation using Method B or C is described in DIN 3990. Method B is much more detailed and is therefore the method we recommend. KISSsoft usually uses Method B. However, we do not consider Method B to be precise enough to calculate the form factors for internal teeth, which is why we recommend Method C.

Changing over to using Method C means that most of the calculation is performed according to Method B and the tooth form factors are only calculated as defined in Method C for the tooth root strength.

Note: The most precise way of calculating internal teeth is to take the exact tooth form into account (see "Tooth form factor using graphical method", section 14.3.16.3).

Bevel gears:

Tooth form factors are calculated according to Method C, taken from the standard.

24.2.4 Required safeties for cylindrical gear units

Defining the necessary safeties (for tooth root, flank, scuffing) for gears in a particular application, for example, in industry standard gear units, vehicles, presses etc., is a very important step in the gear calculation process.

The (DIN 3990 or ISO 6336) standards provide hardly any information about this. DIN 3990, Part 11 (Industrial Gear Units) has this data:

Minimum safety for root:	1.4
Minimum safety for flank:	1.0

AGMA 2001 does not specify minimum safeties. The AGMA 6006 guideline (for gear units in wind power installations) has a note that SFmin = 1.56 is specified for root safety for calculation according to ISO 6336. In contrast, SFmin = 1.0 is sufficient for calculations according to AGMA. This matches our findings that calculations performed according to AGMA give much lower root safeties. Therefore, we recommend a minimum safety of 1.4*1.0/1.56 = 0.90 for industrial gear units calculated according to AGMA.

Scuffing is calculated according to DIN 3990, Part 4:

Minimum safety for scuffing (integral temperature):	1.8
Minimum safety for scuffing (flash temperature):	2.0

The standards do not specify this value for precision engineering (module under 1.5). Despite this, according to empirical values, the required safeties are much smaller than for gears with a larger module (root 0.8; flank 0.6)! The reason for this: The formulae and methods used in strength calculation are all taken from tests with larger gears and only supply very conservative factors (values that err on the side of safety) for small modules.

Defining required safeties for gear calculation

You can use the simple method described here to obtain the required safeties:

1. Examine and define the basic settings of the calculation (e.g. application factor, lubricant, manufacturing quality, processing etc.).
2. Then apply the gear calculation method (without changing the basic settings unless you absolutely have to!) on a known set of gears. You should select gears that run reliably under operating conditions and also gears that have failed.
3. You can then use the resulting safeties calculated with these gear sets to define the point up to which minimum service reliability can be guaranteed.
4. You can then use these parameters to calculate the sizing of new gears. You can, of course, change these minimum safeties to reflect the results of your own tests and experience.

24.2.5 Insufficient safety against scuffing

You can increase the safety against scuffing by:

- Oil selection (higher viscosity at high temperatures)
- Tip relief (profile modification)
- Different distribution of the profile shift

The methods used to calculate the safety against scuffing (unlike those used to determine the tooth root and flank) are still a matter of controversy. For this reason, you should not pay too much attention to this, especially if the results for safety against scuffing at flash temperature and the integral temperature process are very different.

24.2.6 Material hardening factor (for strengthening an unhardened gear)

If you pair a hardened gear with an unhardened gear (e.g. pinion made of 17CrNiMo6 and gear made of 42CrMo4), you get the positive effect of increased load capability on the flank of the unhardened gear. This effect is taken into account by the material hardening factor (factor in the range 1.0 to 1.2). As stated in ISO 6336, the surface roughness of the hardened gear should be low (polished surface), otherwise the load capability will not increase. On the contrary, the tooth of the weaker gear may actually be ground off.

24.2.7 Defining the load stage scuffing (oil specification)

On a test rig, the torque on the test gear is increased gradually until scuffing occurs, as described in Niemann [7], page 166. This load stage is then entered in the oil specification parameters (Example: no scuffing at load stage 10. Scuffing at load stage 11. The load stage scuffing of the oil is therefore 11).

To calculate the scuffing load capacity, you must then enter this load stage (for the oil specification). In the example described above, this is the value 11 (according to Niemann [7], page 341). The safety against scuffing calculation determines the safety against scuffing with predefined safeties greater than 1.0. This creates a necessary reserve, because the gradual increase in torque used in the test only approximates the effective scuffing torque.

24.2.8 The effect of the face load factor KHß for the tooth trace deviation fma is due to a manufacturing error.

In the cylindrical gear calculation defined in ISO 6336, when calculating the face load factor $K_{Hß}$, a higher value was determined for the tooth trace deviation f_{ma} . This was due to a manufacturing error The value for $K_{Hß}$ does not change. Why then, does this value for $K_{Hß}$ not change if a higher value is used for f_{ma}?

Before you can calculate $K_{Hß}$, you must input the position of the contact pattern. If the contact pattern has been defined as "economical" or "optimum", $K_{Hß}$ is calculated in accordance with the formulae in ISO 6336 or DIN 3990. f_{ma} has no influence on the calculation of $K_{Hß}$ and is therefore ignored.

See formulae (53) or (55) in ISO 6336:2006.

The reason for this is that a well designed contact pattern can compensate for manufacturing variations and variations due to deformation. If a higher value for f_{ma} is to be used in the calculation, this means, in reality, that a good contact pattern can never be present. That is why, in this situation, you should select the contact pattern position "not verified or inappropriate" when calculating the face load factor.

24.2.9 Load spectrum with alternating torque

Load bins can also be entered with negative torques.

The problem:

until now, no calculation guidelines have been drawn up to describe how to calculate gears with alternating load spectra.

The only unambiguous case is when a change in alternating torque takes place during every cycle and in each bin of the load spectrum. At this point, a load change corresponds to exactly one double-load with +torque and then with -torque. This instance can be calculated correctly by entering the load spectrum of the +torques and the alternating bending factor Y_M for the tooth root. The flank is also calculated correctly, because the +torques always apply to the same flank.

If, in contrast, the drive runs forwards for a specific period of time and then runs backwards, the experts agree that the tooth root is not subjected purely to an alternating load (and possibly this is the only point at which an alternating load change takes place). However, discussions are still raging as to how this case can be evaluated mathematically. It is even more difficult to define how mixed load spectra with unequal +torques and -torques for the tooth root are to be handled. For this type of case, only the +torques are considered for the flank (with the prerequisite that the +torques are equal to, or greater than, the -torques).

Note about handling load spectra with reversing torque:

A load progression as represented in the figure below, where the tooth is subjected to a load a few times on the left flank, and then a few times on the right flank, can be converted into a load spectrum as shown below. This is represented in an example here.

Load progression (example):

- 13 loads with 100% of the nominal load (100 Nm) on the left flank, then
- 9 loads with 80% of the nominal load (80 Nm) on the right flank, etc.

This results in the following process:

- 11 load cycles with 100% load, positive torque, pulsating; then
- 1 load cycle with 100% load on the left and 80% load on the right; then
- 7 load cycles with 80% load, negative torque, pulsating; then
- 1 load cycle with 80% load on the right and 100% load on the left;

then repeated again from the start.

This can be represented as a load spectrum as follows:

Frequency	torque	Left flank load	Right flank load
11/20 = 0.55	100 Nm	100%	0%
7/20 = 0.35	80 Nm	0%	100%
2/20 = 0.10	100 Nm	100%	80%

Table 24.1: Load progression shown as a load spectrum

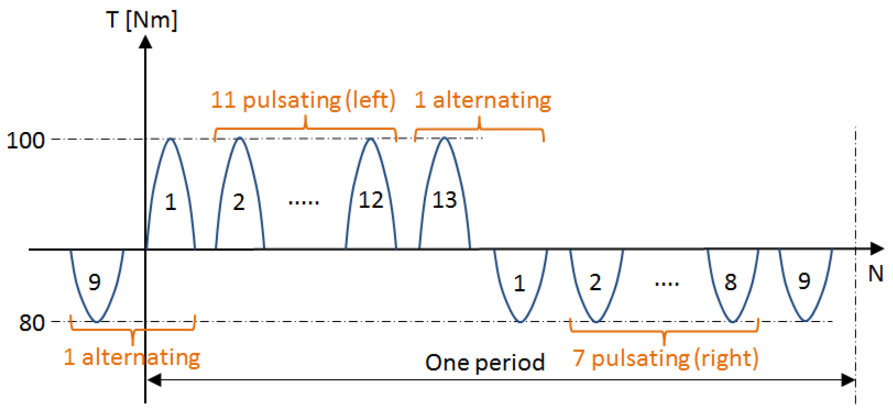


Figure 24.1: Load progression

24.2.10 Strength calculation with several meshings on one gear

How can several simultaneous meshing points on a motor pinion be taken into account in the calculation?

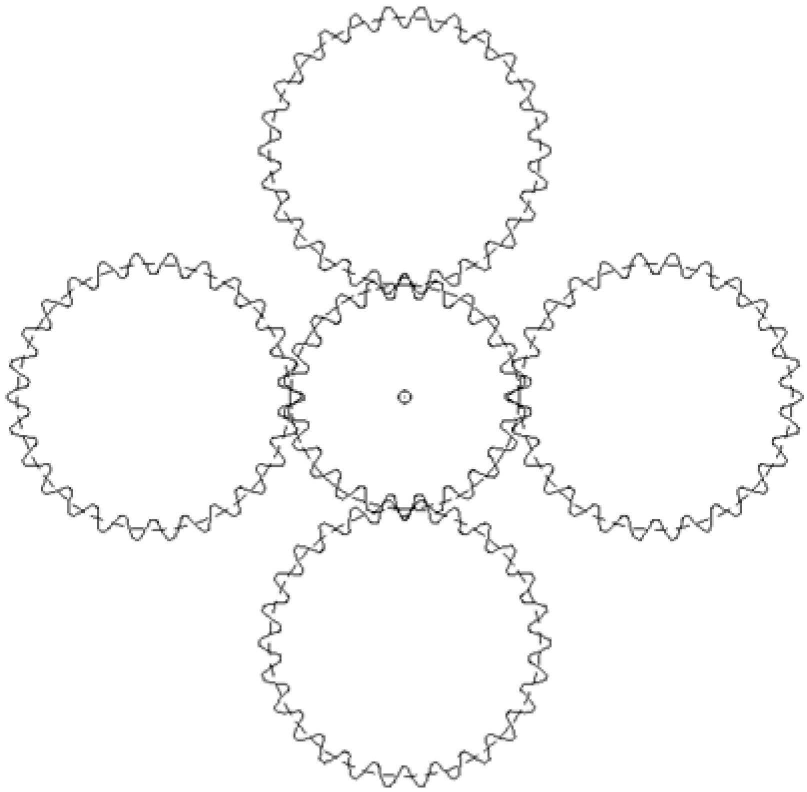


Figure 24.2: Fourfold meshing

You can solve this problem with a normal cylindrical gear pair calculation (Z12).

Simply divide the power by a factor of 4 (reduce by 25%)

To do this, click the Plus button beside the requested service life, in the **Load** tab, to change the number of load cycles for Gear 1 from "Automatically" to 4 "Load cycles per revolution".

24.2.11 Bevel gears: – Determine permitted overloads

Can maximum overloads be taken into account when calculating bevel gears according to ISO standards?

AGMA norms have definitions that allow for a standard overload of 250%. This overload is defined as being present for less than 1 second, not more than 4 times in an 8 hour time period. Does the ISO standard have comparable regulations with regard to overloads (shock)? No references could be found about this subject in the ISO standard.

ISO 10300 does not give any information about permitted overloads. However, ISO has a different Woehler curve (for YNT and ZNT factors) than AGMA. Therefore, in principle if ISO 10300 is strictly adhered to, you must input the total number of load cycle including the overload. The application factor is 2.5 (which corresponds to 250% overload). After this, you must calculate and check the safety factors.

If the load only occurs very infrequently, (less than 1000 times during the entire rating life), this can be handled in a static calculation. KISSsoft has a simplified version of the strength calculation process, specifically to cover this situation. This is based on the ISO method, but only takes into account the nominal stress in the tooth root (without stress correction factor YS). Here, please note that, in this case, you must maintain a minimum safety of 1.5 relative to the material's yield point!

24.2.12 Taking shot peening data into account when calculating gear strength

AGMA 2004-B89 states that shot peening improves the tooth root strength by 25%.

If you use KISSsoft to perform calculations according to DIN or ISO, you can achieve the increase in tooth root strength due to shot peening by inputting the relevant technology factor. To do this, open the **Factors** tab and click on "Z-Y factors..." in the **General factors** group.

You will find the details of useful entries as specified in Linke, Bureau Veritas/RINA or ISO 6336 in the manual. If you want to perform the calculation according to AGMA, you do not have the option of inputting the technology factor. In this case, you must increase the root endurance limit by specifying the relevant percentage rate directly when you enter the material data. To do this, open the **Basic data** tab and then click the Plus button after the material selection. In the dialog window that is then displayed, click on **Own input**. Set the infinite life strength in the calculation data for **Endurance limit root (AGMA 2001)**.

24.2.13 Calculation according to AGMA 421.06 (High Speed Gears)

In KISSsoft, you perform calculations as specified by AGMA 421.06 for high speed gear units in the following way.

AGMA 421 is an old standard (1968), and has long since been replaced by AGMA 6011-I03 (2003), (see chapter 15.2.1, Calculation methods).

24.2.14 Comparison of an FEM calculation with the crossed helical gear calculation

The accepted wisdom is that the differing results in the tooth root stress are primarily due to the lower value of the "Reference Facewidth" in the KISSSOFT calculation.

The effective contact of crossed helical gears is included in our calculation of the "Reference Facewidth". This results from the pressure ellipse (flattening of the point of contact). In addition, if sufficient facewidth is present, 1x module per facewidth is added to each side, as specified in ISO 6336-3.

24.2.15 Determine the equivalent torque (for load spectra)

Some calculation guidelines require you to determine the equivalent torque of a load spectrum and use it to perform sizing. How can I determine the equivalent torque in KISSsoft?

The fundamental issue here is that the verification of a toothing with equivalent torque must give the same safeties as the verification with the actual load spectrum. For this reason, you can follow this procedure:

1. Input the load spectrum and calculate the toothing.

2. Make a note of the lowest root safety and the lowest flank safety for each gear.

3. In the **Module specific settings** window, which you access by selecting **Calculation** > **Settings**, input the safeties you have noted as required safeties in the **Required safeties** tab. At this stage, we recommend you deselect the **Safeties depending on size** flag.

4. Delete the load spectrum by setting it to **Single stage load**.

5. Click the sizing button next to the input for the torque. The equivalent torque is written to the torque input field.

6. Now run the calculation to check the data. The safeties you have now defined for the root or flank of a particular gear must be exactly equal to the previous smallest value (as in step 2). None of the gears can have a safety that is less than the safeties you recorded in step 2.

24.2.16 Check changes in safeties if the center distance changes

Is it possible to check how the safeties change when gears are mounted with a different center distance?

Select **Calculation** > **Settings** > **Module specific settings** in the **Calculations** tab and then click on **Calculation with operating center distance and profile shift according to manufacture**. You can then input the profile shift coefficients and center distance independently of each other. The calculation then uses the circumferential forces in the operating pitch circle instead of the circumferential forces in the reference circle.

24.2.17 Warning: "Notch parameter qs …. outside RANGE (1.0 to 8.0) ..."

The stress correction factor Y_S Stress correction factor YS is calculated with a formula that complies with ISO 6336, Part 3 or DIN 3990, Part 3. This formula uses a notch parameter q_s , which is also documented in these standards.

$q_s = \frac{S_{Fn}}{\frac{r_{OF}}{2}}$	(23.4)

The validity range for the formula for Y_S in accordance with the standard lies in the range 1.0 ...q_s... 8.0. This formula should not be used outside of this range.

If $q_s < 1$, Y_S(calculated with q_s=1), should be rather too large. In this case, the calculation results will fall in the validity area.

If $q_s > 8$, then Y_S will be rather larger (than calculated with q_s= 8). In this case, the calculation results fall into the invalid range, and Y_s is therefore calculated with the effective $q_s$value (>8).

In each case, if q_s exceeds, or falls below, the range 1 to 8, a warning is entered in the report. This report also shows which $q_s$value was used further on in the calculation.

► **Note:**

If you want to change the procedure described here, you can do this either in the setup (STANDARD.Z12 file, etc.) or in a saved file (.Z12, etc.). To do this, open the file in Notepad and change this line:
ZS.qsLIMIT=0;
to: ZS.qsLIMIT=1; (q_s is not changed)
or to ZS.qsLIMIT=2; (q_s<1 is set to q_s=1, q_s>8 is set to q_s=8).

24.2.18 Tooth root stresses in the contact analysis and stress according to FEM – is there a difference?

An FEM-based approach is now available for calculating stresses in virtual toothing (2D). This is an additional option, which is generally good. However, this is only if you can trust the TCA stresses determined by this method – theoretically superfluous?

Calculating tooth root stresses as specified in the ISO or DIN standard only occurs in a cross section in the tooth root, at the point at which the tangents are exactly 30° to the root contour (60° for internal toothing). Investigations have shown that the maximum root stress occurs in this cross section, in standard gear teeth. There are also formulae which can be used to define the rounding radius and the area of the cross section. These values can then be used to determine bending stress. These formulae are based on the assumption that a standard tool is used in a generating procedure. Changes to the tooth contour, for example due to profile modifications, are ignored. The bending stress, consisting of nominal stress (YF coefficient) and the stress correction factor (YS) as specified in the standard, is determined on the basis of measurements taken on a few gears, and is therefore approximated. In special toothings, for example those with deep tooth forms, there may be a significant difference between the theoretical bending stress and the effective bending stress.

The calculation for helical gear teeth, as specified in the standard, is performed with virtualspur gears. The FEM calculation with virtual spur gears therefore uses the same approach as the standard, the only difference being that the exact tooth form is used in the FE calculation. The restriction to the cross section at the 30° tangent and also the formulae for YF and YS no longer apply in this case. The application of the load at the single tooth contact point is treated in the same way. This enables the exact difference between the stress calculated as specified in the standard and in FEM to be determined. As already described above, this is a particularly good approach for special gear teeth or gear teeth with substantial profile modifications in the root area.

The KISSsoft contact analysis (TCA) procedure determines load distribution across the facewidth and then uses this data to calculate the force applied at each individual tooth contact point in every segment across the facewidth. The formulae in the standard are then used to determine tooth root stress in the individual segments. However, KISSsoft's "graphical method" offers a considerable enhancement in functionality compared to the standard. The graphical method applies the stress calculation process using the standard's formulae to all the cross sections in the 30° tangent range (not just at the 30° point), and therefore calculates the cross section (diameter) at the point on the tooth at which the maximum tooth root stress is found (also using the formulae in the standard for the relevant cross section).

The tooth root stress in the TCA result is therefore more accurate than the one calculated using the standard. Despite this, the difference between the root stress according to FEM using the exact tooth form, or the stress calculated using the formulas for YF and YS according to the standard is not taken into account. The FEM calculation can therefore be used to investigate whether the root stress for a specific toothing is very different from the root stress calculated as specified in the standard. If it is, the stress determined using the TCA method can be multiplied by the coefficient (stress according to FEM/stress as specified in the standard) of the 2D FEM results in KISSsoft to achieve the most accurate results.

24.3 Abbreviations used in gear calculation

Abbr. in standards etc.	**Abbr. in KISSsoft**	
a	**a**	Center distance (mm)
a_d	**a.d**	Reference center distance (mm)
A_a	**A.a**	Center distance allowance (mm)
As_e	**As.e**	Tooth thickness allowance at the normal section (mm)
α_{en}	**alf.en**	Load direction angle (degree)

α_n	**alf.n**	Pressure angle at normal section (degree)
α_{Pro}	**alf.Pro**	Protuberance angle (degree)
α_t	**alf.t**	Pressure angle on the reference circle (degree)
α_{wt}	**alf.wt**	Working pressure angle (degree)
b	**b**	Facewidth (mm)
B_M	**B.M**	Thermal contact coefficient (N/mm/s.5/K)
β	**beta**	Helix angle at reference circle (degree)
β_b	**beta.b**	Base helix angle (degree)
c	**c**	Tip clearance (mm)
c'	**c'**	Singular tooth stiffness (N/(mm*µm))
c_γ	**c.g**	Mesh stiffness (N/(mm*µm))
d	**d**	Reference diameter (mm)
d_a	**d.a**	Tip diameter (mm)
d_b	**d.b**	Base diameter (mm)
d_f	**d.f**	Root diameter (mm)
$d_f(x_E)$	**d.f(x.E)**	Root circle with profile shift for A_{se} (mm)
d_i	**d.i**	Internal diameter gear body (mm)
d_{Na}	**d.Na**	Active tip diameter (mm)
d_{Nf}	**d.Nf**	Active root diameter (mm)
$d_{Ff(0)}$	**d.Ff(0)**	Root form circle (mm)
d_{sh}	**d.sh**	External diameter of integral pinion shaft (mm)
d_w	**d.w**	Operating pitch diameter (mm)
D_M	**D.M**	Theoretical ball/pin diameter (mm)
	D.M eff	Effective ball/pin diameter (mm)
e_{fn}	**e.fn**	Normal gap width on the root cylinder (mm)
η_{tot}	**eta.tot**	Total efficiency
ε_α	**eps.a**	Transverse contact ratio
ε_β	**eps.b**	Overlap ratio
ε_γ	**eps.g**	Total contact ratio
f_f	**f.f**	Profile form deviation (mm)
$f_{H\beta}$	**f.Hb**	Helix slope deviation (mm)

f_{ma}	**f.ma**	Tooth trace deviation due to manufacturing tolerances (mm)
f_{pe}	**f.pe**	Single pitch deviation (mm)
f_{sh}	**f.sh**	Tooth trace deviation due to deformation of the shafts (mm)
F_a	**F.a**	Axial force (N)
$F_{\beta y}$	**F.by**	Actual tooth trace deviation (mm)
F_n	**F.n**	Normal force (N)
F_r	**F.r**	Radial force (N)
F_t	**F.t**	Nominal circumferential force in the reference circle (N)
	Fase.d	Tip chamfer (mm)
g_α	**g.a**	Length of path of contact (mm)
Γ	**Gamma**	Gamma coordinates (point of highest temperature)
h	**h**	Tooth height (mm)
h_{aP}	**h.aP**	Addendum reference profile (in module)
h_F	**h.F**	Bending moment arm (mm)
h_{fP}	**h.fP**	Dedendum reference profile (in module)
h_k	**h.k**	Protuberance height (in module)
ha	**ha**	Chordal height (mm)
H	**H**	Service life in hours
I	**I**	AGMA: Geometry factor for pitting resistance
Impulse	**Impulse**	Gear driving (+) / driven (-)
j_n	**j.n**	Normal backlash (mm)
j_t	**j.t**	Rotational backlash (transverse section) (mm)
j_{tSys}	**j.tSys**	Torsional angle of the entire system (°)
k	**k**	Number of teeth spanned
$k * m_n$	**k * m.n**	Tip alteration (mm)
K_A	**K.A**	Application factor
$K_{B\alpha}$	**K.Ba**	Transverse load factor - scuffing
$K_{B\beta}$	**K.Bb**	Width factor - scuffing
$K_{B\gamma}$	**K.Bg**	Helical load factor - scuffing
K_f	**K.f**	AGMA: Stress correction factor
$K_{F\alpha}$	**K.Fa**	Transverse load factor - tooth root

$K_{F\beta}$	**K.Fb**	Width factor - tooth root
$K_{H\alpha}$	**K.Ha**	Transverse load factor - flank
$K_{H\beta}$	**K.Hb**	Width factor - flank
$K_{H\beta be}$	**K.Hbbe**	Bearing application factor
K_V	**K.V**	Dynamic factor
K_{wb}	**K.wb**	Alternating bending coefficient
k_w	**K.w**	Wear coefficient (mm3/Nm)
l	**l**	Bearing distance l on integral pinion shaft (mm)
m_n	**m.n**	Normal module (mm)
m_{Red}	**m.Red**	Reduced mass (kg/mm)
m_t	**m.t**	Transverse module (mm)
M_{dK}	**M.dK**	Diametral measurement over two balls without backlash (mm)
M_{dKeff}	**M.dKeff**	Effective diametral measurement over two balls (mm)
M_{dReff}	**M.dReff**	Effective diametral roller mass (mm)
M_{rK}	**M.rK**	Radial measurement over one ball without backlash (mm)
M_{rKeff}	**M.rKeff**	Effective radial measurement over one ball (mm)
μ_m	**mu.m**	Mean coefficient of friction (as defined in Niemann)
μ_m	**my.m**	Mean coefficient of friction
μ_m	**my.my**	Coefficient of friction
n	**n**	Speed (RpM)
ν_{E1}	**n.E1**	Resonance speed (min-1)
N	**N**	Resonance ratio
N_L	**N.L**	Number of load cycles (in millions)
ν_{100}	**nu.100**	Kinematic nominal viscosity of oil at 100 degrees (mm2/s)
ν_{40}	**nu.40**	Kinematic nominal viscosity of oil at 40 degrees (mm2/s)
p_{bt}	**p.bt**	Base circle pitch (mm)
p_{et}	**p.et**	Transverse pitch on path of contact (mm)
p_t	**p.t**	Pitch on reference circle (mm)
P	**P**	Nominal power (kW)
$P_{V\ Z}$	**P.VZ**	Gear power loss due to tooth load (kW)
$P_{V\ Ztot}$	**P.VZtot**	Total power loss (kW)

P_{WaelzL}	**P.WaelzL**	Meshing power (kW)
R_Z	**R.Z**	Average total height (mm)
ϱ_F	**ro.F**	Tooth root radius (mm)
ϱ_{fP}	**ro.fP**	Root radius reference profile (in module)
ϱ_{Oil}	**ro.Oil**	Specific Oil density at 15 degrees (kg/dm^3)
s	**s**	Distance of integral pinion shaft (mm)
s_{an}	**s.an**	Normal tooth thickness on the tip cylinder (mm)
s_{Fn}	**s.Fn**	Tooth root thickness (mm)
s_{mn}	**s.mn**	Normal tooth thickness chord, without backlash (mm)
	s.mn e/i	Effective normal tooth thickness chord with clearance (mm) (e: upper, i: lower)
S_B	**S.B**	Safety factor for scuffing (flash temperature)
S_F	**S.F**	Safety for tooth root stress
S_H	**S.H**	Safety for pressure at single tooth contact
S_{Hw}	**S.Hw**	Safety factor for contact stress on operating pitch circle
S_{Sint}	**S.Sint**	Safety factor for scuffing (integral temperature)
S_{SL}	**S.SL**	Safety for transmitted torque (integral temperature)
σ_F	**sig.F**	(Effective) tooth root stress (N/mm2)
σ_{F0}	**sig.F0**	Local tooth root stress (N/mm2)
σ_{Flim}	**sig.Flim**	Endurance limit tooth root stress (N/mm2)
σ_{FP}	**sig.FP**	Permissible tooth root stress (N/mm2)
σ_H	**sig.H**	Contact stress on operating pitch circle (N/mm2)
σ_{H0}	**sig.H0**	Nominal contact stress on the pitch circle (N/mm2)
$\sigma_{HB/D}$	**sig.HB/D**	Single tooth contact point contact stress (N/mm2)
σ_{Hlim}	**sig.Hlim**	Endurance limit Hertzian pressure (N/mm2)
σ_{HP}	**sig.HP**	Permissible contact stress (N/mm2)
σ_s	**sig.s**	Yield point (N/mm2)
$\Sigma\ x_i$	**Total x.i**	Total profile shift
T	**T**	Torque (Nm)
θ_B	**the.B**	Highest contact temperature (oC)
θ_{int}	**the.int**	Integral flank temperature (oC)
θ_m	**the.m**	Tooth bulk temperature (oC)

$\theta_{M\text{-}C}$	**the.M-C**	Tooth bulk temperature (oC)
θ_{Oil}	**the.Oil**	Oil temperature (oC)
θ_{s}	**the.s**	Scuffing temperature (oC)
θ_{Sint}	**the.Sint**	Scuffing integral temperature (oC)
u	**u**	Gear ratio
v	**v**	Circumferential speed reference circle (m/s)
v_{ga}	**v.ga**	Maximum sliding velocity on tip (m/s)
	Vqual	Accuracy grade
w	**w**	Nominal circumferential force reference circle per mm (N/mm)
W_{k}	**W.k**	Base tangent length (no backlash) (mm)
	W.k e/i	Effective base tangent length (mm) (e: upper, i: lower)
x	**x**	Profile shift coefficient
x_{E}	**x.E**	Profile shift coefficient at manufacturing for A_{se}
$X_{\alpha\beta}$	**X.alfbet**	Angle factor
X_{B}	**X.B**	Geometry factor
X_{BE}	**X.BE**	Geometry factor
X_{Ca}	**X.Ca**	Tip relief factor
X_{e}	**X.e**	Contact ratio factor
X_{Γ}	**X.Gam**	Mesh load factor
X_{M}	**X.M**	Flash factor
X_{Q}	**X.Q**	Meshing factor
X_{S}	**X.S**	Lubricant factor (scuffing)
X_{WrelT}	**X.WrelT**	Relative structural factor (scuffing)
y_{a}	**y.a**	Running-in value (μm)
y_{b}	**y.b**	Running-in value (μm)
Y	**Y**	AGMA: Tooth form factor
Y_{b}	**Y.b**	Helical load factor
Y_{drel}	**Y.drel**	Notch sensitivity factor
Y_{e}	**Y.e**	Contact ratio factor
Y_{F}	**Y.F**	Tooth form factor
Y_{NT}	**Y.NT**	Limited life coefficient

Y_R	**Y.R**	Surface factor
Y_S	**Y.S**	stress correction factor
Y_{st}	**Y.st**	Stress correction factor test gear
Y_X	**Y.X**	Size factor (tooth root)
z	**z**	Number of teeth
z_n	**z.n**	Virtual gear no. of teeth
Z_β	**Z.b**	Helix angle factor
$Z_{B/D}$	**Z.B/D**	Single contact point factor
Z_E	**Z.E**	Elasticity factor (N1/2/mm)
Z_ε	**Z.e**	Contact ratio factor
Z_H	**Z.H**	Zone factor
Z_L	**Z.L**	Lubricant factor
Z_{NT}	**Z.NT**	Limited life coefficient
Z_R	**Z.R**	Roughness factor
Z_V	**Z.V**	Speed factor
Z_W	**Z.W**	Material hardening factor
Z_X	**Z.X**	Size factor (flank)
ζ_w	**zet.W**	Wear sliding coefficient according to Niemann
ζ_a	**zet.a**	Specific sliding at the tip
ζ_f	**zet.f**	Specific sliding at the root

IV

Shafts and Bearings

Chapter 25 - 31

25 Defining Shafts

This program consists of a base package and different expert add-ins. The following calculations are available:

- Deformation and force, torque and stress curves
- Eigenfrequencies (bending, torsion and axial movements)
- Buckling loads
- Static and fatigue strength
- Rolling Bearing Calculation
- Plain bearing calculation
- Necessary flank line modification for pinions

Base package

In this module, you can input geometry and material data, shaft designations, the drawing number, the support, boundary conditions and external forces and torques (simplified input for couplings, spur and bevel gears, worms, worm gears, belt pulleys etc.).

A shaft with the machine elements mounted on it (for example, gears or bearings) is defined in the graphical shaft editor.
The properties required to define a shaft in this editor are:

- Any dimensions (cylindrical and conical), rotationally symmetrical cross section, solid and hollow shafts, beams (H-, I- or L-profile etc.)
- Integrated drawing tool that enables simple modifications to be made to the shaft contour (diameter, lengths). You can edit these elements by clicking on them.
- Definition of notch geometries for the automatic calculation of notch factors.
  The following notch geometries are available here:
  - Radius
  - Chamfer
  - Relief groove
  - Interference fit
  - Longitudinal key way
  - Circumferential groove
  - Square groove
  - V-notch
  - Spline
  - Cross hole

- You can enter these values for force and torque in any spatial positions, however, the following values are already predefined:
  - Cylindrical gear
  - Bevel gear
  - Worm
  - Worm wheel
  - Coupling
  - Pulley/V-belt
  - Centrical force
  - Eccentric force
  - External masses with moment of inertia (additional mass)
  - Power loss
- Calculation of:
  - Shaft weight
  - Moment of inertia
  - Axial force
  - Static twisting of the shaft due to torsion
- Clear representation of geometry data and the calculated bearing and peripheral forces both on screen and on paper.

25.1 Input window

The KISSsoft system offers a range of different input windows in which you can calculate shafts. The Shaft Editor (see chapter 25.1.1, Shaft editor) displays the shaft system as a graphic. The Element Tree (see chapter 25.1.2, Element Tree) illustrates the structure of the shaft system in a tree structure. The Element editor (see chapter 25.1.4, Element Editor) is where you input parameters for an element.

25.1.1 Shaft editor

The Shaft editor displays the shaft system as a graphic. Enter elements in the Element box here. If your system has several shafts, the new element is always added to the active shaft. A shaft becomes active when one of its elements is selected. If no element has been selected, the last shaft is the active one.

You can select options in the context menu to save the graphic as a picture file, and print it, in the Shaft editor. Each of the different elements also has interactive context menus.

25.1.2 Element Tree

The Element Tree illustrates the structure of the shaft system in a tree structure. Shafts are at the highest level. The connecting elements in systems with several shafts are also shown here. Each shaft groups its main elements by Outer contour (see chapter 25.2.2, Outer contour), Inner contour (see chapter 25.2.3, Inner contour), by Strength (see chapter 25.2.2.1, Defining sub-elements), by Bearings (see chapter 25.2.5, Bearings) and by Cross sections (see chapter 25.2.7, Cross sections). The sub-elements are located beneath the main elements, cylinder and cone.

You can select, copy, insert and delete elements via the Element Tree. You can select options in a context menu to display the actions that are available for each element. Special actions are available, depending on the element type. You can also size shafts, rolling bearings and cross sections. You can also import/export outer and inner contours to DXF (see chapter 25.2.2.2, Importing the shaft geometry)/ (see chapter 25.2.2.3, Export shaft geometry).

25.1.3 Element List

The Element List contains groups of elements in table format. You can edit the parameter listed in the table directly in the Element List. Using the context menu, you can insert elements quickly and easily.

25.1.4 Element Editor

In the Element Editor, you can edit any of the selected element's parameters.

25.2 Element overview

25.2.1 The shaft element

To add a new shaft, click on the can of the appropriate icon in the Element box (see chapter 25.1.1, Shaft editor). You will also find the **Add shaft** option in the Element tree context menu (see chapter 25.1.2, Element Tree). A new entry is displayed at the end of the Element Tree. Click on the element you require in the Element Tree and enter parameters for the shaft in the Element editor (see chapter 25.1.4, Element Editor).

25.2.1.1 Drawing number

In the **Drawing number** input field, you can enter a string of any characters apart from ";" (semicolons). The drawing number you enter here does not affect the calculation.

25.2.1.2 Position

The **Position** input field is where you enter the **Y** coordinate of the starting point of the shaft with regard to the global coordinates system.

► **Note**

Global coordinates are indicated by upper case letters. Lower case letters indicate a shaft's local coordinates system.

25.2.1.3 Temperature

The shaft may undergo thermal expansion if the shaft's temperature is not the same as the reference temperature (see chapter 25.3.6, Reference temperature). In addition to the thermal expansion of the shaft, the thermal expansion of the gear case can also be taken into account by the housing temperature (see chapter 25.3.7, Housing temperature).

25.2.1.4 Ambient density

Bodies placed in hydrostatic fluids become buoyant. The value here is the same as the weight of the displaced medium, and is defined by the volume and the density of the displaced medium. KISSsoft takes this buoyancy effect into account if you enter the appropriate ambient density value. The default setting is for air density. The table below lists technical values for other media.

Medium	**Air**	**Water**	**Oil**
Density ϱ	1.2	998	772

Table 25.1: Densities [kg/m3] of a few important fluids where ϑ = 20oC and p = 1016 mbar

► **Note**

If a shaft is operated in different ambient media, as is the case for input shafts in ships, for example, you can combine two individual shafts, each of which has different ambient density data, by using the **Connections** element in the Element Tree, and then calculate them as a single shaft.

25.2.1.5 Speed

Shaft speed around its longitudinal axis [rpm]. If you click the checkbox to the right of the input field, you can change the speed independently of the other shafts. However, if this checkbox is not active, the value is taken from the Speed input field (see chapter 25.3.4, Speed) in the **Basic data** input window.

25.2.1.6 Direction of rotation

The sense of rotation can influence the way loads are distributed along the shaft, for example, as the result of helical gear teeth, and therefore affect the rating life of the bearing. Click the checkbox to the right of the **Speed** input field to display the drop-down entries and select the one you require. However, if this checkbox is not active, the value is taken from the Shaft rotation input field (see chapter 25.3.5, Direction of rotation) input field in the **Basic data** input window.

25.2.1.7 Material

You can select a shaft material from this selection list and therefore assign a specific material to each individual shaft. If you use this function together with the **Connections** element in the Element Tree, you can generate shafts made of different materials.

25.2.1.8 Base size

The **Raw dimension** input field is decisive for strength calculation. However, if you select the **Pre-machined to actual diameter** option in the **State during heat treatment** drop-down list, in the **Strength** input window, the setting for the raw measure value has no effect on the calculation. In contrast, if the selection is set to **Raw diameter**, the largest rounded shaft diameter is selected, and the strength calculation is performed with this value. Click the checkbox to the right of the input field to specify your own diameter for the blank before it is turned.

25.2.1.9 Hardening depth (FKM)

The hardening depth input field is required for estimating the infinite life strength of surface-treated parts. Hardening depth is used to define the position of the transition surface layer relative to the core. It varies depending on which surface treatment process has been used. This input is not required for the main calculation. You will find another description of this calculation in the "Estimate the fatigue strength of surface treated parts" section.

25.2.1.10 Surface factor

In this selection list, you can define if an additional surface factor should be applied. Here, you can select either **Rollers** or **Shot peening**.

25.2.1.11 State during heat treatment

To define the technological size coefficient $K_{1,\mathrm{deff}}$, select one of these three options:

- Pre-machined to actual diameter. The raw diameter has no influence on the technological size coefficient. The value $K_{1,deff}$ is recalculated for each cross section based on the actual diameter size.
- Raw diameter. $K_{1,deff}$ is determined once from the raw diameter and applied to all cross sections.
- Pre-turned to actual diameter (shoulders K1 from d)

► **Note**

You can also set a value in the **Base size** field in the **Element editor** for the relevant shaft. To do this, input the dimension of the raw material that was used to generate the final material properties during the last heat treatment. If this involves a solid shaft, enter the external diameter of the blank part. For a pipe, enter the wall thickness and, for a cast part, enter the greatest wall thickness.

25.2.1.12 Heat treatment of hollow shafts in the full state

If you do not input the raw diameter for a "heat treatment state", and you are investigating a hollow shaft (di > 0.1*da), you can use this option to specify

whether the coefficients are determined using the solid shaft or hollow shaft. This option is only valid for the FKM and DIN calculation methods.

Table 1.2.3 in the 7th Edition of the FKM Guideline shows data for both a hollow shaft and a solid shaft, and also methods for Case 1 and Case 2. Case 1 is for parts made of treated steel, case hardening steel etc., Case 2 is for parts made of unalloyed steel, normally annealed through hardening steel, etc. KISSsoft automatically calculates values for these 2 cases when you input the material.

According to DIN 743-2, the shaft diameter d_{eff} is used for factor K2 and K3. If a solid shaft is involved, the diameter is used for coefficient K1. If a hollow shaft is involved, the wall thickness s or 2x the wall thickness s is used for d_{eff}, according to FKM.

25.2.1.13 Material properties

Select an entry in the **Material properties** drop-down list to specify how KISSsoft is to define the material characteristic values that are relevant to strength:

1. **with reference diameter:** The R_p and R_m values are taken from the database (at reference diameter) and multiplied by K_1, and the fatigue limit σ_W is determined from the tensile strength R_m according to the standard.
2. **R_p, R_m as stated in the database, σ_W for reference diameter:** The R_p and R_m values are determined depending on size (excluding K_1), and the fatigue limit σ_W is determined at the reference diameter taken from the database and then it is multiplied by K_1.

3. ***R_p*, *R_m* as stated in the database, *σ_W* constant:** The R_p and R_m values are determined depending on size, and the fatigue limit σ_W is taken from the database without being influenced by the geometric size factor. The size factor K_1 is not taken into account here.
4. ***R_p*, *R_m* as stated in the database, *σ_W* calculated from *R_m***: The R_p and R_m values are taken from the database depending on size, and σ_W is determined from the tensile strength R_m according to the standard.

The material's data, used to calculate the shaft strength, is derived from the values in the database as follows:

- Fatigue limit factors (for tension/compression, bending, etc.) are taken directly from the material database. There, these values are defined for every calculation method. If data for these materials has been specified in the calculation method, it is these values that are used.
- Tensile strength values are stored in the database according to their diameter as defined in the specific EN standard. The raw diameter is used to fetch the tensile strength value from the database and use it in the calculation. This method of defining the effective tensile strength is very reliable and can be used for every calculation method. It has the effect that the same values are used for each calculation method.
- When you specify a calculation method, you can decide to use the material database on the basis of the requirements given in the relevant standard. Then, the real tensile strength is defined using the thickness factor taken from the base tensile strength of the sample diameter (normally 10 mm), according to the standard (which must be FKM or DIN. If you use Hänchen, this triggers an error message).
- The yield point or strain limits are taken either from the database or from the standard, in the same way as for the tensile strength.

25.2.1.14 Own data for S-N curve (Woehler line)

Click the **Own data S-N curve (Woehler line)** checkbox to define your own S-N curve (Woehler line). You can also enter values for the sustainable damage or Miner total here. If this option is not selected, the S-N curve (Woehler line) according to DIN 743 or FKM is used.

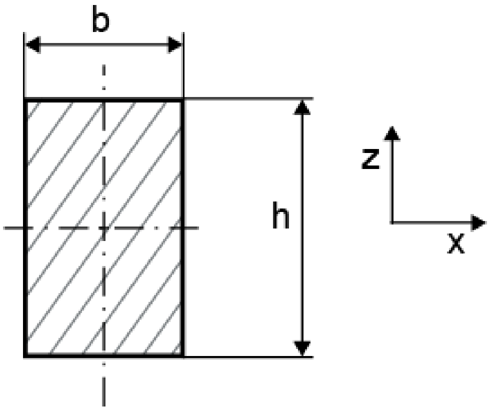
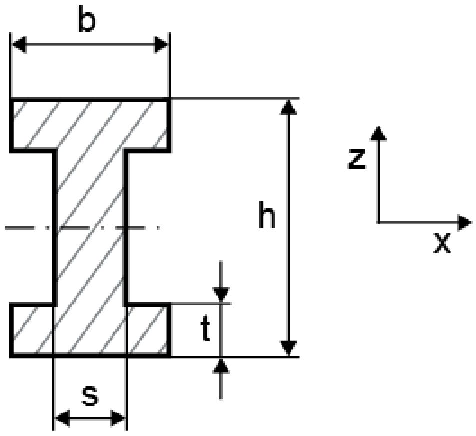
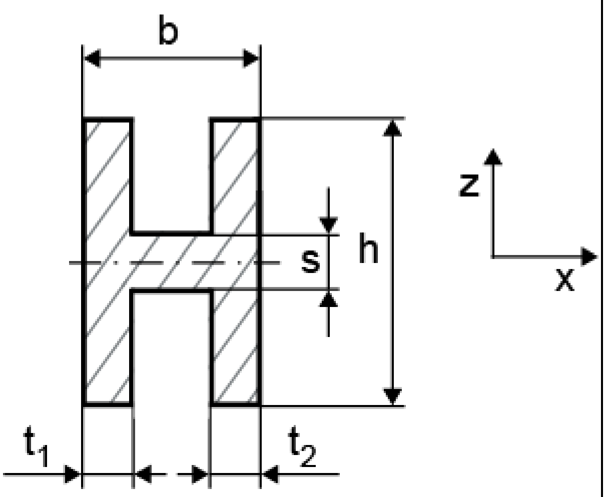
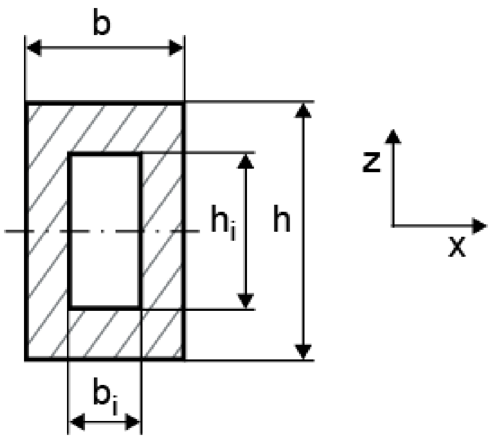
25.2.1.15 Taking the results into account in the report

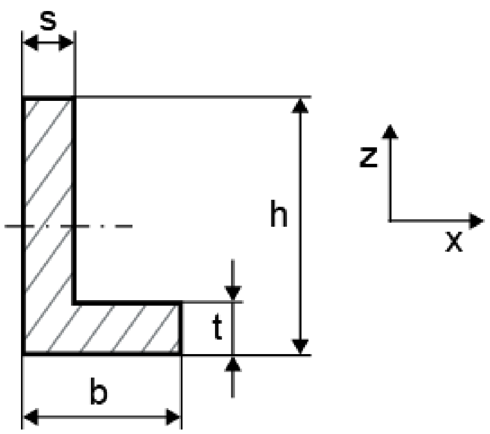
If this option is selected, the corresponding shaft is output in the main shaft report, along with all its elements (outer/inner contour, force elements, bearing). However, this is only valid for inputs and does not affect the results of the calculation.

25.2.2 Outer contour

You can use (hollow) cylinders, (hollow) cones and beams to define the shaft geometry. You can either use the Element box or select "Add" in the context menu to add elements to the Element Tree. If you have selected a contour element in the Shaft editor or Element Tree, select "Insert before" in the context menu to insert a new element to the left of it or select "Insert after" to insert a new element to the right of it. By default, the new element is inserted to the right.

Possible profiles for beams are:

<table>
<tr>
<td>
</td>
<td>
</td>
</tr>
<tr>
<td>Rectangular profile</td>
<td>Double T profile</td>
</tr>
<tr>
<td>
</td>
<td>
</td>
</tr>
<tr>
<td>H profile</td>
<td>Rectangular profile (hollow)</td>
</tr>
</table>

	
L profile	

25.2.2.1 Defining sub-elements

Before you can define a sub-element, first select the main element to which you want to add this sub-element in the Element Tree. Then, right-click to select the sub-element you require. The inserted sub-element is now displayed in the Shaft editor, and its associated notch factors are determined in the strength calculation.

Entering sub-elements:

- **Radius right/left**
  Input values:
  - Radius: Size of the radius
  - Surface roughness: Radius surface
- **Chamfer right/left**
  Input values:
  - Length: Chamfer length
  - Angle: Chamfer angle
- **Relief groove right/left**
  Input values:
  - Relief groove form: Select the relief groove form as defined in DIN 509 or the FKM Guideline.
  - Series (DIN 509): (Selection: Series 1, radii as defined in DIN 250. Series 2, special radius.)
  - Stress (DIN 509): (with conventional stress; with increased fatigue strength)
  - relief groove length: Length of the relief groove in axial direction

    - Transition radius: Radius between the end of the relief groove and the next element
    - Depth of recess: Recess depth
    - Surface roughness: Recess surface
- **Interference fit**
  Input values:
    - Length of Interference fit: Interference fit length
    - Type of interference fit: (Selection: Slight interference fit, interference fit and interference fit with end relief.)
    - Reference measure: this specifies the measurement from the left-hand end of the selected element up to the start of the interference fit
- **Key way**
  Input values:
    - Length: Key way length
    - Standard: Standards used for keys
    - Key way width: Width of the key way (can be entered if "Own Input" is selected)
    - Key way depth: Depth of the key way (can be entered if "Own Input" is selected)
    - Number of keys: (i > 2 not permitted according to standard)
    - Manufacturing process: (Selection: end milling cutter, side milling cutter, combined with interference fit (FKM))
    - Surface roughness: Keyway surface
    - Reference measure: this specifies the measurement from the left end of the selected element up to the start of the keyway
- **Circumferential groove**
  Input values:
    - Depth: Depth of the circumferential groove
    - Rounding in the groove bottom: Radius of the circumferential groove
    - Surface roughness: Surface of circumferential groove
    - Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the circumferential groove
- **Square groove**
  Input values:
    - Width: Width of the square groove
    - Depth: Depth of the square groove
    - Radius: Radius of the square groove
    - Surface roughness: Surface of the square groove

  - Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the square groove
- **V-notch**
  Input values:
  - Depth: Depth of the V-notch
  - Surface roughness: Surface of the V-notch
  - Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the V-notch
- **Spline**
  Input values:
  - Standard: Standard series of spline (click the Plus button to select the required size from a list)
  - Tip circle: you can either select this from a list of standards or input your own value
  - Root circle: you can either select this from a list of standards or input your own value
  - Number of teeth: you can either select this from a list of standards or input your own value
  - Module: you can either select this from a list of standards or input your own value
  - Surface quality: Spline surface quality
  - Length: Spline length
  - Reference measure: this specifies the measurement from the left end of the selected element up to the start of the spline
- **Spline shaft**
  Input values:
  - Tip circle: Tip circle of the spline shaft
  - Root circle: Root circle of the spline shaft
  - Number of keys: Number of keys
  - Straight-sided splines root rounding: (Selection: Shape A, Shape B and Shape C)
  - Length: Length of the spline shaft
  - Reference measure: this specifies the measurement from the left end of the selected element up to the start of the spline shaft (straight-sided spline)
  - Surface quality: Spline shaft surface
- **Cross hole**
  Input values:

        - Bore diameter: Diameter of bore
        - Surface roughness: Cross hole surface
        - Reference measure: this specifies the measurement from the left end of the selected element up to the position of the cross hole
- **Thread**
  Input values:
    - Label: Thread label
    - Thread depth: Thread depth
    - Rounding: Rounding in the notch bottom of the thread
    - Length: Thread length
    - Reference measure: this specifies the measurement from the left-hand end of the selected element up to the start of the thread
    - Surface roughness: Thread surface
- **General notch effect**
  Input values:
    - Width: Width of the overall sub-element
    - Notch factor bending/ torsion/tension-compression/shearing force: you can enter the notch factors directly here.
    - Surface roughness: Surface of the overall sub-element
    - Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the overall sub-element

You can enable the "Conical shoulder" notch type directly in the Strength calculation (see chapter 26.5.14, Cross-section types).

25.2.2.2 Importing the shaft geometry

Right-click on "outside" or "inner contour" in the Element Tree to open a context menu. Click **Import** to import a .ktx or a .dxf file.

Reading (importing) a ktx file:

In KISSsoft, go to the Shaft calculation Element Tree and right-click on the **Outer contour** element to display the context menu. Select the **Import** option in it. Select the required .ktx file and click on **Open**. The shaft contour is now imported into KISSsoft.

Reading (importing) a dxf file:

The outside and inner contour of the shaft (if present) should be output individually by the CAD system.

► **Note:**

You can use the default value **ALL** for the layer name, so that all layers are imported. You can also import the contours as variants in different layers. To do this, enter the layer name in the appropriate input field. If you don't know the exact layer designation, you can input an invalid name as a test (for example, xxx). If you then try to import this file, the resulting error message will list the valid layer names.

- The shaft contour is displayed with a centerline in a CAD system. Use the X-Y plane as the coordinates system (the X-axis is the rotational axis) to ensure the contour is interpreted correctly after it has been imported and so that the shaft is drawn in KISSsoft in the Y-Z plane (the Y-axis is the rotational axis). Save the shaft geometry as a .dxf file.
- In KISSsoft, go to the Shaft calculation Element Tree and right-click on the **Outer contour** element to display a context menu in which you select the **Import** option. Select the .dxf file you require, and click **Open**.
- This opens another dialog, in which you can define the layer, the point of origin (X/Y) and the angle of the symmetry axis. After you have input this data, click OK to close this dialog. The shaft contour is then imported with these details.

25.2.2.3 Export shaft geometry

Right-click on "outside" or "inner contour" in the Element Tree to open a context menu. If you select **Export**, you can create either a .ktx or .dxf file.

25.2.3 Inner contour

The inner contour is built up from left to the right, in the same way as the outer contour. For example, if you want to generate a shaft with an axial hole from the right-hand side, you must first input data for an inside cylinder starting from the left-hand side with a diameter of 0 that extends up to the point where the bore begins.

25.2.4 Forces

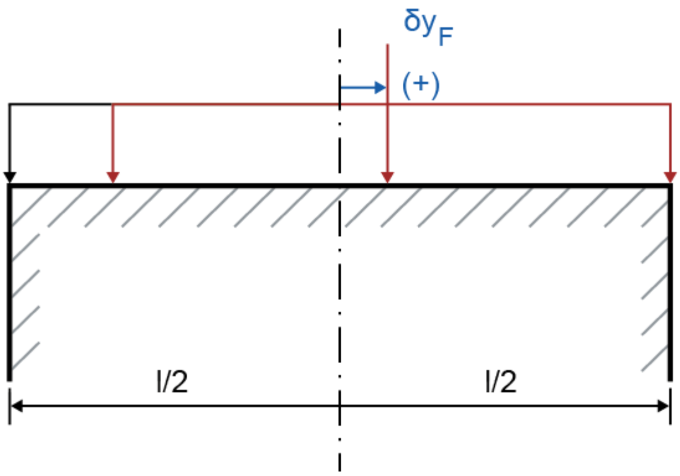
25.2.4.1 Forces

You can add forces to any place on the shaft, even outside of the shaft! Different methods are available for defining force-transmitting elements (such as gears) or even individual forces. In most force elements, the torque direction is defined by setting them as **driving** or **driven**. **Driving** means that the shaft is the driving element or that the moment is counter to the sense of rotation. See (see chapter 25.3.5, Direction of rotation).

Comments about some of the elements:

Cylindrical gear

Position of contact: If you enter the position of contact with the other gear, forces are applied at this point. Instead of simply entering the reference diameter, you get a more accurate result if you enter the operating pitch diameter and the operating pressure angle instead of the nominal pressure angle. Click the Convert button to calculate these values.
If the meshing type is set to "Multiple meshing", you can model several meshings on the same cylindrical gear element. You must then define the position, the active operating pitch diameter and the length of load application for each meshing. The resulting working pressure angle, and the helix angle, are then determined automatically from this data.
By default, the center point for load application is the center of the gear. This can be changed by defining the load application position offset δyF, according to the following formulae:



original starting position of gear load application: L0 = middle of the gear - (gear width/2)
Original final position of the load application on the gear: R0 = middle of the gear + (gear width/2)

* If δyF -> 0
New starting position of the load application on the gear: L1 = L0 + 2 * δyF
New final position of the load application on the gear: R1 = R0

* If δyF < 0
New starting position of the load application on the gear: L1 = L0
New final position of the load application on the gear: R1 = R0 + 2 * δyF

The calculations shown above are only valid if the relevant setting is enabled in the Module specific settings. Use the same process for all the gear elements.

Take stiffness matrix into account: If this checkbox is selected, any gear body definition present in the connected gear calculation is taken into account. Only the gear body of the gear selected in the **Read data from file** option is used. Note that the gear body's stiffness matrix is not used in the standard shaft calculation, i.e. that it doesn't change the shaft stiffness.

Bevel gear
Position of contact: refer to the data for cylindrical gears.
The bevel gear's position can be converted using the bevel gear data. The reference point for positioning is the middle of the bevel gear width on the pitch cone. The bevel gear position can be converted using the position of the axis crossing point on the shaft and other bevel gear data.
The formulae in ISO 23509:2016 Annex D are used to determine the axial and radial forces for bevel gears in the shaft calculation.
An additional force component due to friction is taken into account when calculating hypoid gears. The corresponding coefficient of friction μ can be defined in the module-specific settings.
If the meshing type is set to "Multiple meshing", you can model several meshings on the same cylindrical gear element.

Face gear

The pitch angle for face gears is always set to 90° (this input cannot be changed).

Worm

Position of contact: refer to the data for cylindrical gears. A worm is usually a driving element. Its efficiency is included in the calculation of force components.
If the worm data is read from a Z80 file, select the **Calculation with enhanced formulae (differs from standard)** option in the **Module specific settings**, in the **Calculations** tab, in the worm gear calculation. This ensures that the radial forces in the shaft calculation match up with the radial forces in the worm gear calculation (see chapter 18.5.4.3, Calculating with alternative formulae (differs from standard)).

Worm wheel
Position of contact: refer to the data for cylindrical gears. The worm wheel is usually driven. Its efficiency is included in the calculation of force components.

Rope sheave
Direction of rope sheave: Input the direction of the resulting belt forces as shown in (see Figure 25.6)

The direction of the helix angles and the positions of the elements are defined in Figure (see Figure 25.1).

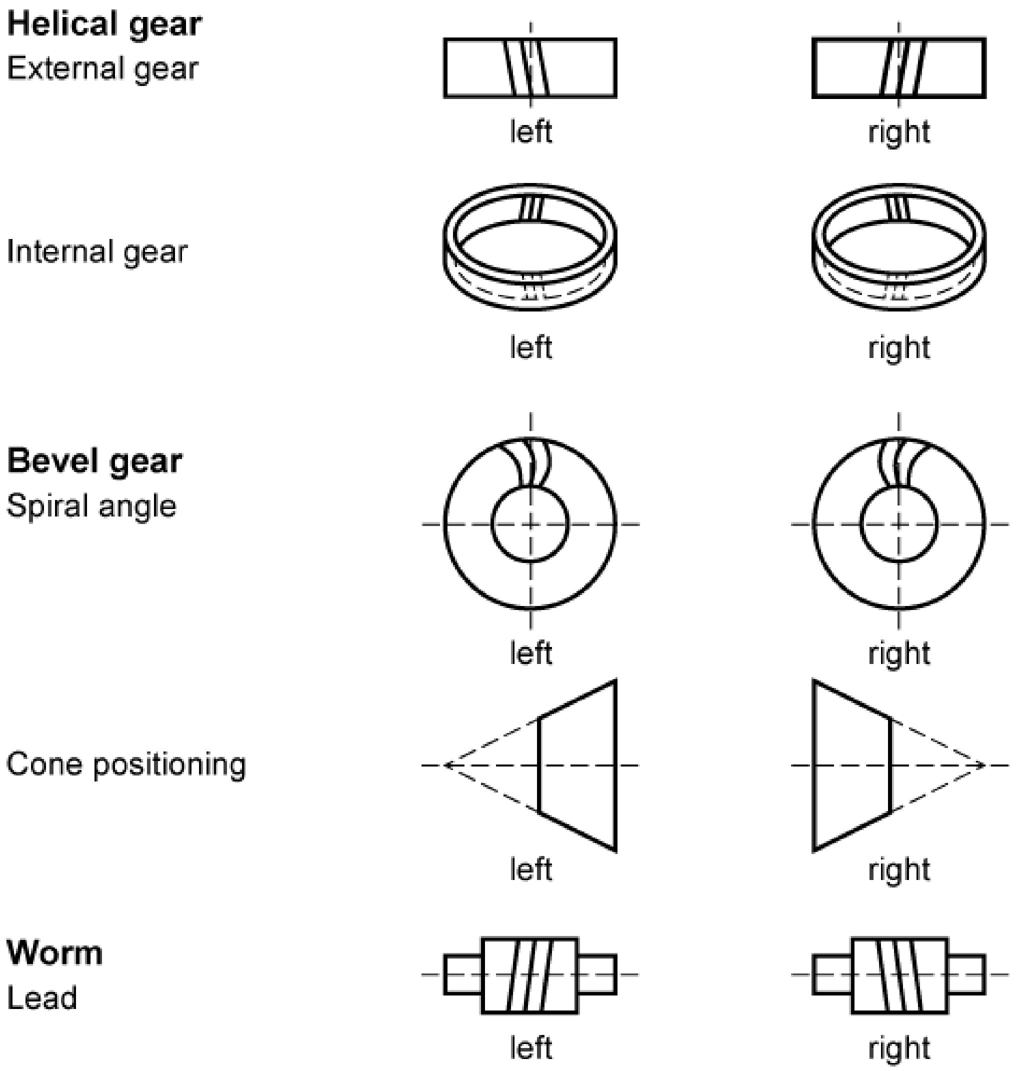


Figure 25.1: For defining the direction for force elements.

Eccentric force

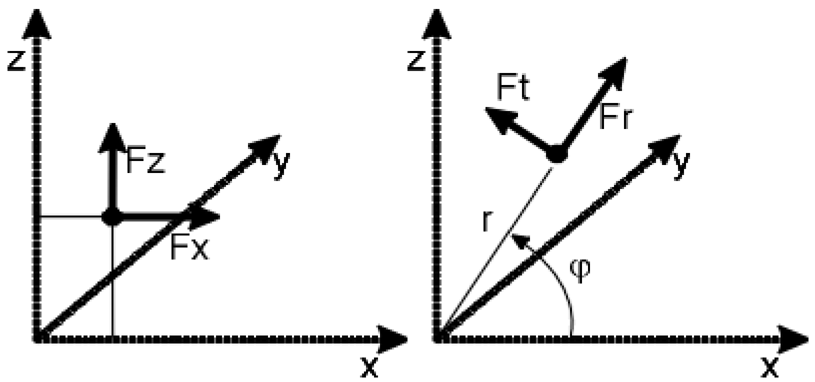


Figure 25.2: Cartesian/polar coordinates for eccentric force

You can enter values for eccentric force either in Cartesian or polar coordinates (see Figure 25.2). You can change the coordinates system in the **Drawings/Settings** screen in the Shaft Editor.

Transferring data from gear calculation

In the **Element Editor**, you can import the data used to define spur and bevel gears from a gear calculation file. Select the element you require in the Element Tree and then click on the **Read data from file** checkbox. Then select the gear number (1 to 4). The data relevant to these gear pairs is then imported directly. In this situation, the data at the pitch point is used instead of the data at the reference circle.

Important: If the **Read data from file** option in this input window remains selected, data will be reimported from the gear calculation every time you call the shaft calculation function. If you then change the gear data later on, the new data will automatically be transferred with it! However, if you only want to import this data once, deactivate this option again once you have imported your data.

25.2.4.2 Coupling

A coupling transmits torque and can also be subject to radial and axial forces. From the torque (or the specified power and speed) you can calculate the circumferential force to

$$F_t = \frac{M_t \cdot 2}{d}$$	(24.2)

F_t	= Circumferential force
M_t	= Torque
d	= Effective diameter

Calculating radial force for a coupling:

$$F^*_R = K_2 \cdot F_t$$	(24.3)

F_t	= Circumferential force
K_2	= Radial force factor

Define the direction of the force in the input window. You are also prompted to enter the mass of the coupling so it can be included in the calculation as a gravitational force.

Calculating axial force for a coupling:

$$F^*_A = K_3 \cdot F_t$$	(24.4)

F_t	= Circumferential force
K_3	= Axial force factor

Axial force acts along the center line of the shaft.

25.2.4.3 Masses

Masses placed on the shaft are used as moments of inertia to determine the critical speeds. They are to be considered as a gravitational force.

25.2.4.4 Magnetic force

The axial force of the magnetic force is shown in the following equation:

$$F_A = -F_A^* \cdot T \cdot k$$

In this case,

F^*_A	=	axial force factor
T	=	torque (with sign)
k	=	groove lead
k= -1	=	groove lead, right
k= +1	=	groove lead, left
k= 0	=	groove lead, straight

This figure is the schematic display for an armature with a groove lead (left).

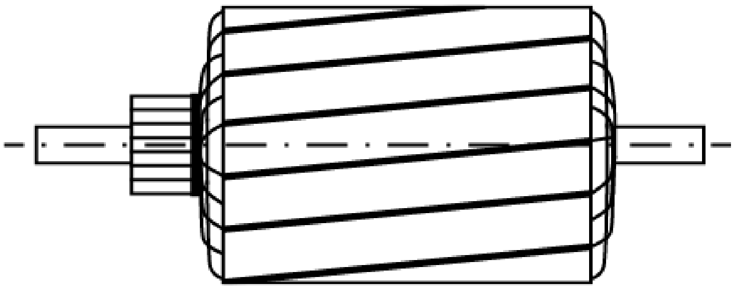


Figure 25.3: Figure: Armature with groove lead (left)

25.2.5 Bearings

In addition to calculating the shaft, you can export rolling bearings, plain bearings and general bearings as separate rolling bearing or plain bearing calculation files (**File > Export**).

25.2.5.1 Support

A support is a generic boundary condition for the associated shaft. You can configure this boundary condition to suit your own requirements. You can model all relevant degrees of freedom as non-locating, elastic or rigid. The relevant degrees of freedom are:

- Displacement in radial direction
- Displacement in axial direction
- Rotation
- Tilting

You can also input the stiffness or clearance as required for these degrees of freedom. The next table lists the different templates that are also available for commonly used bearing types:

Support selection list	**Radial displacement**	**Axial displacement**	**Rotation**	**Tilting**
Own input	Own definition	Own definition	Own definition	Own definition
Non-locating bearing	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on both sides <->	fixed	fixed	non-locating	non-locating
Fixed bearing adjusted on right side ->	fixed	right	non-locating	non-locating
Fixed bearing adjusted on left side <-	fixed	left	non-locating	non-locating
Axial bearing adjusted on both sides <->	non-locating	fixed	non-locating	non-locating
Axial bearing, adjusted on right side ->	non-locating	right	non-locating	non-locating
Axial bearing adjusted on left side <-	non-locating	left	non-locating	non-locating
Fixed	fixed	fixed	fixed	fixed

You can also define a load spectrum for the rotation degree of freedom.

To ensure compatibility with previous versions of KISSsoft, you can also use Cartesian coordinates to configure the degree of freedom (by using Cartesian support):

Support selection list	**ux**	**uy**	**uz**	**rx**	**ry**	**rz**
Own input	Own definition	Own definition	Own definition	Own definition	Own definition	Own definition
Non-locating bearing	fixed	non-locating	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on both sides <->	fixed	fixed	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on right side ->	fixed	right	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on left side <-	fixed	left	fixed	non-locating	non-locating	non-locating
Axial bearing adjusted on both sides <->	non-locating	fixed	non-locating	non-locating	non-locating	non-locating
Axial bearing, adjusted on right side ->	non-locating	right	non-locating	non-locating	non-locating	non-locating
Axial bearing adjusted on left side <-	non-locating	left	non-locating	non-locating	non-locating	non-locating
Fixed	fixed	fixed	fixed	fixed	fixed	fixed

ux, uy, uz: Displacement in x-, y- and z-direction

rx, ry, rz: Rotation about x-, y- and z-direction

25.2.5.2 Rolling bearings

In addition to a general support, you can also select a specific rolling bearing. If you do so, the bearing data is taken from the rolling bearing database.

If a suitable entry is present in the bearings database, the bearing stiffness value is taken from there. Alternatively, you can enter the values directly in the Element editor. You also have the option of defining the radial and axial stiffness, and the tilting stiffness, in a file (see example «Example-Userdefined-Bearing-Stiffness.dat» in the «dat» directory), and then using these values to calculate the local operating stiffness of the bearing. The file should include the bearing clearance in the position contact-load curve, which is why the value input for clearance is set to 0 when the file is imported.

A pretension force, applied on the outer ring, can be used to define the pretension force on the bearing, instead of using the offset. This is only taken into account for bearings with inner geometry, and only if the relevant bearing can accept an axial pretension force. The pretension force is then transformed internally to an equivalent axial offset of the outer ring. For bearings with inner geometry, you can also specify a rotation around the x and z axis of the outer ring. This could then be used, for example, to model the housing deformation, so you can enter the FEM results directly.

$P_d = d_o - d_i - 2 * D_w$

Here, P_d is the diametral pitch bearing clearance, d_o is the outside raceway diameter, di is the internal raceway diameter and D_w is the rolling element diameter. Similar definitions are used for other bearing types, which vary depending on which particular type is involved.

The rolling bearings can be modeled as radial or thrust bearings, each with different mountings, (refer to the next table):

Rolling bearing selection list	**ux**	**uy**	**uz**	**rx**	**ry**	**rz**
Non-locating bearing	fixed	non-locating	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on both sides <->	fixed	fixed	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on right side ->	fixed	right	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on left side <-	fixed	left	fixed	non-locating	non-locating	non-locating
Axial bearing adjusted on both sides <->	non-locating	fixed	non-locating	non-locating	non-locating	non-locating
Axial bearing, adjusted on right side ->	non-locating	right	non-locating	non-locating	non-locating	non-locating
Axial bearing adjusted on left side <-	non-locating	left	non-locating	non-locating	non-locating	non-locating

Table 25.2: Selection options and their definition of the degrees of freedom

25.2.5.3 Plain bearings

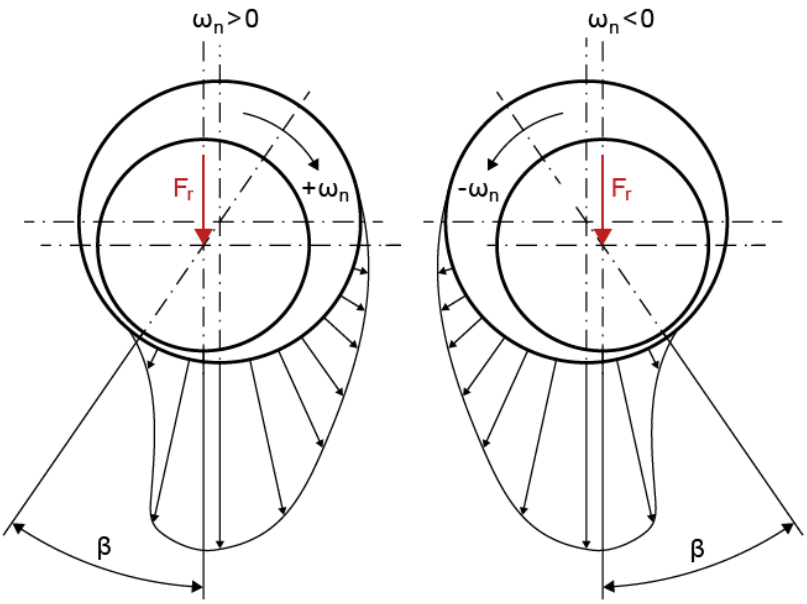
Two types of plain bearings can be defined in the shaft editor:

- Hydrodynamic plain journal bearings

- Journal and thrust plain bearings

25.2.5.3.1 Hydrodynamic Plain Journal Bearings

Hydrodynamic plain journal bearings (see chapter 29, Hydrodynamic Plain Journal Bearings) can be represented with the hydrodynamic plain journal bearing element. In addition to entering the nominal width B, the attitude angle β and the diametral clearance P_d, you can also input the radial stiffness c_r. This affects the shaft after the available clearance has been fully used. You can also enter a damping rate d_r to evaluate the forced vibrations (see chapter 26.8, Forced vibrations).



The plain bearing is implemented as a non-locating bearing which can transmit radial forces but no tilting moments. If a particular configuration results in tilting moments, which the plain bearing must be able to transmit in practice, you must add in a support to ensure the modeling is correct.

25.2.5.3.2 Plain journal and thrust bearings

Non-hydrodynamic plain journal and thrust bearings, which can be used for example to model bushings and washers, can be represented with four different types of plain bearing elements, as shown in the figure below:

a) Plain journal bearing which transmits radial force
b) Plain journal bearing which transmits radial force and tilting moment
c) Plain thrust bearing which transmits axial force on the right
d) Plain thrust bearing which transmits axial force on the left

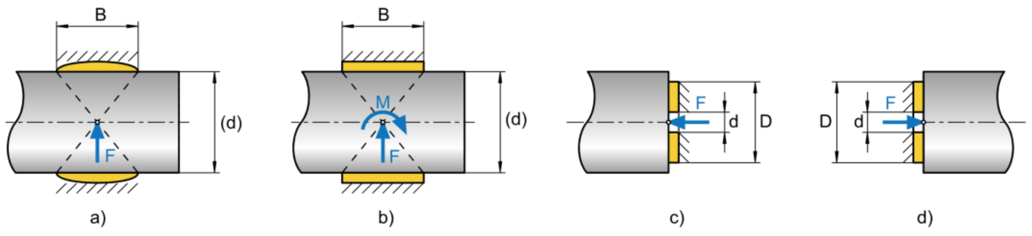


Figure 25.4: Types of non-hydrodynamic plain bearings

Depending on the journal or thrust bearing type, you have to enter the radial or axial friction coefficient, μ_r or μ_a, respectively. Additionally, for journal bearings you have to enter the bearing width B, while for thrust bearings you have to enter the inner and outer diameters of the thrust collar, d and D, respectively. The diameter of the journal bearing does not have to be entered because it is automatically retrieved from the shaft dimensions. All four types of bearings are modeled as infinitely stiff supports without clearance. The point of application of reaction forces and tilting moments is considered as depicted above (see Figure 25.4).

Several results, which are also printed in the report, are calculated for plain bearings. Namely, the friction force F_f, the friction moment M_f and the mean surface pressure p are calculated as:

Journal bearing	**Thrust bearing**
$F_f = F \cdot \mu_r$	$F_f = F \cdot \mu_a$
$r_f = \frac{d}{2}$ $M_f = F_f \cdot r_f$	$r_f = \frac{1}{2}(\frac{d}{2} + \frac{D}{2})$ $M_f = F_f \cdot r_f$
$p = \frac{F}{B \cdot d}$	$p = \frac{F}{\frac{\pi}{2}(D^2 - d^2)}$

Surfaces used for the calculation of the mean surface pressure p are also depicted, as shown below.

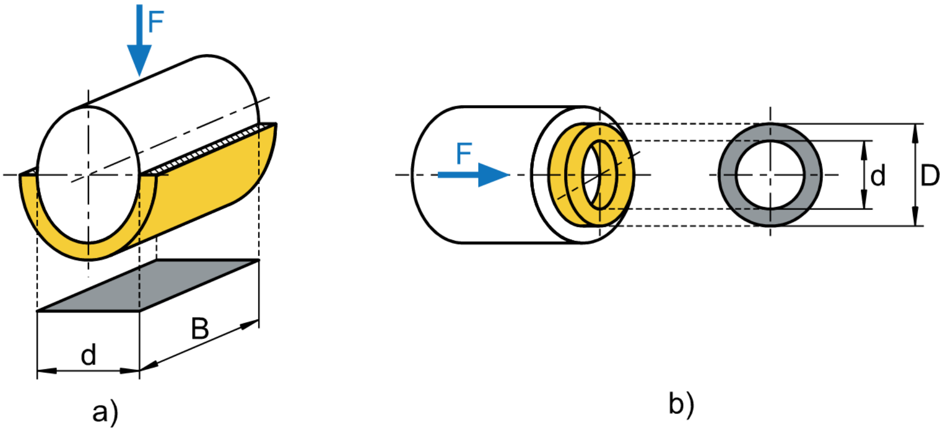


Figure 25.5: Surface used in the calculation of the mean contact pressure for: a) journal bearing, b) thrust bearing.

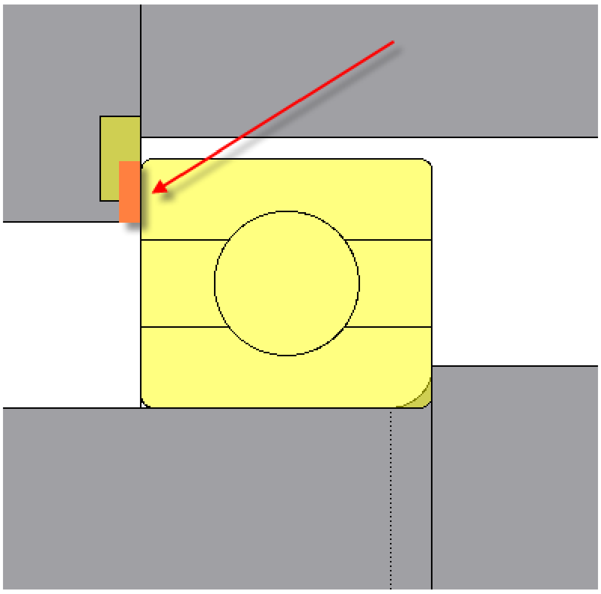
Finally, the frictional power loss P_f is calculated:

$$P_f = M_f \cdot \frac{2 \cdot \pi \cdot n_{rel}}{60}$$

whereby n_{rel} is the relative rotating speed of the shaft.

25.2.5.4 Pure thrust bearing

If a bearing is defined as a pure thrust bearing (left/right/both side layout) no radial forces can be transferred to it. Any radial forces created by the pressure of the outer ring on the housing and the corresponding coefficients of friction will be ignored.



25.2.6 Connection elements

A number of coaxial shafts can be connected by two different connection elements: "General connection", "Connecting rolling bearing" or "Connecting plain bearing".

The connecting element between these shafts defines the shaft's degree of freedom at the connection point for the three relative displacements along the x-, y- and z-axis, and three relative rotations around the x, y and z axis.

25.2.6.1 General connection

A general connection is similar to a support. It is a generic connection for the associated shaft that can be configured to suit your own requirements. You can model all six degrees of freedom as non-locating, elastic or rigid. You can also input the stiffness or clearance as required for all degrees of freedom.

The "Joint" and "Stiff connection" templates are available for typical bearing types.

Joint: All displacements are prohibited, but all rotations are permitted.

Stiff connection: All degrees of freedom (3 displacements and 3 rotations) are prohibited.

25.2.6.2 Connecting rolling bearing

A rolling bearing can be used to connect two shafts. The data you input is the same as that for a rolling bearing for a single shaft, but with the addition of inputs for the internal and external shaft.

The bearing inner ring is assumed to be fixed on the inside shaft, and the bearing outer ring is assumed to be fixed on the outside shaft. The type of the bearing (fixed bearing, placed on left/right side, etc.) defines how the axial forces are transmitted from the shaft through the bearing.

As connection elements are able to transfer not only forces but moments, we strongly recommend that the bearing calculation is performed with internal geometry as defined in ISO/TS 16281.

25.2.7 Cross sections

You will find more examples of cross sections and the strength calculation in the "Calculating Shafts" chapter 26, especially in (see chapter 26.5.12, Cross sections).

25.2.7.1 Free cross section (single notch)

You can use free cross sections to input notch effects, **no matter what the actual definition of the shaft geometry**.

25.2.7.2 Free cross section (multiple notches)

Use this type of cross section to input stress results from a FE calculation at a specific point on the shaft. This is then used to determine a notch factor which can be used to run a nominal stress-based proof according to DIN 743. The standard's application limits also apply to this type of calculation. You will find a more detailed description in (see chapter 26.5.6.4, Calculation of multiple notches according to DIN 743).

25.2.7.3 Limited cross section

You should define the restricted cross section as the preferred cross section type in shaft calculations. The notch effect is determined automatically according to the geometry data input at this position in this cross section. If you make changes to the shaft geometry, you do not need to modify the cross section manually. The changes are transferred automatically. However, if you are working with restricted cross sections, you must input shaft geometry in detail.

25.2.7.4 Documentation point

You can use a documentation point to document the equivalent stress, displacement, rotation, force and moment at any position on the shaft in the editor and in the report.

25.3 Basic data

The **Basic data** input window is where you enter the basic settings for the shaft calculation. The relevant parameters are described below.

25.3.1 Position of shaft axis in space

The position of the shaft axis in space is defined accordingly (see Figure 25.6).

The weight of the shaft (in a horizontal position) is considered a gravitational force in the Z-Y plane when calculating the deflection. However, if the shaft is positioned vertically, the resulting axial force is, for example, included in rolling bearing calculations. If a shaft is positioned at an angle, the relevant components are distributed correctly on the Z-Y plane and on the axial force.

Alternatively, you can use the 3-coordinate format to enter the shaft mass direction vector.

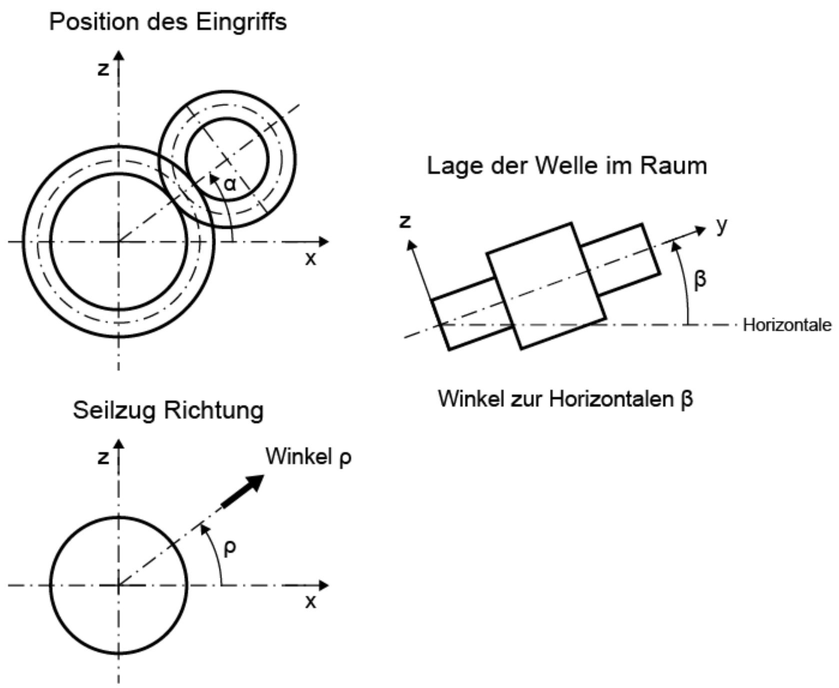


Figure 25.6: Defining the position of the shaft and the position of contact.

25.3.2 Number of eigenfrequencies

Defines the number of eigenfrequencies that KISSsoft is to determine (see chapter 26.2, Eigenfrequencies).

25.3.3 Number of buckling cases

Defines the number of buckling cases that KISSsoft is to determine (see chapter 26.3, Buckling).

25.3.4 Speed

Enter the speed in revolutions per minute (rpm). Click the Plus button to display the **Define speed** window. This is where you define the speed and the sense of rotation for all the shafts.

The shaft speed used for the calculation can also be found in the corresponding report: select "Report" > "Shaft speeds".

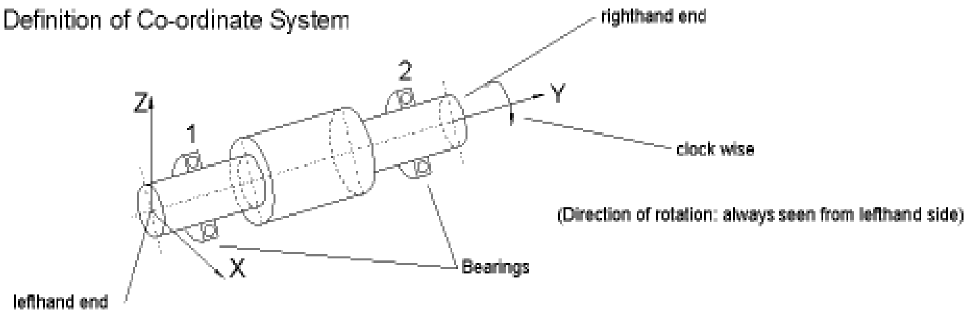
► **Note**

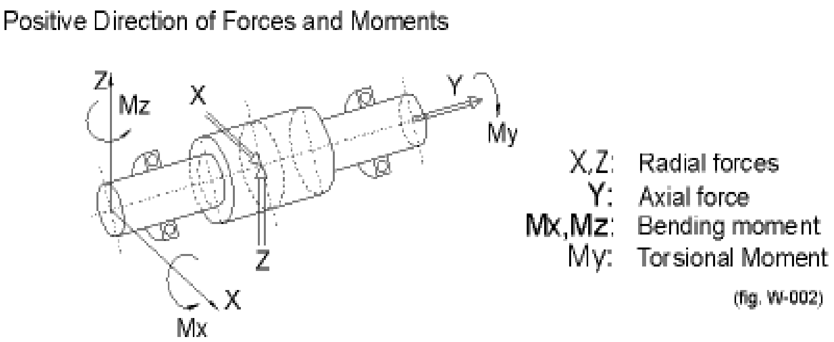
If you change the speed, the effective torques and power change accordingly.

25.3.5 Direction of rotation

The shaft axis runs along the positive y-direction (left to right in the graphical Shaft Editor). In the Shaft Editor, the z-axis points vertically upwards and the x-axis points towards the user. A right-hand rotation of the shaft around the positive Y-axis direction is specified as "clockwise".

The next figure shows the direction of these coordinates and the positive direction of forces and moments. Note that weight has an effect in the negative z-direction if the shaft is positioned horizontally (see chapter 25.3.1, Position of shaft axis in space).





(fig. W-002)

In most force elements, the direction of the torque is defined by setting them as "driving" or "driven". If you enter a "driving" value, this means either that the shaft drives (an external application) or that the moment runs counter to the sense of rotation (i.e. the shaft loses power). If you enter a "driven" value, this means either that the shaft is driven from outside (e.g. by a motor) or that the moment runs in the same direction as the sense of rotation (i.e. the shaft is supplied with power).

25.3.6 Reference temperature

The **reference temperature** is the temperature used in the drawing data or to check the part.

25.3.7 Housing temperature

When used together with the thermal expansion coefficient, the housing temperature causes a general displacement of all the bearing points. It also affects the operating clearance of rolling bearings.

It is assumed that the bearing's outer ring and the housing or the outer ring have the same temperature and that the bearing's inner ring and the inner shaft also have the same temperature.

► **Note**

Take the axial stiffness of the bearings into account if you want to examine the influence of thermal expansions. Otherwise, the load peaks will be too high.

Reference point housing

Reference point for the displacement of bearing points due to the thermal expansion of the housing. For example, if $y_\theta = 0$, this means all thermal expansion is considered relative to the global frame of reference.

The magnitude of the thermal expansion which is applied on the bearing outer ring is given by ΔL, where

$$\Delta L = (T_c - T_r) \cdot \alpha \cdot (y_b - y_\theta)$$

T_c is the housing temperature

T_r is the reference temperature

α is the coefficient of thermal expansion of the housing material

y_b is the global axial coordinates of the bearing (relative to the global frame of reference, not the shaft)

y_θ is the housing temperature thermal reference point used to perform the calculation

For example, if $y_\theta = 0$, this means all thermal expansion is considered relative to the global frame of reference.

25.3.8 Load spectra

If the loads defined in the Shaft Editor have been assigned a load spectrum, this can be taken into account when calculating deformation. Deformation can also be calculated either for the nominal load or for any element of the load spectrum.

To take load spectra into account, select **Load spectra** and then click on the **Consider load spectra** option. However, if you only want to perform the calculation for a single load bin, select **Consider only one load bin of the load spectra**. Enter the appropriate element number in the input field to the right of the drop-down list.

If the **Consider load spectra** option is selected, the following modifications are made if the definition of the load spectra is inconsistent:

- if the frequency H = 0 is set, this is set to the value 10 ^-10
- if the speed factor nfact = 0, this is set to 10^-5 and the torque/load factor is set to 10^-10
- if the torque/load factor is set to Tfact = 0, this is set to 10^-10

25.3.8.1 Load spectra with negative bins

Load spectra with negative load spectrum elements (T < 0 and/or n < 0) are handled as follows:

Torque factor	Speed factor	Shaft direction of rotation	Force element
+	+	-	-
+	-	C	D
-	+	-	D
-	-	C	-

- = unchanged

C = shaft direction of rotation changes clockwise/counterclockwise

D = driving/driven changes

25.3.9 Gears

If the calculation includes **gears**, they can be considered in a number of different ways:

- **Gears are only load applications:** The masses and stiffnesses of the gears are not taken into account.

- **Consider gears as masses:** The gear is handled as a mass in the bending calculation. The mass results from the difference between the operating pitch circle and the external shaft diameter as well as the gear width (same specific weight as the shaft).
- **Consider gears as mass and as stiffness:** The gear is handled as part of the shaft contour (for example, integral pinion shaft).
- **Consider gears mounted by interference fit with stiffness according to ISO 6336-1 (with d_w instead of d_f):** The shaft is stiffened at the mid diameter d_m, with $d_m = (d_1+d_2)/2$, d_1 = shaft diameter, d_2 = the gear's operating pitch circle. The reference diameter is used to calculate the gear's weight. The mean diameter is used in all the other calculations (I_{xx}, I_{zz}, I_p, W_{xx}, W_{zz}, W_p).

► **Note**

If gears have been mounted on shafts by interference fit, it is usually hardly possible to assess the extent to which the gear stiffens the shaft. You cannot use KISSsoft to solve this problem. However, you can estimate the influence the interference fit has: It is sufficient to perform the calculation for **Gear as mass** and for **Gear as mass and stiffness** and note the difference in the diagrams of bending. If the difference is small, the interference fit has no influence. However, if the difference is significant, you must enter more precise information. To do this, you must integrate a part of the gear in the shaft contour in the graphical shaft input.

If multiple identical gear elements are defined at the same position, for a gear with multiple contacts (such as a sun wheel in a planetary system), the weight is only taken into consideration once.

25.3.10 Consider weight

This defines how the shaft's own weight is taken into account in the section dimension calculation. Depending on the orientation of the shaft arrangement (see chapter 25.3.1, Position of shaft axis in space), you will see additional axial and shear forces which may have an influence on the diagrams of bending and/or axial displacement.

► **Note**

In a global coordinates system, gravitational forces act on the shafts in the negative, z-direction.

25.3.11 Consider gyroscopic effect

When calculating the eigenfrequencies, you can also take into account the gyroscopic effect of shafts that have weights attached to one end and either rotate forwards or backwards around the longitudinal axis.

Whereas, in situations that are not technically critical, the eigenfrequency sinks when the speed increases in a counter direction, the eigenfrequency increases when the speed is in the same direction.

25.3.12 Housing material

The housing material value is only used to calculate the thermal expansion of the housing. The materials available for housings are identical to those used for shafts.

25.4 Rolling bearings

25.4.1 Calculation method

If the calculation includes **rolling bearings**, they can be considered in a number of different ways:

- **Stiffness: not calculated. Rating life: ISO 281, with manufacturer's notes:** Calculation using the classic method (as described in manufacturers' catalogs). In this calculation of the way bearings with a contact angle (e.g. tapered roller bearings) react, the bearing reaction is determined at the place where the direction force intersects with the shaft's symmetry axis.
  In this way, the interdependency between the axial and radial forces, such as exists in tapered roller bearings, is included in the calculation. Rolling bearings primarily place constraints on the degree of freedom of movement found in displacement and/or rotation, which is why they are modeled in this way when you select this option. You can input displacement or torsional stiffness (if no values are input, the bearing is assumed to have infinite stiffness). This means the calculation is not affected by the type or size of the bearing.
- **Stiffness: ISO/TS 16281. Rating life: ISO 281, with manufacturer's notes:** If you select this option, the shaft's bending lines are affected by the finite bearing stiffness which is calculated based on the bearing’s geometry. However, the rating life is calculated based on the forces, according to the manufacturer catalog (i.e. tilting moments are ignored in the rating life calculation).
- **Stiffness: ISO/TS 16281. Rating life: ISO/TS 16281:** Both the shaft bending lines and the rating life of the bearing are based on the internal geometry of the bearings according to ISO/TS 16281.

You will find more detailed information about the shaft calculation here: (see chapter 27, Rolling Bearings (Classic Analysis)) and (see chapter 28, Rolling Bearings (Internal Geometry)).

25.4.2 Tolerance field

The definition of the bearing clearance class does not yet provide a definitive statement about bearing clearance because only a range of values has been defined for the bearing clearance class. The **Minimum** and **Maximum** options define the upper and lower limits of the range, whereas the **Mean value** is the arithmetical average of the **Maximum** and **Minimum** for (radial) bearing clearance.

The **Operating bearing clearance** is defined using the selected bearing clearance class (e.g. "C0"), the selected tolerance (e.g. "mean value") and the working conditions, i.e. speed and temperature. For every rolling bearing, the calculation of operating clearance is described below. Starting from the next figure (see Figure 25.7), the following variables are introduced in the calculation:

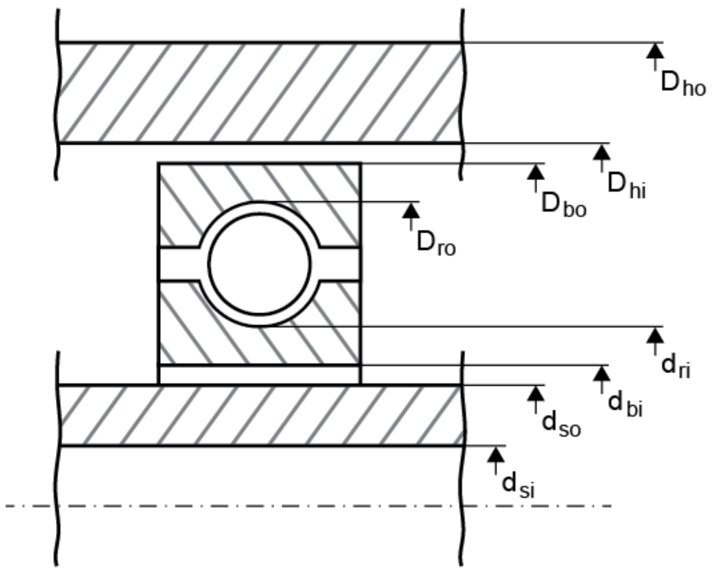


Figure 25.7: Diameters used to calculate bearing clearance

- d_{si}, d_{so} Internal and external diameter of the shaft.
- D_{hi}, D_{ho} Internal and external diameter of the hub. If the bearing is a connecting element, then these values represent the internal and external diameter of the external shaft. For simplicity's sake, the term "housing" is used here to mean either the housing or the external shaft (if present).
- d_{bi}, d_{bo} Internal and external diameter of the bearing, and d_{ri}, d_{ro} for the diameter of the internal and external outer raceway.

All diameter values represent actual diameters, i.e. they take the allowance for each part into account. The calculation steps are as follows:

- The ring race allowance is taken from the corresponding table (e.g. for tolerance "PN"), for the inner ring AB_i and the outer ring AB_o.
- The allowance for the shaft, A_w, and housing, A_n, are calculated from the user-defined data (e.g. "k6"). You can either input the corresponding allowance directly or select the tolerance field from a selection list (the most commonly occurring tolerance fields are included in the list). Alternatively, you can input the tolerance field manually by clicking "Own input" (e.g. ‘k6’).
- The interference is calculated on the inner ring Uwi and on the outer ring U_{wo}.

$$U_{wi} = A_w - AB_i$$
$$U_{wo} = AB_o - A_n$$

- According to DIN 7190, the interference is reduced by the value 0.4*(*R*zA + *R*zB). In this case, *R*zA and *R*zB are the surface roughness of the contact bodies (A: rolling bearing ring, B: shaft/hub). It is assumed that the roughness of the rolling bearing rings is much less than the roughness of the shaft/hub. For this reason, the roughness of the rolling bearing rings is not taken into account (*R*zA = 0).

$$U_{wi} = A_w - AB_i - 0.8R_{zs}$$
$$U_{wo} = AB_o - A_n - 0.8R_{zh}$$

- The effect of temperature is taken into account,

$$U_{wiT} = U_{wi} + (a_s - a_b)(T_s - T_R)d_{nom}$$
$$U_{woT} = U_{wo} + (-a_h + a_b)(T_h - T_R)D_{nom}$$
$$d_{soT} = d_{so}\left[1 + a_s(T_s - T_R)\right]$$
$$d_{biT} = d_{bi}\left[1 + a_b(T_s - T_R)\right]$$
$$D_{hiT} = D_{hi}\left[1 + a_h(T_h - T_R)\right]$$
$$D_{boT} = D_{bo}\left[1 + a_b(T_h - T_R)\right]$$

where α_s, α_h, αb is the thermal expansion coefficient of the shaft, housing and bearing, $T_s T_h T_R$ are the shaft, housing and reference temperatures and d_{nom}, D_{nom} is the nominal diameter of the bearing as defined in the catalog.

- An interference fit calculation is performed for the inner ring if condition A applies and for the outer ring if condition B applies, taking into account the operating speed as well.

$$d_{soT} + U_{wiT} > d_{biT} \, (A)$$
$$D_{boT} + U_{woT} > D_{hiT} \, (B)$$

- The pressure generated in the interference fit changes the operating diameter of the bearing races, and therefore also causes a change, ΔPd, to the nominal bearing clearance.

► **Note**

The selection you make in the Tolerance field has no effect on the general behavior of the bearings.

25.4.3 Axial clearance

The axial clearance for the rolling bearing calculation according to ISO 281 can be defined here. The clearance applies to both directions. As a result, a bearing that is fixed on both sides may deviate either to the right or to the left by this value. This clearance is not used in the calculation if the bearing stiffness from the internal bearing geometry according to ISO/TS 16281 is taken into account.

25.4.4 Failure probability

The value n=10% is used as standard for the failure probability, in the rolling bearing service life calculation. The valid input range is 0.05% < n < 10%.

25.4.5 Required rating life

Preset required rating life for rolling bearings. This value does not actually affect the rolling bearing calculation. However, if the calculated bearing rating life is less than the required rating life, KISSsoft displays a warning message.

25.4.6 Set lubrication for each bearing individually

If this option is selected, and **Modified rating life according to ISO 281** has been selected in the **Basic data** tab, you can enter individual lubricant data for each rolling bearing, in the **Lubrication** sub-tab, in the element in the Element Tree. Otherwise, the contamination is set globally in the **Basic data** tab and applies to all rolling bearings.

25.4.7 Modified rating life according ISO 281

Enables the state of the lubricant to be taken into account when calculating the rating life as a modified service life L_{mn}. This requires the **Lubrication and Contamination** selection lists and the **Lubricant temperature** input field to be configured accordingly.

25.4.8 Lubrication

The choice of lubricant only affects the bearing life calculation. Click the Plus button to enter your own data with **Own input** for the lubricant parameters.

25.4.9 Lubricant temperature

The **Lubricant temperature** changes the lubricant's viscosity. This value is used to determine the extended bearing rating life (aISO) and the moment of friction.

25.4.10 Contamination

As defined in ISO 281, contamination coefficient e_c depends on the type of oil filter, the bearing size, and the viscosity of the lubricant. This value varies within the range 0 (high level of contamination) and 1 (ideal). Select the **Own input** option and then click the Plus button to specify your own e_c values.

► **Note**

Click the Plus button to enter your own values. You can define new values for the **Housing** and **Lubrication** that are based on existing data. However, these values are only data used in the calculation file and are not stored permanently in the database.

25.4.11 Using proprietary bearing internal geometry data

When calculating bearing stiffness and rating life according to standard ISO/ TS16281 an internal bearing geometry, such as number of rolling elements, rolling body diameter, rolling body pitch diameter, osculation, etc., is needed (see also (see chapter 28, Rolling Bearings (Internal Geometry)). If these data are not available in the database, KISSsoft will estimate them. However, some bearing manufacturers do provide these data but are not willing to expose them to the end users. In such case the bearing internal geometry data in KISSsoft are kept in a hidden database which is not exposed to an end user. Since these data are proprietary, they are not shown in the reports or graphics, except number of rolling elements. If this setting is activated, KISSsoft will use - if available - proprietary bearing internal geometry data given in a hidden database, otherwise it will use data provided in a regular KISSsoft database - if stored there. Note that KISSsoft always estimates data that are not available in the regular database or in the hidden database.

25.4.12 Using load spectra: define shaft, oil and housing temperatures individually for each load case

If you use this option, you can input shaft, oil, and housing temperatures individually for each load case. This setting is available only if you choose to take load spectrum into account in the **Basic data** tab. The data can be defined by choosing **Own input** in the load spectrum drop down menu. In there, the data can be defined manually in a table or they can be read from a text file. Appropriate columns in the text file must be defined as shown below. For more details (see chapter 9.5.13, Load spectra) and (see chapter 25.3.8, Load spectra).

```
contents: h T n Temp TempHousing TempOil

// Explanation of variable names (variable names cannot be changed)

// - h … frequency

// - T … torque

// - n … rotating speed

// - Temp … shaft temperature

// - TempHousing … housing temperature

// - TempOil … oil temperature

0.1 1 1 70 70 70

0.2 0.9 1 65 65 65

0.5 0.8 1.1 50 50 50

0.2 0.7 1.1 50 50 50
```

25.4.13 Grease lifetime

In this list, you can select whether an estimated grease lifetime is to be calculated. These methods are described in more detail in the Grease lifetime section (see chapter 27.5, Grease lifetime), in the Rolling Bearings chapter.

25.4.14 Calculation method for friction

Select the method for calculating friction from this list. These methods are described in more detail in the Rolling Bearings section (see chapter 27.4, Moment of friction), in the Moment of Friction chapter.

25.4.15 Oil level

If you select the calculation method described in the SKF catalog 2018 or the Schaeffler catalog 2017 (INA, FAG) to calculate friction, the oil level has an effect on the moment of friction which is determined by the amount of oil lost in the process. This is described in greater detail in the Moment of Friction chapter. (see chapter 27.4, Moment of friction).

You input the oil level with reference to the left-hand end of the first shaft (but only if **Oil bath lubrication** has been specified). The position of the shaft is then used to define a separate oil level for each bearing (h and H) which is then taken into account when calculating the loss. The oil level is displayed in the Shaft Editor, so you can check it.

25.4.16 Type of oil lubrication

The type of oil lubrication used is important if you are using the calculation method described in SKF catalog 2018 to calculate friction. The method differentiates between oil bath and oil injection lubrication (see chapter 27.4, Moment of friction).

25.4.17 Moment of friction, seals

Defines how the moment of friction is to be defined for seals:

- **SKF main catalog according to selected calculation method**
    - **According to SKF main catalog 4000/IV T DE:1994**
      You will find values from the SKF catalog for the seal types used in your bearings integrated in the KISSsoft software. If the KISSsoft system finds a familiar seal label in the bearing label, it calculates the moment of friction for a grinding seal using the coefficients listed in the catalog. Otherwise, the moment of friction is set to zero.
    - **According to SKF main catalog 17000/1 EN:2018**
      You will find values from the SKF catalog for the seal types used in your bearings integrated in the KISSsoft software. If the KISSsoft system finds a familiar seal label in the bearing label, it calculates the moment of friction for a grinding seal using the coefficients listed in the catalog. Otherwise, the moment of friction is set to zero.
      In KISSsoft, the diameter of the mating surface is calculated with the formula ds = d + (D - d) * 0.2
- **according to ISO/TR 13593:1999 Viton** M_{seal}, this diameter is calculated with the formula: M_{seal} = 3,736*10^-3*dsh; M_{seal} in Nm, d_{sh} Shaft diameter in mm
- **according to ISO/TR 13593:1999 Buna N** M_{seal} this diameter is calculated with the formula: M_{seal} = 2,429*10^-3*dsh; M_{seal} in Nm, d_{sh} Shaft diameter in mm

25.4.18 SKF spherical roller thrust bearings: runout error takes into account load distribution

When you select **Calculation > Settings**, you see this option. Use it to specify whether shaft coaxial errors or runout errors influence the way loads are distributed in the bearing. If this option is selected, the influence is not taken into account. If that is the case, a coefficient of 0.88 is multiplied to calculate the dynamic equivalent load P (P=0.88*(Fa+Y*Fr)).

25.4.19 SKF Cloud Services: using the SKF bearing module

Calculations with the SKF bearing module:

- For most SKF bearings, the following bearing performance results are calculated via the SKF cloud service:
    - SKF rating life (L10m)
    - Basic rating life (L10)
    - Equivalent dynamic bearing load (P) & load ratio C/P
    - Minimum load
    - Viscosity ratio (κ) / contamination factor (e_C) / life modification factor (a_{SKF})
    - Friction & power loss
    - Grease life & relubrication interval
    - Bearing frequencies
    - Adjusted reference speed
- A direct connection is made to the SKF bearing database, ensuring accurate and up-to-date bearing data.
- SKF rating life fully accounts for the benefits of SKF Explorer bearings. In general, the SKF rating life is more realistic than the rating life according to ISO 281, but in particular for the SKF Explorer bearings.
- SKF rating life results are rounded down to 100 hours.

Requirements

- Registration is required which can be done via the **SKF Registration tool** in the **Extras** menu.
- Internet connection is required to connect to the SKF server.

Limitations

- The SKF bearing module does not support the following bearings:
    - non-SKF branded bearings

    - SKF super-precision bearings
    - SKF bearings for extreme temperatures
- For these bearings, all results are calculated via the standard methods in KISSsoft.
- The bearing performance results are calculated as listed above. All other bearing performance results are calculated via the standard methods in KISSsoft.

Additional information

More information on the SKF rating life can be found here. More information on the difference between SKF rating life and rating life according to ISO 281 can be found here. If you have any additional questions or need support from SKF, please contact skfbearingmodule@skf.com.

25.4.20 SKF Cloud Services: using the SKF bearing stiffness module

SKF provides its own stiffness calculation for different bearing models, via an interface. Like in ISO/TS 16281, this makes it possible to take non-linear rolling bearing stiffness and the nominal and modified reference rating life into account in the KISSsoft system. Consequently, the shaft calculation is based on the actual internal geometry of the bearing involved. This ensures highly accurate results, benefiting from the expert calculation-related expertise supplied by the bearing manufacturer.

The following values, among other things, can be taken into account via the bearing stiffness module:

- Bearing displacement and torsion
- Reaction forces and reaction moments
- A non-linear stiffness matrix at the operating point
- Nominal and modified reference bearing rating life Lnrh and Lnmrh
- Axial and radial stiffness curves

If this function is selected, KISSsoft attempts to use the bearing stiffness module for all SKF bearings. If there is an error, KISSsoft automatically switches to the ISO/TS 16281 standard, which is implemented in it.

25.4.21 Timken proprietary bearing internal geometry data

If this option is activated, bearing internal geometry data for Timken rolling bearings will be retrieved from Timken Cloud Services. In order to be able to use this functionality you must have a valid Timken account which can be created here, and you must be connected to Timken Cloud Services. This can be done by clicking on the **Timken Cloud Services** button. In addition, you must activate the **Using proprietary bearing internal geometry data** option, otherwise bearing internal geometry data will be estimated by KISSsoft (see chapter 25.4.11, Using proprietary bearing internal geometry data).

25.5 Module specific settings

25.5.1 Calculations tab

25.5.1.1 Non-linear shaft

Click this option to perform a calculation using geometric non-linear beam elements. This can result in either a deflection or a displacement in the axial direction because the arc length remains constant. In most applications where shafts are used, this non-linear model is irrelevant.

► **Example**

A shaft, which is fixed to its mounting on both sides, is subjected to centrical force. The linear beam model does not allow for an elongation of the beam because it ignores axial displacement during shear and moment loads. If you click on the **Non-linear shaft** field, you can select a calculation method that takes into account the bending effect on the shaft and therefore the elongation of the beam. This results in axial forces.

25.5.1.2 Consider deformation due to shearing and shear correction factor

If this checkbox has not been selected, the shaft is modeled to be **infinitely stiff**. In this case, shearing forces have no effect on the diagrams of bending. This model is suitable for all shafts whose length is significantly greater than their cross section. If this is not the case, you can enable **Consider deformation due to shearing.** The associated shear correction factor κ can be modified in the **Module specific settings**.

$$A' = A \cdot \frac{1}{\kappa} \quad (24.1)$$

where

A’	shear section
A	Cross-sectional area

The shear correction coefficient $\kappa \geq 1$ includes the irregular distribution of stress across the cross section and applies to the entire shaft system. $\kappa = 1.1$ applies for circular shaped cross sections, and $\kappa = 1.2$ applies for rectangular shaped cross sections.

► **Note**

The value input here must correspond to the shear correction factor specified in the **valid definition** in KISSsoft, as shown in the equation above. Some sources also use the reciprocal value for the formula symbol.

25.5.1.3 Activating offset of the load application center point

This enables the gear load elements to define their load application center point offsets, as described in the relevant section of the manual (see chapter 25.2.4.1, Forces).

25.5.1.4 Use FEM Solver

The new solver is used by default for shaft calculations. The FEM Solver (CM2 FEM®/CM2 MeshTools®) can also be used here. The new solver is more stable, which is why we recommend it.

The new solver is based on the finite differences (FD) method, with which the equations for the elastic deformation are approximated numerically in a grid (see chapter 25.5.1.10, Node density). In addition, the cylindrical elements of a linear shaft's contours are calculated with the precise analytical formulae in the transfer matrix method [62], and conical elements are calculated with the FD method. For non-linear shafts, the FD method is applied for the entire shaft contour.

25.5.1.5 Saving temporary results in CSV format with .tmp file extension

Results are saved to temporary files (in the TMPDIR). The naming convention is W010-H3_bin_x.tmp, where "x" is the load spectrum's number. For example, the W010-H3_bin_1.tmp, W010-H3_bin_2.tmp and W010-H3_bin_3.tmp files are created for a load spectrum with 3 stages.

1. For rolling bearings
a. General results of the rolling bearing (displacement, tilting, reaction force)
b. Results for each rolling body
c. Results for each slice, if a roller bearing is selected
d. The stiffness matrix

2. For shafts
a. Data for the bending lines

25.5.1.6 Standard radius at shoulders

To calculate the notch effect on shoulders, you require a radius. This can be input as an auxiliary element. If no radius has been defined, the system uses the standard radius defined for calculating the notch effect.

Generally, we recommend you define radii for each shoulder.

25.5.1.7 Equivalent stress for sizings

Defines the equivalent stress used to size a shaft for strength.

25.5.1.8 Maximum deflection for sizings

Defines the maximum permitted deflection for sizing a shaft for deflection.

25.5.1.9 Damping factor for iterating rolling bearing stiffness

When calculating shaft bending that takes rolling bearing stiffness into account, as described in ISO/TS 16281 the non-linear stiffness of every rolling bearing is recalculated in every calculation step. In exceptional cases, this iteration may lead to convergence problems. Setting a damping factor can help because it limits the extent to which stiffness is changed between two calculation steps. A value of 0 corresponds to no damping. A value of 0.5 already corresponds to a significant level of damping. If convergence problems occur when rolling bearings are involved, we recommend you increase the factor in increments, for example, first to 0.1, then 0.2 etc.

25.5.1.10 Node density

The user can influence how many nodes are used to calculate a beam. If you are performing a linear calculation, this has no effect on the result, apart from line moments which are distributed across the existing nodes. The beam elements supply the exact solution in the linear model independently of the length.

Reasons for influencing the density of nodes are, on one hand, to speed up calculations (for example, in series calculations in KISSsys) and, on the other hand, to ensure the accuracy of the display of the bending lines and the corresponding report.

The density of the nodes affects the accuracy of non-linear beam elements. For this reason, the maximum distance between two nodes for non-linear calculations, when compared with a linear calculation, is halved, no matter what value is predefined.

The node density affects only the elements that are modeled with the finite differences method. Elements that are modeled with the transfer matrix method, are not grid-dependent (see chapter 25.5.1.4, Use FEM Solver).

25.5.1.11 Iterative calculation of load distribution

If this option is selected, the load distribution is calculated iteratively for the selected gear in the **Flank line modification** tab. The initial gear is replaced by a specified number of identical gears. The number of gears is set in the "Number of slices" field. The load on each replacement gear is set according to the current load distribution, and the load on each gear is adjusted iteratively until the quadratic mean value (or "root mean square" - RMS) of the error in the line load difference between two sequential calculations is less than 1%.

For details on how to calculate $K_{Hß}$: (see chapter 26.6, Flank line modification)

Note: In the case of bevel gears, the option must be selected so that the effect of the changeable operating pitch circle of the gear can be taken into account. Otherwise, the bevel gear is handled as a cylindrical gear whose pitch circle dw equals the pitch circle in the middle section.

25.5.2 Strength tab

25.5.2.1 Input different numbers of load cycles for bending and torsion (for limited life calculations)

Every time a shaft rotates, the bending load cycle changes. For this reason, the number of bending load cycles is calculated using the rating life and the speed. The number of torsional load cycles is often very much lower, because not every rotation causes a torsional load cycle. For example, a gear unit may be started in the morning and run throughout the day with a constant torque, resulting in exactly one torsional load cycle per day. In contrast, a shaft running at 1000 rpm for 8 hours would be subject to 8000 bending load cycles in the same space of time. As a consequence, in this example, the ratio between the number of bending load cycles and torsion would be 8000: 1. You can enter this ratio here.

25.5.2.2 Continue calculation despite of incorrectly defined cross sections

If a cross section has been defined incorrectly, the entire calculation is interrupted and its status is set to **Inconsistent**. If you set this flag, the safeties for the incorrectly defined cross section are set to 0, the other cross sections are calculated and the calculation is consistent.

25.5.2.3 Continue calculation despite of shaft sections without rotational symmetry

This option has been included so that the shaft can be checked for strength even though non-rotationally symmetrical sections are present. The cross sections to be calculated must not be in the non-rotationally symmetrical element. However, this is only permissible if the non-rotationally symmetrical element has the same moments of inertia Ix = Iz. If not, the stiffnesses and consequently the entire calculation will be incorrect. Other calculations, apart from the strength calculation, must be checked for accuracy.

25.5.2.4 Maximum safety endurance limit

This is where you input the maximum safety endurance limit. This value can then be used to limit the number of safety values in the report so that, for example, only values that are smaller than the maximum safety endurance limit are displayed.

25.5.2.5 S-N curves (Wöhler lines) exponents according to DIN 743-4

If this option is selected for the strength calculation according to DIN 743, the S-N curve (Wöhler line) exponents q_σ and q_τ are set according to DIN 743-4 for steel materials. DIN 743 is only valid for calculating steel.

If this option has not be activated and a material such as aluminum, stainless steel (austenitising) or ADI is selected, the S-N curve (Wöhler line) exponents according to the FKM guideline are used.

2 buckles in the S-N curve (Wöhler line) are displayed for the materials described above, according to the FKM guideline. This means, the S-N curve (Wöhler line) has a different pitch at NL > 10^6 q_σ = 15 and q_τ = 25. The exponents have the formula symbol k in the FKM guideline instead of the formula symbol q which is used in DIN standards.

25.5.3 Rolling bearing tab

25.5.3.1 Save user defined bearings in calculation file

If this option is enabled, the data from all user-defined rolling bearings is saved together with the calculation file. As a result, this calculation can still be performed if this file is opened on a computer whose database does not contain these rolling bearings. If this data is present both in the file and in the database, the data in the database is used and updated in the file when the file is saved.

25.5.3.2 Read user-defined rolling bearings from calculation file

If this option is also enabled, the data from all user-defined rolling bearings stored in the calculation file is used and given priority over the data in the database. If the database contains bearings with identical IDs, this data is ignored.

25.5.3.3 Maximum life modification factor

The maximum life modification factor defines an upper limit for the life modification factor a_{ISO}:

$$a_{ISO} = a_{ISO} \qquad if \qquad a_{ISO} \leq a_{ISO,\max}$$
$$a_{ISO} = a_{ISO,\max} \qquad if \qquad a_{ISO} \geq a_{ISO,\max}$$

The default value, as defined in ISO 281, is $a_{ISO,max}$ = 50.

25.5.3.4 Display critical bearings

The Shaft Editor displays critical rolling bearings with colors to identify their rating life. The color "orange" is used for critical bearings with a rating life which is less than the required rating life. The color "red" is used for bearings with a minimum rating life which is much less than the required rating life. The color "green" is used for bearings whose rating life is longer than the required rating life.

The rating life value used to determine the bearing colors depends on which bearing calculation method has been selected, and whether the user has requested the Modified rating life to be calculated.

	Nominal service time (basic rating life) requested	Modified rating life requested
Rolling bearings, classic (without contact angle) Rolling bearing, classic (contact angle considered) Rolling bearing stiffness calculated from inner geometry	Lnh	Lnmh
Rolling bearing rating life acc. to ISO/TS 16281	Lnrh	Lnmrh

Table 25.3: Table: Lifetime value used for the bearing colors, based on the calculation settings

25.5.3.5 Display service life in scientific notation

When the rating life of a bearing is longer than 1 million hours it is typically truncated to 1 million hours. However, in some cases it is also useful to know the non-truncated rating life. This result can be obtained by activating this option.

25.5.3.6 Housing surface roughness

The housing's surface roughness value is used to calculate the nominal operating clearance for rolling bearings. The pressure is calculated for a housing with an infinitely large external diameter.

Additional shafts can be defined here if different roughnesses are required for different bearings or if the external diameter must be defined.

25.5.3.7 Bearing manufacturer

Only bearings made by the selected bearing manufacturers are available in the selection list.

25.5.4 Reliability tab

25.5.4.1 Calculation method for reliability

You can select one of the following methods for calculating reliability: **No calculation**, **Bertsche**, **AGMA 6006** or **VDMA 23904**.

A required system reliability can be predefined for each of the three calculation methods. The Weibull shape parameter and a vector for failure-free time can also be defined.

Use the information in AGMA and VDMA for rollers and ball bearings for these definitions. The shaft reliability can also be calculated if the method according to Bertsche is selected.

When calculating shaft reliability, the service life per cross section is determined by performing a strength calculation. A reliability per cross section can then be calculated

This reliability is used to derive a reliability for the individual shafts and for the system. These different reliabilities (bearing, cross sections, shafts and system)

can then be displayed as graphics.

25.5.5 Shaft editor and 3D view tab

25.5.5.1 Show coordinate system

This option displays/hides the coordinate system in the Shaft editor.

25.5.5.2 Show automatic dimensioning

This option displays/hides the dimension lines in the Shaft editor.

26 Calculating Shafts

Once you have finished defining the shafts, either click the Σ button in the tool bar or press **F5** to calculate all the shaft-specific values. The results are then made available as graphics and tables, and in different reports.

26.1 Deflection and bearing forces, distribution of force and torque

The stress, displacement and tilting calculation are based on the Finite Difference Method (FDM).

You can use this calculation to:

- Calculate the diagrams of bending, course of transverse force and torque diagram in the XY and ZY plane (the shaft rotational axis is always Y), with or without taking the dead weight into account.
- Calculate the axial force taking into account the weight (depending on the length of the shaft).
- Display all essential values as graphics: course of deflection, shearing force, bending moment in different levels, torsional moment and static equivalent stress (VM).
- Calculate the forces and moments in bearings (and ends of shafts) for an unlimited number and any type of bearing.
- The utilization and damage of a rolling bearing is calculated as follows:

  $$Ausnutzung = \frac{P}{P_{req}} = \left(\frac{L_{req}}{L}\right)^{1/k}$$

  $$Schädigung = \frac{L_{req}}{L}$$

  The utilization of a rolling bearing is calculated as follows: in this case *L*req is the required rolling bearing service life, *P*ref is the equivalent load which corresponds to *L*req, *L* is the achieved service life and *k* is a coefficient that depends on the type of rolling bearing (*k = 3* for ball bearings, *k = 10/3* for roller bearings).
- Bearing clearance is always considered. If the bearing calculation method according to inner geometry is selected, then the bearing stiffness at the operating point and the static safety are also reported. 2 static safeties - S0r and S0r - are calculated.
  S0w is calculated as

  $$S_{0_w} = (p_0 / p_{\max})^n$$

where p_{max} is the maximum Hertzian pressure on the ring race. For ball bearings p_0 = 4200 N/mm2 and n = 3. For roller bearings p_0 = 4000 N/mm2 and n = 2.

S0r is calculated with the following formula

$$S_{0r} = C_0 / P_{0r}$$

where C0 is the basic static load rating of the bearing, and P0r is the equivalent nominal load (i.e. tilting moments are ignored), which causes the same maximum surface pressure. The same calculations are available for standalone bearing calculations with internal geometry.

- The relative displacement and torsion of the inner ring to the outer ring is calculated and recorded.

Note:
the calculation assumes that the inner ring of the bearing is connected to the shaft. If a hollow shaft is connected to the inside of a rolling bearing, the bearing displacement and rotation are documented with the reversed prefix operator.

- Calculate the inclination of the diagrams of bending in bearings, e.g. when calculating cylindrical roller bearings. The progression of the angle of inclination can also be displayed on screen and printed out.
- The diagrams of bending can be calculated with or without taking shear deformation into account.

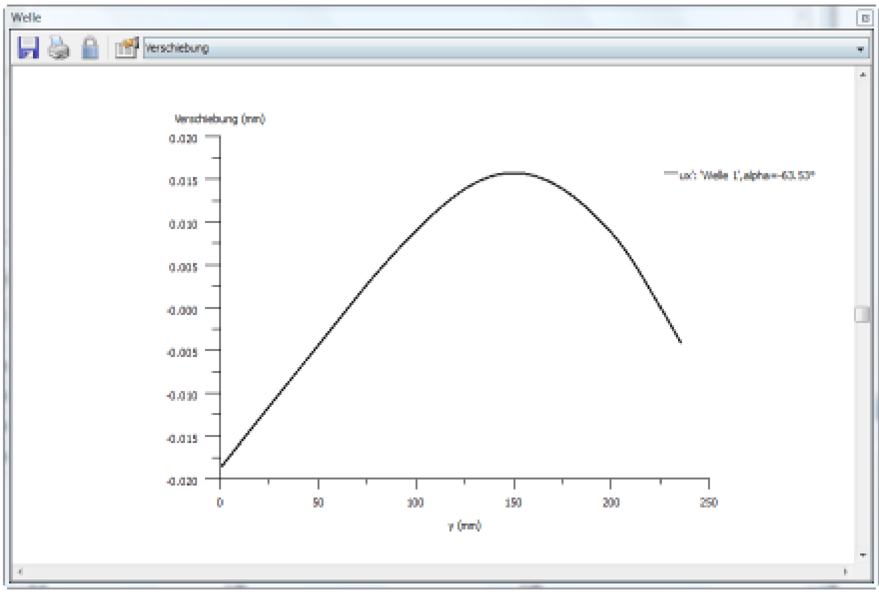


Figure 26.1: Displacement graphic displaying the bending lines in the plane α = 63.53°

► **Note**

Although the data about equivalent stress gives an initial indication of the static strength of a shaft, it cannot be used to calculate infinite life strength. To do this, you must perform the actual strength calculation. However, this equivalent stress data is useful for beams, because the load they are subjected to is usually only a static load. If the moment of resistance in torsion has not been defined for beams, torsional stress is not included in the equivalent stress calculation. Despite this, you can still perform the calculation.

26.1.1 Calculating force on bearings with a contact angle

Bearings with a contact angle must be handled as a special case when calculating shafts and bearings. The effective active bearing center used to calculate the bearing reactions is determined at the point at which the compression force line of action intersects with the shaft centerline. In the rolling bearing catalogs, this is described as the axial force resulting from the oblique position of the bearing housing. You can use this to define the data (radial and axial loads) required to calculate the rating life.

It is more difficult to calculate the load progression in the shaft, and this is also not documented clearly in the technical literature. Here, two modeling types are possible:

- **Approach 1:** In bearings that have a contact angle, the effective bearing force line of action passes through the pressure center point. For this reason, you can calculate the bearing forces because, for calculation purposes, the bearing can be considered as being at the pressure center point. This matches the procedures used to define the rolling bearing load.
- **Approach 2:** However, you cannot introduce the bearing force on the shaft outside the bearing width. This is why KISSsoft places the bearing force in the center of the bearing. At the same time, the eccentric load application creates an additional bending moment which equals the product of the distance of the bearing- and pressure center point, times the radial force. **This is also the approach used in KISSsoft.**

Both approaches return the same progression of bending moment between the pressure centers. There is, however, a difference in the area of the pressure/bearing centers.

In real life, the load is not necessarily applied to the center of the bearing but to the entire area of the bearing. Therefore, the bending moment can be placed precisely on the shaft shoulder. However, this then causes a problem in the strength calculation if the load application acts directly on the proof point (i. e. when the proof point lies between the bearing center and the shaft shoulder).

The calculation of the diagrams of bending produces a difference in that, in approach 1, the deflection is zero in the pressure center and, in approach 2, it is at the bearing position. Here, approach 2 is certainly more precise, especially when large contact angles, where the pressure center lies outside the bearing width, are involved. Only approach 2 enables the calculation to include cases in which the pressure center point lies outside the shaft.

As often happens in such cases, the reality lies somewhere between the two approaches. More precise calculations can only be performed using time-consuming FEM calculations which take into account the properties of the bearing housing.

Approach 2 is more precise and convenient for shaft calculations (because it allows for pressure center points being outside the shaft), which is why this variant has been included in KISSsoft shaft calculation functions.

Notes about the strength calculation: Any strength verification based on the nominal stress concept (DIN 743, FKM, etc.), has limited validity, in the force application zone (e. g. internal bearing ring on the shaft shoulder) when the local stress distribution does not correspond to the estimated nominal stress. In practice, the results calculated on these points must be interpreted with caution.

In KISSsoft, the additional internal axial force that is present in the case of bearings with a contact angle is calculated as Fr * 0.5/Y, as described in "Die Wälzlagerpraxis" and different bearing product catalogs. (FAG as here, NSK with a factor value of 0.6 instead of 0.5, SKF for tapered roller bearings, as here, and for angular contact ball bearings with a factor value of 1.14 (Catalog 2004 as a function of Fa/C)). If factor Y is not present in the bearing database, no additional axial force is taken into consideration. Consequently, the calculation process is the same as the KISSsoft bearing calculation.

26.2 Eigenfrequencies

Click on **Graphic > Shaft > Eigenfrequencies** to display the results of the eigenfrequencies calculation for the modeled shaft system. The calculation is based on a one-dimensional Finite Element Method (FEM) which takes into account the support type and its stiffnesses.

The calculation enables you to:

- calculation of any number of eigenfrequencies
- display natural modes
- takes into account the gyroscopic effect large spinning masses. The critical speeds (bending mode) are calculated for forwards and backwards whirl. During synchronous forwards whirl, an unbalance increases the bending oscillations, because the angular speed of the rotating shaft and angular speed of the shaft's peripheral center point are the same. However, synchronous backwards whirl is, in most cases, not technically important.
- For beam profiles, the critical bending mode (eigenfrequencies) is calculated in both main planes.
- Gears can automatically be handled like masses. In this situation, KISSsoft takes into account the mass and the moments of inertia of the gear located on the shaft.

26.2.1 Bending critical speed

The calculation of critical speeds takes into account any masses located on the shaft. However, applied forces have no effect on the calculation. For this reason, additional masses must be modeled as masses and not as loading forces.

The nodal points of the bending eigenmodes (vibration on plane x-z) are also documented in the report: select "Report" > "Nodal points".

26.2.2 Torsion critical speed

- Calculation of the critical rotating eigenfrequencies of shafts.
- Calculation of any number of eigenfrequencies.
- Graphical display of natural oscillation.

26.3 Buckling

Use this function to calculate the buckling load for shafts and beams. All boundary conditions, bearings and effective axial forces (point or line loads) are taken into account in the calculation. Only the axial forces you specify are used to calculate the buckling load. This function calculates the factor by which all these forces have to be multiplied to create a situation under which buckling occurs. This factor therefore corresponds to the safety against buckling.

26.4 Rough sizing of shafts

The rough sizing of shafts is based on equivalent stress. A number of options affect the behavior of this functionality:

1. **Equivalent stress**: the maximum equivalent stress to which the shaft material is subjected.
2. **Change only cylinder diameters:** If this option is selected, the length of the cylinders that form the outer contour is retained and only their diameter is changed. Otherwise, KISSsoft sets both the length and diameter of the cylinders. In this case, the inner contour is deleted.
3. **Do not delete cross sections A-A etc.:** If this option is selected, the user-defined cross sections for the strength calculation (A-A, B-B etc.) are deleted, and KISSsoft attempts to find the most critical cross sections in the new design.
4. **Consider bearings in sizing:** If this option is selected, rolling bearings are sized according to their required rating life.

5. Match shaft diameter to bearing bore: If this option is selected, KISSsoft rounds up the final cylinder diameter to match the bearing's internal diameter.
6. **Take bearing type from model:** If this option is selected, existing bearings are retained. Otherwise, you can replace the bearings in the model with a specific bearing type as required.
7. **Move bearing if needed:** When a bearing is being sized, it may happen that a larger, wider bearing is selected, and this covers the neighboring cylinder. If this option is selected, the bearing is moved so that it does not cover the cylinder.

Once the calculation has finished, the old shaft contour is displayed so you can compare the old and new data more easily.

26.5 Strength

To enter values for the strength calculation, click on the **Strength** tab in the **Shaft calculation** module user interface.

In KISSsoft, you can use these methods to calculate shaft and axle strength:

- Hänchen & Decker
- DIN 743:2012-12
  Load capacity of shafts and axes [63] including FVA proposed update concerning fatigue strength and tensile strength []
- FKM Guideline (2020)
  Analytical strength verification of steel, cast iron and aluminum materials in mechanical engineering, 7th Edition 2020
- AGMA 6101-F19/AGMA 6001-F19
  Design and Selection of Components for Enclosed Gear Drives
- No strength calculation
  In this case, strength verification is not performed. However, all the other results (diagrams of bending, equilibrium of forces, bearing reactions etc.) will still be calculated.

A static proof and proof of fatigue strength can be applied in each case. The proof according to FKM, DIN and AGMA can also be performed using a load spectrum.

Some of the shaft-specific data for the strength calculation can be defined for a particular shaft in the Element editor.

26.5.1 Calculation method

You can select the calculation method you prefer for strength verification from this list.

26.5.1.1 Hänchen & Decker

Strength calculation according to R. Hänchen and H. K. Decker [64] is an old, but well established method. If insufficient notch factor data is present, the equations produced by TÜV in Munich, Germany, are used: they are derived from known test results.

Material values

As shown in Figures 52, 56, 60 in accordance with [64] for construction, heat treated and case hardened steels. The empirical formula used is in accordance with Hänchen [64], page 37

$$\sigma_{bw} = 0.4 \cdot \sigma_b$$

You can input the material data in the database (see chapter 9, Database Tool and External Tables).

Calculation of equivalent stress

In the case of bending and torsion, KISSsoft calculates the equivalent stress value σ_V according to the hypothesis of the largest distortion energy (see [64], section 3.2.5.).

Calculation of safety against fatigue failure

- Maximum load according to [64], Equation (4a). Operating factor as defined in [64] Table 1 (page 24).
- Design bending fatigue limit under reversed bending according to [64], Equation (42a).
- Safety against fatigue fracture according to [64], Equation (46).
- Required safety against fatigue failure according to [64], Figure 156, depending on the frequency of the maximum load.
- The result of the calculation is the ratio of the required safety to the calculated safety, expressed as a percentage.

Important formulae

A)= Equivalent stress (infinite life strength)

$\sigma_v = \sqrt{\sigma_b{}^2 + 3 \cdot (\alpha_0 \cdot \tau_t)^2}$	(25.1)

$\sigma_b = \frac{M_b}{W_b}$	(25.2)
$\tau_t = \frac{M_t}{W_t}$	(25.3)

A1) Equivalent stress (strength against overload failure and deformation) (τ_t = 0)

$\sigma_v = \sqrt{(\sigma_z + \sigma_b)^2 + 3 \cdot (\alpha_0 \cdot \tau_q)^2}$	(25.4)
$\sigma_z = \frac{F_z}{A}$	(25.5)
$\tau_q = \frac{F_q}{A}$	(25.6)

B) Calculation of safety against fatigue failure:

$\sigma_{bWG} = \sigma_{bW} \cdot b_O \cdot \frac{b_d}{b_{kb}}$	(25.7)
$S_D = \frac{\sigma_{bWG}}{(\sigma_v \cdot f)}$	(25.8)

α_0	a.0	Stress ratio factor	
A	A	Cross section area	*(cm3)*
b_d	b.d	Thickness number	
b_{kb}	b.kb	Notch factor (bending)	
b_o	b.o	Surface number	
f	f	Total load factor	
F_q	F.q	Shearing force	*(N)*
F_z	F.z	Tension/Compression force	*(N)*
M_b	M.b	Bending moment	*(Nm)*

M_t	M.t	Torque	*(Nm)*
σ_b	s.b	Bending stress	*(N/mm*2)
σ_{bW}	s.bW	Fatigue strength under reversed bending stresses	*(N/mm*2)
σ_{bWG}	s.bWG	Deformation strength under reversed bending stresses	*(N/mm*2)
σ_v	s.v	Equivalent stress	*(N/mm*2)
S_D	S.D	Safety against fatigue failure	
τ_q	t.q	Shear stress (force)	*(N/mm*2)
τ_t	t.t	Torsional stress	*(N/mm*2)
W_b	W.b	Axial moment of resistance	*(cm*3)
W_t	W.t	Polar moment of resistance	*(cm*3)

Stress ratio factor

Values for the stress ratio factor are displayed in the next table (see Table 26.1).

Bending	alternating	alternating	static	static	static	static
Torsion	pulsating	alternating	pulsating	alternating	static	static
Structural steel	0.7	0.88	1.45	1.6	1.0	1.0
Case hardening steel	0.77	0.96	1.14	1.6	1.0	1.0
Through hardening steel	0.63	0.79	1.00	1.6	1.0	1.0

Table 26.1: Stress ratio factor α0 according to Hänchen, p. 28 [64] or Niemann, I, p. 76 [8]

26.5.1.2 DIN 743 (2012)

The German DIN 743 standard [63] uses the most up to date information to calculate shafts and includes the following points:

- Consistent distinction between the different load classifications (tension/compression, bending, torsion) and between mean stress and stress amplitude.
- Surface factor: The influence on the strength is documented when using thermal methods (nitriding, case hardening) and mechanical processes (shot peening, rolling).

- Notch factors: Data for construction elements other than the usual notch factors is mentioned in all the specialist literature. This data, such as relief grooves, interference fit with relief groove or square notches (recesses for snap rings) is widely used nowadays but has only been poorly documented until now. All notch factors are documented for tension/compression, for bending and for torsion.
- Materials: it includes an extensive list of materials, and also instructions about how to derive estimated values for undocumented steels.
- Limited life: the calculation of load strength according to the "Miner extended" method is described in Part 4 of the standard.

The critical limitations of the DIN 743 standard are:

- Shearing load (shearing forces) is not included. This is not a disadvantage except for shafts with a very short distance between bearings.
- It only applies to steels and operating temperatures between -40oC and +150oC.
- As defined in the standard, the minimum safety margins for deformation and fatigue failure are defined as stated in 1.2. However, these safety factors only cover the lack of precision in the calculation method, and do not cover the problems encountered in load assumptions or the consequences if the material fails. The required safety factors must therefore be checked or agreed by both the customer and contractor.

26.5.1.3 FKM Guideline, 2020 Edition

The FKM guideline (FKM: Forschungskuratorium Maschinenbau e.V., Frankfurt (Board of Research in Mechanical Engineering)) is based on the former GDR standards and includes the latest knowledge on workshop theory. It will probably form the basis of a new VDI guideline. The FKM Guideline is long (running to approximately 175 pages plus 400 pages of commentaries), and includes not only the classic range of endurance limit calculation, but also fatigue strength calculations and rating life calculations, taking into account load spectra. It also provides calculation approaches for special problems such as operating temperatures above 100°C.

If the user selects **Define data for each load case individually** from the list and defines a stress ratio for each load spectrum, the proof of fatigue strength can be performed as an equivalent stress verification, according to the FKM guideline. This proof performs the same function as the amplitude proof which is normally performed. To take into account the effect of different stress ratios on the load side of the proof, the load spectrum is converted to the mean stress Sm = 0 and the stress ratio R = -1. This enables a strength calculation to be performed for a shaft, according to the FKM guideline, with the effect of a Rainflow matrix load spectrum.

The calculation is performed according to the 7th edition (2020) of the FKM Guideline, using the solutions proposed by Haibach.

Fatigue strength

The service strength coefficient $K_{BK,S}$ is determined according to chapter 2.4 of the guideline. The number of cycles at knee point N_D is 106.

$K_{BK,S}$ is greater than 1.0 if the number of load cycles is less than N_D. Above N_D, $K_{BK,S}$ usually equals 1.0.

Normal calculations with a given load (without load spectrum) are referred to as an "individual load". This is calculated in accordance with section 2.4 of the guideline. Three different processes can be used for load spectra. See (see chapter 26.5.2, Type of calculation).

26.5.1.4 AGMA 6101-F19/AGMA 6001-F19

AGMA 6101-F19/ 6001-F19 [63] describes how to calculate a closed gear unit. Calculations are described for shafts, interference fits, keys, bearings, housings and bolts in this AGMA standard.

- It distinguishes between the different load classifications (tension/compression, bending, torsion and shearing) and between mean stress and stress amplitude.
- Notch factors: the few notch factors given here are only for bending. The same ones are used for the other loads.
- Materials: it includes an extensive list of materials, and also instructions about how to derive estimated values for undocumented steels. The permitted values are converted from the core hardness value entered by the user.
- In KISSsoft, load spectra are not taken into consideration when the AGMA method is applied (as it is not described adequately).

The critical limitations of the AGMA standard are:

- Only for cylindrical steel shafts, but could maybe also be used for other materials.
- The only notch types defined in detail are shoulder, circumferential groove and cross hole.
- According to the standard, the set minimum safeties against peak load and fatigue are 1.0. However, these safety factors only cover the lack of precision in the calculation method, and do not cover the problems encountered in load assumptions or the consequences if the material fails. The required safety factors must therefore be checked or agreed by both the customer and contractor.

26.5.2 Type of calculation

You can perform a safety analysis using one of these four different methods:

- **Static**. Calculates the safety against yield safety.

- **Infinite life strength**. Calculates the safety against the infinite life strength (in the horizontal section of the S-N curve (Woehler lines), no load spectrum used)
- **Limited life**. Calculates the safety against fatigue for a given number of cycles. Here, a constant load is used (no load spectra). The calculation method according to AGMA only defines a one-step load spectrum. A one-step load spectrum is handled separately in the FKM method. According to the methods defined in DIN 743 and FKM, the S-N curve (Woehler lines) runs horizontally after it reaches the number of load cycles limit ND (10^6).
- **Miner consequent/elementary/extended**. These methods differ in the way they calculate the inclination of the S-N curve (Woehler lines) above the number of breakpoint cycles.

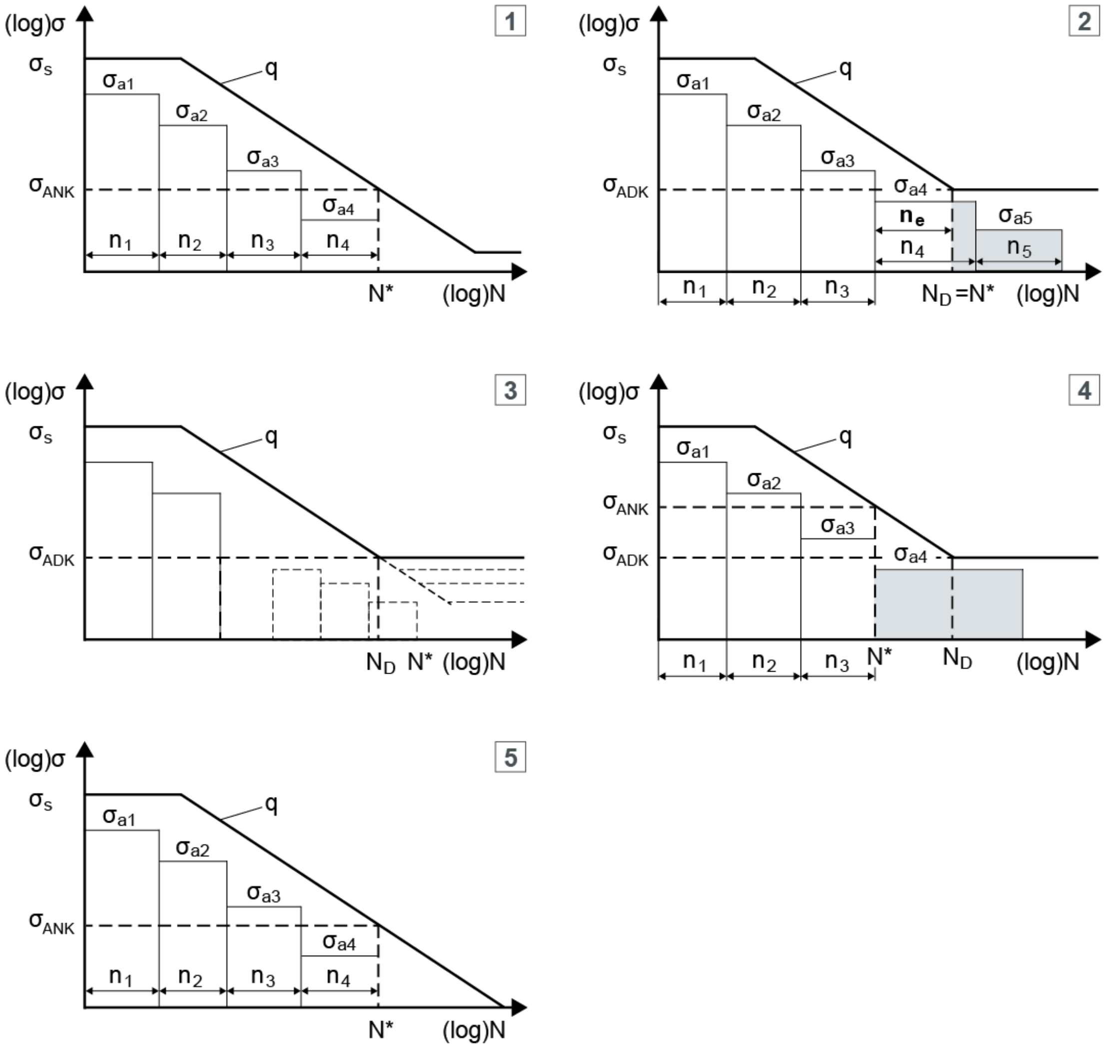


Figure 26.2: Miner hypotheses

- Legend:

- 1) Miner elementary following FKM
- 2) Miner extended in acc. with DIN 743-4:2012
- 3) Miner consequent following FKM guideline
- 4) Miner original according to Haibach
- 5) Miner elementary according to Haibach
- The gray fields are the fractions that are ignored.

► **Note**

The calculation methods according to Miner are only available if you have selected the **Consider load spectra** option in the **Load spectra** drop-down list in the **Basic data** input window. Load spectra (see chapter 15.2.8, Define load spectrum) can be defined in the KISSSOFT database tool. You then only need to select them in the calculation.

26.5.3 Rating life

The required rating life in number of revolutions is calculated from the required rating life in hours.

26.5.4 Strength parameters according to Hänchen and Decker

26.5.4.1 Frequency of load

This value refers to the load value you entered previously (such as torque). If a load applies to the whole rating life of the shaft, the frequency is 100%, otherwise it is correspondingly lower.

26.5.4.2 Notch factors

- Thickness number: according to [64], Image 120.
- Surface number: as stated in [64], Figure 119, Definition of the associated machining process in [64], Table 4.
- The following curves have been pre-programmed:

	Coarsely cut out	Curve with b_o = 0, 50 at 150 kp/mm2
	Milled/finely turned	Curve with b_o = 0, 70 at 150 kp/mm2
	Ground	Curve with b_o = 0, 94 at 150 kp/mm2
	Polished	Curve with b_o = 0, 97 at 150 kp/mm2

- Shoulder notch effect coefficient during bending according to [64], Figure 131.
- Wheel seat with key: proposed values after consulting with TÜV, Munich. Only very few details given in [64], section 6.4.

- Interference fit: Proposed values after consulting with TÜV, Munich. Details given in [64], section 6.4.
- Bearings are handled as weak interference fits. Only very few details given in [64], section 6.4.
- Shaft-hub connections (multi-wedge toothing): Stress concentration factor and section modulus according to [64], section 8.5. Conversion of the stress concentration factor into the notch effect coefficient according to [64], section 5.6, Formula (36) and (37b) or (37c) with the radius for the substituting notch according to [64], Figure 112.
- Thread: Diameter quotient according to [64], Figure 123. Conversion into notch effect coefficient as shown above.

26.5.4.3 Safety against deformation/fracture

KISSsoft calculates the required safety against fatigue fracture, depending on the frequency of the maximum load, using Hänchen's definitions. If the frequency is 100%, the specified safety is 2.0. At 0%, it is 1.0. However, the safety does not follow a linear progression between these two extremes.

The required safety against overload failure is 3.5 to 5.0, depending on the type of application or guideline involved. The required safety against deformation (yield point) is usually 2.0 to 3.5.

26.5.5 Strength parameters according to FKM

26.5.5.1 Temperature duration

The FKM guideline takes into account thermal creep in various materials. Constant high temperatures will reduce the shaft's strength and therefore also reduce its safeties.

Part temperatures in the range from -40°C - +500°C are taken into consideration according to the FKM Guideline. For temperatures above 100°C (or above 60 degrees C, for fine grain steels), temperature factors (for the tensile strength, yield point, and resistance to change) are used to take the reduction in strength into account.

► **Note**

You can define the shaft temperature in the **Element Editor**. To do this, click on the shaft you require, in the **Element Tree**, and then enter the appropriate value in the **Temperature** field.

26.5.5.2 Protective layer thickness

This input field is where you define the thickness generated as a result of electro galvanization, hot-dip galvanizing or zinc flake coating, for steel, cast iron, or the aluminum oxide layer, for aluminum materials.

26.5.5.3 Enter safeties

If this option is selected, you can enter the required safeties on the right-hand side. Alternatively, click the Plus button to display the **Define safeties** dialog window in which you can specify safeties as defined in FKM.

The safety factors for the static strength calculation, j_m (for overload failure) and j_p (for deformation), are determined in accordance with section 1.5 of the guideline, and the safety factor for fatigue resistance, j_D, is determined in accordance with Part 2.5 of the guideline. You will find detailed comments in the Guideline.

Steel		j_m = 2.0	j_p = 1.5	j_F = 1.5	j_F = 1.5
GS, GJS	-not checked	j_m = 2.8	j_p = 2.1	$j_G * j_F$ = 2.6	$j_G * j_F$ = 2.6
	-non-destruction-tested	j_m = 2.5	j_p = 1.9	$j_G * j_F$ = 2.4	$j_G * j_F$ = 2.4
GJL, GJM	-not checked	j_m = 3.3	j_p = 2.6	$j_G * j_F$ = 3.1	$j_G * j_F$ = 3.1
	-non-destruction-tested	j_m = 3.0	j_p = 2.4	$j_G * j_F$ = 2.9	$j_G * j_F$ = 2.9

j_m, j_p: The values apply for	-	severe damage as the result of failure
	-	high probability of load occurrence

If only minor damage results from the fracture, the safety factors can be reduced by about 15%. Provided the probability of the same load occurring again is low, the safety factors can be reduced by about 10%.

$j_G * j_F$: The values apply for	- severe damage as the result of failure
	- irregular inspection

If only minor damage results from the fracture, the safety factors can be reduced by about 15%. Provided inspections are carried out regularly, safety factors can be reduced by about 10%.

26.5.5.4 Stress case

The stress case can identify four scenarios for the development of the stress ratio σ_a/σ_m with continued increase in load, starting from the operating point.

26.5.5.5 Estimation of the infinite life strength for surface-treated parts (section 5.5)

This calculation should only be used for surface-treated rolled steel. The surface treatments include the following treatment methods applied to the materials:

- case-hardened
- nitrided, gas-nitrided, nitro-carburated
- induction hardened
- rolling
- shot peening

These types of treatment can be defined either when you input the material, or when you input the surface factor for the shaft in the Element Editor.

This process is based on the concept of a local infinite life strength. Two points on the part are considered. The first point is on the part's surface, and the second point is at the transition point between the hard surface layer and the core. The resulting stresses are converted into main stresses $\sigma 1$ and $\sigma 2$. Only the largest main stress $\sigma 1$ is then used for subsequent calculation.

You can also input a hardening depth for this calculation in the Element Editor. The hardening depth is then used to calculate the distance from the component's surface to the transition point between the hard surface layer and the core.

The Strength tab is where you define whether the constant Kf is to be calculated according to formulae 4.3.2 and 4.3.3 or taken from Table 4.3.1. You also have the option of inputting the core hardness when you specify the material. Alternatively, this can also be estimated from the tensile strength.

This approach is used to calculate the internal stress, which is included in the calculation of mean stress sensitivity. In this case, the degree of utilization for the point on the component's surface is calculated first, followed by the degree of utilization at the transition point between the hard surface layer and the core. The greater of the two degrees of utilization is then used for the proof. Both degrees of utilization should be < 1.

The results are only displayed in the report if this calculation method has been selected for rolled steel with the predefined treatment types.

26.5.6 Strength parameters according to DIN

26.5.6.1 Stress case

The stress case can identify two scenarios for the development of the stress ratio σ_a/σ_m with continued increase in load, starting from the operating point.

26.5.6.2 Calculation with experimental data

Use this option to define a Haigh diagram which has been determined from experimental data. If you input a file name (e.g. WMAT-001.dat) in the **Experimental data** field for module-specific material data as defined in DIN 743, a selection list appears in the **Strength** tab.

- **Do not take into account**: the data is ignored.
- **Use in DIN 743 (KFσ according to DIN 743)**: the data is imported from the file which was defined for the materials under Experimental data, and the KFσ coefficient is defined according to DIN 743.
- **Use in DIN 743 (KFσ=1)**: the data is imported from the file which was defined for the materials under Experimental data, and the KFσ coefficient is always set to 1.

Instructions about how to define the data can be requested from KISSsoft. The measured Haigh diagram is not interpreted exactly as described in DIN 743. The overall influence coefficient divides the Haigh diagram into x- and y-coordinates so that the results are much lower.

The influence of mean stress as defined in DIN 743 increases as the notches become sharper, and should not decrease. This modification ensures that this influence always increases.

If the comparative medium stress is σmv <0, the line of the Wöhler diagram is extended into the negative area of the Haigh diagram, and the curve is then broken in the negative area by compression yield point σDk. For more information, see also the description in DIN 743-1, p. 21.

26.5.6.3 Safety against fatigue/deformation

Enter the required safeties for endurance/yield in these input fields. A warning is displayed if these values are not achieved for one or more of the specified cross sections.

26.5.6.4 Calculation of multiple notches according to DIN 743

The procedure for creating the model in a FE program (modeling guideline) and for defining a notch factor from the stresses (evaluation guideline) is described in FVA research project No. 700 I ‘Berechnung von Mehrfachkerben nach DIN 743 durch Einbindung von FEM-Ergebnissen’.

The methods defined here are limited to notches with component surfaces that are not subject to force. You will still have to use a notch effect coefficient determined by experimentation to calculate critical failure points with fretting fatigue over time in the contact zone (e.g. interference fits or key connections).

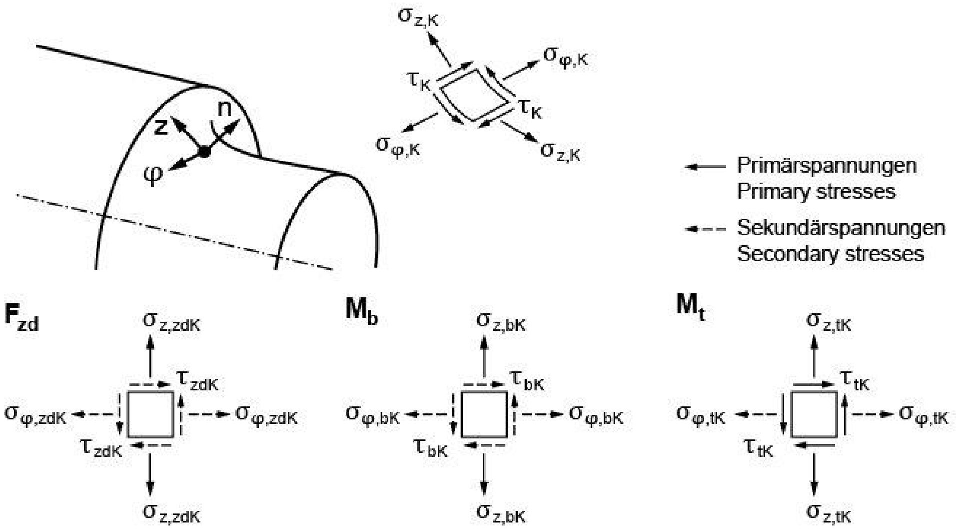


Figure 26.3: Local notch stresses in a general case

There are two different methods described: Method A (a comprehensive calculation method for multiple notches) and Method B (a simplified method involving multiple notches).

KISSsoft does not use Method B.

Method A:

Method A takes into account all the effects that affect strength for multiple notches.

The stresses at the critical proof point are defined at the point where the equivalent stress amplitude is greatest on the component surface. The von Mises stress (VM) is then applied to these values to calculate the equivalent stress amplitude.

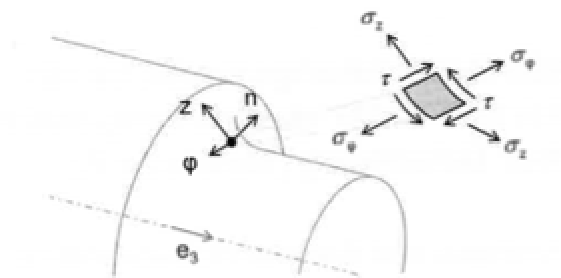


Figure 26.4: Components of local notch stress

The FE analysis usually supplies a spatial stress tensor in the global coordinate system, which still has to be transformed into the plane stress tensor.

The notch factor is defined as a quotient of the local stress peaks at the notch root and of the nominal stress in the notch cross section.

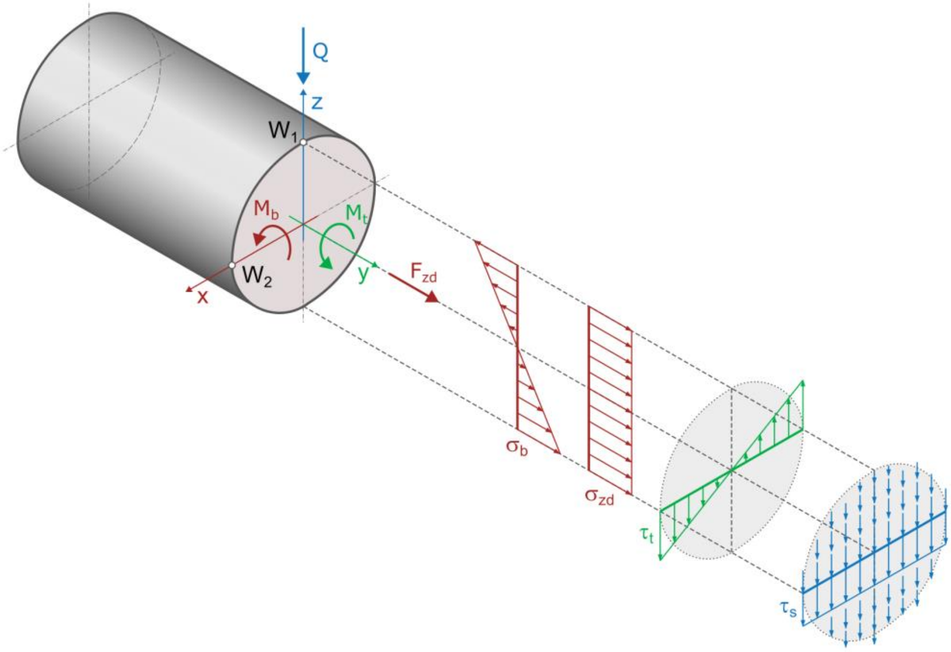


Figure 26.5: Bending, tension/compression, torsion and shearing stresses

To define the notch factor, the local stress components are separated into primary and secondary stresses. A load type's primary stress is the nominal stress associated with the base load case.

You must define the stress gradient before you can determine the theoretical stress concentration factors. This requires the stress values that occur at a neighboring node in the interior of the component.

The notch factor is determined using the equations in the next table (Method A). The notch factor can also have values that are less than 1. The secondary stresses are ignored in Method B.

Load type	Stress concentration factors		
Tension/Compression	$\alpha_{zd\sigma_z} = \dfrac{\sigma_{zaK}}{\sigma_{zdn}}$	$\alpha_{zd\sigma_\varphi} = \dfrac{\sigma_{\varphi aK}}{\sigma_{zdn}}$	$\alpha_{zd\tau} = \dfrac{\tau_{aK}}{\tau_{zdnv}}$
Bending	$\alpha_{b\sigma_z} = \dfrac{\sigma_{zaK}}{\sigma_{bn}}$	$\alpha_{b\sigma_\varphi} = \dfrac{\sigma_{\varphi aK}}{\sigma_{bn}}$	$\alpha_{b\tau} = \dfrac{\tau_{aK}}{\tau_{bnv}}$
Torsion	$\alpha_{t\tau} = \dfrac{\tau_{aK}}{\tau_{tn}}$	$\alpha_{t\sigma_z} = \dfrac{\sigma_{zaK}}{\sigma_{tnv}}$	$\alpha_{t\sigma_\varphi} = \dfrac{\sigma_{\varphi aK}}{\sigma_{tnv}}$
NOTE	$\sigma_{nv} = \sqrt{3} \cdot \tau_n$ und $\tau_{nv} = \sigma_n / \sqrt{3}$		

Figure 26.6: Table 4 - Calculating the notch factor for components (Method A)

The notch effect coefficient β is calculated separately for each component present in the local stress amplitude. The stress gradient required to do this is not defined separately.

Special case: interference fit (mean stress without base load case):

If mean stress without a base load case is included in the calculation as nominal stress, it can be handled in the same way as mean stress. Firstly, the equivalent stress is calculated from the resulting local stresses. Then, the relevant notch factor can be used to determine the nominal mean stress.

The safety verification is then performed component by component, because the notch factor was also evaluated individually for each component.

The component safeties are calculated as follows:

	Tension/Compression	Bending	Torsion
Fatigue limit of part	$\sigma_{z,zdWK} = \dfrac{\sigma_{zdW}(d_B) \cdot K_1(d_{eff})}{K_{zd\sigma_z}}$	$\sigma_{z,bWK} = \dfrac{\sigma_{bW}(d_B) \cdot K_1(d_{eff})}{K_{b\sigma_z}}$	$\sigma_{z,tWK} = \dfrac{\sigma_{bW}(d_B) \cdot K_1(d_{eff})}{K_{t\sigma_z}}$
	$\sigma_{\varphi,zdWK} = \dfrac{\sigma_{zdW}(d_B) \cdot K_1(d_{eff})}{K_{zd\sigma_\varphi}}$	$\sigma_{\varphi,bWK} = \dfrac{\sigma_{bW}(d_B) \cdot K_1(d_{eff})}{K_{b\sigma_\varphi}}$	$\sigma_{\varphi,tWK} = \dfrac{\sigma_{bW}(d_B) \cdot K_1(d_{eff})}{K_{t\sigma_\varphi}}$
	$\tau_{zdWK} = \dfrac{\tau_{tW}(d_B) \cdot K_1(d_{eff})}{K_{zd\tau}}$	$\tau_{bWK} = \dfrac{\tau_{tW}(d_B) \cdot K_1(d_{eff})}{K_{b\tau}}$	$\tau_{tWK} = \dfrac{\tau_{tW}(d_B) \cdot K_1(d_{eff})}{K_{t\tau}}$

$$K_{(i)\sigma_z} = \left(\frac{\beta_{(i)\sigma_z}}{K_2(d)} + \frac{1}{K_{F\sigma}} - 1\right) \cdot \frac{1}{K_V} \quad \text{und} \quad K_{(i)\sigma_\varphi} = \left(\frac{\beta_{(i)\sigma_\varphi}}{K_2(d)} + \frac{1}{K_{F\sigma}} - 1\right) \cdot \frac{1}{K_V}, \quad K_{(i)\tau} = \left(\frac{\beta_{(i)\tau}}{K_2(d)} + \frac{1}{K_{F\tau}} - 1\right) \cdot \frac{1}{K_V}$$

Figure 26.7: Table 5 - Calculating the part's fatigue limit

The mean stress sensitivity is also calculated here. Adjust the mean stress sensitivity of the ranges in DIN 743 to ensure this additional calculation does not return unexpected results. The safety is determined using the formulae for combined load and phase equivalence in the plane stress state.

Method B:

In Method B, the mechanical stress is simplified when determining the notch factor. This method is not suitable for a superimposed dynamic load.

The equivalent stress (VM) is calculated from the primary and secondary stress.

This method can only be used to a limited amount for a combined load. Limits:

σa2/σa1 < 0.2 and τa2/σa1 < 0.2

The notch factor is using the notch stresses for the load cases (tension/compression, bending, or torsion amplitude) from a FE provision analysis.

Load type	Tension/Compression	Bending	Torsion
Stress concentration factor	$\alpha_{zd} = \frac{\sigma_{va}}{\sigma_{zdn}}$	$\alpha_b = \frac{\sigma_{va}}{\sigma_{bn}}$	$\alpha_t = \frac{\sigma_{va}}{\sqrt{3} \cdot \tau_{tn}}$

Figure 26.8: Table: Table 7 - Calculating the notch factor with equivalent stress (Method B)

The supporting effect and notch effect coefficient are defined in the same way as specified in the formulae in DIN 743.

Mean stresses that are not subject to base load (interference fit) are not included in the simplified Method B.

The safety verification is performed in the same way as in DIN 743.

26.5.7 Strength parameters according to AGMA

26.5.7.1 Coefficients used to determine fatigue strength

The following coefficients are required to calculate the modified fatigue strength σ_f:

Surface finish coefficient k_a

You can set the surface finish for each individual element in a list. Coefficient k_a is then determined from the list entry, using the method shown in Figure 4, AGMA 6101-F09.

Size factor k_b

The size factor k_b is determined on the basis of the shaft diameter, using the method shown in Figure 5, AGMA 6101-F09.

Reliability coefficient k_c

The **Strength** tab is where you define the reliability (default 0.99%). This value is used to determine reliability coefficient k_c, using the method shown in Figure 6, AGMA 6101.

Temperature coefficient k_d

The coefficient k_d =1 is set for temperatures between -30°C and 120°C. The entries for the shaft can be modified if the shaft temperature lies outside this range.

Life factor k_e

This coefficient is determined on the basis of stresses σ_e and σ_u and the number of load cycles NL.

Surface processing coefficient k_f

For key ways, this coefficient is taken directly from Table 2 in AGMA 6101-F09.

For straight-side or normal splines, a K_t coefficient of 2.0 is assumed.

If a combination of a key and an interference fit is present, a k_f coefficient of 0.33 to 0.4 is specified in the standard. We recommend to enter the value 0.33.

The notch effect type of the circumferential groove is used for V-notches and threads. It has a fixed ratio of h/r = 20.

The system takes an individual K_t coefficient from the diagrams for the shoulder, circumferential groove and cross hole (Figures 8 to 10, AGMA 6101-F09) and uses them, together with coefficient q (Figure 7, AGMA 6101-F09) to calculate coefficient k_f.

Miscellaneous effects coefficient k_g

This coefficient can be used to take into account, for example, heat treatments, residual stresses (shot peening, cold rolled, etc.), corrosion, and surface coatings.

The default value set for this coefficient is k_g =1, however, it can be overwritten with the values entered for the shaft.

26.5.7.2 Calculating the Safeties

Load spectra are not taken into consideration when the AGMA method is applied.

To calculate the fatigue safety FSf, the average stress and the amplitude are determined using the von Mises stress.

These values, together with the modified fatigue strength and the tensile strength, are required later to determine the fatigue safety.

The total stress is determined using the von Mises stress to calculate peak load safety. The peak load safety coefficient is calculated using the total stress, the tensile strength, a coefficient for the yield point, and a peak load coefficient.

The peak load coefficient can be defined individually in the **Strength** tab for all loads (tension/compression, bending, torsion, thrust). In the KISSsoft system, the default setting for the peak load coefficient is 1.0. As specified in the standard, the setting for this coefficient is 2.0 for cylindrical gears with straight and helical flanks, bevel gears, and double helical gearings. For worm gears, the setting for this coefficient is 3.0. The following rules apply to multi-level gear units with a cylindrical or globoid worm gear stage that are connected to a non-worm gear stage (cylindrical gear-, angled-, or bevel gear stage):

- Fp = 2.0, if the stages are to withstand 200% of the current load
- Fp = 3.0, if the stages are to withstand 300% of the current load

The yield point coefficient can be also defined in the **Strength** tab. In the KISSsoft system, the default setting for the yield point coefficient is 0.75. (As specified in the standard, it is 0.66 to 0.8).

According to the standard, the resultant safety factors for fatigue FSf and peak load FSp must be >= 1.

26.5.8 Stress

This is where you define how the stresses calculated by KISSsoft (e.g. the bending moment) are to be converted into mean stresses and stress amplitude. You can select usual loads (alternating, pulsating, static load) from the list. For special cases, open the **Stress** selection list and select the **Own Input** option. Then, enter a suitable value in the **Stress ratio** field (see chapter 26.5.9, Stress ratio). Rotating shafts normally have alternating bending and pulsating or static torsion.

26.5.8.1 Selection list data for stress ratios and load factors

If a calculation is to be performed for several shafts, a selection list with these options is displayed in the **Strength** tab when you define the stress ratio:

- Details for all shafts and cross sections (definitions in the **Strength** tab)
- Enter details for all shafts separately (definition in the element editor for the relevant shaft)

- Enter details for all cross sections separately (definition in the element editor for the relevant cross section (see chapter 26.5.8.2, Define data for each cross section individually))
- Define data for each load case individually (definition in load spectrum, the stress ratios are specified in each load case (see chapter 26.5.8.3, Define data for each load case individually))

If you select the **Enter details for all shafts separately** option, you can define the stress ratios and load factors for each shaft.

26.5.8.2 Define data for each cross section individually

When calculating a shaft, you can enable this option by selecting this list item.

Use this option to define the stress ratios and load factors for every individual cross section. This is a useful option if a shaft is subjected to a special load. In other words, if an eccentric force occurs and this force either remains static while the shaft rotates or rotates along with the shaft.

You cannot use the entry in the shaft editor to define this. However, this option does enable you to perform a strength calculation for the relevant cross section. Example applications are radial piston pumps, washing machines, centrifuges, vibrator units, etc.

26.5.8.3 Define data for each load case individually

When calculating a shaft with a load spectrum, you can enable this option by selecting this list item.

Use this option to define the stress ratios for each individual load spectrum case in the load spectrum definition.

This data can be used to perform a proof of fatigue strength as an equivalent stress verification, according to the FKM guideline. This proof performs the same function as the amplitude proof which has normally been performed up to now. In contrast to this, different stress ratios can be defined for each load case in an equivalent stress verification.

To take into account the effect of different stress ratios on the load side of the proof, the load spectrum is converted to the mean stress Sm = 0 and the stress ratio R = -1. This enables a strength calculation to be performed for a shaft, according to the FKM guideline, with the effect of a Rainflow matrix load spectrum.

26.5.9 Stress ratio

You must also enter the stress ratio, because KISSsoft requires this value to split the load on the corresponding cross-section into mean load and load amplitude.

Maximum stress per load cycle:		σ_o
Minimum stress per load cycle:		σ_u
Stress ratio		$R = \sigma_u/\sigma_o$
Mean stress:	σ_m	$= (\sigma_o + \sigma_u)/2$
		$= (\sigma_o + R \cdot \sigma_o)/2$
		$= \sigma_o \cdot (1 + R)/2$
Stress amplitude:	σ_a	$= (\sigma_o - \sigma_u)/2$
		$= (\sigma_o - R \cdot \sigma_o)/2$
		$= \sigma_o \cdot (1 - R)/2$

For:

Pure alternating stress	$(\sigma_u = - \sigma_o)$	$R = - 1$
Pulsating stress	$(\sigma_u = 0)$	$R = 0$
Static stress	$(\sigma_u = \sigma_o)$	$R = 1$

Normally valid for rotating shafts or axes:

Bending and shearing force:	$R = -1$
Torsion and tension/compression:	$R = 0$ (ev. R = 0 to 1)

► **Note**

In contrast to the calculation according to DIN or FKM, where there is a clear differentiation between the mean stress and amplitude stress, when a strength calculation in accordance with Hänchen (see chapter 26.5.1.1, Hänchen & Decker) is performed, the loads that are entered are converted into a comparative stress that is then compared with the fatigue limit for bending. For this reason, if you select this method, the stress ratio only affects the value of the stress ratio factor α_0.

26.5.10 Load factor for static analysis

The static calculation normally uses the greatest possible load. The maximum load factor covers the difference between the load value you specified and the peak value.

Maximum stress: $\sigma_{max} = \sigma_o \cdot f_{max}$

You can specify individual factors for every type of stress (bending, tension/compression, etc.).

The load factor is not used if the forces or power ratings are specified in free cross sections.

► **Example**

Electric motor with a permanent torque 100 Nm, starting torque 170 Nm. When you specify the shaft data, enter 100 Nm and set the maximum load factor to 1.7.

26.5.11 Load factor endurance calculation

If necessary, the mean stresses and the stress amplitudes can be multiplied by a load factor. As the DIN 743 standard does not include this factor, you should generally predefine it as 1.0. Using a factor > 1 is a good idea if you specify the nominal torque in a shaft calculation without taking into account the increases in torque due to the vibrations caused when the shaft rotates.

The load factor is not used if the forces or power ratings are specified in free cross sections.

The calculation according to Hänchen includes the following information:

The total load factor f according to Hänchen [64], p. 24:

$f = f_{un} \cdot f_{betr} \cdot f_{leb}$	(25.9)

f_{un}	Uncertainty in maximum load (1.0 or 1.2 to 1.4)
f_{betr}	Operational approach (shocks) (1.0 to 3.0)
f_{leb}	Importance of part (1.0 or 1.2 to 1.5)

► **Note:**

The Hänchen method uses only one load factor, which is the larger of the two values entered for bending and torsion.

26.5.12 Cross sections

The yield safety and safety for fatigue fracture are determined at specific cross sections along a shaft that are defined by you. In the Element Tree, you will see **Cross sections** at group level. In the context menu, you can either add a **Free cross section** or a **Limited cross section**.

26.5.12.1 Surface roughness

If you enter a value for surface roughness as defined in ISO 1302, the corresponding surface roughness, R_Z , is displayed in the selection list. This value, R_Z , is then used in the calculation. In the calculation according to DIN or FKM, surface roughness has already been included in the notch factor in some cases. In these cases, the surface factor is always 1.0, no matter what value you input as the surface roughness.

26.5.13 Sizing

You can select the **Sizing** option in the context menu for the **Cross section** entry in the Element Tree, to make it easier for you to define the cross sections that need to be recalculated.

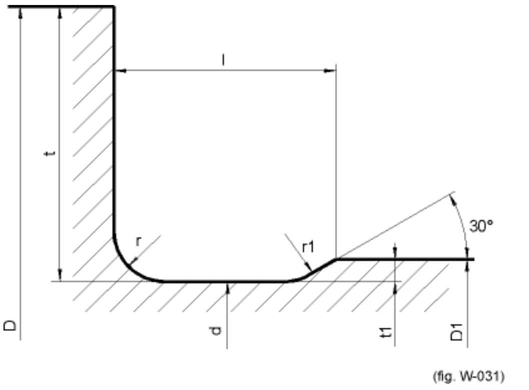
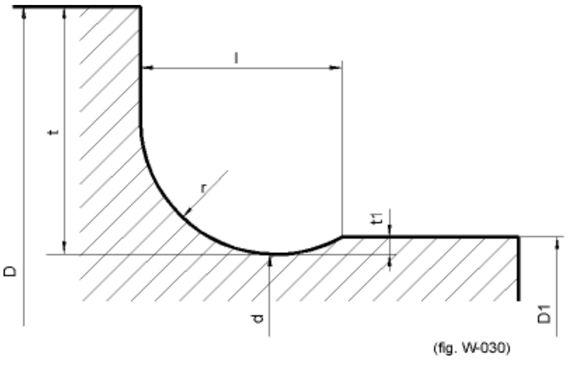
In this sizing process, KISSsoft automatically finds cross sections (shaft shoulders, interference fits in bearings, key-grooves and other notch effects) which have been defined in the **graphical shaft input**, and in which a notch effect occurs. It displays the cross sections that have the lowest safeties. You must check these cross sections carefully.

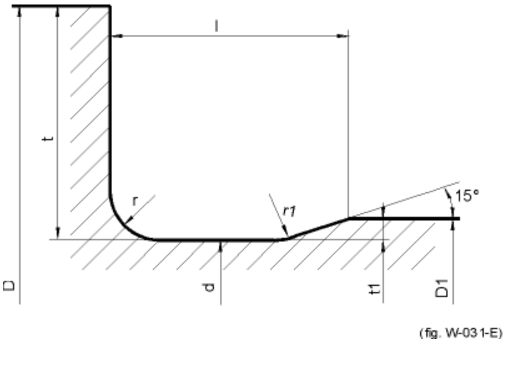
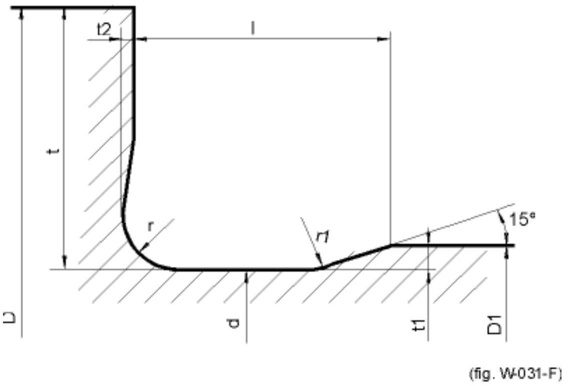
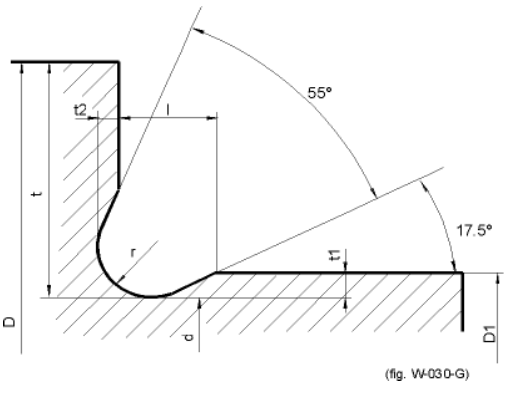
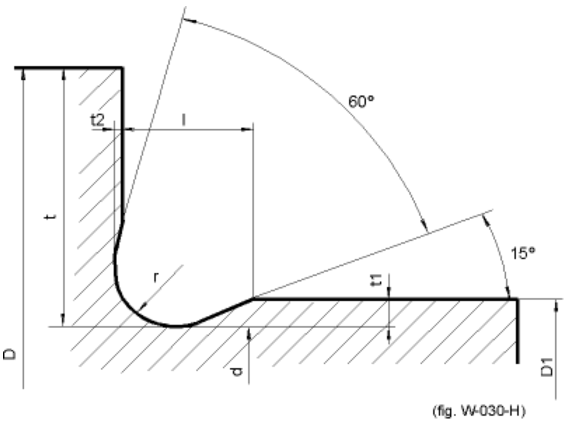
▶ **Note**

It is essential that you check the model for other notch effects that KISSsoft might not be able to find, for example threads or cross holes.

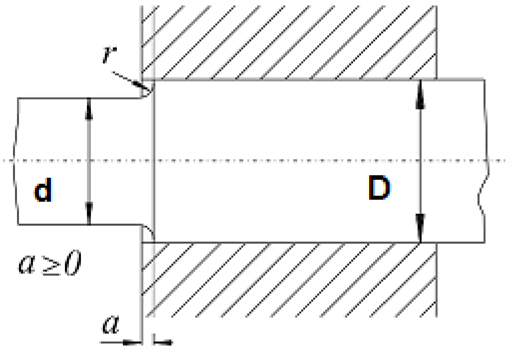
26.5.14 Cross-section types

- **Shoulder**
- **Shoulder with relief groove**

	
FKM Shape B	FKM Shape D

	
DIN 509 Shape E	DIN 509 Shape F
According to FKM, these shapes are handled like shape B.	
	
DIN 509 Shape G	DIN 509 Shape H
According to FKM, these shapes are handled like shape D.	

- **Shoulder with interference fit**

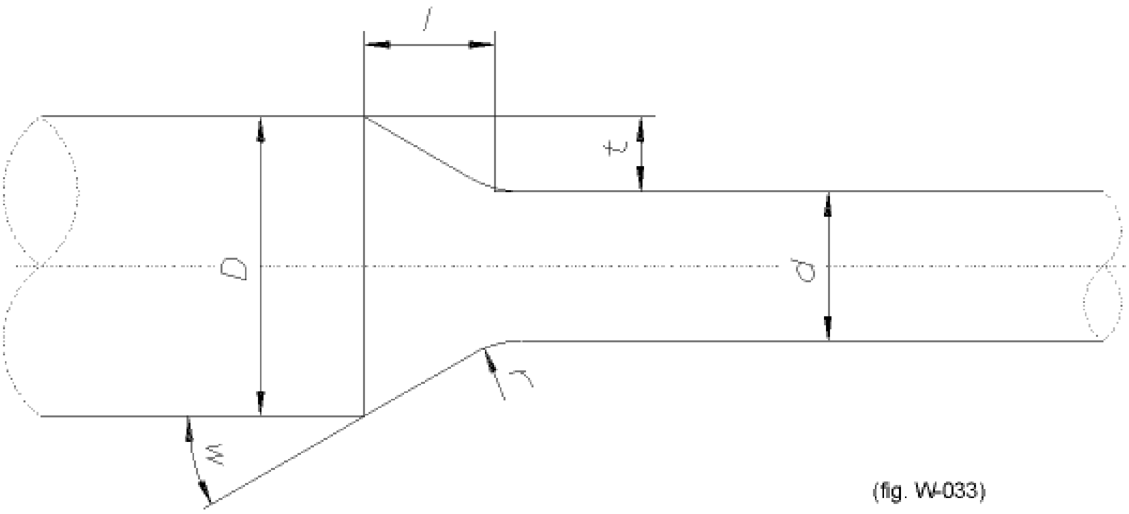


In Hänchen+ Decker and AGMA 2101:	not possible.
In DIN 743:	The notch factor is calculated like a shoulder, but with the ratio d/(1.1*D). The maximum transmission for D/d ~ 1.1 and for r/(D/d) ~2.

	This condition is only applied if D/d = 1.1, otherwise the notch effect of the shoulder is used.
In the FKM guideline:	The notch effect coefficient is determined for the fit H7/n6. The notch effect coefficient is also calculated for a shoulder and then used, in the worst case, in subsequent calculations.

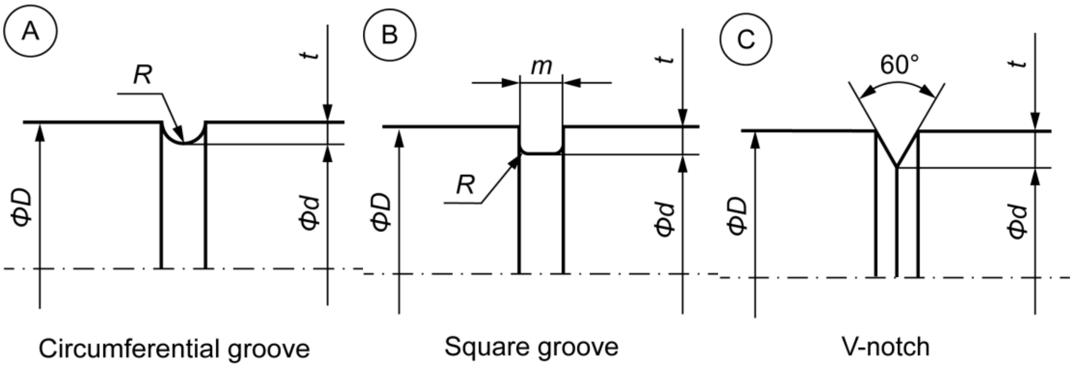
Notch factors are documented in the different methods. The notch factors calculated in FKM are usually significantly larger than in DIN.

- **Shoulder with conical transition**



(fig. W-033)

- **Shaft grooves**
  With the following variants:



- **Thread**
  Notch factors for threads are not supplied separately in the specialist literature. For this reason, notch factors for threads are handled like those for V-notches.
- **Interference fit:** Interference fit (tight interference fit, slight interference fit, interference fit with relief grooves)

Only notch factors for the tight interference fit are defined in DIN 743, so the FKM Guideline is used to define the factors for the other types of interference fit.

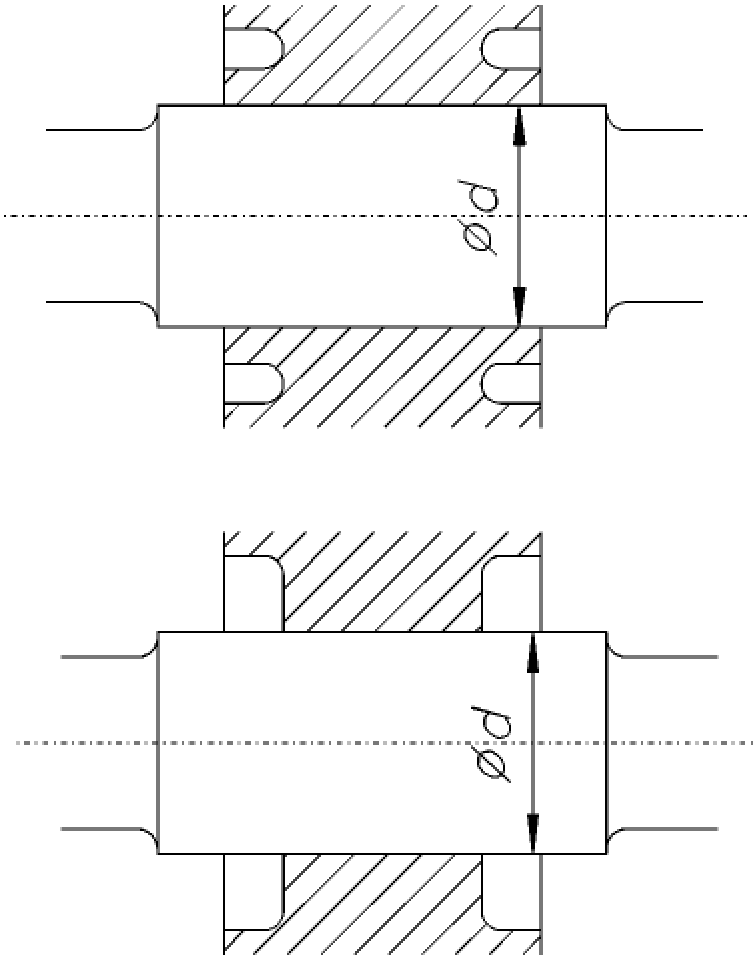
► **Note:**

Interference fit types "Slight interference fit" and "Interference fit with relief grooves" are no longer present in the latest version of the FKM Guideline, the 7th edition (2020). They are determined according to the old FKM Guideline (2012). The "Interference fit" interference fit type should be used for upgrades or new developments.

Defining notch effect coefficients for different types of interference fit:
Slight interference fit according to FKM and DIN 743: The notch effect coefficients for bending and tension/compression are defined according to Kogaev, Figure 5.3-11. Formula 5.3.16 is used to define torsion. Formula βQ = 1+(βT-1)/2, an assumption according to Prof. Haibach, is used to define shearing force.
This type of interference fit is described in the FKM Guideline (2012) and should no longer be used.

Interference fit acc. to FKM: The notch effect coefficients for bending are calculated as shown in Figure "Interference fit without hub overhang" in Table 5.3.1. The values for torsion are calculated from the value for bending. **Interference fit acc. to DIN 743:** The notch effect coefficients for bending and torsion are taken from Table 1., Case 2., in DIN 743-2.

Interference fit with relief grooves acc. to FKM and DIN 743: The notch effect coefficients for "Interference fit with exceptions" are determined according to Table 5.3.1, Figure 3, in the FKM Guideline for bending. The value for torsion is determined from the value for bending with the formula from the FKM Guideline.
This type of interference fit is described in the FKM Guideline (2012) and should no longer be used.



(fig. W-021)

Top: Interference fit with relief grooves.
Below: Interference fit with end relief.

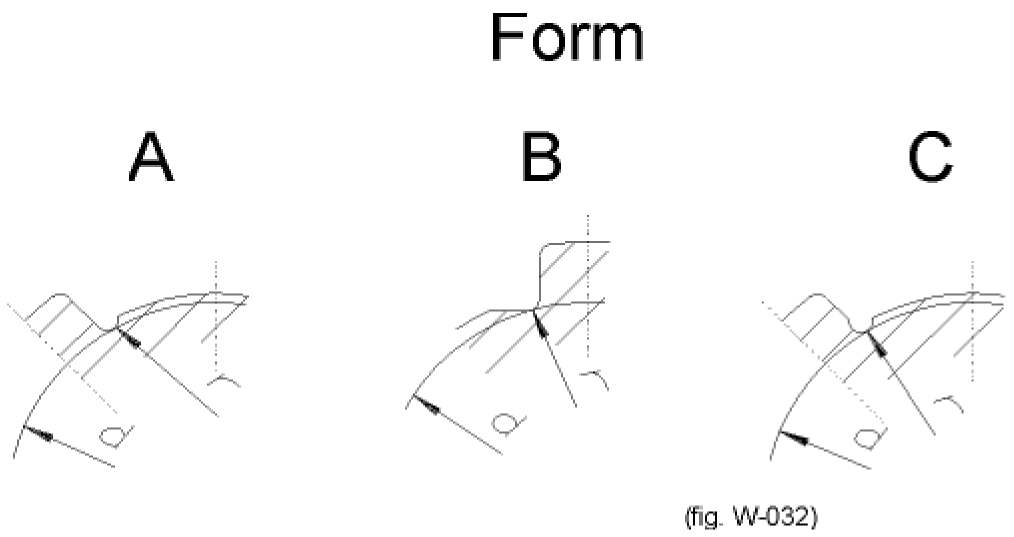
Key

In each method, the moment of resistance for bending is determined from shaft diameter d. As described by Hänchen, the moment of resistance for torsion is calculated from the incorporated circle d - t. According to FKM, DIN and AGMA, it is calculated from the outer shaft diameter d.

The notch factors are documented in the different methods. However, Hänchen provides very little information about this that can be used to extrapolate values for steel of higher strength (with the appropriate comment about the calculation). In contrast, these values are well documented in the DIN standard and the FKM Guideline (in the tables for Interference fit with key). Two different production methods for keys are described in AGMA 6101 (side milling cutter or keyway cutter). This standard also distinguishes between 2 different hardness ranges.

The program includes tables for cross sections with keys. The data is imported from a data file which includes the DIN 6885.1 (corresponds to ISO/R 773), DIN 6885.2 and DIN 6885.3 standards. You can also specify other standards.

- **Serration and straight-sided splines**



Straight-sided spline shapes

To calculate serrations or straight-sided splines, you must first enter the tip circle and root diameter data. All other values are used purely for documentation purposes.

To calculate the moments of resistance:

In Hänchen & Decker and FKM:	From the mean value ($d_a/2 + d_f/2$)
In DIN 743 and AGMA 6101:	From the root circle

Notch factors are documented in the different methods.

An exception to this is the calculation according to FKM, where the root diameter of straight-sided splines (in this case: d) is used to calculate the notch radius.

- **Cross hole**
- **Smooth shaft**
  If you select **Smooth shaft** the notch factor is set to 1. You should select this for the cross section with the maximum stress.
- Input your own notch factors (see chapter 26.5.12, Cross sections)
- Intersecting notch effects (see chapter 31.1, Intersecting notch effects)

26.5.15 General entries

26.5.15.1 Thickness factors from the shaft diameter

You can derive material values that depend on the diameter either from the effective shaft diameter (d or D) or from the thickness of the raw material. The choice based on the effective shaft diameter gives more reliable safety results, but can only be used if the shaft is through hardened before it is turned.

However, if you select **Pre-machined to actual diameter (for shoulders K1 from d)**, the material data for shoulders is derived from the smaller diameter (d). If you select **Pre-machined to actual diameter**, it is derived from the larger diameter (D). Although deriving these values from D gives slightly lower strength values, the results are therefore somewhat safer. The standard does not comment on this.

26.5.16 Thermally safe operating speed

The definition of the thermally safe operating speed is described in DIN 732 [65]. The calculation of the thermally safe operating speed is based on the thermal balance in the bearing. The thermally safe operating speed is derived from the thermal reference speed, using the speed ratio. The result of this calculation is the speed that will be reached by the bearing running at the permitted temperature in an actual situation. In order to define the thermally safe operating speed, you must first define the thermal reference speed for each case.

The thermal reference speed is defined in DIN ISO 15312 [66]. The thermal reference speed is the bearing-specific speed calculated under a given set of nominal operating conditions, such that equilibrium is achieved between heat development (friction) and heat dissipation (through bearing contact and lubricant).

You can enter the values for the calculation in the special "Thermally safe operating speed" tab and in the relevant rolling bearing in the Element Editor.

The calculation is also available for use in the rolling bearing calculation module [W050], where the calculation process and the values you enter are described in more detail. (see chapter 27.3, Thermally safe operating speed)

26.6 Flank line modification

For various purposes, it is important that you know how much a specific point in the shaft cross section moves in a particular direction due to elastic deformation (bending and torsion). An example of this is calculating the gaping between the two halves of a coupling that are mounted on each end of the same shaft. In this situation, the displacement of a point on the shaft cross section is calculated in the axial direction.

The most important application of this calculation is to determine shaft deformation in the meshing area. The deformation for the pitch point is calculated along the facewidth. In this situation, the displacement of the pitch point due to bending and torsion is calculated only in the direction of the normal to the flank. A displacement parallel to the flank only results in a very minimal change in sliding velocity and can therefore be ignored.

In the **Flank line modification** tab, you can directly select the toothing currently present on the shaft. Based on previously entered data, you can determine the necessary specifications for the calculation (facewidth from and to, coordinates of meshing point, direction of the normal to the tooth flank in the pitch point) which are displayed in the user interface. Therefore, assuming that the counter gear has infinite stiffness, the progress of the pitch point displacement due to deformation can be determined along the facewidth.

► **Note:**

When calculating the flank line modification, a possible offset of the load application on the gear selected for the calculation (calculation A or B) is temporarily disabled. This means the gear load application offset of gear A is disabled when calculation A is performed, but is re-enabled when calculation B is performed.

This deformation, also known as gaping, can be displayed by selecting **Graphic > Flank line modification > Deformation**.

This shows the deformation in the pitch point. A proposal for an optimal flank line modification is also displayed. This type of modification would ensure a homogenous load distribution across the facewidth.

You can input the tooth contact stiffness cγ in another input field. For steel gears, the tooth contact stiffness per mm facewidth is approximately 20 N/mm/°. The values of cγ are calculated precisely and documented in the cylindrical gear calculation. This stiffness can then be used to calculate the load distribution along the facewidth. Click **Graphics > Flank line modification > load distribution** to see the result.

Calculating the load distribution coefficient $K_{H\beta}$ for gear calculation

The results window also shows the load distribution coefficient $K_{H\beta}$, calculated according to ISO 6336, with the equation $K_{H\beta}$ = wmax/wm from the average line load (wm) and the maximum line load (wmax). This calculation enables the face load factor to be estimated with significantly more accuracy, similar to Method B in ISO 6336. The procedure is basically similar to Annex E of ISO 6336. However, you must be aware that the shaft of the counter gear used here is assumed to have infinite stiffness. This is permitted if the shaft of the counter gear has much greater stiffness. Manufacturing deviations are also only included if, for example, they have been defined by inputting an angular deviation of the shaft (bearing offset) as part of the shaft data.

The gear body can also be taken into account as a stiffness matrix. To do this, select the **Take stiffness matrix into account** option in the cylindrical gear force element to perform the corresponding calculation (see chapter 25.2.4.1, Forces).

► **Note:**

If $K_{H\beta}$ is to be determined while taking into account the deformation of the two shafts: The deformation components of two shafts can be combined in the cylindrical gear calculation in the **Contact analysis** tab.

Sizing the flank line modification

This calculation module has been designed to enable you to define the best possible flank line modification both quickly and accurately. To do this, you can input a modification consisting of flank line crowning or end relief and flank angle deviation. You can specify the flank angle deviation either as a positive or negative number, depending on the required progression. The modification input here is then also displayed in the "Deformation" graphic. In the "Load distribution" graphic you can then clearly see the improved load distribution achieved by this calculation. Click **Graphic > Flank line modification > flank line diagram** to call the graphic for manufacturing the modification (gear drawing).

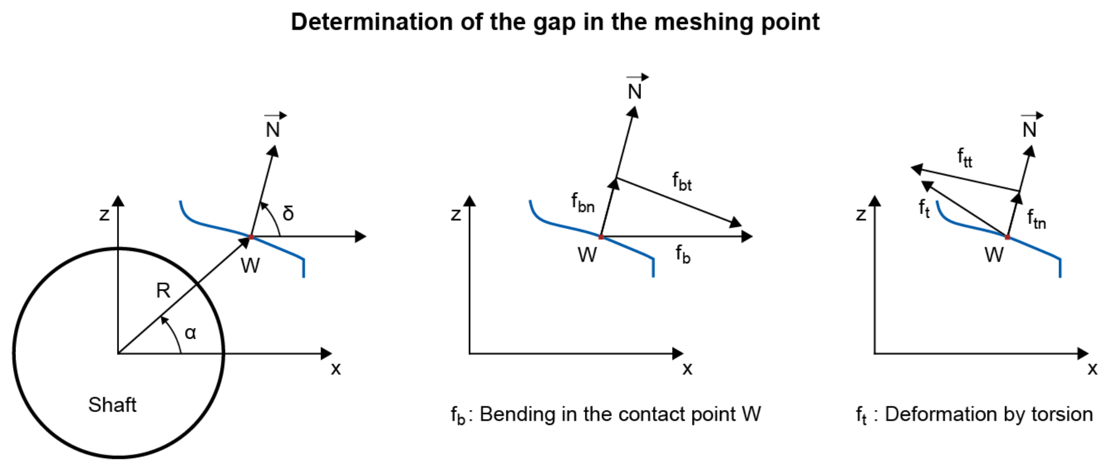


Figure 26.9: Determining gaping in the meshing point

26.7 Campbell diagram

Call this function from the **Calculations > Campbell diagram** menu option. In the relevant tab, you can select the shaft to be analyzed, the range of shaft speeds, the number of calculation steps with which the speed is calculated, and the number of resonance curves (forwards whirl) to be displayed.

The Campbell diagram shows the eigenfrequencies in a wider range of shaft speeds, which enables you to follow the forward and backwards whirl associated with the eigenmodes. To calculate the Campbell diagram, set the number of eigenfrequencies in the Basic data tab. The gyroscopic effect causes large changes in the eigenfrequencies. You can take it into consideration by clicking selecting the **Consider gyroscopic effect** option in the Basic data tab.

In normal cases, the backwards whirl drops in frequency, while the forwards whirl increases in frequency. In the case of forwards whirl, as shaft speed increases, the gyroscopic effect increasingly affects the spring stiffness and increases the eigenfrequencies. The effect is reversed for backward rotation, so increasing shaft spin speed reduces the effective stiffness, and thus reduces the eigenfrequencies. The eigenfrequencies are also affected by the stiffness of the bearings.

26.8 Forced vibrations

Use this module to calculate forced vibrations for the shafts defined in KISSsoft. The dynamic excitation of an unbalance mass is used to dynamically excite the shaft.

26.8.1 Calculation procedure

The first step in the calculation procedure is to enter this special calculation by selecting **Calculation** > **Forced response**. Select the shaft whose speed will be changed during the calculation, the relevant speed range and the number of calculation steps, here. Another input is the material (structural) damping for torsional, axial and bending vibrations. Note that the viscous damping of bearings must be defined separately for each bearing. To ensure that the calculation can be performed, at least one unbalance mass in a shaft must be specified ("Additional mass" element, "Mass", "Eccentricity" and "Angular position of the eccentric mass" input fields).
Note that results of completed calculations can only be requested at predefined documentation points. The documentation points in this case are used as measurement probes for the dynamic response of the shafts. Note that the resulting responses from different dynamic forces are added in the time domain, and the final result is given based on the maximum value found during this operation.
Apart from the calculation for a range of speeds of the reference shaft, there is also an option for performing a calculation for a specific running speed of the reference shaft and determining the dynamic response results along its length.

26.8.2 Results

Once the calculation is finished, you can display the results by selecting **Graphics > Shafts > Forced response**.

27 Rolling Bearings (Classic Analysis)

Manufacturer catalogs (such as SKF) include fairly comprehensive methods for verifying the rating life and static load capacity of rolling bearings. Specialized technical literature is also available to help you resolve more detailed problems [2].
KISSsoft includes data from well-known bearing manufacturers. to which you can add your own information.

The calculation module is integrated in the Shafts Module and can also be started separately by clicking **Shafts and Bearings > Rolling bearing ISO 281, ISO 76**.

27.1 Selecting the type of rolling bearing

27.1.1 Properties of the most important bearing types

Selecting the most suitable type of rolling bearing is sometimes no easy matter. The table below presents an overview of the critical properties of the most important types of rolling bearing:

- **Deep groove ball bearing (DIN 625):**
  The single row radial deep groove ball bearing is the most commonly used, because it is both extremely versatile and also the most inexpensive form of rolling bearing, because of its simple design. This bearing can withstand relatively high radial and axial forces in both directions.
- **Single row angular contact ball bearing and four-point contact bearing (DIN 628):**
  Each ring of a self-holding single row angular contact ball bearing has one lower shoulder and one higher shoulder. The grooves on the higher shoulder are positioned so that the contact angle is normally α = 40°. The higher number of balls in this configuration means it can withstand not only radial forces but also larger axial forces in one direction (towards the higher shoulder) than deep groove ball bearings. Axial reaction forces due to the angle of the groove are generated when the bearing is subjected to a radial load. You must take this into account when sizing the bearing. Because of their one-sided axial loading capacity, these types of bearings are usually installed in pairs in which the second bearing is mounted in the opposite direction. The axial load that acts on the bearing in the case of an O- or X-arrangement is calculated and displayed in the mask.
- **Double row angular contact ball bearing (DIN 628):**
  The double row angular contact ball bearing corresponds to a pair of mirror image compounded single row angular contact ball bearings (back-to-back arrangement) with α = 25° or 35°, and can therefore withstand radial and high axial forces in both

directions.

Use: To support the shortest possible bending-resistant shaft that is subject to strong radial and axial forces: integral worm shafts, shafts with helical gears and bevel gears.

- **Double row self-aligning ball bearing (DIN 630):**
  The self-aligning ball bearing is a double row bearing with a cylindrical or conical bore (bevel 1:12). It can compensate for shaft displacement and misalignment (up to approximately 4° angular deviation) thanks to its hollow sphere raceway in the outer ring. It can be subjected to radial loads and axial loads in both directions.

  Use: bearings which are inevitably subject to mounting inaccuracies and bending of the shaft, e.g. transmissions, conveyors, agricultural machinery, etc.
- **Cylindrical roller bearing (DIN 5412):**
  Cylindrical roller bearings can support larger radial loads than ball bearings of the same size (point contact area!) because the contact between the rollers and the races is made along a line. Demountable cylindrical roller bearings can only support small axial forces (if at all) and require accurately aligned bearings.
  The different construction types can be identified by the rim arrangement: construction types N and NU have an unconfined outer and inner ring and can be used as non-locating bearings, construction type NJ can be used as a step bearing and construction types NUP and NJ can be used as a guide bearing for axial shaft support in both directions.

  Use: in gear units, electric motors, for axles of rail vehicles, for rollers in a rolling mill. In general, suitable for bearing applications that are subject to large radial loads.
- **Needle roller bearing (DIN 617):**
  Needle roller bearings are a special type of cylindrical roller bearing in which a cage separates the needle rollers to keep them at a specific distance from, and parallel to, each other. The bearing is supplied with or without an inner ring, and is only suitable for radial forces. Its significant features are: small overall diameter, high degree of rigidity in the radial direction and a relative insensitivity to uneven loading.

  Use: Predominantly used at low to medium speeds and when oscillatory motion is present, e.g. as connecting rod bearings, rocker-shaft bearings, swivel arm bearings, jointed cross-shaft axle bearings (vehicles), spindle bearings, etc.
- **Tapered roller bearing (DIN 720):**
  The ring raceways in tapered roller bearings are cone-shaped shells which must converge into one point due to the action of kinematic forces. The bearings with α = 15°(30°) can support high loads both in the radial and axial directions. The detachable outer ring makes them easy to assemble and dismantle. Tapered roller bearings are installed in mirror image pairs. The bearing clearance can be set and adjusted as

required. Due to the angle of the race, a radial force produces an axial reaction force.

Use: hub bearings of vehicles, cable pulley bearings, spindle bearings in machine tools, shaft bearings in worms and bevel gears.

Calculation: the axial force that you must specify when calculating dynamic equivalent loads is defined in several theories (see, for example, page 296 of FAG Wälzlager Catalog WL 41520DE (1995)). The axial force acting on the bearing is displayed in the screen. The bearing forces that include the contact angle can be calculated directly.

- **Barrel-shaped bearing (DIN 635), toroidal roller bearing (CARB), and double row self-aligning ball bearing (DIN 635):**
  Spherical raceways in the outer rings and barrel-shaped rollers (toroidal-shaped for CARB bearings), as in double row self-aligning ball bearings, enable barrel-shaped, toroidal roller (CARB) and double row self-aligning roller bearings with a cylindrical or conical bore (1:12) to compensate for misalignment and for the angular dislocation of the shaft (oscillating angle 0.5° to 2°). Barrel roller bearings (single row self-aligning roller bearings) are suitable for high radial loads, but can only withstand low axial forces. In contrast, double row self-aligning roller bearings (α = 10°) can be used for the highest radial and axial forces. Toroidal roller bearings (CARB) have an extensive range of uses in many load applications. Toroidal roller bearings combine the angular flexibility of double row self-aligning roller bearings with the axial displacement options of cylindrical roller bearings.

  Use: for heavy wheels and cable pulleys, propelling shafts, rudder posts, crank shafts, and other heavily loaded supports. Toroidal bearing: paper making machinery, blowers and, generally, in planetary gear units.

27.1.2 Comparing types

Selecting the most suitable type of rolling bearing is sometimes no easy matter. The table below gives an overview of the most important properties. The rolling bearing you select for specific operating conditions has often already been determined by its properties and properties. You can use this information to select the bearing you require for frequently occurring working cases and for specialized requirements. However, results may overlap, and therefore the cost factor may be decisive.

In addition to the specified rolling bearing types, some hybrid bearings (with ceramic rolling bodies) have been included, for some types. The special properties of these bearings are described in the "Hybrid bearing" section.

Radial bearing:

Features	**a**	**b**	**c**	**d**	**e**	**f**	**g**	**h**	**i**	**j**	**k**	**l**	**m**	**n**
Radial load capability	⊗	⊗	⊗	Ø	⊗	+	+	+	+	+	+	+	+	+
Axial load capability	⊕	⊗	⊗	⊗	Ø	-	⊕	⊕	-	⊕	⊕	+	⊕	⊗
Inside position adjustment	-	-	-	-	-	+	Ø	-	+	Ø	-	-	-	-
Mounting position adjustment	⊕	⊕	⊕	-	⊕	-	-	⊕	-	⊕	⊕	⊕	⊕	⊕
Dismountable bearings	-	-	⊕	⊕	-	+	+	+	+	⊕	-	+	-	-
Alignment error adjustment	Ø	-	-	-	+	Ø	Ø	Ø	-	Ø	-	Ø	+	+
Increased precision	⊕	⊕	⊕	Ø	-	⊗	⊕	⊕	+	-	-	⊗	-	-
High speed running	+	+	⊕	Ø	⊗	+	⊗	⊗	+	-	-	⊕	⊕	⊕
Quiet running	+	⊗	Ø	Ø	Ø	⊕	Ø	Ø	⊕	-	-	Ø	Ø	Ø
Conical bore	-	-	-	-	+	⊗	-	-	+	-	-	-	+	+
Seal on one/both sides	⊕	-	⊕	-	⊕	-	-	-	-	-	⊕	-	-	⊗
High stiffness	⊗	⊕	⊕	⊗	Ø	⊕	⊕	⊕	+	+	+	+	⊕	⊕
Low friction	+	⊕	⊗	⊕	+	⊕	⊕	⊕	+	-	-	⊕	⊗	⊕
Fixed bearing	⊕	+	⊕	⊕	⊗	-	⊗	⊕	-	⊗	⊗	+	⊕	⊕
Non-locating bearing	⊗	⊗	⊗	-	⊗	+	⊗	Ø	+	⊗	⊗	Ø	⊗	⊗

+ very good ⊕ good ⊗ normal/possible Ø with restrictions- not suitable/no longer relevant

a	Deep groove ball bearing
b	Angular contact ball bearing (single row)
c	Angular contact ball bearing (double row)
d	Four-point contact bearing
e	Double row self-aligning ball bearing
f	Cylindrical roller bearing NU, N
g	Cylindrical roller bearing NJ
h	Cylindrical roller bearing NUP, NJ+HJ
i	Cylindrical roller bearing NN
j	Cylindrical roller bearing NCF, NJ23VH
k	Cylindrical roller bearing NNC, NNF

l	Tapered roller bearing
m	Barrel roller bearing
n	Double row self-aligning roller bearing

not listed here: Needle roller bearing, needle cage

Thrust bearing:

Features	**o**	**p**	**q**	**r**	**s**	**t**
Radial load capability	-	-	Ø	-	-	Ø
Axial load capability	⊗	⊗	⊗	⊗	⊗	+
Inside position adjustment	-	-	-	-	-	-
Mounting position adjustment	-	-	-	-	-	-
Dismountable bearings	+	+	-	+	+	+
Alignment error adjustment	⊕	⊕	Ø	-	-	+
Increased precision	⊗	-	+	+	⊕	-
High speed running	⊕	Ø	⊗	+	Ø	Ø
Quiet running	Ø	-	Ø	Ø	-	-
Conical bore	-	-	-	-	-	-
Seal on one/both sides	-	-	-	-	-	-
High stiffness	⊕	⊕	⊗	+	⊕	⊗
Low friction	⊗	Ø	⊕	⊕	-	-
Fixed bearing	⊕	⊕	+	+	⊗	⊗
Non-locating bearing	-	-	-	-	-	-

+ very good ⊕ good ⊗ normal/possible Ø with restrictions- not suitable/no longer relevant

o	Deep groove thrust ball bearing (one-sided)
p	Deep groove thrust ball bearing (double direction)
q	Axial angular contact ball bearing (one-sided)
r	Axial angular contact ball bearing (double direction)
s	Cylindrical roller thrust bearing
t	Axial spherical roller bearing

not listed here: Thrust needle cages, angular contact thrust roller bearings, cross roller bearings

27.1.3 Hybrid bearings

In a hybrid bearing, the race is made of rolling bearing steel, and the rolling bodies are made of a ceramic material (silicon nitrite, Si_3N_4).

Hybrid bearings are included in the databases of standard rolling bearings. The rolling bearing database has a particular setting which identifies hybrid bearings.

The calculation basis is the same as for standard types of rolling bearing.

However, the thermal reference speed and thermally safe operating speed cannot be determined, because hybrid bearings are not covered by the standards.

The moment of friction for these bearings cannot be determined because the calculation methods used in rolling bearing catalogs do not cover hybrid bearings.

The most important benefits of hybrid bearings are:

- higher stiffness
- suitability for use at higher speeds
- reduced inertia and centrifugal forces in the bearing
- reduced frictional heat
- lower energy consumption
- longer bearing life and grease lifetime

27.2 Load capacity of rolling bearings

We distinguish between the dynamic load capacity of the rotating bearing and the static load capacity in idleness (at standstill), at very slow speed or if very small oscillations are present, relative to the working state, but not to the effect of the load.

27.2.1 Dynamic load capacity

The dynamic load capacity is a property of the entire bearing. ISO 281 describes a number of various properties of a rolling bearing that occur if the bearing experiences specific mechanical loading under specific conditions at specific speeds. This data is then used to calculate the number of operating hours (this is usually based on a failure probability of 10%).

27.2.2 Static load capacity

The static load capacity includes properties that a rolling bearing must display in order to withstand certain mechanical loading situations in idleness (at standstill), at very low speeds (n < 20 rpm) or during oscillatory motion.

Plastic deformation (indentation) occurs between the rolling bodies and the races when the bearing is subjected to a moderate static stress due to the weight of the shaft and the other elements. This value gradually increases as the stress increases. However, the plastic deformation must not be so great that it influences the operational properties of the bearing in its rotational movement. As defined in ISO 76, the static characteristic value $S_0 = C_0/P_0$ is a safety factor against detrimental plastic deformation which is a measure of the sufficient static load capacity.

The static load number, which is used to determine the bearing size, can be determined by taking into account the safety factor which depends on the operating conditions:

$S_0 > 2$	for shocks and impacts as well as exacting requirements for smooth operation and for axial spherical roller bearings
$S_0 = 1$	for normal operation and low noise requirements
$S_0 = 0.5...0.8$	for smooth and non-impact operation with few requirements (non-loaded bearing with adjusting or swivel motion)

27.2.3 Rolling bearing calculation with internal geometry

The rolling bearing reference rating life calculation is based on ISO/TS 16281.

The additional results of this calculation are the maximum Hertzian pressure on the inner and outer ring (right and left ring for a thrust bearings), the static safety, the reference and modified reference rating life in hours, the stiffness matrix at the operating point, and the load distribution or pressure curve on each rolling element.

If the calculation of rolling bearings is performed with internal geometry, the bearing frequencies can be determined. The calculated values (over-rolling frequency of the outer ring, over-rolling frequency of the inner ring, rotational frequency of the rolling elements and the cage frequency) are documented in the report and shown in a graphic.

For more detailed information, see (see chapter 28, Rolling Bearings (Internal Geometry)).

f the rolling bearing inner geometry is provided by the manufacturer, then it is used in the calculation. If these data are unknown, then KISSsoft runs an approximation method that tries to determine the internal geometry using the rolling bearing load ratings (the static load rating C_0 and dynamic load rating C). This procedure is based on ISO 76 and ISO 281, and normally produces quite useful results.

If individual values such as the number of rolling bodies are known, only the remaining values are approximated.

If the inner geometry you entered in the database is either insufficient or incorrect, this data is then ignored, and the inner geometry is approximated. A note is then printed in the report, stating that an approximation of the inner geometry has been used.

Internal geometry cannot always be taken into account when calculating bearing types (see chapter 9.5.37.2, Rolling bearing Internal geometry).

27.3 Thermally safe operating speed

The method for defining the thermally safe operating speed is described in DIN 732 [65], based on the heat levels in the bearing. The thermally safe operating speed is derived from the thermal reference speed, using the speed ratio. The result of this calculation is the speed that will be reached by the bearing running at the permitted temperature in an actual situation. This thermally safe operating speed may differ greatly from other operating speed limits, depending on lubrication type, because the reference conditions only apply to quite specific cases. Before you can determine the thermally safe operating speed, you must first define the thermal reference speed for each case.

► **Note:**

Calculations cannot be performed for barrel roller bearings (single row self-aligning roller bearings), angular contact thrust roller bearings, cross roller bearings, and all hybrid bearings, because none of the relevant standards have values for them.

27.3.1 Thermal reference speed

The thermal reference speed is defined in DIN ISO 15312 [7]. The thermal reference speed is the bearing-specific speed calculated under a given set of nominal operating conditions, such that equilibrium is achieved between heat development (friction) and heat dissipation (through bearing contact and lubricant). Mechanical or kinematic criteria are not taken into account for this speed. The reference values (temperatures, load, viscosity of the lubrication, reference face of the bearing,. . .) are fixed so that the reference speed using oil- or grease-lubricated bearings will result in identical values.

27.3.1.1 Dissipated Heat Flows

The heat flow Qr is calculated from the reference heat flow density qr that is specific to a rolling bearing (for heat flow dissipated through bearing contact and lubricant), and from heat dissipation via the reference surface Asr.
Qr = 10-6 * qr * Asr
qr, Asr are defined under reference conditions according to DIN ISO 15312.

27.3.1.2 f0r and f1r coefficients

The coefficients f0r and f1r used to define the thermal reference speed are different, depending on which bearing type/series (also lubrication type for f0r) is used. They are shown in Table A.1 of the standard. Not all bearing variants are listed in the table.

The following values have been assumed for bearings and bearing types for which no data has been defined in the standard:

	f0r (tabular value)	f1r
Ball bearing	1.7	0.00015
Roller bearing	3	0.0003
Thrust ball bearing	1.7	0.00015
Thrust roller bearing	3.5	0.0015

27.3.1.3 Calculating the thermal reference speed

The dissipating heat flows and the friction power are set as equal values so that the energy balance of the bearing is correct. The equation for the energy balance is:

NFr = 103 * *Qr*

NFr: Friction power [W]
Qr: dissipated heat flow: [kW]

The subsequent equation becomes:

(π *nθr)/30 * (10-7 *f0r * (vr*nθr)2/3 *dm^3 + f1r *P1r *dm) = qr *ASr

nθr: thermal nominal speed [rpm]
f0r: coefficient from Table A.1, DIN ISO 15312 [-]
r: reference viscosity [mm2/s]
dm: average rolling bearing diameter [mm]
f1r: coefficient from Table A.1, DIN ISO 15312 [-]
P1r: reference load [N]
qr: rolling bearing-specific reference heat flow density (bearing contact, lubricant) [kW/m2]
ASr: Heat-transferring reference surface [mm2]

nθr can be determined using this equation.

27.3.2 Process for calculating thermally safe operating speed (DIN 732-2)

As, when calculating the thermal reference speed, this calculation is based on the thermal balance in the bearing. dissipated heat flow:
$Q = Q_S + Q_L + Q_E$

Q_S: heat flow dissipated from the bearing contacts
Q_L: heat flow dissipated by lubrication (only when there is circulatory lubrication) (the lubricant's density ϱ = 0.91 kg/dm^3 and specific heat capacity cL = 1.88 KJ/(kg *K) are predefined.)
Q_E: additional heat flows (the calculation, Q_E = 0 is assumed.)

27.3.2.1 Friction coefficients f0 and f1

Coefficients f0 and f1 and the dynamic equivalent load P1 are only needed to define the load and lubrication parameters. These values differ depending on the specific bearing type/model, lubrication type, or load direction. They are listed in Table A.1 in the standard. Not all bearing variants are listed in the table. The values for various types of lubrication below have been defined (and incorporated in KISSsoft). They are based on the notes about f0 in Table A.1 in the standard.

- Oil, bath lubrication, bearing in oil mist: f0 = 0.5 * f0 (tabular value)
- Oil, bath lubrication, oil level up to middle bearing: f0 = 2.0 * f0 (tabular value)
- Oil, bath lubrication, oil level up to middle of the lowest rolling element: f0 = 1.0 * f0 (tabular value)
- Oil, circulatory lubrication: f0 = 2.0 * f0 (tabular value)
- Grease, run-in bearing: f0 = 1.0 * f0 (tabular value)
- Grease, newly greased: f0 = 2.0 * f0 (tabular value)

The following values have been assumed for bearings and bearing types for which no data has been defined in the standard:

	P1	f0 (tabular value)	f1
Ball bearing	3.3*Fa - 0.1*Fr (P1 <= Fr) where P1 = Fr	1.7	0.0007*(P0/C0)^0.5
Roller bearing	Fr	3	0.0003
Thrust ball bearing	Fa	1.7	0.0007*(P0/C0)^0.5
Thrust roller bearing	Fa	3.5	0.0015

27.3.2.2 Calculating the thermally safe operating speed limit

The thermally safe operating speed is derived from the thermal reference speed, using the speed ratio. nθ = fn * nθr

The load and lubrication parameters must be calculated before the speed ratio can be iterated from this equation:

KL * *f*n5/3 + *KP* * *f*n = 1

Load parameter *KL*:

KL = 10-6 * (π/30) * *n*θ*r**10-7 * (*f*0*r* * n2/3 * nθr2/3 * dm3)/*Q*

Lubricant film parameter *KP*:

KP = 10-6 * (π/30) * *n*θ*r**(*f*1 * *P1* * dm)/*Q*

nθr: thermal nominal speed [rpm]
f0: Coefficient of friction from Table A.1, DIN 732 [-]
f1: Coefficient of friction from Table A.1, DIN 732 [-]
n: Lubricant viscosity [mm2/s]
dm: average rolling bearing diameter [mm]
P1: Reference load [N]
Q: Total dissipated heat flows [kW]

27.4 Moment of friction

In a rolling bearing, the friction between individual components has a decisive influence on heat development and therefore the operating temperature. The appropriate heat value can be estimated from the moment of friction of the rolling bearing. Select **Calculation > Settings** to display the Settings tab. In it, you can select this calculation method.

You can only perform this calculation by clicking the **Modified service life according to ISO 281** option in the Strength tab (to display it, select **Basic data > Strength**).

27.4.1 Calculation according to SKF Catalog 1994

The prerequisite for calculating the moment of friction is that the bearing rotating surfaces must be separated by a lubricant film. The total bearing moment of friction results from the sum:

$M = M_0 + M_1$	(27.1)

M_0: load-independent friction moment

M_0 is determined by the hydrodynamic losses in the lubricant. It is especially high in quickly rotating, lightly loaded bearings. The value M_0 depends upon the quantity and viscosity of the lubricant, as well as the rolling speed.

M_1: load-dependent friction moment

M_1 is determined by the elastic deformation and partial sliding in the surfaces in contact, especially due to slowly rotating, heavily loaded bearings. The value M_1 depends on the bearing type (bearing-dependent exponents for the calculation), the decisive load for the moment of friction and the mean bearing diameter

For axially loaded cylindrical roller bearings, an additional axial load-dependent moment of friction, M_2 , is added to the formula.

$M = M_0 + M_1 + M_2$	(27.2)

M_2: axial load-dependent friction moment

M_2 depends on a coefficient for cylindrical roller bearings, the axial loading and the bearing's mean diameter.

For sealed rolling bearings, an additional axial load-dependent moment of friction, *M*3 is added to the formula.

$M = M_0 + M_1 + M_3$	(27.3)

M_3: **Moment of friction for grinding seals**
The moment of friction for grinding seals depends on the bearing type, the bearing size, the diameter of the seal-lip mating surface, and the layout of the seal. As the type of seal, the diameter of the seal-lip mating surface, and the seal layout, differ from one manufacturer to another, it is difficult to define a generally applicable moment of friction.

Select **Calculation > Settings** to display the Settings tab. In it, you can choose different options for determining this reference size:

- **according to SKF main catalog in selected calculation method**
  - *according to the Hauptkatalog (main catalog) 4000/IV T DE: 1994:*
    You will find values for the seal types used in your bearings in the SKF catalog, which is integrated in the KISSsoft software. If the KISSsoft system finds a familiar seal label in the bearing label, it calculates the moment of friction for a grinding seal using the coefficients listed in the catalog. Otherwise, the moment of friction is set to zero.

Example of a seal label in the name of a rolling bearing:
SKF: 623-2RS: this means that the bearing has a RS1-type seal on both sides. The KISSsoft system then searches for names with "-2RS1" in them. If this label is present, the coefficients from the SKF catalog are applied and the moment of friction for grinding seals is calculated.

- **according to ISO/TR 13593:1999 Viton** M_{sea} calculated with the formula: M_{sea} = 3,736*10^-3*dsh; M_{sea} in Nm, d_{sh} Shaft diameter in mm
- **according to ISO/TR 13593:1999 Buna N** M_{sea} calculated with the formula: M_{sea} = 2,429*10^-3*dsh; M_{sea} in Nm, d_{sh} Shaft diameter in mm

Coefficients f0, f1 (see chapter 27.3.2.1, Friction coefficients f0 and f1) and P1 (values that depend on the bearing type and bearing load) used for the calculation have been taken from ISO 15312. The formulae, exponents and coefficients have been taken from the SKF Catalog, 1994 Edition.

27.4.2 Calculation according to SKF Catalog 2018

As this calculation has to take into consideration a myriad of factors and influences, it is only performed if selected as an option in the modified rating life calculation. However, this calculation can also be performed without these default values. The calculation of the total moment of friction according to the 2018 SKF catalog is determined by a combination of rolling and sliding friction in the roller contacts (between rolling bodies and cage, the bearing surface, the lubricant, and the sliding friction from grinding seals caused in sealed bearings). The calculation of the moment of friction depends on various coefficients:

- Rating (load)
- Type of bearing
- Bearing size
- Operating speed
- Lubricant properties
- Lubricant quantities
- Seals

The following working conditions must be present for the calculation to be performed:

- Grease or oil lubrication (oil bath, oil mist, or oil injection process)
- Load equal or greater than minimum load
- Load constant in size and direction
- Nominal operating clearance

If the load is less than the minimum load, the calculation continues using the minimum load. If a minimum load value has been entered in the database, this value is used. If not, the software will determine this value. In the case of radial bearings, the minimum load is converted into a minimum radial force. In thrust bearings, the minimum axial force is defined by the software. The value for the minimum load is not used here.

The formula for the total moment of friction is:

$M = M_{rr} + M_{sl} + M_{seal} + M_{drag}$

M_{rr}: **Rolling moment of friction**
The rolling moment of friction depends on the bearing type, mean diameter, radial and axial loading, rotation speed, and lubricant viscosity. The design coefficients required to calculate the rolling moment of friction are defined using the rolling bearing's series. The design coefficients and coefficients used in the calculation are taken from the SKF Catalog 2018.

Coefficients used for rolling friction:

- ϕ_{ish}: **Lubricant film thickness factor**
  In a lubricant flow, the lubricant is exposed to shear forces caused by the movement of the rolling bodies. This produces heat and therefore reduces the rolling moment of friction.
- ϕ_{rs}: **Lubricant displacement factor**
  The constant rolling action squeezes excess lubricant away from the contact zone of the rolling body. This reduces the lubricant film thickness and therefore reduces the rolling moment of friction.

Assumptions have been made for bearing types and bearing series for which no design coefficients have been defined in the catalog, so that the rolling moment of friction can still be calculated despite their absence.

M_{sl}: **Sliding moment of friction**
The sliding moment of friction depends on the bearing type, mean diameter, radial and axial loading and lubricant viscosity. The design coefficients required to calculate the sliding moment of friction are defined using the rolling bearing's series. You will find the factors used for this calculation in the SKF 2018 catalog.

M_{seal}: **Moment of friction for grinding seals**
The moment of friction for grinding seals depends on the bearing type, bearing size, diameter of the seal-lip mating surface, and the seal type. As the type of seal, the diameter of the seal-lip mating surface, and the seal layout, differ from one manufacturer to another, it is difficult to define a generally applicable moment of friction.

Select **Calculation > Settings** to display the Settings tab. In it, you can choose different options for determining this reference size:

- **according to SKF main catalog in selected calculation method**
  - *PUB BU/P1 17000/1 EN: October 2018*
    You will find values for the seal types used in your bearings in the SKF catalog, which is integrated in the KISSsoft software. If the KISSsoft

system finds a familiar seal label in the bearing label, it calculates the moment of friction for a grinding seal using the coefficients listed in the catalog. Otherwise, the moment of friction is set to zero.
Example of a seal label in the name of a rolling bearing:
SKF: 623-2RS: this means that the bearing has a RS1-type seal on both sides. The KISSsoft system then searches for names with "-2RS1" in them. If this label is present, the coefficients from the SKF catalog are applied and the moment of friction for grinding seals is calculated.
In KISSsoft, the diameter of the seal-lip mating surface is calculated with:
ds = d + (D - d) * 0.2

- **according to ISO/TR 13593:1999 Viton** M_{sea} calculated with the formula: M_{sea} = 3.736*10^-3*dsh; M_{sea} in Nm, d_{sh} Shaft diameter in mm
- **according to ISO/TR 13593:1999 Buna N** M_{sea} calculated with the formula: M_{sea} = 2.429*10^-3*dsh; M_{sea} in Nm, d_{sh} Shaft diameter in mm

M_{drag}: **Moment of friction caused by lubrication losses**
This moment of friction is caused by flow, splash or injection losses during oil bath lubrication. To calculate this moment, you must also input the oil level depth (h_{Oil}), which you can specify by selecting **Calculation > Settings**. You will find a more detailed description of this entry in the Oil level and lubrication type section (see chapter 27.12). The design coefficients KZ and KL for rolling bearings with a cage are also applied to toroidal roller bearings (CARB).

27.4.3 Calculation according to Schaeffler 2017 (INA, FAG)

To define the total moment of friction, the speed, load, lubrication type, lubrication method and viscosity of the lubricant at operating temperature must be known.

Formula used for the total moment of friction:

$M_R = M_0 + M_1$	(27.1)

M_0: speed-independent (load-independent) moment of friction

M_0 is determined by the hydrodynamic losses in the lubricant. It is especially high in quickly rotating, lightly loaded bearings. The value M_0 depends upon the quantity and viscosity of the lubricant, as well as the rolling speed.

M_1: load-dependent friction moment

M_1 is determined by the elastic deformation and partial sliding in the surfaces in contact, especially due to slowly rotating, heavily loaded bearings. The value M_1 depends on the bearing type (bearing-dependent exponents for the calculation), the decisive load for the moment of friction and the mean bearing diameter

For axially loaded cylindrical roller bearings, an additional axial load-dependent moment of friction, M_2 , is added to the formula.

$M_R = M_0 + M_1 + M_2$	(27.2)

M_2: axial load-dependent friction moment

M_2 depends on a coefficient, k_B for cylindrical roller bearings, the axial loading and the bearing's mean diameter.

For bearings with a TB design (better axial load capacity achieved using new calculation and production methods), bearing factor f2 is displayed in a special diagram in the main catalog.

Coefficients f0, f1 (see chapter 27.3.2.1, Friction coefficients f0 and f1) and P1 (values that depend on the bearing type and bearing load) used for the calculation have been taken from DIN ISO 15312. The formulae, exponents and coefficients have been taken from the Schaeffler Catalog, 2017 Edition.

27.5 Grease lifetime

Grease lifetime is the length of time that a bearing remains adequately lubricated without having to be regreased. If the grease lifetime is reached without regreasing, it can be expected that the bearing will fail.
By its very nature, the grease lifetime is very application-specific. However, there are approaches which can be used to estimate its approximate value. In the KISSsoft system, methods from the Schaeffler (INA/FAG) and SKF catalogs can be used to calculate a guide value. Select **Calculation > Settings** to display the Settings tab. In it, you can select the appropriate method. The calculation is performed when you select the Enhanced service life calculation (see chapter 27.7.1, Modified rating life calculation according to the Supplement to DIN ISO 281 (2007)) and select a grease as the lubricant in it.

27.5.1 Calculation according to Schaeffler 2018 (INA, FAG)

According to the Schaeffler catalog 2018 (INA, FAG), a guide value for the grease lifetime can be determined on the basis of a speed-dependent base grease lifetime. This guide value can then be adjusted to suit the particular application and environment in which the bearing is working:

$$t_f g = t_f * K_T * K_P * K_R * K_U$$

t_f: base grease lifetime
K_T: correction factor for increased temperature
K_P: correction factor for increased load

K_R: correction factor for oscillating mode
K_U: Correction factor for environmental influences

The lower guide value of the order of magnitude specified in the relevant diagram is used as the reference for the base grease lifetime t_f.

The correction factor K_T for increased lubricant temperature is used above the lubricant-specific operating temperature. In oscillating mode, the correction factor K_R is assumed to be 1.

27.5.2 Calculation according to SKF Catalog 2018

The SKF catalog 2018 states that a guide value for the grease lifetime can be determined on the basis of a speed- and load-dependent base grease lifetime. Like the Schaeffler approach, this can then be adjusted by a number of factors, depending on how the bearing is used and its environment.

In the KISSsoft system, the lubrication lifetime is determined as the guide value for the lubrication interval and a modification, K_S, is applied for vertical shafts and a modification, K_T, is applied for lubricant temperatures greater than 70°C. The catalog lists a number of additional influencing factors and provides qualitative descriptions for them. These factors must then be evaluated and taken into account on an application-specific basis.

$$t_f g = K_T * K_S ...$$

t_f: base grease lifetime
K_T: correction factor for increased temperature
K_S: Correction factor for a vertical shaft

27.6 Maximum Speeds

Rolling bearings are reliable and can be expected to reach their calculated rating life as long as the maximum speed (speed limit) is not exceeded. This depends on the type, size and lubrication.

A warning message is displayed if the maximum permissible speed is exceeded.

The permitted maximum speed can be much lower, depending on the lubrication type used (see chapter 27.3, Thermally safe operating speed).

27.7 Rating life

The nominal rating life is calculated using the formulae given in ISO 281, and corresponds to the formulae that can also be found in the manufacturers' catalogs. Usually the rating life is calculated at

90% (10% probability of failure), in hours. The label used here is L_{10h} (h: hours. 10: probability of failure).

27.7.1 Modified rating life calculation according to the Supplement to DIN ISO 281 (2007)

ISO 281 includes the regulations for "modified rating life" which take into account the influence of load, lubricant conditions, materials specifications, type, material internal stresses and environmental factors.

The life modification factor: a_{ISO} can be defined as follows:

$$a_{ISO} = f\left(\frac{ec \cdot Cu}{P}, \kappa\right)$$	(27.3)

a_{ISO}:	Life modification factor from diagram [-]
ec:	Contamination characteristic value [-]
Cu:	fatigue load limit [N]
P:	Dynamic equivalent load [N]
κ:	viscosity ratio = n_u/n_{u1}
n_{u1}:	reference viscosity diagram [mm2/2]
n_u:	VT diagram for the lubricant [mm2/2]

The fatigue load limit Cu is specified by the bearing manufacturers. If none of these values are known, you can calculate them with the approximate formula as defined in ISO 281.
The contamination factor ec (between 0 and 1) is taken directly from the degree of cleanliness.

27.7.2 Rating life calculation with load spectra

The load spectrum on the bearing has these values:

k:	Number of elements in the load spectrum
q_i:	Frequency (load bin i) (%)
n_i:	Speed (load bin i) (rpm)
F_{ri}:	Radial force (load bin i) (N)
F_{ai}:	Axial force (load bin i) (N)

You can take this load spectrum data from the shaft calculation, in which case you may obtain different load spectra for radial and axial forces. Alternatively, you can select a load spectrum from the database. For bearing forces, the important factor here is the torque factor (not the load factor) and a negative prefix operator will only affect the axial force.

Achievable rating life with simple calculation approach:

You calculate the rating life by defining an equivalent design load and the average speed. You can then use the usual formulae to calculate the rating life.

$$n_m = \frac{n_1 \cdot q_1}{100} + \frac{n_2 \cdot q_2}{100} + \cdots + \frac{n_k \cdot q_k}{100}$$	(27.4)
$$P_m = \sqrt[p]{\frac{P_1^p \cdot q_1}{100} + \frac{P_2^p \cdot q_2}{100} + \cdots + \frac{P_k^p \cdot q_k}{100}}$$	(27.5)

n_m:	Average speed
p:	Exponent in the rating life formula (3.0 or 10/3)
P_i:	Dynamic equivalent load (load bin i)
P_m:	Average dynamic equivalent load

Achievable rating life with the modified rating life calculation:

When the modified rating life calculation is used, the rating life is calculated separately for every equivalent load bin. The result is then used to determine the total service life:

$$L_{nx} = \frac{100}{\frac{q_1}{L_{nx1}} + \frac{q_2}{L_{nx2}} + K + \frac{q_k}{L_{nxk}}}$$	(27.6)

L_{nxi}:	Service life (load spectrum bin i) in the case of speed n_i and load F_{ri}, F_{ai}
L_{nx}:	Total rating life
	Index x can be h or rh

27.8 Failure probability

Normally, the failure probability is assumed to be 10%. This means there is a 90% probability that the nominal rating life will be achieved. In this case, the coefficient a_1 = 1.0. If the failure probability value has to be lower, this coefficient must also be lower (at 1%, a_1 = 0.21).

To define the failure probability, select **Calculation > Settings**.

27.9 SKF spherical roller thrust bearings: runout error takes into account load distribution

When you select **Calculation** > **Settings**, you see this option. Use it to specify whether shaft coaxial errors or runout errors influence the way loads are distributed in the bearing. If this option is selected, the influence is not taken into account. If that is the case, a coefficient of 0.88 is multiplied to calculate the dynamic equivalent load P (P=0.88*(Fa+Y*Fr)).

27.10 Bearing with radial and/or axial force

For every bearing, you can specify whether it is subject to a radial or axial force. If the bearing is subject to axial force, you must also specify whether the force is applied in both directions (<>), in the direction of the y-axis (– >) or in the opposite direction (> –).

27.11 Calculating axial forces on bearings in face-to-face or back-to-back arrangements

Because of the inclination of the raceways in the bearing, a radial load generates axial reaction forces in taper roller bearings, angular contact spindle ball bearings and angular contact ball bearings. This data must be taken into account when the equivalent design load is analyzed.

Axial reaction forces are calculated in accordance with SKF (rolling bearing catalog) which exactly match the values defined in FAG.

For bearings in a back-to-back arrangement, left bearing A, right bearing B, outer axial force in A-B direction, the following data applies:

Condition	Formula

$\frac{F_{rA}}{Y_A} \leq \frac{F_{rB}}{Y_B}$	$F_{aA} = F_a + \frac{F_{rB}}{2 \cdot Y_B}$ $F_{aB} = F_a - F_{aA}$
$\frac{F_{rA}}{Y_A} > \frac{F_{rB}}{Y_B}$ $F_a > \frac{1}{2} \cdot \left(\frac{F_{rA}}{Y_A} - \frac{F_{rB}}{Y_B} \right)$	$F_{aA} = F_a + \frac{F_{rB}}{2 \cdot Y_B}$ $F_{aB} = F_a - F_{aA}$
$\frac{F_{rA}}{Y_A} > \frac{F_{rB}}{Y_B}$ $F_a \leq \frac{1}{2} \cdot \left(\frac{F_{rA}}{Y_A} - \frac{F_{rB}}{Y_B} \right)$	$F_{aA} = F_a + F_{aB}$ $F_{aB} = \frac{F_{rA}}{2 \cdot Y_A} - F_a$
F_{rA}, F_{rB}	Radial force on bearing A, B
Y_A, Y_B	*Y* coefficient of bearing A, B
F_a	External axial force
F_{aA}, F_{aB}	Axial force on bearing A, B

For all other cases (face-to-face arrangement or axial force in the other direction), simply reverse the formula.

These calculated internal tension values are displayed in the main window. If the actual internal forces are higher, for example, due to the use of spring packages, you can change the value manually.

27.12 Oil level and lubrication type

To input the oil level and lubrication type, select **Calculation > Settings**. These entries are required to define the moment of friction due to lubrication losses. The value h is given in the shaft calculation and results in the following formula for every bearing:

$$H = \frac{D}{2} - h \geq 0$$

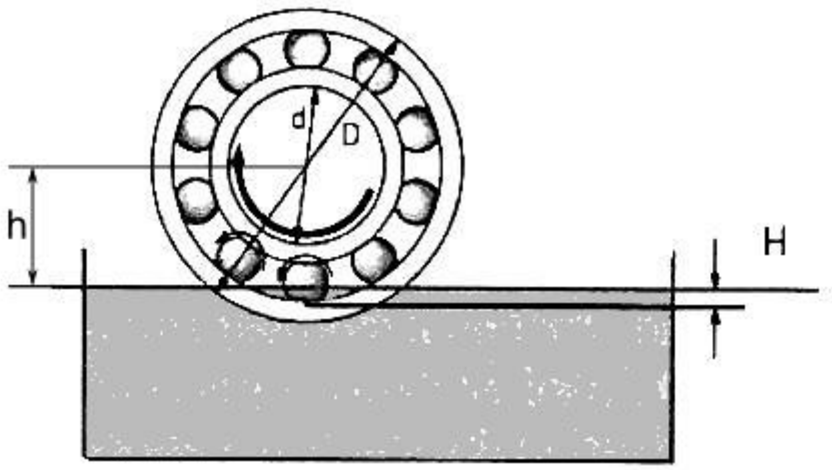


Figure 27.1: Oil level in the bearing

Two different types of lubrication can be defined:

- Oil bath lubrication
- Oil injection lubrication

If you select the **Oil injection lubrication** (spray lubrication) option, the value determined for the flow loss-dependent moment of friction for oil bath lubrication is multiplied by 2.

28 Rolling Bearings (Internal Geometry)

In addition to the classic bearing calculation (see (see chapter 27, Rolling Bearings (Classic Analysis))), KISSsoft also provides a calculation according to ISO/TS 16281. Here, the internal bearing geometry (number of rolling elements, rolling body diameter, etc.) is used to calculate the bearing load and the service life.

This method is integrated in the Shaft calculation and is also available as a separate KISSsoft module. Unless otherwise indicated, the separate KISSsoft module is described below. The module is designed to be used by bearing experts, or users who know the internal geometry of their bearings.

Notes:

EHL lubricant film thickness
The minimum EHL lubricant film thickness is calculated for rolling bearings with a known internal geometry, using the methodology described in [67]. The effect of pressure on viscosity is taken into account using the Barus' equation, as documented in the same reference.

Spin to roll ratio
The spin to roll ratio of ball bearings is calculated on the basis of the equations in [67]. The assumption of "outer raceway (ring) control" is used, meaning that no spin of the ball is present on the outer raceway (ring). It is a well-known fact that this assumption primarily applies for lightly loaded high speed bearings. Ball gyroscopic motions and cage effects are not considered any further than that.

28.1 Bearing data tab

28.1.1 File interface

Use this module to link to a shaft calculation file. This means bearing information is automatically transferred from the shaft calculation file without you having to reenter the data. The user must input the:

- File name: name of the shaft calculation file (extension .W10), from which the selected bearing data will be extracted.
- Element type: Here, you select whether the bearing is a rolling bearing that belongs to a shaft, or a connecting rolling bearing

- Shaft no.: if the bearing belongs to a shaft, the user must input the shaft number here. The program then runs through the shafts Element Tree from top to bottom 5ba0bcd6a1d96.
- Bearing no.: number of the selected bearing, either on the corresponding shaft or from the list of connecting elements. The program runs through the shafts Elements tree from top to bottom 5ba0bcd6a1d96.
- Data exchange: determines how data is exchanged between the shaft file and this module. In each case, the geometry of the selected bearing is transferred from the shaft file.
  - Bearing load: the information transferred from the shaft file is the applied force and torque of the bearing as well as the lubricating conditions
  - Bearing displacement: the information transferred from the shaft file is the displacement and rotation of the inner ring of the bearing as well as the lubricating conditions
  - Own Input: only the bearing geometry is transferred. You can specify your own load and lubrication conditions.

28.1.2 Bearing data

This is where the geometry of the bearing is defined (see chapter 27.2.3, Rolling bearing calculation with internal geometry). In addition to the geometry data, you can also enter the basic dynamic load rating. If you do not know this value, the rating is calculated using the current geometry data as specified in ISO 281. If you require a modified rating life (see chapter 27.7.1, Modified rating life calculation according to the Supplement to DIN ISO 281 (2007)), input the fatigue load limit Cu. If Cu is not known, it is also calculated on the basis of ISO 281.

Note for the shaft calculation: In this module, the effect of surface hardness on the static capacity can be taken into account by entering the Vickers hardness. You will find the formulae for this in [68]. The hardness value of every bearing calculated with their inner geometry is predefined as HV 660 for the shaft calculation.

28.1.2.1 User-defined roller profile

A logarithmic profile as specified in ISO 16281 is usually used for roller bearings. However, a user-defined roller profile can be used instead, if required. The expected structure of this file is as follows:

-- This is a comment

DATA

1 -0.45 0.000581256

2 -0.41 0.000390587

3 -0.37 0.000277616

4 -0.33 0.000200197

...

...

21 0.33 0.000200197

22 0.37 0.000277616

23 0.41 0.000390587

24 0.45 0.000581256

END

Notes:

- Lines that start with "--" are comments and are ignored.
- The profile function definition starts with the keyword "DATA" and ends with the keyword "END".
- Each line must have three columns. The first column is the index. It is only included as a reference source for the user. Its values have no effect, and are ignored. The second column is the non-dimensional position x/L_{we} for which the profile is defined in mm/mm. The values in this column should range between -0.5 and +0.5. The third column is the non-dimensional profile f_r/D_w in mm/mm. The values in this column cannot exceed 0.5.
- If the profile is not defined over the entire rolling body width, the value is extrapolated quadratically for these areas.
- To save space, the data represented by "..." has been omitted.

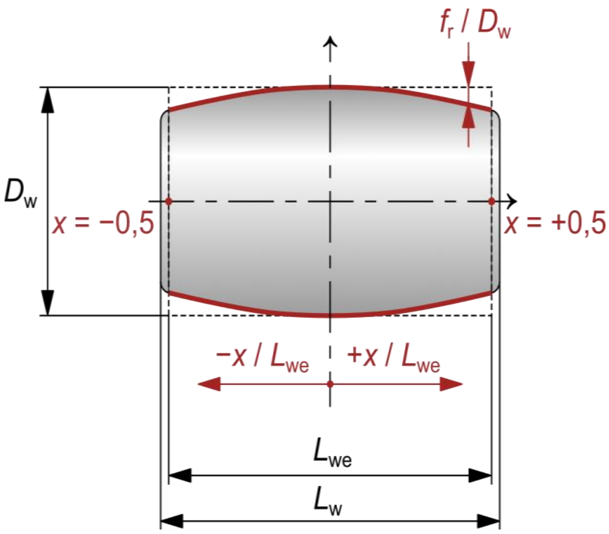


Figure 28.1: Coordinate frame used to define the user-defined roller profile

28.1.2.2 User-defined raceway profiles

Roller bearings usually have straight inner and outer raceway profiles. However, if needed, these can be modified by specifying user-defined raceway profile modifications, similar to user-defined roller profiling. Raceway profile modification is defined as the multiplication of a constant modification value and a modification ratio for a specific relative position along the effective roller length.

Constant modification values are set for the inner and outer raceway, r_{mod_i} and r_{mod_o}, in the KISSsoft user interface and should always be positive. Modification ratios f are set in the DAT file. The file should have the following structure:

-- This is a comment
-- Index | `Relative position | Modification ratio`

DATA

1 -0.50 1.00

2 -0.30 0.75

3 0.00 0.00

4 0.30 0.75

5 0.50 1.00

END

Notes about the DAT file:

- Lines that start with "--" are comments and are ignored.
- The lines with profile modification ratios for a specific relative position along the effective roller length start with the keyword "DATA" and end with the keyword "END".
- Each line must have three columns. The first column is the index. It is only included as a reference for the user. Its values have no effect and are ignored. The second column is the non-dimensional relative position along the effective roller length x/L_{we}, in mm/mm, for which the modification ratio f is defined. Usually, the values in the second column would extend from -0.5 to +0.5, but they can also exceed this range. The third column contains the modification ratio, which defines the amount of modification as $f \cdot r_{mod}$, at a specific relative position along the effective roller length.
- A positive modification ratio $f > 0$ will result in increased bearing clearance at the specified position (or the removal of raceway material), while a negative modification ratio $f < 0$ will result in reduced bearing clearance at the specified position (or the addition of raceway material). At least one value in the modification ratio should be $f \leq 0$, otherwise the modification will increase the bearing clearance.
- There should be at least 2 lines with the data in the file. The amount of modification between given positions is calculated by means of linear interpolation. The amount of modification outside given positions is calculated by means of linear extrapolation.

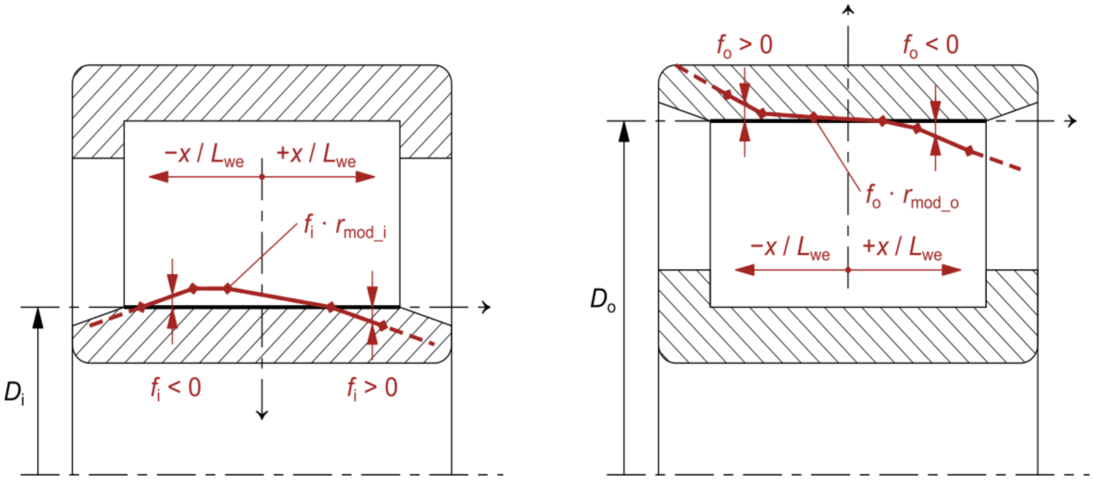


Figure 28.2: Parameters and coordinates used to define inner and outer raceway profile modifications

28.2 Rating (load) tab

Define the bearing's working conditions in this window.

28.2.1 Rating (load)

Four combinations of data can be entered here:

- (A) Force and Tilting moment
- (B) Force and Tilting
- (C) Displacement and Tilting moment
- (D) Displacement and Tilting

Speed: the speed of the inner ring relative to the outer ring. The outer ring is always assumed to be fixed (non-rotating).

Oscillating angle: the oscillating angle for partially rotating bearings. The rating life in million oscillation cycles is determined according to [2].

Note for the shaft calculation: The default setting for the shaft calculation process is combination D.

Note: A complete oscillation is $2 \cdot \varphi_s$

28.2.2 Modified rating life calculation in accordance with ISO 281

The effect of lubrication, filtration and contamination on bearing rating life can be taken into account here.

Lubricant: the lubricant used

Operating temperature: the temperature of the lubricant

Contamination: the class of the contamination

28.3 Graphics

The following graphics are provided:

1. Load distribution
This shows the load distribution on the rolling bearings (balls/rollers). For thrust bearings, the magnitude of the reaction force is used for the plot.

2. Deformation (elastic rings)
Shows the radial and axial deformation of the inner and outer ring.
3. Stress distribution on raceway
Shows the stress distribution on the inner and outer raceways.

4. Pressure curve
This shows how the pressure develops along the length of each roller, or at every contact point in a ball bearing.

5. Pressure curve for each rolling body
This shows the pressure curve on each roller element along the roller profile.

6. Stiffness curve
This shows the force-displacement curve of the bearing. Both radial and axial stiffness are shown.

7. Bearing frequencies
This shows the calculated bearing frequencies (over-rolling frequency of the outer ring, over-rolling frequency of the inner ring, rotational frequency of the rolling elements and the cage frequency) depending on the relative speed for each bearing.

28.4 Fine sizing

After you have loaded the "Roller bearing ISO/TS 16281" module, click on the Fine sizing option  to open a window in which you can perform fine sizing on the internal geometry of rolling bearings. In the same way as for fine sizing gears (see chapter 15.19, Fine sizing macrogeometry), you can vary the geometry parameters to generate a multitude of different bearing variants. The type you selected in the bearing data now determines which specific input parameters (for example, number of rolling bodies, radial clearance, osculation etc.) are available. Any existing bearing data for the bearing type is transferred directly to the fine sizing function.

You need a certain amount of experience before you can use the fine sizing function effectively. This is because the process may generate bearing variants whose internal geometry does not match either the currently applicable standards or roller bearings that have actually been manufactured. The system does check some of the parameters you input, and displays warning messages when it encounters implausible data.

You input the external dimensions of the roller bearing in the upper third of the first tab in the window. The middle third is where you input the parameters for the internal bearing geometry (according to the bearing type). In each case, the number of variants for the parameters are determined from the start values, the final values and the intervals (for example: if between 11 and 15 rolling bodies are involved, and the interval is set to two rolling bodies, the program calculates exactly three variants, if all the other intervals are set to zero).

The lower third of this tab is where you define the boundary conditions that each of the rolling bearing variants the system generates has to fulfill. You can set the density of the rolling bodies to a number between 0% and 100% (if you set it to 100%, the bearing will have so many rolling bodies of this type that they will touch each other along the whole length of the operating pitch circle). The minimum wall thicknesses you enter here apply to the inner and outer ring races. Any bearing variants that fail to meet the boundary conditions are ignored in the rest of the calculation.

Now, click on the "Calculate" button in the bottom area of the window to generate the bearing variants. The system now calculates the detailed inner geometry and the load rating and rating life. In this case, the load data is taken from the "Rating" tab in the "Rolling bearing ISO/TS 16281" module. A progress bar shows you how the calculation is progressing.

The bearing variants determined here are then displayed in a table in the "Results" tab in the Fine sizing window. These results can also be output in the report. The bearing variants are also displayed as a graphic in the "Graphic" tab (in a similar way to the results of the Gear fine sizing function (see chapter 15.19.6, Graphics)). Here you can display the values of different parameters along the horizontal and vertical axes. You can also use a color scale to make the results easier to understand.

29 Hydrodynamic Plain Journal Bearings

Niemann [8] provides a very accurate method for calculating plain radial bearings that can run at high speeds. This also produces good results for oval-clearance or tilting pad plain bearings.

ISO 7902 [69] or DIN 31652 include a very good and detailed method for calculating stationary hydrodynamic plain radial bearings that are to run at low to medium speeds. For those running at high speeds, use the equally excellent DIN 31657 [70].

29.1 Calculation methods

You can use one of these four methods to calculate oil-lubricated, hydrodynamic plain journal bearings:

- a) According to G. Niemann, Maschinenelemente I, 1981, [8].
  This method is very suitable for quickly rotating bearings. This also produces quite good results for special construction types such as tilting pad or oval-clearance plain bearings.
  This method calculates the power loss, oil flow, oil temperature, and minimum lubricant gap thickness, according to [8] and [71]. This calculation can only be used for pressure-lubricated bearings (circulatory lubrication) when the service reliability is also tested.
- b) According to ISO 7902, Parts 1 to 3, 2020 [69].
  This method is very suitable for slowly rotating bearings. It also determines the oil consumption, the oil flow and the entire heat balance.
  Calculation according to ISO 7902, Parts 1 to 3 (2020) for pressure-less and pressure-lubricated bearings. This takes into account the way in which lubricant is applied (lubrication holes, lubrication groove, lubrication glands). It calculates all the operating data as defined in ISO 7902, including the operating temperature, minimum lubrication gap width, power loss, oil flow etc. It also checks service reliability.
- c) According to DIN 31652, Parts 1 to 3, 2015, [72].
  This method is very suitable for slowly rotating bearings. It also determines the oil consumption, the oil flow and the entire heat balance.
  Calculation according to DIN 31652, Parts 1 to 3 (2015 Edition) for pressure-less and pressure-lubricated bearings. This takes into account the way in which lubricant is applied (lubrication holes, lubrication groove, lubrication glands). It calculates all the operating data as defined in DIN 31652, including the operating temperature, minimum lubrication gap width, power loss, oil flow etc. It also checks service reliability.
- d) According to DIN 31657, Part 1-4, 1996, [70].
  This method is very suitable for quickly rotating bearings. It also determines the oil consumption, the oil flow and the entire heat balance. The calculation is suitable for

multi-lobed plain bearings and tilting pad plain bearings.
Complete calculation according to DIN 31657, Parts 1 to 4 (1996 Edition) for pressure-lubricated bearings. It calculates all the operating data according to DIN 31657, including the operating temperature, minimum lubrication gap width, power loss, oil flow etc. It also checks service reliability.

29.2 Module specific entries

Calculating the lubricant volume-specific heat c.

The lubricant's volume-specific heat can be calculated in two ways:

- **Take into account dependence on temperature**
- **Simplified assumption (as in ISO 7902/DIN 31657)**: $1.8 \cdot 10^6$ J/(m^3·K)

If the **Take into account dependence on temperature** option has been selected, the specific heat capacity of the lubricant can also be specified, if it is known. For example, you must overwrite this value if you want to perform a calculation for a water-lubricated plain bearing, otherwise you will get incorrect results.

Run calculation with critical Reynolds number

If this option has been selected, the calculation which checks for the critical Reynolds number (laminar to turbulent transition) continues even if an error message is output. Otherwise, the calculation is interrupted.

29.3 Coefficients of thermal expansion

To calculate the clearance, you need the coefficients of thermal expansion of the shaft and hub.

These are the thermal expansion coefficients for the most important materials:

Steel	$11.5\ 10^{-6}$/°C
Cast iron	$11.0\ 10^{-6}$/°C
White metal	$18.0\ 10^{-6}$/°C
Composite bronze	$18.0\ 10^{-6}$/°C

29.4 Average surface pressure

You will find the permitted values in:

- Niemann, Volume I, Table 15/1, [8]
- ISO 7902, Part 3, Table 2, [69]
- DIN 31657, Part 4, Table 1, [70]

Permitted maximum values for the surface pressure, depending on operating temperature (ISO 7902/DIN 31652):

- Pb and Sn alloys: 5 (15) N/mm2
- Cu Pb alloys: 7 (20) N/mm2
- Cu-Sn alloys: 7 (25) N/mm2
- Al Sn alloys: 7 (18) N/mm2
- Al Zn alloys: 7 (20) N/mm2

the values shown in brackets were recorded under special working conditions.

Permitted maximum values for the surface pressure, depending on operating temperature (DIN 31657):

- Lead alloys: 16 to 25 N/mm2
- Tin alloys: 25 to 40 N/mm2
- Copper alloys (bronzes): 25 to 50 N/mm2

29.5 Geometries according to DIN 31657

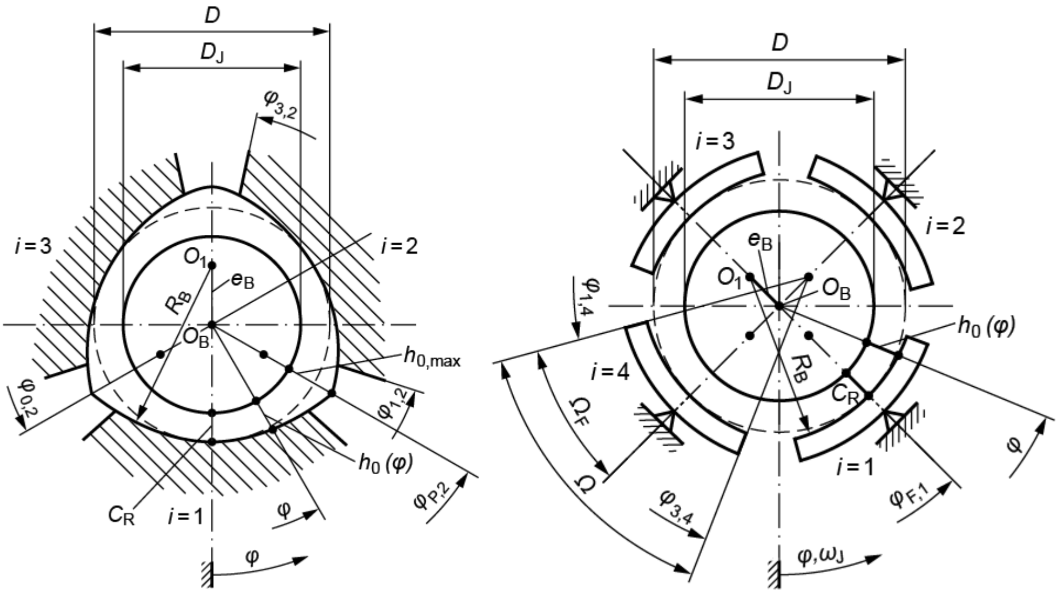


Figure 29.1: Display a multi-lobed plain bearing (left) and a tilting pad plain bearing (right) according to DIN 31657-1

$$C_R = \frac{D - D_j}{2} - R - R_j$$

$$\triangle R_B = R_B - R_j$$

$$e_B = R_B - \frac{D}{2} - R_B - R$$

$$\psi = \frac{C_R}{\frac{D}{2}} - \frac{C_R}{R}$$

$$\frac{\triangle R_A}{C_R} = \frac{R_B - R_j}{C_R}$$

$$\varepsilon = \frac{e}{C_R}$$

Different load cases and arrangements of the multi-lobed plain bearings, as shown in DIN 31657-2 and as are present in the tables.

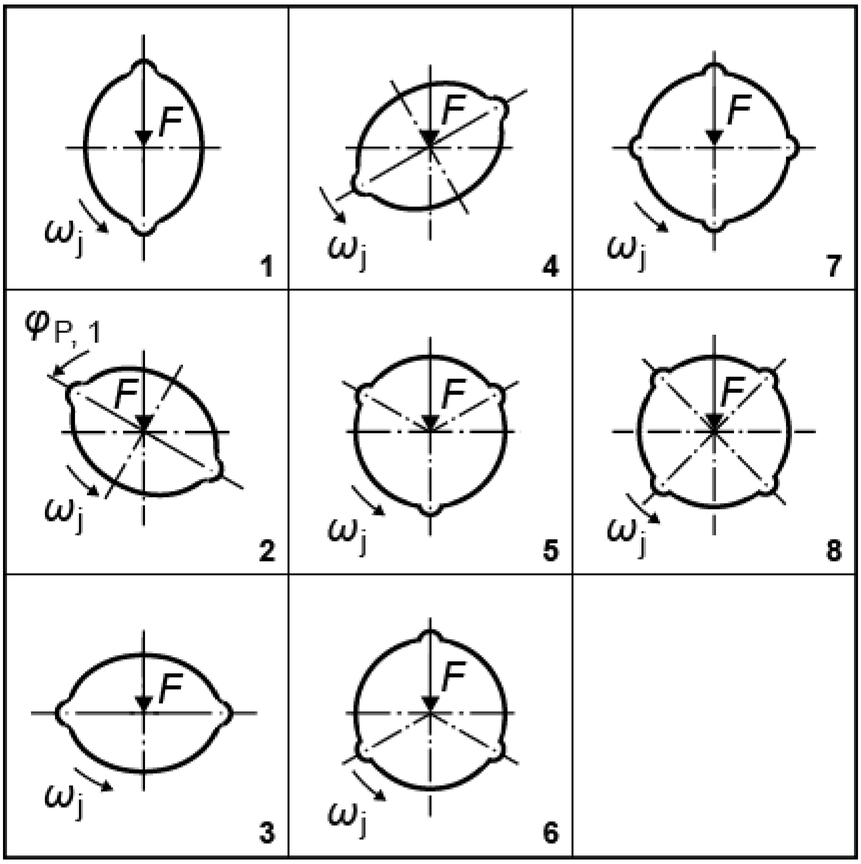


Figure 29.2: Arrangements of multi-lobed plain bearings

1) Z=2;Ω=150°;φP,1=180°;h*0,max=3,5;B/D=0.75

2) Z=2;Ω=150°;φP,1=240°;h*0,max=3,5;B/D=0.75

3) Z=2;Ω=150°;φP,1=270°;h*0,max=1,3,5;B/D=0.5,0.75,1

4) Z=2;Ω=150°;φP,1=300°;h*0,max=3,5;B/D=0.75

5) Z=3;Ω=100°;φP,1=240°;h*0,max=3,5;B/D=0.75

6) Z=3;Ω=100°;φP,1=300°;h*0,max=1,3,5;B/D=0.5,0.75,1

7) Z=4;Ω=70°;φP,1=270°;h*0,max=3,5;B/D=0.75

8) Z=4;Ω=70°;φP,1=270°;h*0,max=1,2,3,4,5;B/D=0.5,0.75,1

Different load cases and arrangements of the tilting pad plain bearings, as shown in DIN 31657-3 and as are present in the tables.

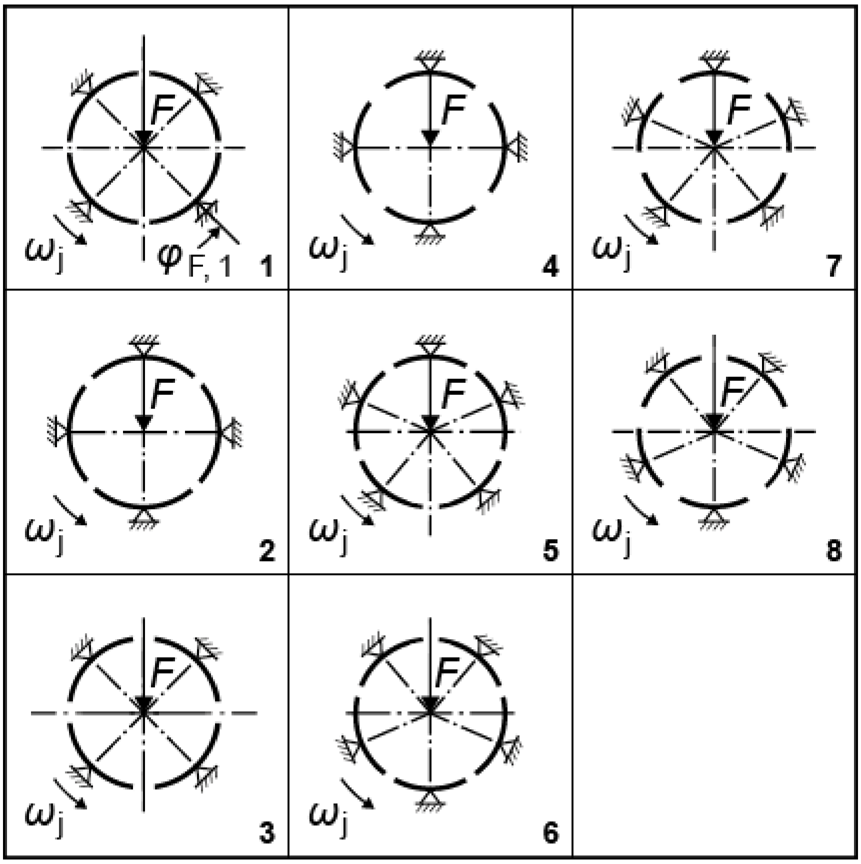


Figure 29.3: Arrangements of tilting pad plain bearings

1) Z=4;Ω=80°;φF,1=45°; ΔRB/CR=2,3,5;B/D=0.5,0.75,1

2) Z=4;Ω=80°;φF,1=0°; ΔRB/CR=3;B/D=0.75

3) Z=4;Ω=60°;φF,1=45°; ΔRB/CR=2,3,5;B/D=0.5,0.75

4) Z=4;Ω=60°;φF,1=0°; ΔRB/CR=3;B/D=0.5

5) Z=5;Ω=60°;φF,1=36°; ΔRB/CR=2,3,5;B/D=0.5,0.75

6) Z=5;Ω=60°;φF,1=0°; ΔRB/CR=3;B/D=0.5

7) Z=5;Ω=45°;φF,1=36°; ΔRB/CR=2,3,5;B/D=0.5

8) Z=5;Ω=45°;φF,1=0°; ΔRB/CR=3;B/D=0.5

29.6 Stiffness

To calculate the stiffness of the plain bearing, the maximum force at the narrowest point (operating point) is determined.

This maximum force can then be used to calculate the stiffness at the narrowest point. The diametral clearance (eccentricity) is also produced as a result of the plain bearing calculation.

The results for the stiffness cr, the diametral clearance Pd and the misalignment angle β are listed in the report. These results can then be entered in the shaft calculation to determine the stiffness of the plain bearing.

29.7 Lubrication arrangement

The different lubrication arrangements are shown in the next three figures.

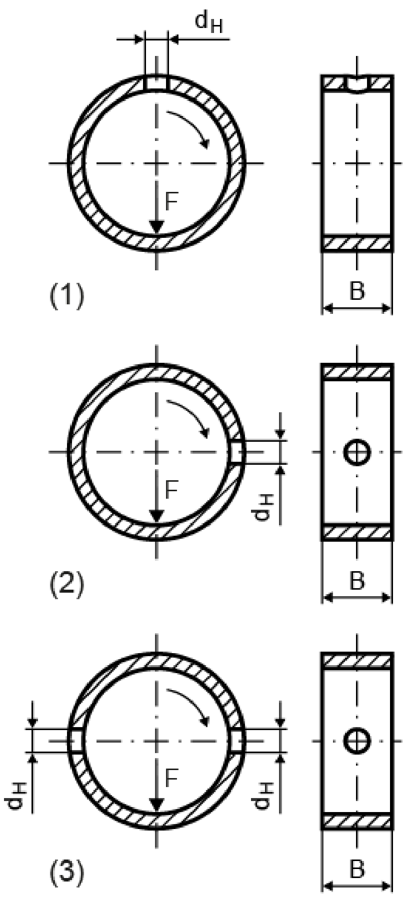


Figure 29.4: 1: One lubrication hole, opposite to load direction. 2: One lubrication hole, positioned at 90° to the load direction. 3: Two lubrication holes, positioned at 90° to the load direction.

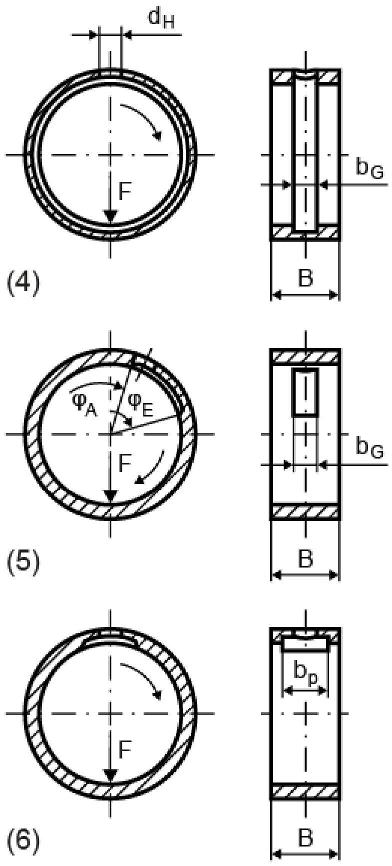


Figure 29.5: 4: Lubrication groove (ring groove). 5: Lubrication groove (ring groove). 6: Lubrication pocket opposite to load direction.

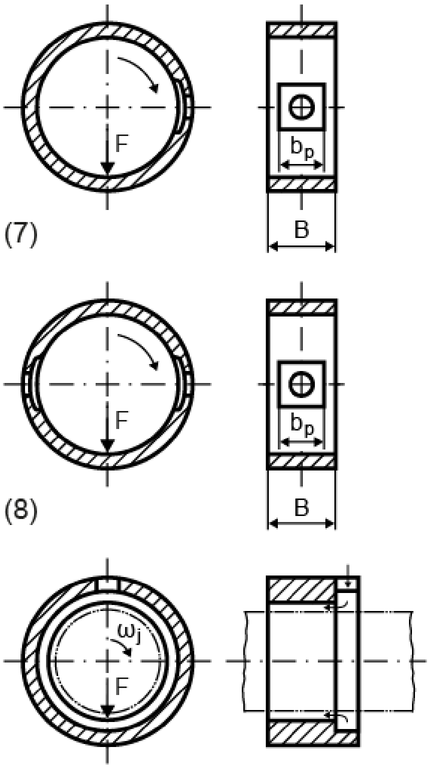


Figure 29.6: 7: One lubrication pocket positioned at 90° to the load direction. 8: Two lubrication pockets positioned at 90° to the load direction. 9: From one bearing edge across the entire perimeter of the bearing (only Draft DIN 31652)

29.8 Heat transfer coefficient

If the heat transfer coefficient value is not known, you can take 15 to 20 (W/m^{2}k) as a guide value.

29.9 Heat transfer surface

If the values of the heat transfer surface are not known, you can take 10 * d * b to 20 * d * b as a guide value. This value is only needed if heat is lost due to convection.

d: Bearing diameter
b: Bearing width

29.10 Oil temperatures

Oil exit temperature:

- Normally approximately 60°C
- Upper limit for usual mineral oils: 70° to 90°C

Oil inlet temperature:

- With the usual cooler: 10°C lower than the output temperature
- With a very efficient cooler: 20°C lower than the output temperature

29.11 Mixture factor

The mixture factor that is used for the calculation according to DIN 31657 should lie between 0.4 and 0.6.

If the mixture factor M=0, this would mean that there is no mixture in the lubrication pockets, or that the exiting lubrication flow rate Q2 flows entirely into the next lubrication gap.

If the mixture factor M=1, this would mean complete mixture in the lubrication pockets.

29.12 Sizing the bearing clearance

Bearing clearance = d_bore - d_shaft

In general, a larger bearing clearance makes the bearing more stable and allows it to cool more effectively. However, it also results in reduced load capacity.

- **Suggestion according to Niemann**
  Proposal for metal bearings in mechanical engineering according to Niemann, Volume I, Table 15/2, [8].

  The following values should be applied for other materials:

Cast iron bearing	0.001 * d
Light metal bearing	0.0013 * d
Sintered bearing	0.0015 * d
Plastic bearing	0.003 * d

d : Bearing diameter

- **Proposal according to ISO 7902**
  Proposal for metal bearings in mechanical engineering according to ISO 7902, Part 3, Table 4, [69].
  In this sizing method, you can either use the proposal according to ISO 7902 or calculate the clearance from a predefined output temperature (only where the lubricant is used to dissipate the heat).
- **Proposal according to DIN 31652**
  Proposal for metal bearings in mechanical engineering according to ISO 7902, Part 3, Table 4, [69], as the DIN standard does not contain a proposed value.
  In this sizing method, you can either use the proposal according to ISO 7902 or calculate the clearance from a predefined output temperature (only where the lubricant is used to dissipate the heat).
- **Proposal according to DIN 31657**
  Proposal for plain bearings in mechanical engineering according to DIN 31657, Part 4, [70].
  In this sizing method you can either use the proposal according to DIN 31657 or calculate the clearance from the entered output temperature.
- **Proposal according to K. Spiegel**
  Proposal for clearance according to K. Spiegel: Goettner equation
  Relative bearing clearance = (2.5+50.0/D)/1000.0 "with d [mm]"
  Bearing clearance in mm: (2.5+50.0/D)/1000.0*D

29.13 Sommerfeld number

You must calculate the Sommerfeld number because it is an important characteristic value for plain bearings.

A Sommerfeld number > 1 occurs in heavily loaded bearings at the
limit for b/d: $0 < b/d \leq 2$

A Sommerfeld number < 1 occurs in quickly rotating bearings at
the limit for b/d: $0.5 < d/b \leq 2$

d: Bearing diameter
b: Bearing width

29.14 Bearing width

Reference value for bearing width as defined in Niemann, Volume I, Table 15/1, [8]
Normal range: b/d = 1 to 2

Reference value for bearing width according to ISO 7902, [69]
Normal range: b/d = 0.125 to 1

Reference value for bearing width according to DIN 31652, [72]
Normal range: b/d = 0.01 to 5

Reference value for bearing width according to DIN 31657, [69]
Normal range: b/d <= 0.8, b/d = 0.8 is assumed for the calculation of Qp

d: Bearing diameter
b: Bearing width

29.15 Permissible lubricant film thickness

The suggested value is taken from ISO 7902, Part 3, Table 1, [69] or DIN 31652, Part 3, Table 1, [72].

The values in this table are all empirical values. They assume that the average roughness height is < 4*μ*, there is a small form error and that the lubricant is to be filtered appropriately.

The proposed value according to Niemann is taken for DIN 31657, because no value is present in this standard.

A value is determined according to the diameter, in the case of the proposed value according to Niemann.

30 Hydrodynamic Plain Thrust Bearings

The DIN standard has two different methods for calculating hydrodynamic plain thrust bearings, according to type.

- Calculation of pad thrust bearings according to DIN 31653 [73]: This standard applies to bearings that have fixed sunken surfaces for lubrication (see Figure 30.1) which are separated from the rotating disks by a film of lubricant.
- Calculation of tilting-pad thrust bearings according to ISO 12130, DIN 31654 [74]: This standard applies to bearings that have fixed sunken surfaces for lubrication (see Figure 30.2) which are separated from the rotating disks by a film of lubricant.

If you do not consider the influence of the center of pressure on the tilting-pad thrust bearings, the same calculation procedure is described in both standards, which is why it is described here only once. However, any significant variations to these two standards will get a special mention here.

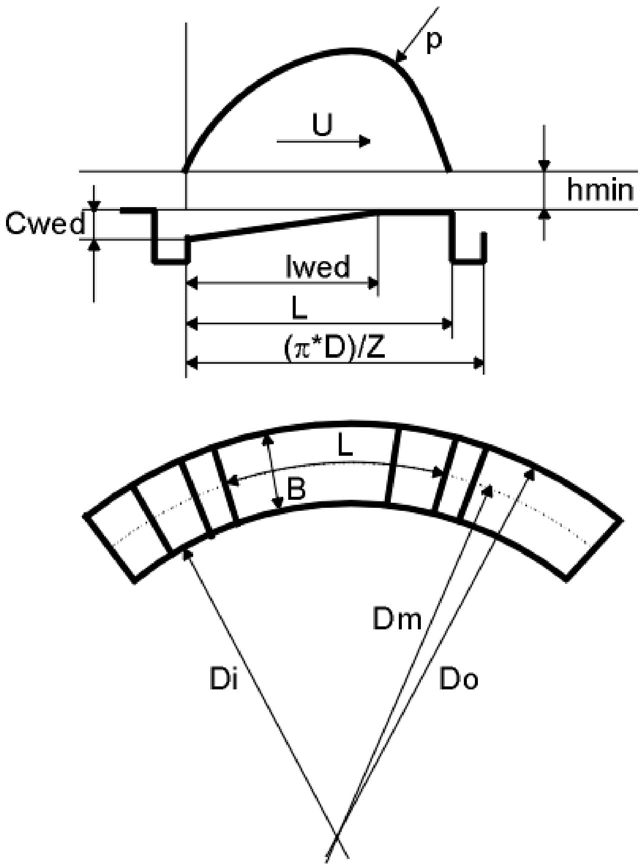


Figure 30.1: Pad thrust bearings as described in DIN 31653

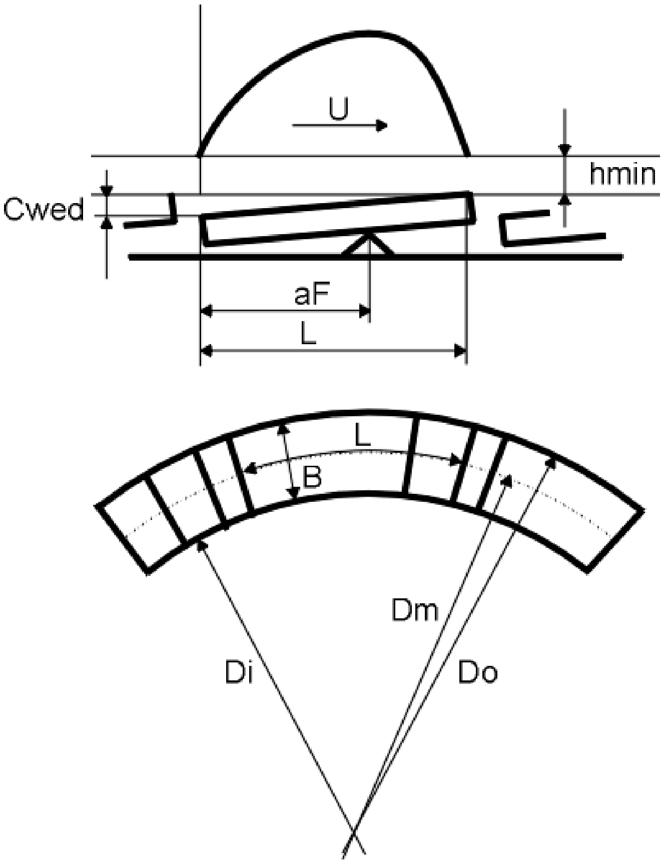


Figure 30.2: Tilting-pad thrust bearings according to ISO 12130, DIN 31654

30.1 Calculation

Essentially, both calculation procedures are based on the equation used to ascertain the thermal balance in the bearing. You can use either convection or circulatory lubrication in this calculation.

- Non-pressure-lubricated (self-lubricating) bearings dissipate heat out to the surrounding environment by convection. According to the standard, the coefficient of thermal expansion kA lies between 15 . . and 20 W/(m^2*K). In the program, the default value is 20W/(m^2*K), but you can change this as required.
- Pressure-lubricated bearings mainly dissipate heat through the lubricant. In this case, you must define a mixture factor M, which must lie between 0 and 1. Experience shows that this usually lies between 0.4 and 0.6. The default setting is 0.5, but this can be changed.

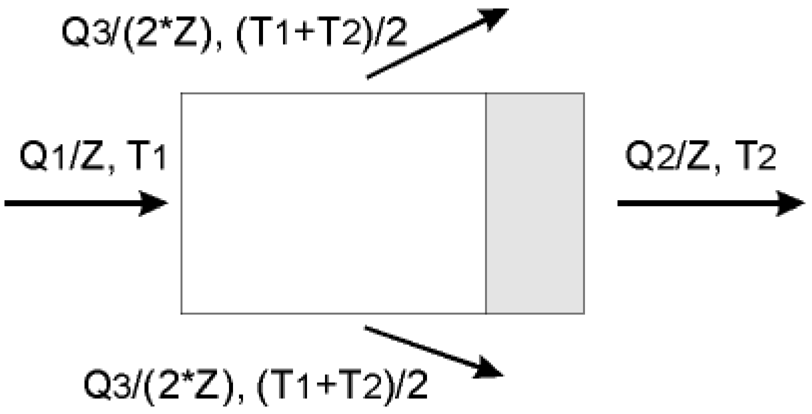


Figure 30.3: Segment lubricant and heat levels

These calculations provide values for the friction power, the lowest lubricant film thickness and the operating temperature. They also calculate the lubricant flow rate for circulatory lubrication.

The bearing force (in idleness) is only used to determine the lowest admissible lubricant film thickness, and is otherwise irrelevant. The load coefficient, friction coefficient and lubrication flow rates are calculated according to the formulae (not the diagrams or tables) provided in DIN 31653/31654 Part 2. The hmin/Cwed ratio for tilting-pad thrust bearings is calculated from the support position of the tilting-pad aF*. The formula for this is given in ISO 12130 Part 2 or in DIN 31654 Part 2.

30.2 Sizings

You can also calculate the bearing force (nominal load), but before you can do this, you must enter all the other input values. The bearing force is then calculated using the value you specified for the lowest lubricant film thickness, hlim.

The lowest possible lubricant film thickness hlim can be calculated according to DIN 31653 or 31654. This lowest possible lubricant film thickness is calculated depending on sliding velocity, average diameter and loading, according to the formula.

For convection:
If you do not know the value for the heat transfer surface, you can use a formula for approximation as defined in the standard:
A = (15 to 20) * B * L * Z
Click the Sizing button next to the surface value input field to calculate this value using the formula A = 15 *B *L *Z.

For circulatory lubrication:
Experience shows that the exit temperature is between 10 and 30 K higher than the entry temperature. Click the Sizing button next to the exit temperature to calculate a default value with a 10 K temperature difference.

30.3 Calculation of volume-specific heat

In the dialog you see by selecting **Calculations > Settings**, there are two methods you can use to calculate volume-specific heat:

- **Take into account dependence on temperature**
- **Simplified assumption (as in ISO 7902)**: 1.8*10^6J/(m3*K)

30.4 Limiting values in the calculation

The standards only apply to laminar flow in the lubrication gap. For this to happen, the Reynold number must lie below the critical value of 600.
These results are also checked for highest permissible bearing temperature, Tlim, the smallest possible lubricant film thickness, hlim, and the specific bearing load. These limiting values are defined in DIN 31653/ISO 12130/DIN 31654 Part 3.

31 Answers to Frequently Asked Questions

31.1 Intersecting notch effects

If at all possible, notch effects, for example in a shoulder with an interference fit, should not be overlapped when the shaft is designed. However, if this does happen, in the worst case scenario, the FKM Guideline should be applied to calculate the overall notch effect coefficient K_f:

$$K_f = 1 + \left(K_{f1} - 1\right) + \left(K_{f2} - 1\right)$$

from part notch effect coefficients K_{f1} and K_{f2} . In KISSsoft, this situation can be resolved by selecting **Own input** for the notch effect (see chapter 26.5.14, Cross-section types) of a free cross section (see chapter 25.2.7.1, Free cross section (single notch)).

The overall notch effect coefficient can then be calculated as follows:

1. Two cross sections (for example, A-A and B-B) are defined with the same Y-coordinate.
2. Cross section A-A is calculated by selecting notch type Kf1 (for example, shoulder). The notch factors are displayed directly in the Elements Editor (see chapter 25.1.4, Element Editor).
3. The process described in 2. is then repeated for cross section B-B.
4. The resulting notch factors for both these notches are noted down, and the notch factors Kf are calculated according to the formula given above.
5. Now, both cross sections (A-A and B-B) are deleted, and a new free cross section C-C with the same Y-coordinate is added. Open the Element Editor and select **Own Input** for the notch effect. Then, enter the overall notch effect coefficients calculated in point 4.

31.2 Notch effects on hollow shafts

All the notch factors described in the standards have been determined for solid shafts. No data is available for hollow shafts. KISSsoft calculates the nominal stresses for hollow shafts using the section modulus and taking into account the internal diameter.

31.2.1 Notches on the outer contour

For "small" internal diameters, the error due to calculating notch effect values for solid shafts is relatively small. You can then use the results as approximations. However, when "large" internal diameters are involved, you must correct the notch effect values.

According to the FKM Guideline 2020, 7th Edition, you cannot accurately calculate the notch effect values for a round shaft that has a longitudinal bore for bending and tension/compression using the notch effect coefficients for a round solid shaft. You should use the notch effect coefficient for a round solid shaft for torsion and round shafts that have a circumferential notch, shoulder or cone. Use the nominal stress value for a round shaft that has a longitudinal bore.

31.2.2 Notches on the inner contour

You cannot use these calculation methods to determine the notch factors of notches on the inner contour.

31.3 Fatigue Limits for New Materials

If you want to add a new material to the database, you must enter its infinite life strengths, and also the yield point and tensile strength.

Hänchen gives

$$\sigma_{bW} \approx 0.495 \cdot \sigma_B$$

an approximation of the bending fatigue limit, and also other approximations from different sources. For the tension/compression fatigue limit, this states

$$\sigma_{zdW} \approx 0.45 \cdot \sigma_B \quad oder \quad \sigma_{zdW} = 0.5\mathrm{K}\ 0.6 \cdot \sigma_B$$

, and for the torsion fatigue limit it states

$$\tau_{tW} \approx 0.55 \cdot \sigma_{bW} \quad oder \quad \tau_{tW} = 0.4\mathrm{K}\ 0.5 \cdot \sigma_{bW}$$

.

According to DIN 743, the following approximations can be made:

$$\sigma_{bW} \approx 0.5 \cdot \sigma_B \quad \sigma_{zdW} \approx 0.4 \cdot \sigma_B \quad \tau_{tW} \approx 0.3 \cdot \sigma_B$$

For through hardened steels (there can be different values for other material types), the FKM Guideline proposes:

$$\sigma_{W,zd} \approx 0.45 \cdot R_m \qquad \tau_{W,s} \approx 0.26 \cdot R_m$$

31.4 Taking double helical gearings into account in the shaft calculation

In the shaft analysis process, when you input cylindrical gear data in **Hand of gear**, you can select double helical gearing from the **selection list**. A gear with this characteristic always has an axial force 0 N. When double helical gearings are transferred from the gear calculation (the **Read data from file checkbox** is selected), the total width (= left side + gap + right side) is also transferred, as is the total power. The shaft calculation then takes both the gap and the effective gear teeth into account. This generally results in a very useful model.

If you require a more precise model, input the two halves of the gear individually, one angled to the right and the other angled to the left. Unfortunately, you cannot do this by transferring the data directly from the gear calculation.

V

Connections

Chapter 32 - 45

32 Cylindrical Interference Fit

The calculation is based on the complete DIN 7190-1 standard. It takes into account elastics with longitudinal, radial and oil interference fits. The following influences are included or can be calculated:

- Load in circumferential and axial directions.
- Load with bending moment and radial force.
- Calculate the maximum torque for a non-slipping fit. Note: If slip occurs in the fit, micro sliding will cause corrosion due to friction.
- Influence of centrifugal force.
- Verification of an elastic-plastic loaded interference fit as specified in DIN 7190-1 with predefined interference (stresses and strains are calculated only for the purely elastic case)
- Analysis of hubs with multiple interference fits
- Display stress curves (equivalent, tangential and radial stresses)
- Display tolerance fields:
  - only take into account allowances
  - take into account temperature and centrifugal force (without pressure)
  - take into account temperature, centrifugal force and pressure.

You can calculate the safety of the interference fit against sliding and the safety of the shaft material and the hub against fracture and yield point. The calculation also takes into account the effect of centrifugal force on the expansion of the interference fit and on the stresses in the shaft and hub. The tolerance system specified in DIN 7151 (e.g. input of a diameter with tolerance 60 H7/f6) has been integrated to make it easier to input data. In addition to entering the tolerance directly, the tolerance pairing can be calculated on the basis of the minimum safety against sliding and permissible types of material stress. Input the surface roughness based on a quality as defined in ISO 1302.

Calculating the pressure: For elastic materials, this is calculated according to the theory of mechanics for thick cylinders under internal pressure and thick cylinders under external pressure (e.g. [75], page 399, or [8]).

The influence of speed:
Speed is taken into account in accordance with the theory of a rotating cylinders, as defined by Dubbel [26], C41.

The amount of embedding is based on the formulae in DIN 7190-1.

Calculation of equivalent stress:

You can select the hypothesis of equivalent stress calculation method by selecting **Calculations > Settings** . The "Settings" chapter provides more information about this function.

Bending moment and radial force: this takes into account the effect of a bending moment and a radial force on the pressure. The additional amount of pressure is calculated as follows:

$p_b = \frac{9}{2}\frac{M_b}{d_F l_f^{\,2}} \qquad p_r = \frac{F_r}{d_F l_F}$	(32.1)

So that there is no gaping, this additional pressure must also be lower than the minimum pressure on the connection ((p_b + p_r) < p_{min}).

These formulae have been taken from the technical literature according to Prof. Hartmann, "Berechnung und Auslegung elastischer Pressverbindungen", pages 6 and 7.

Other data is taken into account in accordance with the following table:

Dismounting force: According to Niemann [76] and [77], p. 363

Transmission without slip: According to Kollmann [78], p. 59-64

$T_{grenz} = \frac{\pi}{2} D_F^3 \tau_R \frac{\tan h(k \frac{l}{D_F})}{k}$	(2.110)
$\tau_R = v_{ru} p$	(2.93)
$k = \sqrt{\frac{32}{1 - Q_A^2}\frac{G_A}{G_I}}$	(2.107)

T_{grenz}: Limit torque [Nmm]
D_F: Diameter of joint [mm]
τ_R: Shear stress [N/mm^2]
l: Length of interference fit [mm]
v_{ru} = adhesive coefficient [-]
p: Joint pressure (N/mm^2]
G_A, GI: Shear modulus shaft/hub [N/mm^2]
Q_A: Diameter ratio
k: Constant

Micro sliding: If the torque of an interference fit is increased continuously until it exceeds the micro sliding limiting value, local slip will occur at the position at which the torque is applied. As torque decreases continuously in the interference fit, the slip occurs only in one part of the interference fit length, even if the torque then increases again. This effect is called micro sliding (it involves one-sided movement of the shaft, back and forth in the hub). It can cause fretting. Further explanations and details about this calculation are given in "Welle-Nabe-Verbindungen" (Shaft-Hub Connections), by Kollmann [78].

Note about the calculation as defined by Kollmann:
The limit torque for micro sliding is calculated using equations 2.93, 2.107 and 2.110.

Mounting: You will find information about mounting in the report. The temperature difference for mounting is calculated in such a way that, even if the maximum interference is reached (worst case scenario) there will still be enough clearance in the joint. Define mounting clearance under "Settings". Here, the values for mounting the shaft at ambient temperature and for a deeply cooled shaft (shaft at approximately -150°C) are calculated.

Verification of an elastic-plastic loaded interference fit according to DIN 7190-1:

Requirements: EI = EA, nyI = nyA, n = 0, diI = 0

If all the prerequisites are fulfilled, as defined in DIN 7190-1, you can calculate the plasticity diameter DPA (diameter at which the plastic range ends) of the external component that is to be mounted. The corresponding joint pressure, and the ratio between the ring surface qpA and the overall cross section qA are also calculated. (Experiential limit according to DIN 7190-1 for heavily loaded interference fits in mechanical engineering qpA/qA <= 0.3)

32.1 Inputting Tolerances

Tolerances according to ISO/DIN: To input tolerances in this format: 60 H7/f6, take the following steps into consideration:

- Enter 60 (mm) as the "Diameter of joint".
- Enter H7 as the manufacturing tolerance for the hub or f6 as the manufacturing tolerance for the shaft.
- The program automatically checks whether the tolerances you specified actually exist, and whether you entered the data in the correct format.

Define own tolerances: Select the Plus button next to the tolerance field to display the current allowances for shaft and hub. You can then change these values as required.

32.2 Coefficients of friction

Tables 32.2 and 32.4 contain coefficient of friction values as defined in DIN 7190-1.

Materials	Coefficient of friction			
	dry		lubricated	
	v_{ll}	v_{rl}	v_{ll}	v_{rl}
E 335	0.11	0.08	0.08	0.07
GE 300	0.11	0.08	0.08	0.07
S 235JRG2	0.10	0.09	0.07	0.06
EN-GJL-250	0.12	0.11	0.06	0.05
EN-GJS-600-3	0.10	0.09	0.06	0.05
EN-AB-44000 and following	0.07	0.06	0.05	0.04
CB495K	0.07	0.06	-	-
TiAl6V4	-	-	0.05	-
v_{ll}: in longitudinal direction - loosening				
v_{rl}: in longitudinal direction - sliding				

Table 32.1: Coefficients of friction for longitudinal fits under continuous stress as defined in DIN 7190-1.

Material pairs, Lubrication, Joining	**Coefficients of friction** v_r, v_{rl}, v_u
Steel-steel pairing:	
Oil pressure connection, normally joined with mineral oil	0.12
Oil pressure connection with degreased contact surfaces	0.18
joined with glycerin	
Shrink fit normally after warming the	0.14
External component up to 300°C in electrical oven	
Shrink fit with degreased contact surfaces after	0.20
Heating up to 300°C in electrical oven	
Steel-cast iron pairing:	
Oil pressure connection, normally joined with mineral oil	0.10
Oil pressure connection with degreased contact surfaces	0.16
Steel-MgAl pairing, dry	0.10 to 0.15
Steel-CuZn pairing, dry	0.17 to 0.25
v_r: Sliding	
v_{rl}: in longitudinal direction - sliding	

v_u: in circumferential direction	

Table 32.2: Coefficients of friction for radial interference fits in the longitudinal and peripheral direction subjected to sliding as defined in DIN 7190-1.

32.3 Variable external diameter of the hub

In the case of a stepped external diameter, a single equivalent diameter is determined from the diameters and lengths. This value is then used to calculate the stiffness of the external component. To input this value, click on the Plus button to the right of the "Hub outside diameter Da" input field.

32.4 Convert external pressure with multiple interference fit

The effect of outside pressure can be taken into account for superimposed interference fits. This additional pressure is defined through a series of sequential interference fits and is calculated by the software (no direct user input necessary). Input the value in the **Multiple interference fit** window by clicking the appropriate flag. Click the "Definition" button to open the input window in which you define the interference fits.

General remarks:

- the first two lines correspond to the innermost interference fit. The lines are merely displayed in the table for completeness' sake. The values users can edit are the hub's external diameter and tolerances.
- The external diameter of each ring is also used as the internal diameter of the next ring.
- When a new ring is added, the following default values are used:
  - External diameter = internal diameter + 50 mm
  - The Material and Surface roughness are the same as for the original hub
  - The operating temperature is the temperature of the hub
  - The inner diameter has the same tolerance class as the shaft (for example, s6)
  - The external diameter has the same tolerance class as the hub (for example, H7)

The pressure on the hub is calculated for the following three cases of tolerance values of all the elements:

- The average tolerance at each contact surface
- The worst case, i.e. the maximum oversize at each contact surface

- The best case, i.e. the minimum oversize at each contact surface

32.5 Materials

In the selection list, you can select materials as specified in the standard.
Select the Plus button next to the selection list to open the Materials window. Select **Own input** to open a new screen in which you can enter the material data manually.

► **Note**

You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that they can also be used in subsequent calculations.

32.6 Settings

You make the basic entries for the calculation, joining temperature, safeties etc. in the module-specific settings (**Calculations**> **Settings**).

The input and selection fields in the module-specific settings are explained below:

- **Select equivalent stress** according to [6], with von Mises stress:

  $$\sigma_v = \sqrt{\sigma_1^2 + \sigma_2^2 - \sigma_1 \cdot \sigma_2}$$

  According to [78], page 13, with the shear stress hypothesis:
  $\sigma_v = \max(|\sigma_{\phi\phi}\ \sigma_{rr}|,|\sigma_{\phi\phi}\sigma_{rr}|)$

- **Assembly temperature shaft/hub (press on)**
  The press-on force can be calculated from the assembly temperatures.
- **Coefficient of friction (press on)**
  You can define the μe/μa ratio here. This ratio describes the relationship between the coefficient of friction for press on and the axial coefficient of friction. The default value is 1.3.

  Note:

  Only values for forcing off are given in DIN 7190-1. Niemann Volume I (1981 edition), Table 18/3 p. 363, also gives values for press on.

- **Coefficient of friction (press off)**
  You can define the μll/μa ratio here. This ratio describes the interrelationship between the coefficient of friction for press off and the axial coefficient of friction. The default value is 1.6.
- **Calculation of joining temperature(lateral interference fit)**
  The mounting clearance can be entered either depending on the diameter of the joint df (modified if warming is taken into account) or as a constant mounting clearance. The joining temperature of the external component is then calculated accordingly. You can also input the temperature of shaft during joining.

  This temperature and the mounting clearance are then used to calculate the hub joining temperature. The hub joining temperature is only output in the report if the shaft temperature during joining lies between -273 °C and 20 °C.

  In the settings you can select whether the shaft is sub-cooled and/or the hub is heated before joining. If you have marked the checkbox, the respective value for the coefficient of thermal expansion according to DIN 7190 is used. By default, the checkbox is set to sub-cooling of the shaft.
- **Required safeties**
  In the Settings, you can input the required safeties against sliding, yield point and fracture. The required safeties are used to determine the searched values in the sizings. The required safeties against plastic deformation are used to define the plasticity diameter that must be set when an interference fit is placed under plastic-elastic stress.
- **Calculate material strength with wall thickness as raw diameter**
  If you set this checkbox, the strength of the hub material is determined using the wall thickness instead of the raw diameter.
- **Allow calculation of elastic-plastic loaded interference fits**
  If you set this checkbox, the calculation is also performed for elastic-plastic stress (according to DIN 7190-1), otherwise, only elastic stress is included.
- **For negative interference during operation, continue the calculation and document the allowances**
  If there is a negative allowance during operation, an error message is displayed. This usually interrupts the calculation, because this value would mean that the fit was no longer an interference fit. Set this flag to force the calculation to continue despite the error message so that you can display allowances for higher operating temperatures. Only the section with the allowances calculated for operation is then displayed in the report.
-

32.7 Sizings

Possible shaft/hub tolerance pairs can be displayed and adopted as specified in ISO/DIN.

KISSsoft has a very convenient sizing function that you can use for suitable tolerance pairs. Standardized tolerance pairs are stored in the M01-001.dat file. Click the Sizing button next to the tolerance input field to open the sizing screen.

Based on the nominal required safety (which you can change in Settings), you can determine all the tolerance pairs which fulfill the requirements (sufficient safety against sliding, sufficient safety against fracture and yield point). The results are then displayed in a list.

Torque, axial force, joint diameter, and length of interference fit
KISSsoft can size the maximum transmittable torque, transmittable axial force, required length, and diameter (according to the required safety values you have entered in the Settings).

33 Conical Interference Fit

Use this module to calculate the service reliability of a conical interference fit. Determining mounting conditions.

The following **calculation methods** are available:

- Method as defined by Kollmann [78], Verification and sizing.

- Method as defined in DIN 7190-2:2017

► **Note:**

The cone angle is the angle β between the flank of the cone and its axis. The opening angle α of the cone is twice the size of the angle of taper.

Notes for the calculation:

- All known investigations focus on external and internal components made of materials that have the same Young's modulus. (Kollmann)
- If you select the method described in DIN 7190-2, and the shaft and hub are made of different materials, the material with the higher Young's modulus should be used for the shaft material.
- Conical interference fits must always have a stop at the upper end. For this reason, the program only deals with this situation.
- Conical interference fits are normally clamped axially with a bolt. You must check the joint carefully by measuring the length. Simply tightening it with a torque wrench is not accurate enough. Conical interference fits are only joined by pressing them on in exceptional circumstances.
- To perform the calculation according to Kollmann, without clamping using a bolt, a special coefficient of friction is used in the longitudinal direction. The coefficient of friction is calculated according to a series of articles in Antriebstechnik 12 (1973), issued by the company Voith.

$$\mu^* = \mu \cdot \sqrt{1 - \frac{\tan^2 \beta}{\mu^2}}$$

Adhesive coefficient for sliding in the longitudinal direction: Coefficient of friction as detailed in investigations by Galle (see Kollmann [78], Table 2.20):

Material pairing	Previous load	Coefficient for stiction
Ck60/16MnCr5	-	0.299
42CrMo4/16MnCr5	-	0.269
31CrMoV9/31CrMoV9	-	0.247
Ck60/16MnCr5	U	0.407
42CrMo4/16MnCr5	U	0.297
31CrMoV9/16MnCr5	U	0.375
31CrMoV9/31CrMoV9	U	0.468
Ck60/16MnCr5	W	0.357
42CrMo4/16MnCr5	W	0.472
31CrMoV9/31CrMoV9	W	0.387

Load classifications:

-	none
U	Circumferential bending load
W	Fatigue torsion load

No adhesive coefficients for other combinations of materials are available, so you will have to estimate them.

Adhesive coefficients for radial interference fits in longitudinal and circumferential direction of sliding: Coefficient of friction as described in DIN 7190 [79], Table 4 (see chapter 32.2, Coefficients of friction).

33.1 Calculation

Sizing according to Kollmann: If you select this method, the maximum angular deviation γ_{max}, the pressing distance for mounting a_f, and the joining force, can be sized using the required safeties and the specified load and geometry.

Verification according to Kollmann: The transmittable torque is calculated for no angular deviation and for maximum angular deviation.

You can also input the pressing distance for mounting af or the mounting pressing force Ffmin.

The safeties against the yield point and against sliding are also calculated in the verification process.

The safeties defined by Kollmann include the required safeties, which you can define in the Settings tab. To display it, select **Calculation** → Settings.

Method as defined in DIN 7190-2:2017: In this calculation method, the sliding moment, joining force, and axial force are calculated for the state at which all the elements are already joined, from the input torque, mounting without force and all the geometry data. The safety against sliding is calculated in this way. If different external hub diameters or different internal shaft diameters are present, these can also be defined as slices (max. 7). The values are calculated individually for each slice and then added together. The maximum value is used for the pressure in the joint.

$$S_R = T_R / T_{\max}$$

In the case of a central load application, the safety against sliding S_R should have a minimum value of $S_{Rmin} > 1.3$.

In addition, the effective equivalent stresses are calculated if you enter a joining force manually for mounting p_{FA}. The maximum equivalent stress is compared with the yield point of the particular material.The safety is defined in this way. The maximum equivalent stresses appear every time for the inner diameter of the hub or shaft when a hollow shaft is being used.

$$S_P = R_{p0,2} / \sigma_{v\max}$$

The Safety against yield point S_P (plastic flow) should reach a value of $S_{Pmin} > 1.3$ so that the parts do not undergo plastic deformation when being assembled or disassembled.

33.2 Application factor

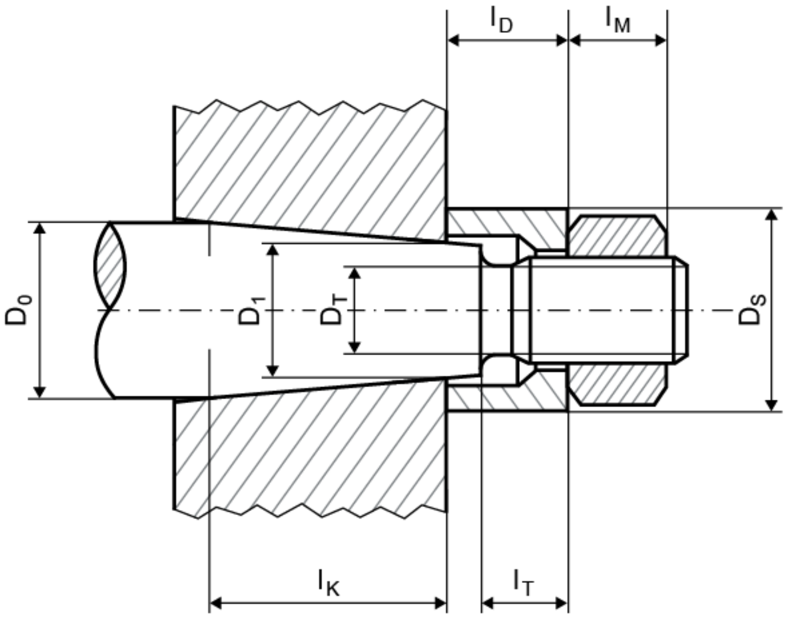
The application factor is defined in the same way as in the cylindrical gear calculation:

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 33.1: Application factor as used in calculations according to DIN 6892. You will find more detailed comments in DIN 3990, DIN 3991 and ISO 6336.

33.3 Axial spanning with nut

Axial spanning (tightening the nut) produces relative axial displacements which are applied to the individual parts. This causes transversal strain. and therefore increases the compacting pressure on the active surface. The required input values can be seen in the diagram below.



33.4 Variable external diameter of the hub

In the case of a stepped external diameter, a single equivalent diameter is determined from the individual diameters and lengths. This value is then used to calculate the stiffness of the outer part. To input the diameter, click the Plus button to the right of the input field for the external diameter of the hub D_{aA}.

33.5 Conicity

The cone angle can be determined in two ways, by clicking on the Plus button to the right of the input field for the half cone angle β:

- **by setting a conicity:**
  The conicity is defined as follows:
  $x = l/(D_0-D_1)$. x corresponds to the value to be input.

- **by selecting a morse taper shank:**
  Morse tapers are defined in DIN 228 and have a conicity of between 1:19.212 and 1:20.02.

33.6 Materials

In the selection list, you can select materials as specified in the standard.
If you select the Own input option, you can enter the material data manually. You can also enter and save your own materials directly in the database (see chapter 9, Database Tool and External Tables). This means they can also be used in other calculations.

33.7 Settings

Define the required safeties against sliding and against yield point in the module-specific settings (Calculation → Settings). The following aspects can also be taken into account:

If you have selected Calculate material strength with wall thickness as raw diameter, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

If you select the Consider pressure at both diameters (Kollmann) flag, the pressure at both the large and small cone diameters is taken into account. Otherwise, only the pressure at the largest diameter is used.

► **Note:**

This note only applies for the method according to Kollmann. The average diameter is used for the calculation according to DIN 7190-2.

Enter the **required safeties for sliding and yield point** under Settings. The required safeties define the values the system searches for during sizing.

For the method according to **Kollmann**, a minimum safety of 1.0 is used for the required safety against sliding. The default required safety against yield point value is 2.0.

DIN 7190-2 recommends that the safety against the yield point is set to 1.2, to ensure that the elements are not affected by plastic flow when they are disassembled hydraulically.

The safety against sliding for a central load application should be set to at least 1.2.

33.8 Sizings

KISSsoft can size the maximum transmittable torque, the permitted cone angle (for self-locking) and the length of interference fit for transmitting the maximum torque.

The torque and the length of interference fit are sized using the defined required safeties.

In a verification according to **Kollmann**, you can size either the pressing distance for mounting or the mounting pressing force, using the required safety against sliding.

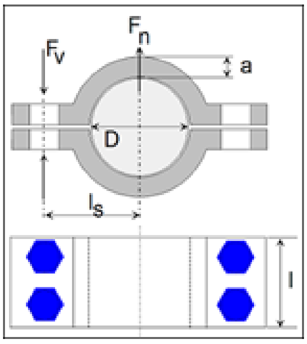
As specified in **DIN 7190-2**, the sizings are calculated using the required safety against sliding, apart from the joining pressure, which is sized using the required safety against yield point.

34 Clamped Connections

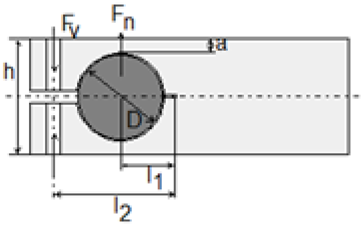
Clamped connections are only used to transfer low or medium torque (minor fluctuations).

There are two different configurations of clamped connections that can be calculated:

- Split hub
  In the case of a split hub, it is assumed that pressure is distributed uniformly across the whole joint. The pressure can be uniform surface pressure, cosine-shaped surface pressure or linear contact.



- Slotted hub
  We recommend you use as narrow a fit as possible, to ensure that the pressure is mostly of a linear nature (as hubs are also subject to bending). The calculation is performed for the least practical case of linear pressure.



▶ **Note**

Calculations of safety against sliding and surface pressure are described in literature by Roloff Matek [80]. The calculation of bending is performed as specified by Decker [81].

34.1 Calculations

Split hub:

Depending on the type of surface pressure, an additional factor for surface pressure and safety against sliding is used to calculate a split hub: This varies according to the type of surface pressure:

- K = 1 → uniform surface pressure
- K= π^2/8 → cosine-form surface pressure
- K = π/2 → linear contact

In KISSsoft, you can select the type of pressure in a selection list in the Basic data window.

Formula for surface pressure:

$$pF = \frac{2 \cdot KA \cdot T \cdot SH \cdot K}{\pi \cdot D^2 \cdot ls \cdot \mu}$$

Formula for safety against sliding:

$$SH = \frac{Fkl \cdot i \cdot \pi \cdot D \cdot \mu}{2 \cdot KA \cdot T \cdot K}$$

Formula for calculating bending:

$$\sigma B = \frac{0{,}2 \cdot Fkl \cdot i \cdot ls}{2 \cdot Wb}$$

Slotted hub:

Formula for surface pressure:

$$pF = \frac{i \cdot Fkl \cdot l2}{D \cdot l \cdot l1}$$

Formula for safety against sliding:

$$SH = \frac{Fkl \cdot i \cdot D \cdot \mu \cdot l2}{KA \cdot T \cdot l1}$$

Formula for calculating bending:

$$\sigma B = \frac{0{,}3 \cdot Fkl \cdot i \cdot (l2 - l1)}{Wb}$$

Description of codes:

pF: Surface pressure [N/mm2]

KA: Application factor

T: Nominal torque [N]

SH: Safety against sliding

K: Correction factor surface pressure

l: Joint width [mm]

D: Diameter of joint [mm]

lS: Distance bolt to shaft center [mm]

l1: Distance normal force to center of rotation [mm]

l2: Distance clamp load to center of rotation [mm]

µ: Coefficient of friction

σB: Bending stress [N/mm2]

Fkl: Clamp load per bolt [N]

i: Number of bolts

Wb: Moment of resistance [mm3]

34.2 Sizings

In these calculations, you can size the transmissible nominal torque, clamp load per bolt F_{Kl} and number of bolts to suit a predefined required safety value.

34.3 Settings

Required safety against sliding SSH: You can predefine the required safety against sliding SSH in the module-specific settings. It is used for sizing the transmissible nominal torque T, the clamp load per bolt F_{Kl} and the number of bolts used i.

Coefficient of permissible surface pressure p_{Fact}: If you have selected cast iron as the hub material, the tensile strength is multiplied by the coefficient p_{Fact} to calculate the permitted pressure.

pzul =pFact*Rm (default value ~ 0.35 for an interference fit)

For all other materials, this coefficient multiplied by the yield point is used to calculate the permitted pressure.

pzul =pFact*Rp (default value ~ 0.35 for an interference fit)

34.4 Materials

In the selection list in the Materials window, you can select materials as specified in the standard. Select the Plus button next to the selection list to open the Materials window. If you have selected Own input, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that they can also be used in subsequent calculations.

35 Keys

Keys are by far the most commonly used shaft-hub connections. In particular, they help to transmit the torque. Their geometry has long been standardized according to DIN 6885 [82]. However, to ensure adequate safety levels are achieved when transmitting torque, you always had to contact secondary sources of technical literature [8]. The DIN standard 6892 [83] documents the different calculation methods that can be used for feather key connections.

You must perform **two checks** for key connections:

1. Check the torque transmission by monitoring surface pressure on the shaft, hub and key.
2. Check the endurance limit of the shaft due to the notch effect caused by the key way. This effect is already described in DIN 743 [63]. We recommend you use this standard to analyze the shaft strength rather than DIN 6892.

Special characteristics of calculations according to DIN 6892:

- Key connections are usually combined with a light interference fit. The calculation takes into account the decrease in torque on the key due to the interference fit.
- The calculation proves the nominal torque as well as the actual pitch torque over the entire operating period. The endurance limit calculation based on the nominal torque also includes the number of load changes, which experience has shown to have a significant and damaging effect on the key.
- The way in which load is applied or discharged has a considerable effect on the operating safety of keys. This effect is taken into account by using a wide range of load distribution coefficients.
- The values for the permitted pressure are derived from the yield point. As a result, you can derive this for common and more unusual materials as specified in the standard. The hardness influence coefficient is used to take the surface treatment into account.

Calculation Method B as defined in DIN 6892 recommends you use a differentiated calculation method to prove the service reliability of key connections. Method C has been greatly simplified.

35.1 Main window

In the Geometry input screen, select Standard in the selection menu to select these **calculation standards**:

- Own input

- DIN 6885.1: 1968 (Standard)
- DIN 6885.1: 1968 Form G, H, J
- DIN 6885.2: 1967
- DIN 6885.2: 1956
- ANSI B17.1-1967 (R1998) Square
- ANSI B17.1-1967 (R1998) Rectangular

When you select a calculation method and define the load, the system calculates the safeties for the shaft, hub and key (surface pressure) and the key (shearing).

The following **calculation methods** are available:
DIN 6892 B [83]
DIN 6892 C [83]

The calculation takes into account the tolerances of the key radii and the direction of force. You can also enter your own value for the number of keys and the application factor.

Explanatory information about the figure (see Figure 35.1):

→	Application or removal of torque
o	Start of key
F_u	Center of force application point on hub

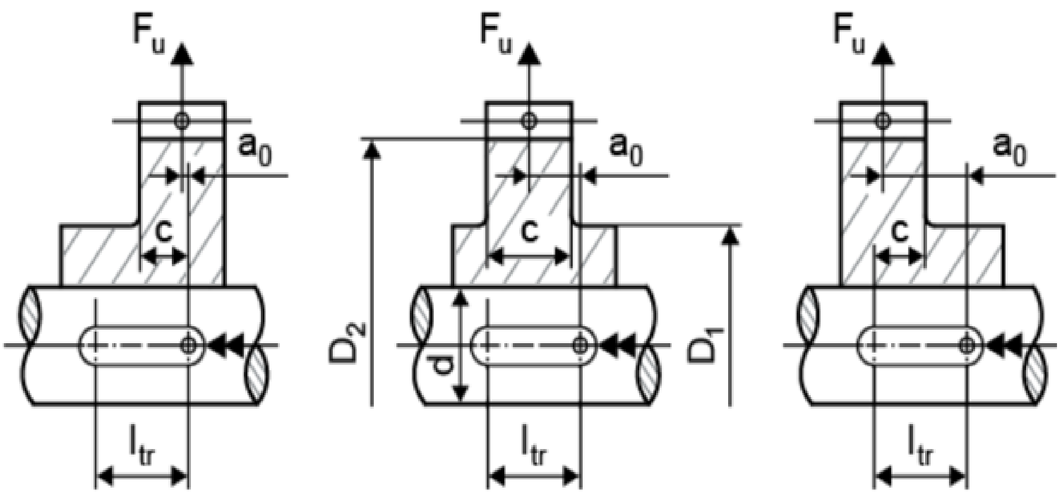


Figure 35.1: Key: load introduction

Supporting key length: The supporting key length is defined according to DIN 6892 as follows:
helical key - (A, E, C according to DIN 6885) --> $l_{tr} = l_{eff}$ - b
Spur gear key - (B, D, F, G, H, J according to DIN 6885) --> $l_{tr} = l_{eff}$

	l_{eff}	Actual key length
	l_{tr}	Supporting key length
	b	Key width

Frictional torque: Key connections are usually combined with a light interference fit. The calculation takes into account the decrease in torque on the key due to the interference fit. This influence is only taken into account for calculations according to DIN 6892 B.

Number of load peaks: To determine the safety for the maximum torque, you must enter the approximate number of load peaks. This influence is only taken into account for calculations according to DIN 6892 B.

35.1.1 Additional inputs for DIN 6892 Method B

If you select the calculation method according to DIN 6892 B, you can input these values in the Geometry screen:

- Chamfer on shaft
- Chamfer on hub
- Small external diameter of hub D_1
- Large external diameter of hub D_2
- Width c for external diameter D_2
- Distance a_0 (see Figure 35.1)
- Torque curve: indication of whether this is alternating torque.

► **Note:**

If **alternating torque** is present, you can also define the backwards torque here. If this backwards torque is greater than the minimum effective frictional torque (T_{maxR} > T_{Rmin}*q; q=0.8), the load direction changing factor f_w is set to 1.
If (T_{maxR} > T_{Rmin}*q and T_{max} > T_{Rmin}*q; q=0.8), the maximum torque is therefore also greater than the minimum effective frictional torque. In this case, the frequency of the changes in load direction is taken into account when defining the load direction changing factor (from diagram: f_w<1)

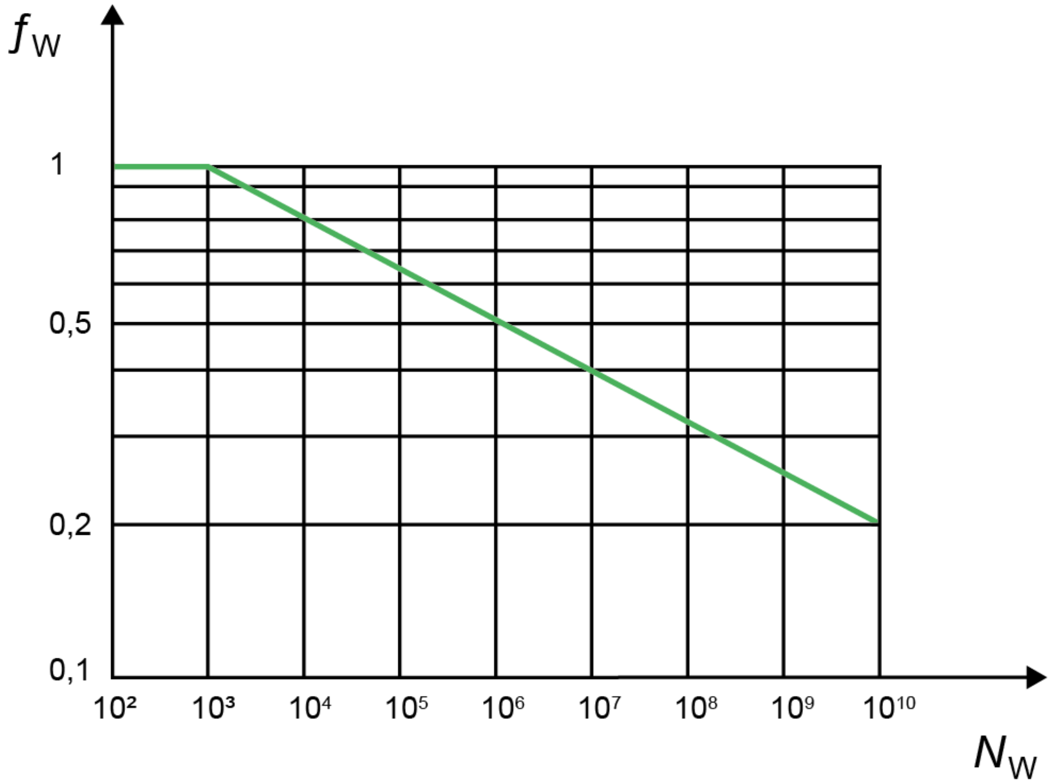


Figure 35.2: Load direction changing coefficient for alternating load, Figure 6, DIN 6892

- Use coefficient q = 0.5 to perform a calculation with the equivalent torsional moment. The formula corresponds to the formula with maximum torsional moment, therefore (T_{eqR} > T_{Rmin}*q; q=0.5).
- **Frequency of the changes in load direction:** In this case, you input the number of torque changes throughout the entire service life (but only if alternating torque is present).

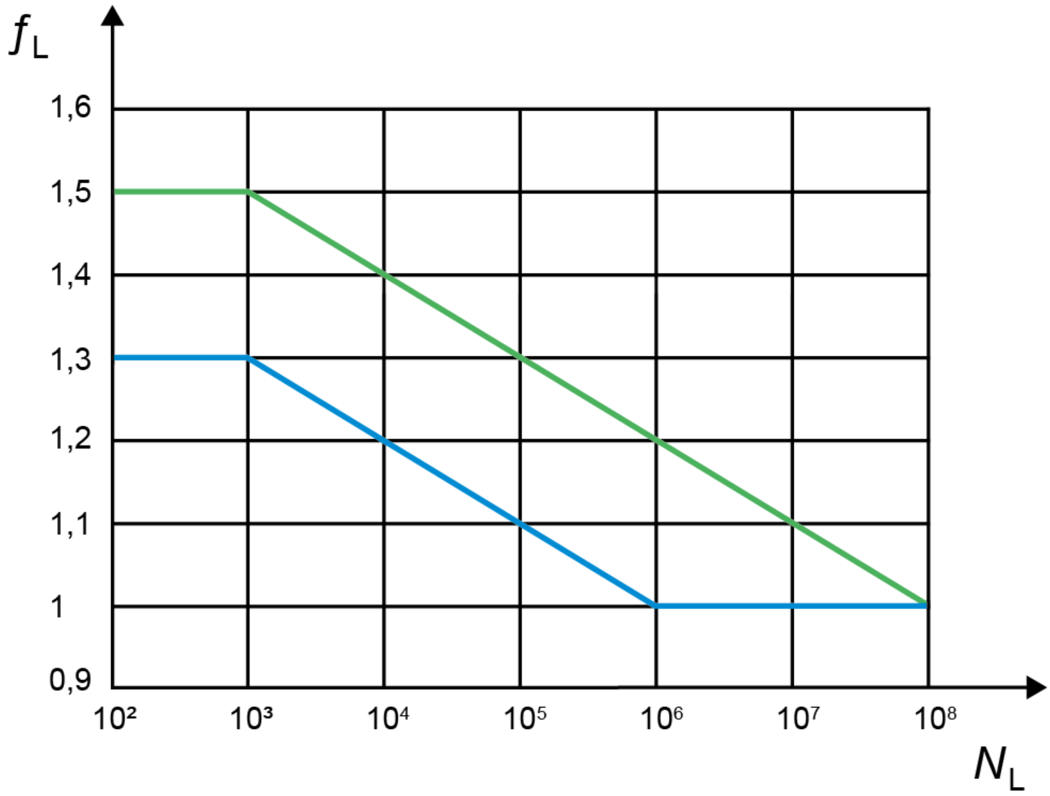


Figure 35.3: Peak load factor, Figure 7, DIN 6892

► **Note:**

The green line represents a ductile material and the blue line represents a brittle material.

35.2 Application factor

You define the application factor here in the same way as in the cylindrical gear calculation:

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25 or higher

Table 35.1: Proposals for the application factor for calculations according to DIN 6892. You will find more details in DIN 3990, DIN 3991 and ISO 6336.

Suggestions for the application factor from other sources: (see Table 35.2), (see Table 35.3)

Type of Machine	**Characteristic operational behavior**	**Type of Shocks**	**Operating factor**
Turbine, blower	Uniformly rotating movements	Slight	1.0. . . 1.1
Internal combustion engine	Reciprocating movements	Medium	1.2. . . 1.5
Presses, saw frames	Reciprocating impacting movements	strong	1.6. . . 2.0
Hammers, stone crushers	Impacting movements	Very heavy	2.1. . . 3.0

Table 35.2: Application factor according to Roloff/Matek [84].

	surfaces pressed together	**surfaces sliding against each other without load**	**surfaces sliding against each other under load**
constant load	1.0	2.0	6.0
Pulsating load, average shocks	1.5	3.0	9.0
Alternating load, average shocks	3.0	6.0	18.0
Pulsating load, heavy shocks	2.0	4.0	12.0
Alternating load, heavy shocks	6.0	8.0	36.0

Table 35.3: Application factor that takes into account the load behavior as described by Professor Spinnler [85].

35.3 Contact coefficient

Contact coefficient as defined in DIN 6892 [83]:

$K_v=1/(i*\phi)$

$\phi = 1$	for one key
$\phi = 0.75$	for two keys, to calculate the equivalent surface pressure
$\phi = 0.9$	for two keys, to calculate the maximum surface pressure
More than two keys are unusual.	

KISSsoft calculates the contact coefficient based on the number of keys.

35.4 Own inputs

If you set the Own input flag, you can enter your own data for the keys' geometry, that differs from the values specified in DIN 6885. To do this, open the Geometry input screen and set the standard to Own input. Then click on the Plus button.

► **Note**

If you already know the upper and lower allowance, you must enter the mean value for the chamfer and the two groove depths. The value for the peak incline a is only to be defined for key forms G, H, and J, according to the DIN 6885.1 standard.

35.5 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials).

35.6 Materials

You can select materials according to the standard in the selection list in the Materials screen.
If you have set the Own input flag, a new dialog is displayed. This contains the material data used in the calculation, which you can specify to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that they can also be used in subsequent calculations.

35.7 Settings

- Calculation method: Here you can select either **DIN 6892 Method B or Method C**. The default setting is Method B, because Method C has been greatly simplified.
- **Required safety:** In the Settings, you can specify the required safety for the individual parts of the connection. According to Niemann, Volume I, 5th Edition (2019), the minimum safety for ductile materials is set in the range 1.0…1.3 and the minimum safety for brittle materials is set in the range 1.3…2.0. The required safeties define the values the system searches for during sizing.
- Support factors for shaft, hub and key: The support factors given in DIN 6892 differ according to the materials and parts involved. The factors vary within a specific range. The minimum values are used in KISSsoft. These values can be overwritten by setting the flags.

- Take pressure on key into account: When sizing the transmittable torque (Sizing button) you can also take the permitted pressure on the key into account.
- Calculate material strength with wall thickness as raw diameter: When the strength values for the hub are being set, either the external diameter is used (hub was turned from solid) or the wall thickness of the hub is used (hub was heat treated as a ring).
- Symmetrical arrangement of 2 keys: In the key arrangement graphic, the default setting is for 2 keys to be displayed, offset by 120° from each other. You can use this option to arrange the 2 keys symmetrically.

35.8 Sizings

During the sizing process, the required value is defined such that the required safety (specified in the dialog you see by selecting Calculations→ Settings) is only just achieved. To view the results in the lower part of the main window, run the calculation after the sizing process is complete.

Possible sizings:

- transmittable nominal torque
- necessary length of key way in shaft and hub

The "Keys" tutorial has been created specially to describe how you verify these keys.

36 Straight-sided splines

Straight-sided spline shaft connections are often used for adjustable, form-closed shaft-hub connections.

Main areas of use: Vehicle gear trains, machine tools.

KISSsoft calculates the load on the shaft and hub (surface pressure) for straight-sided splines. This calculation, along with defining the safeties, is performed as described in classic technical literature ([8]). The calculation defined by Niemann forms the basis of DIN 6892 (key calculation).

36.1 Standard profiles

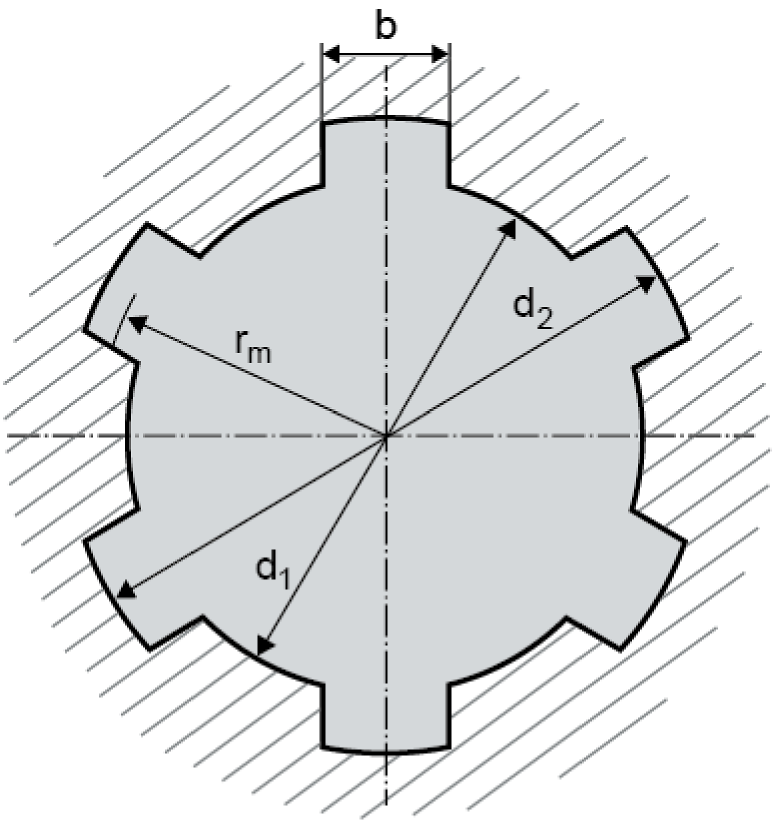
You can select one of these standards from the selection list:

- **DIN ISO 14: 1986 (light series)**
- **DIN ISO 14: 1986 (medium series)**
- **DIN 5464: 2010 (heavy, for vehicles)**
- **DIN 5471: 1974 (for machine tools)**
- **DIN 5472: 1980 (for machine tools)**
- **Own input**

In a splined shaft connection, after you select a standard, the program displays the corresponding external and internal diameters, and the number of splines, along with their width.

► **Note:**

select the Own input option to define your own splined shaft profile. The geometric parameters are based on the following figure.



36.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior of the driving machine	Operational behavior of the driven machine			
	uniform	moderate shocks	average shocks	heavy shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

36.1 table: Application factor according to DIN 6892

36.3 Torque curve/Number of changes of load direction

When selecting the Type of loading, you can select:

1. No alternating torque
2. With alternating torque

If you select **With alternating torque**, the calculation not only defines the frequency of change of load direction, as defined in DIN 6892, Figure 7, but also the frequency of change of load direction coefficient f_w. If you select **No alternating torque**, the coefficient will be set to 1.0.

36.4 Occurring contact stress

This formula is used to calculate occurrences of contact stress. The formula is used both for the equivalent load and for the maximum load:

p(eq,max)=$k_{\varphi\beta}$(eq,max) * $k_{\lambda e}$ * T * 2000/(d_m * l_{tr} * h * z)

$k_{\varphi\beta}$: Share factor	l_{tr}: supporting length
$k_{\lambda e}$: Length factor	h: spline height
T: torque	z: Number of splines
d_m: average diameter	

36.5 Length factor

A **length factor $K_{\lambda e}$** is multiplied by the loading. This takes into account how the load is distributed across the supporting length as a consequence of the twist between the shaft and hub.

The length factor depends on the equivalent diameter derived from the bearing length, the small and large external hub diameter and the width c to the external diameter. The distance a_0 is also used to determine the length factor. This factor is shown in a diagram in Niemann.

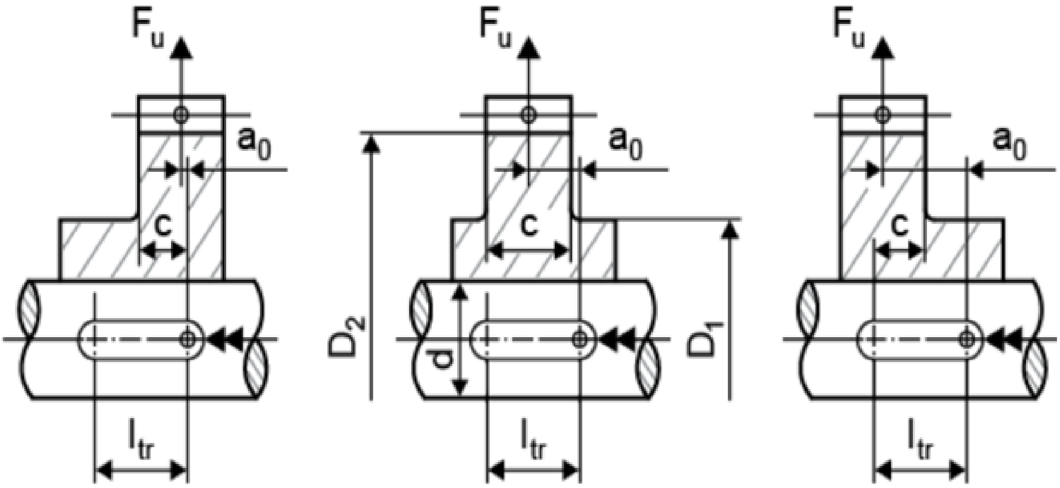


Figure 36.1: Straight-sided splines: load introduction

36.6 Share factor

A **share factor, $k_{\varphi\beta}$,** is taken into account in the calculation of the occurring contact stress. This is then multiplied by the load. Intermediate sizes not shown in the table are interpolated linearly.

Form-closure connection	Spline connection with involute flank - tolerance zones according to DIN 5480					
	H5/IT4	H7/IT7	H8/IT8	H9/IT9	H11/IT11	Maximum value
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 36.2: Share factor according to Niemann

36.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

for **ductile materials**:
$p_{eq}=f_s * f_H * R_p$

for **brittle materials**:
$p_{eq}=f_s * R_m$

Structural steel	Material	f_S
Shaft	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.3
	GJL	1.1
Hub	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 36.3: Support factor according to DIN 6892

KISSsoft always uses the minimum value for the support factor fs.

A **notch factor, f_S**, takes into account the supporting effect that occurs when bodies are subjected to pressure.
The **hardness influence coefficient, f_H**, is calculated from the surface to core hardness ratio for surface hardened parts.
The hardness influence coefficient for case-hardened steel is 1.15. Otherwise, it is set to 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$$p_{max}=f_L * p_{eq}$$

f_L is the **load peak coefficient**, which depends on the material type and the number of load peaks. This coefficient is shown in a diagram in DIN 6892.

36.8 Materials

The selection list contains materials from the standard.
If you have set the Own input flag, a new dialog is displayed. This contains the material data used in the calculation, which you can specify to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that they can also be used in subsequent calculations.

36.9 Settings

Select Calculations → Settings to enter the required safeties for the individual parts of the connection. According to Niemann, Volume I, 5th Edition (2019), the minimum safety for ductile materials is set in the range 1.0…1.3 and the minimum safety for brittle materials is set in the range 1.3…2.0. The required safeties define the values the system searches for during sizing.

The **support factors** for the shaft and the hub in DIN 6892 differ according to the materials and parts involved. The factors vary within a specific range. The minimum values are used in KISSsoft. These values can be overwritten.

If you have selected Calculate material strength with wall thickness as raw diameter, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

36.10 Sizings

During the sizing process, the required value is defined such that the required safety (specified in the dialog you see by selecting Calculations → Settings) is only just achieved. To display the results in the lower part of the main window, run the calculation after the sizing process is complete. Possible sizings:

- Transmissible nominal torque T_n
- Transmissible maximum torque T_{max}
- Supporting length l_{tr}

37 Spline (strength only)

Splines are spur gear meshings that have a shortened tooth height and a large pressure angle (usually 30°). In KISSsoft, you can use one of two different calculation modules to calculate splines. The geometry and tolerances required for manufacture, and the strength calculation, are described in the Connections section in the Splines (Geometry and Strength) chapter 38.

For splines, you must calculate the load on shaft and hub (surface pressure). You can also add additional standards. Toothing data is defined in the database and therefore you can make the use of in-house profiles mandatory. You can also use the KISSsoft Spline (geometry and strength) module, to calculate the manufacturing mass and the tolerances. This calculation, along with defining the safeties, is performed as described in classic technical literature ([8]).

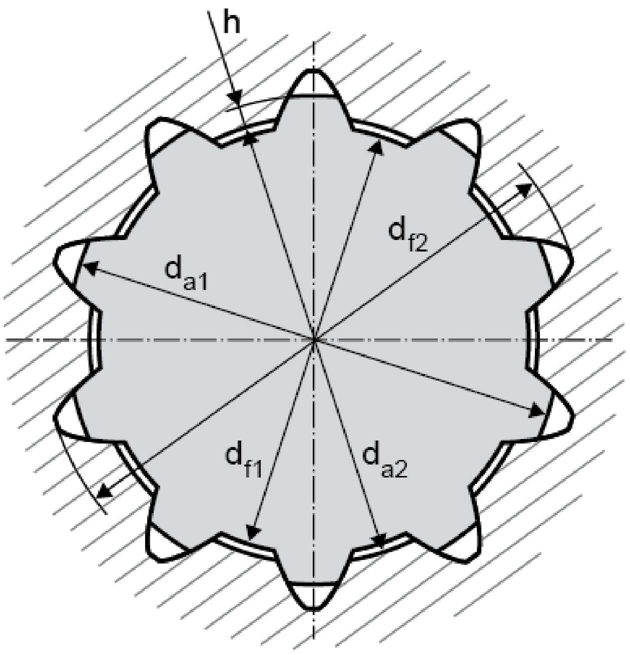
37.1 Standard profiles

You can choose one of these standards from the selection list:

- **Own input according to DIN**
- **DIN 5480**
- **DIN 5481 (Serration profile)**
- **DIN 5482 (Serration profile)**
- **Own input according to ISO**
- **ISO 4156**
- **Own input according to ANSI B92.1**
- **Own input according to ANSI B92.2M**
- **ANSI B92.1**
- **ANSI B92.2M**

For splines, the corresponding values are displayed in the list after the norm selection.

d_{a1}: Tip diameter of the shaft	z: Number of teeth
d_{a2}: Tip diameter of the hub	x: Profile shift coefficient
m: Module	



► **Note:**

Select the Own input option to enter your own data for the straight-sided spline. The critical factor here is that the tip diameter of the shaft is greater than the tip diameter of the hub. If not, an error message is displayed.

37.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

37.1 table: Application factor according to DIN 6892

37.3 Torque curve/Number of changes of load direction

When selecting the Type of loading, you can select:

1. No alternating torque
2. With alternating torque

If you select **With alternating torque**, the calculation not only defines the frequency of change of load direction, as defined in DIN 6892, Figure 7, but also the frequency of change of load direction coefficient f_w. If you select **No alternating torque**, the coefficient will be set to 1.0.

37.4 Occurring contact stress

This formula is used to calculate occurrences of contact stress. The formula is used both for the equivalent load and for the maximum load:

$p(eq,max)=k_{\varphi\beta}(eq,max) * k_{\lambda e} * T * 2000/(d_m * l_{tr} * h * z)$

$k_{\varphi\beta}$: Share factor	l_{tr}: supporting length
$k_{\lambda e}$: Length factor	h: spline height
T: torque	z: Number of splines
d_m: average diameter	

► **Note:**

If pmax < peq, pmax = peq is set.

37.5 Length factor

A length factor $K_{\lambda e}$ is multiplied by the loading. This takes into account how the load is distributed across the supporting length as a consequence of the twist between the shaft and hub.

The length factor depends on the equivalent diameter derived from the bearing length, the small and large external hub diameter and the width c to the external diameter. The distance a_0 is also used to determine the length factor. The length factor is according the diagrams in figure 4 - figure 6 in DIN 6892:2012.

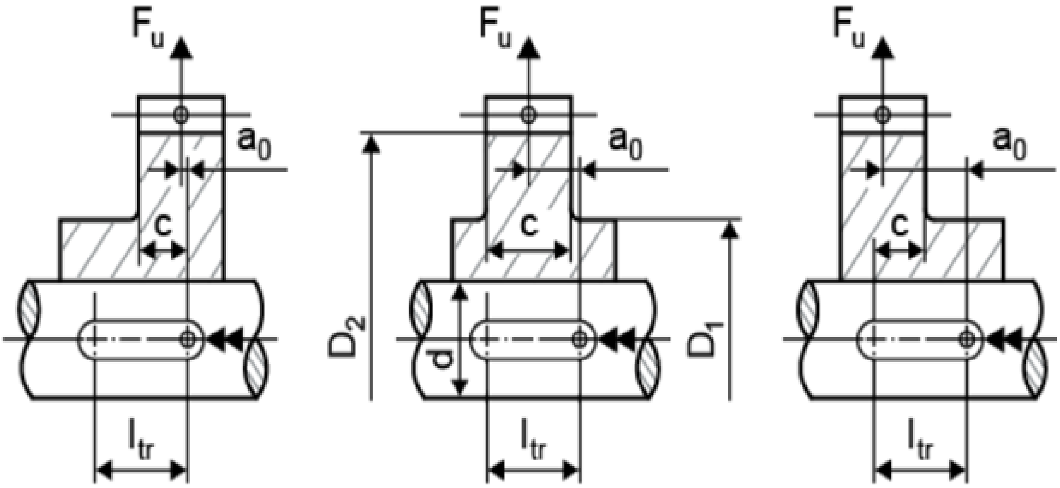


Figure 37.2: Splines: load introduction

► **Note:**

The recommended values for $K_{\lambda e}$ according to Niemann apply to splined joints which are subject to torsion. Any additional stress caused by bending or shearing forces can create a large number of high local loads. More precise methods must be used to investigate these cases.

37.6 Share factor

If you are working with DIN profiles, the "Manufacturing quality according to DIN" can be used directly as a "Tolerance field according to DIN 5480" to define the value of $k_{\varphi\beta}$ according to the table shown below.

If you are working with ISO and ANSI profiles, the specified manufacturing quality +4 (see "Tolerances" section) is taken and used as the "Tolerance field according to DIN 5480". This value is then used to determine $k_{\varphi\beta}$.

Form-closure connection	**Spline connection with involute flank - tolerance zones according to DIN 5480**					
	H5/IT4	**H7/IT7**	**H8/IT8**	**H9/IT9**	**H11/IT11**	**Maximum value**
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 37.2: Share factor according to Niemann

► **Note:**

The recommended values for $k_{\varphi\beta}$ according to Niemann apply to splined joints which **are only subject to torsion**. Any additional stress caused by bending or shearing forces can create a large number of high local loads. More precise methods must be used to investigate these cases.

37.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}, this applies:

for **ductile materials**:
$p_{eq}=f_S * f_H * R_p$

for **brittle materials**:
$p_{eq}=f_S * R_m$

Structural steel	Material	f_S
Shaft	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.3
	GJL	1.1
Hub	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 37.3: Support factor according to DIN 6892

KISSsoft always uses the minimum value for the notch factor fs.

A **notch factor, f_S**, takes into account the supporting effect that occurs when bodies are subjected to pressure.
The **hardness influence coefficient, f_H**, is calculated from the surface to core hardness ratio for surface hardened parts.
The hardness influence coefficient for case-hardened steel is 1.15. Otherwise, it is set to 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the **load peak coefficient**, which depends on the material type and the number of load peaks. This coefficient is shown in a diagram in DIN 6892.

37.8 Materials

The selection list contains materials from the standard.
If you have set the Own input flag, a new dialog is displayed. This contains the material data used in the calculation, which you can specify to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that they can also be used in subsequent calculations.

37.9 Settings

Select Calculations → Settings to enter the required safeties for the individual parts of the connection. According to Niemann, Volume I, 5th Edition (2019), the minimum safety for ductile materials is set in the range 1.0…1.3 and the minimum safety for brittle materials is set in the range 1.3…2.0. The required safeties define the values the system searches for during sizing.

The **support factors** for the shaft and the hub in DIN 6892 differ according to the materials and parts involved. The factors vary within a specific range. The minimum values are used in KISSsoft. These values can be overwritten.

If you have selected Calculate material strength with wall thickness as raw diameter, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

37.10 Sizings

During the sizing process, the required value is defined such that the required safety (specified in the dialog you see by selecting Calculations → Settings) is only just achieved. To display the results in the lower part of the main window, run the calculation after the sizing process is complete. Possible sizings:

- Transmissible nominal torque T_n
- Transmissible maximum torque T_{max}
- Supporting length l_{tr}

38 Splines (Geometry and Strength)

You can calculate the geometry and the control measurement values of splines and hubs according to DIN 5480 (1986 Edition), ISO 4156, ANSI B92.1 or ANSI B92.2M. Strength calculations according to Niemann, DIN 5466 and AGMA 6123-C16 are also included.

The geometry profiles according to DIN 5481 (2019) and DIN 5482 (1973) are saved in files.

When you open the file for the profile you require, the KISSsoft screens are filled with all the necessary geometry settings.

38.1 Underlying principles of calculation

38.1.1 General

Involute short cut teeth are often used for couplings. Teeth with a very large pressure angle $\alpha_n = 30°$. To increase the strength, in comparison to normal gear teeth, the calculation is performed with a tooth height that is reduced by about a half. Couplings with teeth as defined in DIN 5480 are very widespread. This standard gives precise descriptions of the geometry and tolerance calculation. The strength calculation is performed in accordance with the usual methods described in technical literature [86], [64].

► **Note**

The moment of inertia is calculated as follows: the inside diameter of the shaft is $d_i = 0$, and the hub external diameter is the rounded result of $d_i = d_f + 4m_n$. The moment of inertia is then determined for the cylinder between d_i and $(d_a + d_f)/2$.

38.1.2 Calculation of spline connections as described in DIN 5480 with diameter centering

Diameter-centered connections are centered in the outside or inside diameters. The hub root diameter with the shaft tip circle is used for outside centering. The hub tip diameter with the shaft root circle is used for inside centering. Here, the gear toothing is only used for rotational synchronization. The connection must therefore have sufficient **backlash** to prevent the center holes from intersecting. Due to the small tolerances of the centering diameter, diameter-centered connections require increased manufacturing effort to limit the central displacement. This is why they are only used in exceptional circumstances.

To calculate diameter-centered connections:

1. Go to the Calculation module and click on Connections > Splines (Geometry and Strength), then click on the Reference profile tab. In the Reference profile drop-down list, select the DIN 5480 Major diameter fit option for the shaft or the Major diameter fit option for the hub.
2. Click the Tolerances tab to open the Tolerances input window. Check that no flag has been set in the checkbox to the right of Tip diameter deviation (upper/lower) and Root diameter deviation (upper/lower) input fields, for Shaft or for Hub. Then the recommendations according to DIN 5480 are used as tolerances.

For the **tip circle**, the following apply:

- for outside centering, H6 for the shaft tip diameter and H11 for the hub tip diameter
- for inside centering, h11 for the shaft tip diameter and H7 for the hub tip diameter

For the **root circle**, the following apply:

- for outside centering, h14 for the shaft root diameter and H7 for the hub root diameter
- for inside centering, h6 for the shaft root diameter and H14 for the hub root diameter

9H/9e is recommended for the **tooth thickness allowances**.

38.1.3 Calculating spline connections according to DIN 5480 with flank centering

In **flank-centered connections**, the flanks are used for centering and rotational synchronization. The nominal lengths of the root diameter used for the shaft and the hub in flank-centered connections are the theoretical root diameter calculated with h**fP** = 0.55*m. All reference profiles without the note Major diameter fit or Minor diameter fit are for flank-centered splined joints.

The allowances that apply where cutting manufacturing processes are involved cover the appropriate greatest possible dedendum (hfP=0.65*m) and the root diameter allowances. The tolerance field for the root diameter of the tooth space is 9H. For the hub root diameter it is 11a.

The following values apply for **flank-centered splines**:

- Hub tip diameter da2 with tolerance field H11
- Shaft tip diameter da1 with tolerance field h11
- Allowance for the hub root diameter df2, calculated with Adf2 = (0.2*m+1.73*(Ae+TG)).
- Allowance for the hub root diameter df1, calculated for machining: with Adf1 = -(0.2*m+(1.73*As+TG)). For cold roll forming: with Adf1 = -0.76*m.

This process for calculating root circle tolerances according to the base standard, DIN 5480-1, Table 5, is described below.

Select Calculations→ Settings → General and then select whether root circle tolerances are to be calculated according to the manufacturing process (DIN 5480-16, 4.3.1) or according to the base standard (DIN 5480-1, Table 5).

When the root circle tolerances are calculated according to the basic standard DIN 5480-1 (Table 5), the root form diameter is calculated as specified by the formulae on page 13:

Shaft root form diameter: dFf1 = |da2| - 2*cFmin

Hub root form diameter: dFf2 = -(da1 + 2*cFmin);

cFmin comes from Table 4 in the DIN 5480-1 standard.

If either the reference diameter or the module lie outside of the defined range, the adjacent edge value in the table is used as cFmin.

The tolerance band calculated for the root diameter is also used here.

38.1.4 Calculating the root diameter tolerance ΔDre, ΔDri according to ANSI B92.1

ANSI B92.1:1996 has a printing error on page 145, where ΔDre, ΔDri are calculated. According to the gear teeth theory, factor 2 should not be present in the formula. This error has been corrected in KISSsoft. As this error has resulted in some tools being sized incorrectly at companies in the past, this option (checkmark in module specific settings) has been added in KISSsoft so that the calculation can also be performed with a tolerance that is too high.

38.2 Basic data

38.2.1 Geometry standards

The complete standard and preference sequences are also available for most of the standards in this list. Use the database tool (see chapter 9, Database Tool and External Tables) to add your own standards to the list or extend existing guidelines. For example, the DIN 5480 preference sequence is stored in the M02C-001.dat file in the dat folder in your KISSsoft installation directory. Each line corresponds to an entry in the Define profile list and uses the following syntax:

d_{a1}	d_{a2}	m_n	z	x*

where

d_{a1}	Tip diameter, shaft
d_{a2}	Tip diameter, hub
m_n	Normal module
z	Number of teeth
x^*	Profile shift coefficient shaft

▶ **Example:**

```
#Units:mm|mm|mm||
3.50|2.72|.5|6|0.00
4.00|3.18|.5|7|0.00
4.50|3.66|.5|8|0.00
5.00|4.14|.5|9|0.00
5.50|4.62|.5|10|0.00
6.00|5.11|.5|11|0.00
```

Figure 38.1: Example entry in M02C- 001.dat

The selected entry in Figure (see Figure 38.1) stands for d_{a1} = 5.5mm, d_{a2} = 4.62mm, m_n = 0.5mm, z = 10 and x^* = 0.

▶ **Note**

You can only edit the Normal module, Number of teeth and Profile shift coefficient input fields if you first select Own input in the drop-down list for geometry standards.

38.2.2 Normal module

Enter the normal module. However, if you know the pitch, transverse module or diametral pitch instead of this, click on the Convert button to open a dialog window, in which you can perform the conversion.

38.2.3 Pressure angle at normal section αn

The normal pressure angle at the reference circle is also the reference profile flank angle. For splines, the pressure angle is usually $_{\alpha n}$ = 30°.

38.2.4 Number of teeth

For internal toothed gears, you must enter the number of teeth as a negative value as stated in DIN 3960. The shaft and the hub must have the same number of teeth, but different prefix operators.

38.2.5 Profile shift coefficient

The tool can be adjusted during the manufacturing process. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a **positive profile shift**, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a **negative profile shift**, the tool is pushed further into the material, with the result that the tooth is narrower and undercutting may occur sooner. For pinion and gear factors:

$$x_1^* = -x_2^*$$

If you click the Convert button, KISSsoft will determine whether the profile shift coefficients are to be taken from measured data or from values given in drawings.

The following options are available:

- **Base tangent length**: Enter the base tangent length and the number of teeth spanned. This option cannot be used for (internal) helical gear teeth because their base tangent length cannot be measured.
- **Measurement over balls:** Enter the dimension and the diameter of the ball. In a gear with helical gear teeth and an odd number of teeth, the measurement over balls is not the same as the measurement over two pins. See Measurement over pins.
- **Measurement over pins:** Input the dimension and the diameter of the pin. You must also enter a minimum span for helical gear teeth and gears with an odd number of teeth, so the measurement can be performed. The measurement over pins cannot be measured in internal helical gears.
- **Tip circle:** This is a rather imprecise calculation because the tip circle does not always depend solely on the profile shift.
- **Tooth thickness at reference circle:** Here, you specify the tooth thickness. You can also enter the arc length or chordal length, and specify whether the value is in transverse or normal section.
- **Tooth thickness at arbitrary diameter:** Here you specify the chordal tooth thickness in normal section at an arbitrary diameter that is defined by the height above chord ha (from da without allowances). Use this option if the tooth thickness is not present at the reference diameter, but only at an arbitrary diameter. It can be used for the pinion type cutters as well to obtain the profile shift coefficient. Although this conversion is accurate,

the fact that it is based on an iterative process may result in minor deviations in some cases.

► **Note**

The profile shift coefficient of the shaft and hub must be the same value.

38.2.6 Quality and tolerances

The manufacturing qualities that can be achieved are displayed in the next table.

Manufacturing process	Quality according to DIN/ISO		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 38.1: Accuracy grades for different manufacturing processes

Tolerances for splines

As a comparison, we have calculated an example for the tolerances specified in DIN 5480, ISO 4156 and ANSI B92.1 (spline) and for the tolerances given in ISO 1328 (cylindrical gear pair) with accuracy grade 6 (external) and accuracy grade 7 (internal).

The results for the ISO 4156 accuracy grade are 3 to 4 accuracy grades higher than the same results for the DIN 5480 accuracy grade.

ANSI B92.1 returns the same results as ISO 4156.

This is all very confusing, because all other gears (cylindrical gears with straight or helical teeth, bevel gears, worms, etc.) return accuracy grades that are mostly very similar for DIN, ISO and AGMA tolerance results.

38.2.7 Niemann geometry data

This is geometry data that is only used to calculate **strength as specified by Niemann**:

- Diameter of shaft bore d_I
- Large outside diameter of hub D

You will need to enter more values to perform the calculation as defined by Niemann. Depending on the position of the load, you can enter the value a0. If a shouldered hub is present, you must also enter the small hub external diameter D and the width of the center part c (with D). The following diagram shows how to define these values:

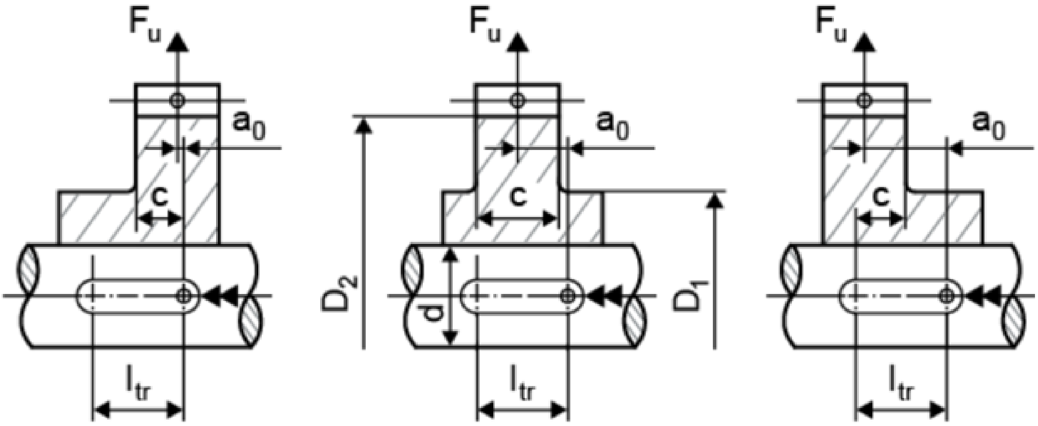


Figure 38.2: Parameter definition acc. to Niemann

38.2.8 Geometry details

Select the **Details** button on the top right in the Geometry input window to open a new window, the Define details of geometry window. You can enter drawing numbers for the shaft and the hub there. You can also enter a toothing runout.

38.2.9 Define details of strength

The strength calculation is then performed either according to Niemann [8], DIN 5466 or AGMA 6123-C16. As DIN 5466 is still being developed, it is not described in any further detail here. To perform the calculation according to DIN 5466 and Niemann, you must enter additional data (see chapter 37, Spline (strength only)).

38.2.9.1 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. The next table provides information about the coefficient values. You will find more details in ISO 6336, DIN 3990 and DIN 3991.

Operational behavior of the driving machine	Operational behavior of the driven machine			
	uniform	moderate shocks	average shocks	heavy shocks
uniform	1.00	1.25	1.50	1.75

light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 38.2: Assignment of operational behavior to application factor

38.2.9.2 Strength method as defined by Niemann/Winter

38.2.9.2.1 Type of loading/Number of changes of load direction

When selecting the **Type of loading**, you can select:

1. No alternating torque
2. With alternating torque

If you select **With alternating torque**, the calculation not only defines the **frequency of change of load direction**, as defined in DIN 6892, Figure 7, but also the frequency of change of load direction coefficient fw. If you select **No alternating torque**, the coefficient will be set to 1.0. This data is only used for calculations as described in Niemann.

$p_{eq}=f_w * p_{zul}$

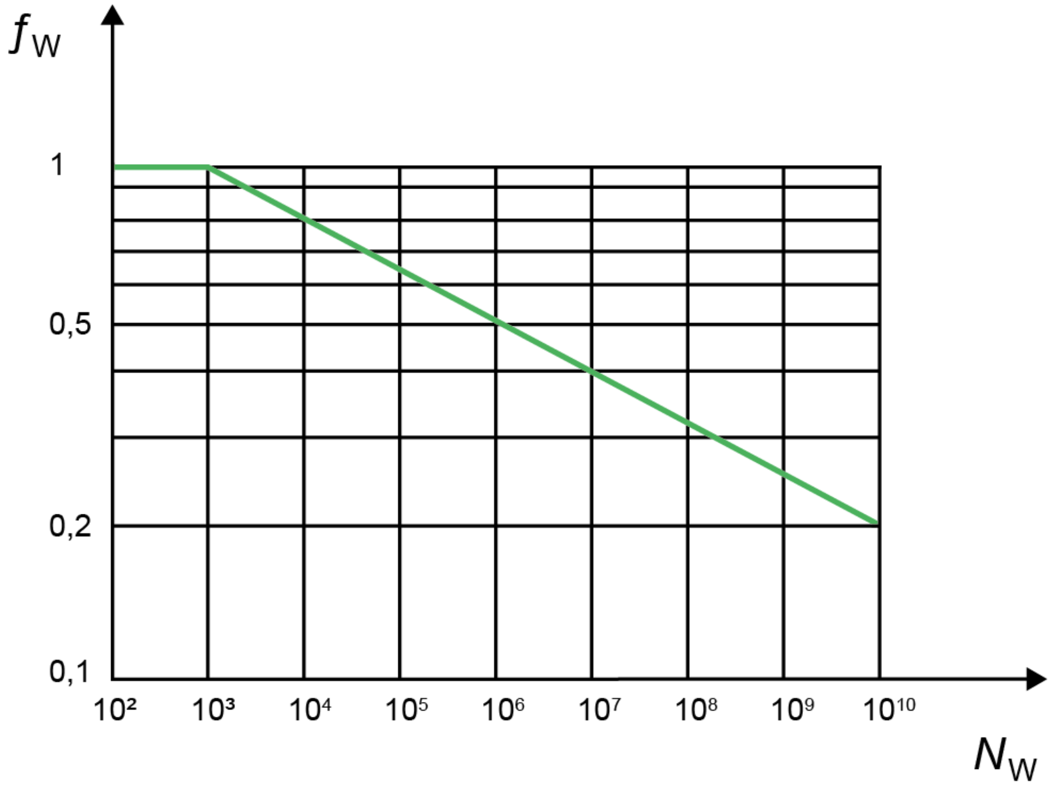


Figure 38.3: Graphic as described in DIN 6892 Figure 7: Load direction changing coefficient for alternating load

38.2.9.2.2 Number of load peaks

fL is the Load peak coefficient, which depends on the material type and the Number of load peaks NL. This coefficient is shown in a diagram in DIN 6892 and is required as an input value for performing the calculation according to Niemann.

For calculation with **peak torque**:

$p_{max}=f_L * p_{zul}$

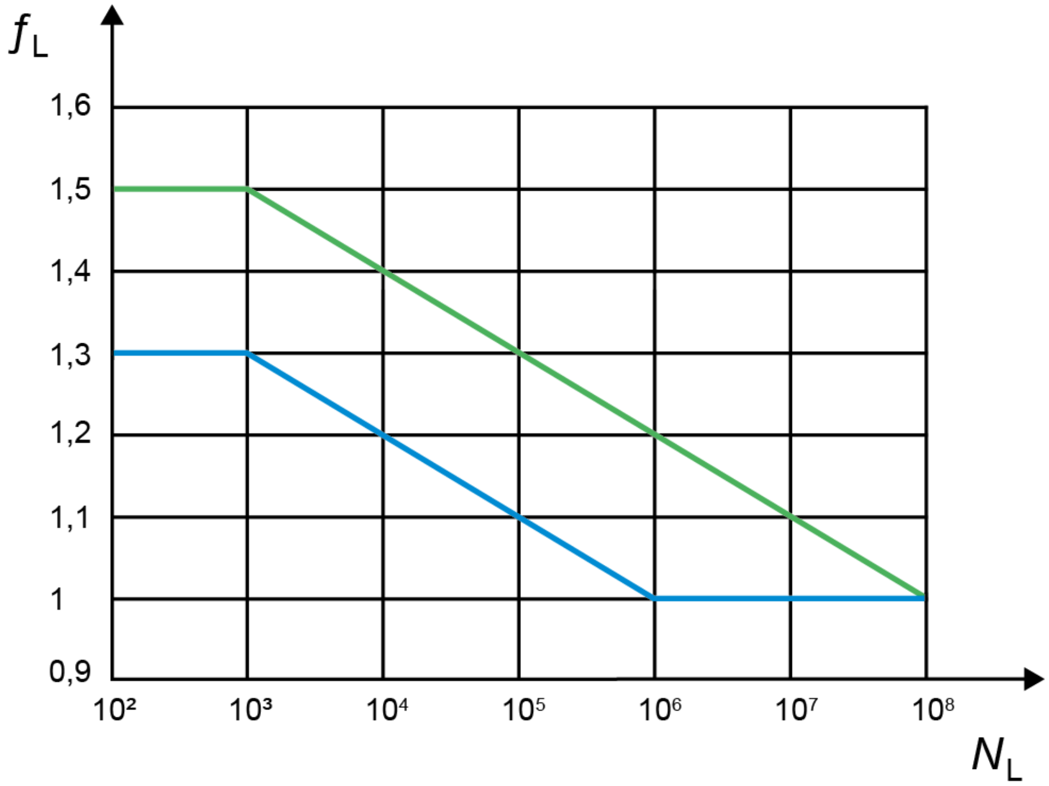


Figure 38.4: Graphic as described in Niemann (DIN 6892 Figure 8): Load peak coefficient

- Green line: ductile material
- Blue line: brittle material

38.2.9.2.3 Support factors

The support factors for the shaft and the hub in DIN 6892 differ according to the materials and parts involved. The factors vary within a specific range. The minimum values are used in KISSsoft, and these values can be overwritten in this window.

38.2.9.3 Geometries according to DIN 5466

38.2.9.3.1 Resulting shearing force

Shearing forces vertical to the shaft axis cause flank contact on both sides of the opposing side of the contact point.

38.2.9.3.2 Stress ratio R

Stress ratios are the ratios between under and over stress with regard to a particular type of load, such as torque. Here *R* = -1 and defines a pure alternating stress ratio, *R* = 0 defines a pure pulsating stress ratio.

38.2.9.3.3 Width and circumferential factor

If you select the checkbox on the right of the input field for these coefficients, you can overwrite the values manually. Otherwise, this value is calculated automatically and may vary within the range [3, 5]. As these are multiplied together to define the overload, you can achieve safeties of up to 20 times smaller than is possible with the calculation method defined in Niemann.

38.2.9.4 Strength method AGMA 6123-B06

This standard is intended to be used for sizing a closed planetary gear unit. However, it also includes a part in which the spline is calculated (section 10.4).

This calculation method calculates a permissible torque for contact stress and a permissible torque for scuffing and wear resistance.

It can be used to calculate both the permitted shearing stress, ssA, and the permitted contact stress, scA, from the material's core hardness (HRC value).

The calculated permitted torques are compared with the maximum torque, Tmax (Tnenn*KA). These values are then used to calculate the safety.

Select Calculations > Settings to display the Settings tab. In it, you can define the required safeties. These are then used to determine the safety factors and calculate the sizings.

Optionally, in this calculation method, you can also define the misalignment type and misalignment angle.

You can use these values to define the face load factor Km.

► **Note:**

The prerequisite for calculating shearing stress is that the load is carried by half the number of teeth. The prerequisite for performing a calculation against scuffing and wear resistance is that the load (wear) is carried by all the teeth.

If a thinner ring which runs at a faster speed is used, it must be checked for ring bursting. AGMA 6123 has a calculation for doing this. This calculation is split into three parts: the radial component, the tensile stress and the centrifugal ring stress.

These components can then be used to calculate the overall tensile stress on the ring, which is then compared with the permitted stress (converted from the core hardness). These values are then used to calculate the safety.

A prerequisite for calculating the ring's safety against ring bursting is that the load is carried by half the teeth.

The formulae for the individual calculations are listed at the end of the calculation report.

38.2.9.5 Crowned splines Dudley

For crowned splined shafts, where the spline has a curved tooth modification, the surface pressure can be verified using the Dudley calculation method [87].

This calculation method requires the additional input of the Number of revolutions, if the alignment type not aligned, crowned is selected in the AGMA calculation method. The values from this calculation are given in a separate block in the report, along with the results from AGMA 6123.

38.2.10 Materials

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select Own input from the list, to create a new entry, or enter the material in the database (see chapter 9, Database Tool and External Tables) first. To do this, click the Plus button. This opens the Material hub/shaft window, in which you can select your material from a list of materials that are available in the database. Select the Own input option to enter specific material characteristics. This option corresponds to the Create a new entry window in the database tool.

38.3 Tolerances

38.3.1 Tooth thickness tolerance

Select one of the options from the Tooth thickness tolerance drop-down list.

The allowances for **Actual** (smax, smin, emax, emin) correspond to the individual measurements (base tangent length or measurement over pins measured on the gear teeth). The allowances for **Effective** (svmax, svmin, evmax, evmin) correspond to the measurement with gauges (all teeth checked together). The gear backlash of a spline connection is therefore derived from the "Effective" allowances. The effective allowances include not only the tooth thickness allowances of individual teeth but also a pitch and form error component. The effective allowances are the following **theoretical values**. They are smaller (the tooth is thicker) than with the actual allowances.

► **Note**

According to the standard, the allowances for tooth thickness (smax, smin) are predefined for the shaft. In contrast, for the hub, the allowances apply to the tooth space (emax, emin).

If the tooth thickness tolerance has been set to your own input, you can input svmax for the shaft (**Effective** maximum allowance) to calculate svmin, because the relationship applies in this case:

svmin – smin = svmax – smax

In addition, you can then use the flag to predefine the individual measurement allowances for **Actual**. However, if this flag is not set, the difference svmax–smax (pitch and form error component), and the tolerance interval smax-smin are set according to the standard for the selected accuracy grade.

The same also applies to the hub.

► **Note**

The **circumferential backlash** and **normal backlash** are calculated using the formulae for gear pairs, where jt.act and jn.act are the values for **Actual** and jt.eff and jn.eff are the values for **Effective**. In the case of radial clearance, we have to consider that all the teeth can have contact when they are moved. The first gear pair to make contact is the one whose direction of movement most closely matches the normal angle to the flank. The radial clearance therefore corresponds to the normal clearance in transverse section (jr = jt * cos(alfa)). The function also checks whether the tip clearance is so small that it also restricts the radial clearance.

38.3.1.1 DIN 5480

Unlike ISO 4156 or ANSI 92.1, DIN 5480 has the special feature that sveffmin = svmax always applies to the shaft and eveffmax = svmin to the hub. For this reason, sveffmin and eveffmax are not displayed.

► **Note**

The tolerance widths in the specified values for gauges are larger because of Taylor's formula [88].

38.3.1.2 ANSI 92.1 and ISO 4156/ANSI 92.2M

The following points must be taken into consideration if the tooth thickness tolerance is to be defined as Own input:

- Enter the tooth thickness allowance sv for the **Effective** tooth thickness for the overall measurement (caliber) to suit the tolerance system that you are using to calculate cylindrical gears.

- The **Actual** tooth thickness s for single measurements is defined using these equations. These equations apply to the shaft tooth thickness or to the hub tooth space.

$s_{\max} = s_{v,\max} - \lambda$	(38.1)
$s_{\min} = s_{v,\min} - \lambda$	(38.2)

38.3.2 Effective/Actual

Click the Convert button in the Tolerances tab next to the **Effective (max/min)** input field to open the Convert total Effective (Actual) deviation of tooth thickness for shaft/hub dialog. The values input in this screen can then be used to convert the **Effective/ Actual tooth thickness allowances**. Here, you can enter values either for the base tangent length, measurement over balls or pins, or the tooth thickness.

38.3.3 Shaft/hub: diameter of ball/pin

The implemented DIN 5480, Part 1, contains an extract of the measuring roller diameter as specified in DIN 3977 that must be used here. You can decide whether to extend the list of available roller diameters in the Z0Rollen.dat file in the dat folder in your KISSsoft installation directory.

38.4 Gauges

Spline connections are often checked using gauges.

Go gauges are always fully toothed (teeth all around the perimeter) and are used to test the **effective** tolerance limit. For hubs, this is the **min. effective** tooth space and for shafts this is the **max. effective** tooth thickness.

No-go gauges are always toothed by sector (depending on the number of teeth on the test piece, 2 to 7 teeth located opposite each other) and are used to test the **actual** tolerance limit. For hubs, this is the **max. actual** tooth space and for shafts, this is the **min. actual** tooth thickness. The externally located flanks of each sector are given sufficient clearance (flank relief, see 1 in the Figure), as they cannot be measured exactly.

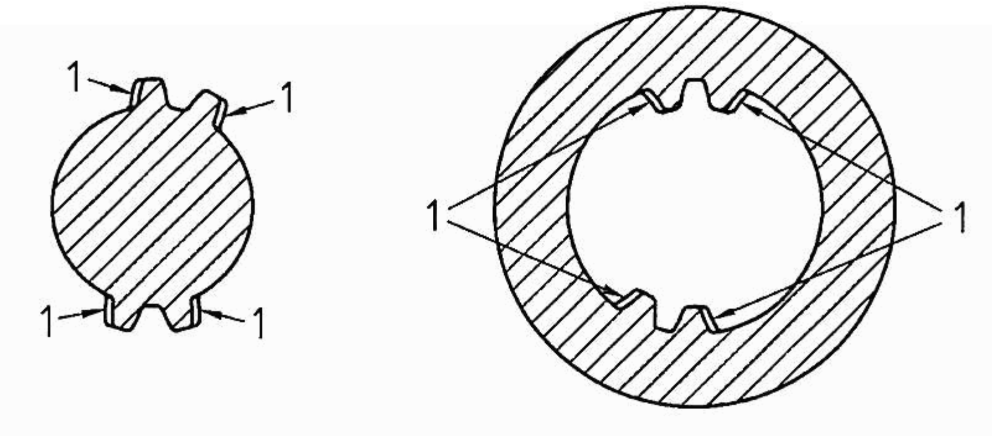


Figure 38.5: Displaying gauges

KISSsoft can calculate all the gauge allowances detailed in ISO 4156 and DIN 5480-15. To do this, select to Report and then click on the Construction of gauges option. The system does not automatically calculate the gauge dimensions for profiles that comply with ANSI.

DIN 5480-15 is limited to a pressure angle of 30°. Pressure angles 37.5° and 45° are defined in ISO 4156. DIN 5480-15 has data for a module range of 0.5 to 10 mm and a maximum number of 100 teeth on the sample. No information is provided in the report for values that exceed the data in DIN 5480-15.

According to DIN 5480-15, a calculated **allowance coefficient AF1** should be entered for the measuring rollers, if the measuring rollers do not exactly match the specified dimension. Then, the distance **AF1** is calculated as a multiplication coefficient of the mean value of the allowances for the two measuring rollers. This enables the tolerance for the distance over rollers to be determined. As this value is not known, this calculation is not performed in KISSsoft.

38.5 Curved tooth

It is possible to define the curved tooth geometry of the shaft in the shaft/hub connections. Currently only the 3D geometry of the shaft is generated. Inputs for the curved tooth definition are in the Modificationstab. Such splines can be used as couplings to compensate potential axial misalignment of the shafts. An example of a curved tooth shaft geometry is shown in the following figure.

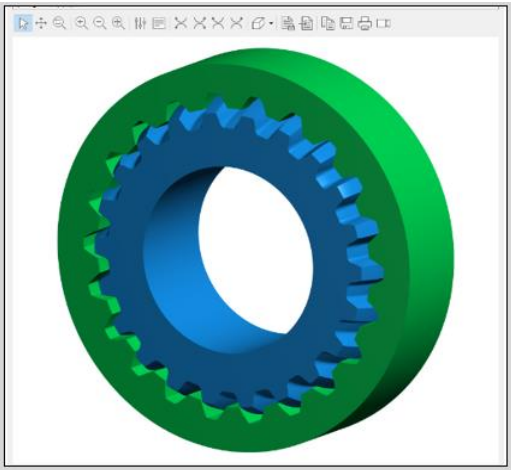


Figure 38.6: Curved tooth shaft in spline/hub connection

38.6 Tooth form

The Tooth form tab has six different tolerance fields for tooth thickness values (actual, effective) and four different diameter tolerance fields that you can select to generate the tooth forms on splined joints.

The default settings here are the average allowance for tooth thicknesses and diameter. After the calculation has been performed, the resulting diameter and tooth thickness are output in the tooth form report.

► **Note**

Set the flag next to the Root diameter deviation field in the Reference profile tab to ensure that the diameter tolerance you have selected has an effect on the root diameter. Otherwise, only the tooth thickness tolerance will be used to calculate the root diameter.

If this flag is not set, the calculation uses a default tolerance for the root diameter as defined in the standard. In other words, the diameter tolerance you selected will not be used to calculate the root diameter.

39 Polygons

Polygon connections are used to create shaft-hub connections that can withstand very heavy loads. In particular, the low notch effect present in this type of connection does not reduce shaft strength.

For polygon shafts, you must calculate the load on the shaft and hub (surface pressure). You can also add additional standards.

You can use one of these two methods to calculate the load on the shaft and hub (surface pressure) and to define the safeties:

- Niemann/Winter, Volume I, 4th Edition [8].
- DIN 32711-2 (P3G-Profile) [89]/ DIN 32712-2 (P4C-Profile) [90]

Only the static load case is considered in the calculation according to DIN. In Niemann's method, you can also calculate the influence of an alternating torque or the load peaks.

39.1 Standard profiles

You can choose one of these standards from the selection list:

- **Polygon P3G profile DIN 32711-1:2023**
- **Polygon P4C profile DIN 32711-1:2009**

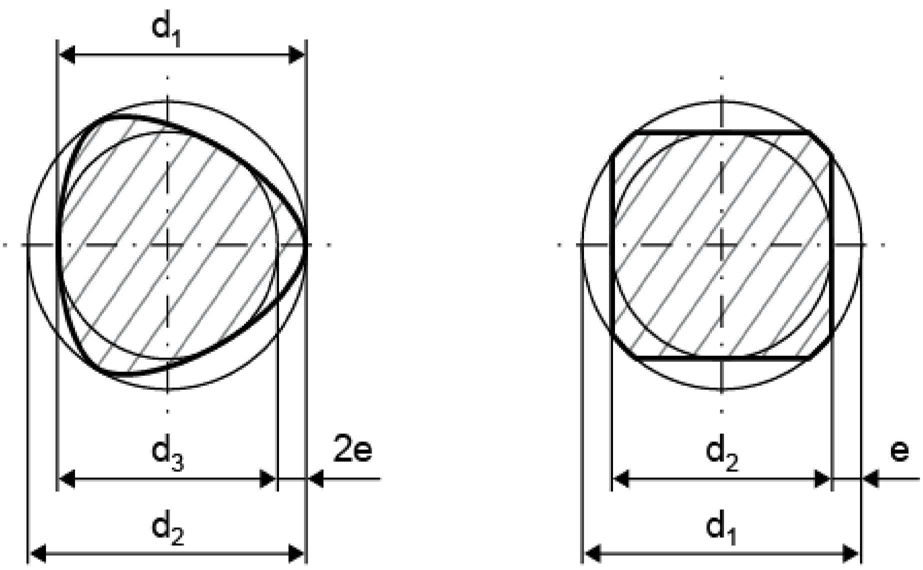


Figure 39.1: Polygon profiles

For the **P3G profile**, [91] Part 1-2, after selecting a standard from the list, the mean circle diameter d1, the diameter of outer circle d2, the diameter of inner circle d3, the eccentricity e and the hub wall factor y are displayed.
For the **P4C profile**, [92] Part 2, [90] Part 1, the diameter of outer circle d2, the diameter of inner circle d3, the eccentricity e and the hub wall factor y are displayed in the list.

39.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

39.1 table: Application factor according to DIN 6892

39.3 Torque curve/Number of changes of load direction

This influence can only be made to apply using the Niemann calculation method.

When you select the torque curve, you can select one of two positions:

1. No alternating torque
2. With alternating torque

If you select **With alternating torque**, the calculation not only defines the frequency of change of load direction, as defined in DIN 6892, Figure 7, but also the frequency of change of load direction coefficient f_w. If you select **No alternating torque**, the coefficient will be set to 1.0.

39.4 Occurring contact stress

Method according to Niemann: This formula is used to calculate occurrences of contact stress. The formula is used both for the equivalent load and for the maximum load:

Profile P3G:

p(eq,max)=T * 1000/(l_{tr} * d1 * (0.75 * π * e + 0.05 * d1))

Projection area = l_{tr} * n * 2 * e; (n = 3)

d_1: Mean circle diameter	T: Torque
l_{tr}: supporting length	e: Eccentricity

Profile P4C:

er = (d2 - d3) / 4; dr = d3 + 2 * e

p(eq,max)=T * 1000/(l_{tr} * (p *dr* er + 0.05 * d2^2))

Projection area = l_{tr} * n * 2 * er; (n = 4)

d_2: Diameter of outer circle	T: Torque
l_{tr}: supporting length	e: Eccentricity
d_r: mathematical theoretical diameter	e_r: mathematical eccentricity
d_3: Diameter of inner circle	

Method according to DIN: The following formula is used to calculate the occurrence of contact stress:

Profile P3G:

p=T * 1000/(l_{tr} * d1 * (0.75 * π * e + 0.05 * d1))

d_1: Mean circle diameter	T: Torque
l_{tr}: supporting length	e: Eccentricity

Profile P4C:

er = (d2 - d3) / 4; dr = d3 + 2 * e

p=T * 1000/(l_{tr} *dr (π * er + 0.05 * dr))

d_2: Diameter of outer circle	T: Torque
l_{tr}: supporting length	e: Eccentricity
d_r: mathematical theoretical diameter	e_r: mathematical eccentricity

d_3: Diameter of inner circle	

39.5 Permissible pressure

Method according to Niemann: The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}, this applies:

for **ductile materials**
$p_{eq}=f_s * f_H * R_p$

for **brittle materials**:
$p_{eq}=f_s * R_m$

Structural steel	**Material**	$\mathbf{f_s}$
Shaft	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.3
	GJL	1.1
Hub	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 39.2: Notch factor according to DIN 6892

KISSsoft always uses the minimum value for the notch factor fs.

A **notch factor, f_s**, takes into account the supporting effect that occurs when bodies are subjected to pressure.
The **hardness influence coefficient, f_H**, is calculated from the surface to core hardness ratio for surface hardened parts.
The hardness influence coefficient for case-hardened steel is 1.15. Otherwise, it is set to 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the **load peak coefficient**, which depends on the material type and the number of load peaks N_w. This coefficient is shown in a diagram in DIN 6892.

Method according to DIN: The permissible surface pressure on the shaft or hub for polygon profiles P3G and P4C is:

$p_{zul} = 0.9 * R_{p0.2}$

39.6 Materials

The selection list contains materials from the standard.
If you have set the Own input flag, a new dialog is displayed. This contains the material data used in the calculation, which you can specify to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that they can also be used in subsequent calculations.

39.7 Settings

Select Calculations -> Settings to enter the required safeties for the individual parts of the connection. According to Niemann, Volume I, 5th Edition (2019), the minimum safety for ductile materials is set in the range 1.0…1.3 and the minimum safety for brittle materials is set in the range 1.3…2.0. The required safeties define the values the system searches for during sizing.

The **support factors** for the shaft and the hub in DIN 6892 differ according to the materials and parts involved. The factors vary within a specific range. The minimum values are used in KISSsoft. These values can be overwritten.

If you have selected Calculate material strength with wall thickness as raw diameter, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

39.8 Sizings

During the sizing process, the required value is defined such that the required safety (specified in the dialog you see by selecting Calculations --> Settings) is only just achieved. To display the results in the lower part of the main window, run the calculation after the sizing process is complete. Possible sizings:

- Transmissible nominal torque T_n
- Transmissible maximum torque T_{max} (only for Niemann)
- Supporting length l_{tr}

39.9 Graphics

The polygon form is defined using the formulae in the relevant DIN standard (32711-1/32712-1) and is displayed as a graphic which can be exported either as a picture file or as a DXF file.

Polygon curve equation (profile P3G, DIN 32711-1):

$$x(\alpha) = [Dm/2 - e \cdot cos(3 \cdot \alpha)] \cdot cos(\alpha) - 3 \cdot e \cdot sin(3 \cdot \alpha) \cdot sin(\alpha)$$
$$y(\alpha) = [Dm/2 - e \cdot cos(3 \cdot \alpha)] \cdot sin(\alpha) + 3 \cdot e \cdot sin(3 \cdot \alpha) \cdot cos(\alpha)$$

Polygon curve equation (profile P4C, DIN 32712-1):

$$x(\alpha) = [Dm/2 - e \cdot cos(4 \cdot \alpha)] \cdot cos(\alpha) - 4 \cdot e \cdot sin(4 \cdot \alpha) \cdot sin(\alpha)$$
$$y(\alpha) = [Dm/2 - e \cdot cos(4 \cdot \alpha)] \cdot sin(\alpha) + 4 \cdot e \cdot sin(4 \cdot \alpha) \cdot cos(\alpha)$$

40 Woodruff Keys

Connections that use Woodruff keys are no longer commonly used, because the deep key way in these keys causes too great a notch effect. However, this connection still widely used in precision mechanics.

For Woodruff keys, you calculate the load on the shaft and hub (surface pressure). You can also add additional standards. The calculation of the load placed on the shaft and hub (surface pressure), together with determining the safeties, is performed as described in classic technical literature ([8]. The calculation defined by Niemann is based on DIN 6892 (key calculation).

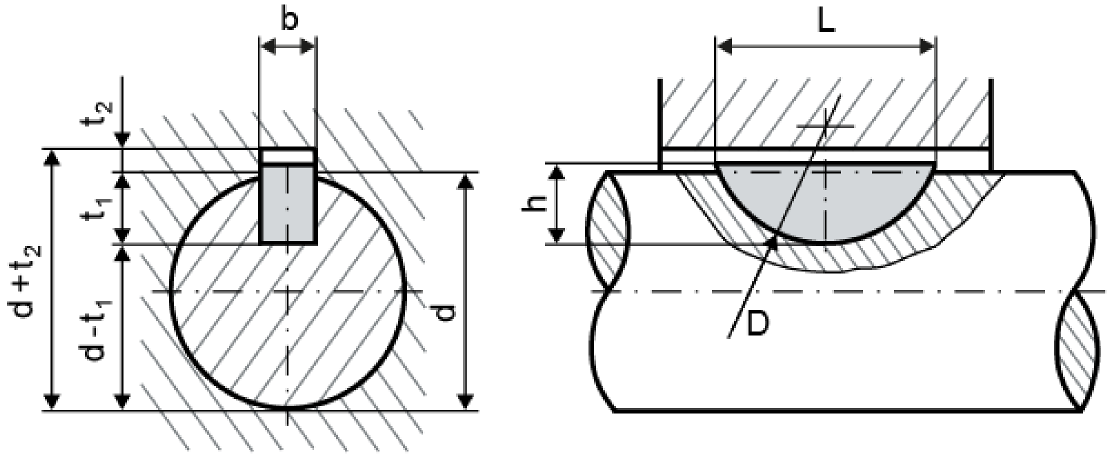
40.1 Standard profiles

You can select one of these standards from the selection list:

- DIN 6888, series A (high hub groove)
- DIN 6888, series B (low hub groove)
- Own input

After you select the standard for calculating the Woodruff key, a list of corresponding values is displayed.

b: Width	d: Diameter
h: Height	t_1: Shaft groove depth



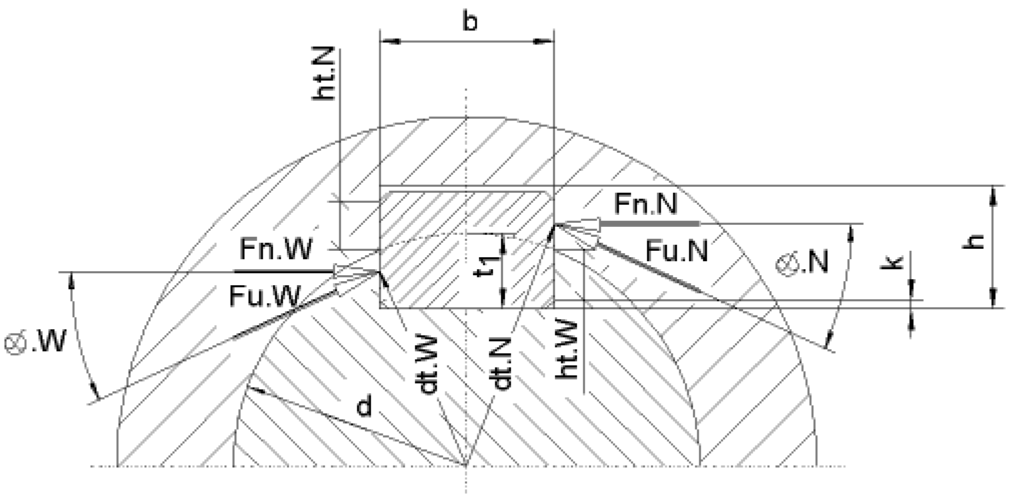


Figure 40.1: Woodruff key with circumferential and normal forces for the calculation as defined in Niemann

► **Note:**

Select the Own input option to define your own Woodruff keys.

40.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**average shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

40.1 table: Application factor according to DIN 6892

40.3 Torque curve/Number of changes of load direction

When you select the torque curve, you can select one of two positions:

1. No alternating torque
2. With alternating torque

If you select **With alternating torque**, the calculation not only defines the frequency of change of load direction, as defined in DIN 6892, Figure 7, but also the frequency of change of load direction coefficient f_w. If you select **No alternating torque**, the coefficient will be set to 1.0.

40.4 Occurring contact stress

This formula is used to calculate occurrences of contact stress. The formula is used both for the equivalent load and for the maximum load:

$p(eq,max)=k_{\varphi\beta}(eq,max) * k_1 * T * 2000/(d * l_{tr} * h_{tw} * z)$

$k_{\varphi\beta}$: Share factor	l_{tr}: supporting length
k_1: Length factor	h_{tw}: supporting height (shaft)
T: Torque	z: Number of Woodruff Keys
d: Shaft diameter	

40.5 Length factor

A **length factor, k_1**, is multiplied by the loading that takes into account how the load is distributed across the bearing length as a consequence of the torque action of the shaft and hub. The length factor depends on the equivalent diameter derived from the bearing length, the small and large external hub diameter and the width *c* to the external diameter. The distance a_0 is also used to determine the length factor. This factor is shown in a diagram in Niemann.

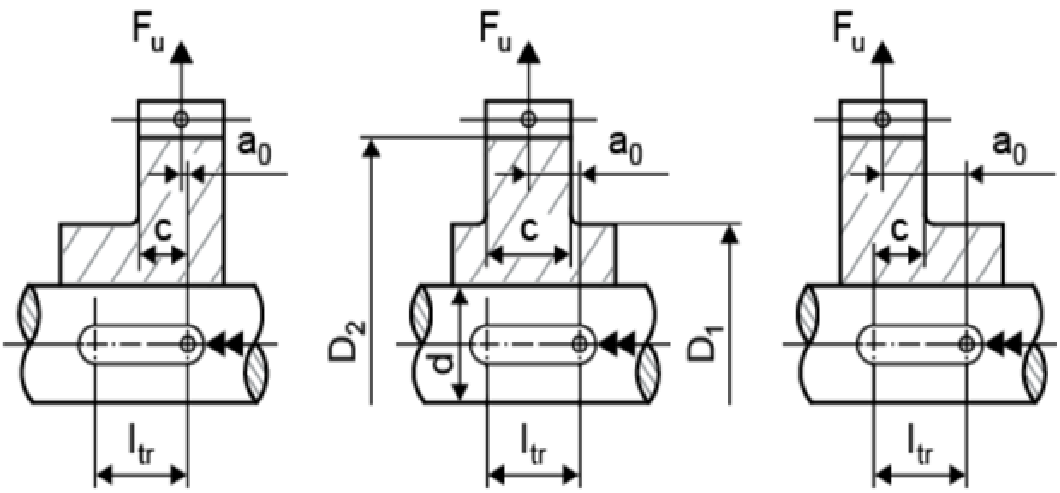


Figure 40.2: Woodruff key: load introduction

40.6 Share factor

A share factor of $k_{φβ}$ is taken into account in the calculation of the occurring contact stress. This is then multiplied by the load. Intermediate sizes not shown in the table are interpolated linearly.

Form-closure connections	**Spline connection with involute flank - tolerance zones according to DIN**					
	H5/IT4	**H7/IT7**	**H8/IT8**	**H9/IT9**	**H11/IT11**	**Maximum value**
$k_{φβ}$eq	1.1	1.3	1.5	2	4	z/2
$k_{φβ}$max	1	1.1	1.3	1.7	3	z/2

Table 40.2: Share factor according to Niemann

40.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}, this applies:

for **ductile materials**:
$p_{eq}=f_S * f_H * R_p$

for **brittle materials**:
$p_{eq}=f_S * R_m$

Structural steel	**Material**	**f_s**
Keys	Structural steel, bright steel, through hardening steel, case-hardened steel	1,1
Shaft	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.3
	GJL	1.1
Hub	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 40.3: Notch factor according to DIN 6892

KISSsoft always uses the minimum value for the notch factor fs.

A **notch factor of f_s** takes into account the supporting effect that occurs when bodies are subjected to pressure.
The **hardness influence coefficient f_H** is calculated from the surface to core hardness ratio for surface hardened parts.

The hardness influence coefficient for case-hardened steel is 1.15. Otherwise, it is set to 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the **load peak coefficient**, which depends on the material type and the number of load peaks. This coefficient is shown in a diagram in DIN 6892.

40.8 Materials

The selection list contains materials from the standard.
If you have set the Own input flag, a new dialog is displayed. This contains the material data used in the calculation, which you can specify to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that they can also be used in subsequent calculations.

40.9 Settings

Select Calculations -> Settings to enter the required safeties for the individual parts of the connection. According to Niemann, Volume I, 5th Edition (2019), the minimum safety for ductile materials is set in the range 1.0…1.3 and the minimum safety for brittle materials is set in the range 1.3…2.0. The required safeties define the values the system searches for during sizing.

The **support factors** for the shaft, hub and Woodruff key in DIN 6892 differ according to the materials and parts involved. The factors vary within a specific range. The minimum values are used in KISSsoft. These values can be overwritten.

If the Take pressure on key into account flag is set, the values of the Woodruff key are included in sizings. Otherwise, the sizing procedure will be carried out using the values for shaft and hub.

If you have selected Calculate material strength with wall thickness as raw diameter, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

40.10 Sizings

During the sizing process, the required value is defined such that the required safety (specified in the dialog you see by selecting Calculations→ Settings) is only just achieved. To display the results in the lower part of the main window, run the calculation after the sizing process is complete.

Possible sizings:

- Transmissible nominal torque T_n

41 Bolts and Pins

Five types of calculation have been created to categorize bolt/pin connections, depending on where they are used:

- Cross pin under torque
  The surface pressure of the shaft and hub, and the shearing of the pin, are checked in cross pin connections where large forces are involved.
- Longitudinal pin under torque
  Longitudinal pin connections are subject to surface pressure in the shaft and hub and shearing on the pin.
- Guide pin under bending force
  Guide pin connections are subject to bending stress due to a moment and to shearing force caused by transverse forces. The pin's shearing force, surface pressure and bending, and the surface pressure on the part, are calculated in this case.
- Bolt connection under shearing action (in double shear)
  The bolt is subject to bending and shear stress and to surface pressure in this arrangement. You can use different calculation methods, depending on the fit of the rod/bolt and fork/bolt connections. Experience shows that the decisive value in sizing non-sliding surfaces is the bending stress. In sliding surfaces, it is the surface pressure.
- Bolts in a circular layout (in single shear)
  In this arrangement, the effective torque is distributed uniformly over the individual bolts/pins. Therefore, the shaft and hub are subject to surface pressure from the individual bolts/pins, creating a shearing force. The maximum shear stress (calculated as specified in Roloff Matek [80], page 254) and the minimum safety for the bolts are also output.

The load placed on the bolts, shaft and hub (or part) is calculated, and the safeties are determined, as described in classic technical literature (Niemann, Maschinenelemente I, 4th Edition, 2005 [8]), **with the exception of Bolts in a circular layout**.

The cross-sectional area and moment of resistance to bending of spiral and slotted spring pins (bushes) is calculated according to Decker [81]. In configurations in which the bolts, spiral and slotted spring pins (bushes) are only subjected to shearing, the permitted shearing force specified in the relevant DIN EN ISO standard can be applied to the pins.

41.1 Influence factors

When calculating individual connections you must include a number of influencing factors which are defined depending on the load, stress, type, etc.:

- Application factor K_A
- Dynamic factor:
  Fixed load: Cd = 1; pulsating load: Cd = 0.7; alternating load: Cd = 0.5;
  for slotted spring pins and spiral pins (bushes)
  Fixed load: Cd = 1; pulsating load: Cd = 0.75; alternating load: Cd = 0.375;
- Reduction factors for full/grooved pin
  Full pin: Ck = 1; grooved pin (bending, shear): Ck = 0.7; grooved pin pressure: Ckp = 0.8;

Since the permissible stress values in the literature are very low, other material factors (the factors for permissible surface pressure/shear stress and bending stress) have been added to obtain the values in the table. You can define these factors by selecting Calculation → Settings.

41.2 Materials

In the selection list, you can select materials according to the standard.

If you have selected Own input, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that they can also be used in subsequent calculations.

41.3 Settings

Select Calculation → Settings to define factors for permissible surface pressure/shear stress/bending stress and the required safeties.

The factors you enter are multiplied with the tensile strength Rm for all parts, bolts and pins, except split spring pins and spiral pins (bushes). This calculation defines the permitted values.

In the case of split spring pins and spiral pins (bushes), the permitted values are taken directly from the file and do not depend on tensile strength Rm.

41.4 Permitted values

Full pins/bolts/grooved pins: The factors you enter are multiplied with the tensile strength Rm for all parts, bolts and pins, except split spring pins and spiral pins (bushes). To enter these values, select Calculation → Settings. This calculation defines the permitted values.

Split spring pins and spiral pins (bushes): The permitted values for split spring pins and spiral pins (bushes), are imported from a file.

For configurations that are only subject to shearing, the permitted values can be taken from the relevant DIN EN ISO standard for the pins.

These values for the minimum safety against shearing are displayed in the standards for a case involving double shear. If the calculation involves single shear, the minimum shearing force (double shear) used in the standard can be halved. This was described in the old standard (e.g. DIN 1481:1978).

The permitted values for shear and bending stress under different loads can be taken from the technical documentation provided by Niemann.

The permitted values for the material E295 were set as default values and assigned to the coefficients.

Material E295:

Tensile strength: Rm = 490 N/mm2

Surface pressure: $\rho_{zul,r}$ = 126 N/mm2

Shear stress: $\tau_{szul,r}$ = 70 N/mm2

Bending stress: $\sigma_{bzul,r}$ = 112 N/mm2

This results in:

Surface pressure coefficient: 0.25

Shear stress coefficient: 0.14

Bending Stress Coefficient: 0.22

41.5 Sizings

Click the Sizing button next to the diameter and nominal torque to size values based on the required safeties.

42 Bolts

KISSsoft calculates bolted joints according to VDI 2230 (2015). In addition to providing tables with standard values, the program also has a range of options that enable you to enter your own definitions for most of the constraint values (such as geometry and material data). Although the VDI 2230 standard does not have iteration functionality, i.e. it can be calculated manually, the flexible input and modification options give you a user-friendly software solution at your fingertips. However, you must be familiar with VDI 2230 before you can interpret the results and enter the required values correctly in the program.

VDI 2230 compares the assembly pretension force (FM/FMzul) with the minimum and maximum required assembly pretension force (FMmax and FMmin). FMzul is based on the permitted load on the bolt. FMmax and FMmin describe the minimum necessary pretension forces that ensure the connection will function correctly.

The minimum necessary mounting pretension force, FMmin, is calculated from the operating force FA and the resilience of the parts and the bolt, the embedding loss FZ, the thermal forces Fvth and the required clamp load FKerf. FMmax can be calculated while taking into consideration the scatter of the tightening technique (tightening factor αA) from FMmin.

$F_{M\max} = \alpha_A \cdot F_{M\min}$	(42.1)
$F_{M\max} = \alpha_A \cdot \left[F_{Kerf} + (1-\phi) \cdot F_A + F_Z + \Delta F_{vth}\right]$	(42.2)

The maximum required mounting pretension force FMmax must be less than the permitted pretension of the bolt FMzul. A similar comparison to this comparison is the one between the minimum required mounting pretension force FMmin and the minimum pretension force achieved by tightening FM/αA:

$F_{M\max} \overset{!}{<} F_M \qquad F_{M\min} \overset{!}{<} \frac{F_M}{\alpha_A}$	(42.3)

42.1 Special features in KISSsoft

In VDI 2230, the values for pretension force FM when utilizing 90% of the yield point, and for the tightening torque, are listed in Tables A1 to A4. These values are rounded (rounding error <= 1%). In contrast, KISSsoft uses the equations on which the tables are based to calculate the values. The results are therefore more general than the ones that use the values given in the tables and might therefore also differ from them slightly.

42.2 Values input in the "Load" tab

The bolted joint configuration, with loads, is defined in this tab. In addition, a load application, eccentric load/pretension, temperatures and a "Proof for bolts with FEM results", if selected, can be entered in this tab. The values apply for the working conditions.

42.2.1 Operating data

Enter operating data in the Basic data tab. In this tab, you can select one of these bolting configurations:

1. Bolted joint under axial load (single bolt)
2. Bolted joint under axial load and shearing force (single bolt)
3. Flange connection with torque and forces (multiple bolts)
4. Multi-bolted joint with any bolt position
5. Proof for bolts with FEM results

VDI 2230 assumes that the operating forces that will affect the bolted joint are already known. You can enter external forces and torques, depending on which configuration you select in KISSsoft. In configurations with several bolts, KISSsoft uses the values input for the entire joint to calculate the axial force and the pretension force for an individual bolt.

For a bolted connection under shear load, this shear load is received by the friction force between the bolted parts. The friction force is determined by the coefficient of friction and the pretension force. The torsional moment is then applied at the location of the friction radius. This radius is determined from the dimensions of the clamped parts (connecting solids).

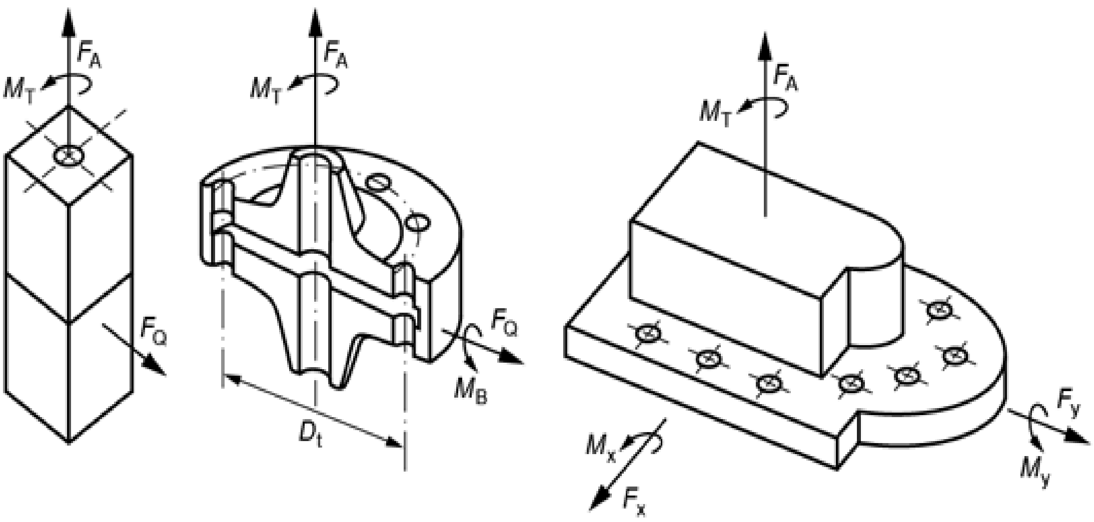


Figure 42.1: Bolting configurations: 1/2, 3 and 4

42.2.1.1 Bolted joint under axial load and shearing force (single bolt)

In the second configuration, the required clamp load for axial load transmission is calculated from the shearing force F_Q, the torque M_T, the coefficient of friction μ_T, the average friction radius r_a and the number of force transmitting parting lines q_T:

$$F_{KQ} = \frac{1}{\mu \cdot q_T} \cdot \left(F_Q + \frac{M_T}{ra} \right)$$	(42.5)
$$F_{Kerf} \geq \max\left(F_{KA} + F_{KP}, F_{KQ}\right)$$	(42.6)

F_{KQ}	Clamp load required to transmit a shearing force and/or a torque through friction grip
F_{KP}	Clamp load required to guarantee sealing (required when internal pressure is present)
μ_T	Interface coefficient of static friction (when shearing force or torques are present), → (see Figure 42.2).
ra	The friction radius resulting from the dimensions of the clamped parts to which the torsional moment is applied.

42.2.1.2 Bolted joint under axial load (single bolt)

The occurring axial forces F_{Amax} and F_{Amin} are entered directly. The necessary clamp load F_{Kerf} is defined in accordance with

$$F_{Kerf} = \max\left(F_{KA} + F_{KP}, F_{KQ}\right)$$	(42.4)

based on the required clamp load for axial load transmission F_{KQ} and the sealing function F_{KP} are calculated. F_{KA} is present to prevent gaping in the required clamp load and is calculated by the program.

42.2.1.3 Flange connection with torque and forces (multiple bolts)

The forces on the single bolt in the case of flanged joints (with stress from torque and/or shearing force and/or bending moment and/or axial force) are calculated according to [93], and also partially according to [84], for example, 8.4:

$$F_{KQe} = \frac{1}{n \cdot \mu_T} \left(\frac{2000 \cdot M_T}{dt} + F_Q \right)$$	(42.7)

$$F_{KPe} = \frac{F_{KP}}{n}$$	(42.8)
$$F_{Kerf} = \max\left(F_{KA} + F_{KP}, F_{KQ}\right)$$	(42.9)
$$F_{Bo} = \frac{1}{n}\left(F_{A\max} + \frac{4000 \cdot M_B}{dt}\right)$$	(42.10)
$$F_{Bu} = \frac{1}{n}\left(F_{A\min} - \frac{4000 \cdot M_B}{dt}\right)$$	(42.11)

dt	Pitch circle diameter
n	Number of bolts
μ_T	Coefficient of friction between the bolted parts, (see Figure 42.2)
F_Q	Shearing force on configuration
F_{Amax}	Axial force on configuration (maximum)
F_{Amin}	Axial force on configuration (minimum)
F_{Bo}	Upper operating force on the bolt that is subject to the greatest stress
F_{Bu}	Lower operating force on the bolt that is subject to the greatest stress
F_{KP}	Configuration sealing load
M_B	Bending moment on configuration
M_T	Torque on configuration
F_{Kerf}	Required clamp load
F_{KQ}	Required clamp load (e.g. for friction grip)
F_{KP}	Clamp load required to ensure sealing (at internal pressure)
F_{KA}	Clamp load required to prevent gaping under eccentric load

If you select a flanged joint configuration, we strongly recommend that you define the geometry of the clamped parts as **individual annulus segments**.

Experience shows that the results of VDI 2230 are usually very conservative for flanged connections. To achieve realistic results, you should increase the coefficient of friction between the parts.

Approximate values for coefficient of static friction μ_T in the interface (VDI 2230)

Material pairing	Coefficient of static friction μ_T	
	dry	lubricated
Steel – steel/cast steel (general)	0.10 ... 0.30	0.07 ... 0.12
Steel – steel; cleaned	0.15 ... 0.40	/
Steel – steel; case-hardened	0.04 ... 0.15	/
Steel – GJL	0.11 ... 0.24	0.06 ... 0.10
Steel – GJL; cleaned	0.26 ... 0.31	/
Steel – GJS	0.10 ... 0.23	/
Steel – GJS; cleaned	0.20 ... 0.26	/
GJL – GJL	0.15 ... 0.30	0.06 ... 0.20
GJL – GJL; cleaned/degreased	0.09 ... 0.36	/
GJS – GJS	0.25 ... 0.52	0.08 ... 0.12
GJS – GJS; cleaned/degreased	0.08 ... 0.25	/
GJL – GJS	0.13 ... 0.26	/
Steel – bronze	0.12 ... 0.28	0.18
GJL – bronze	0.28	0.15 ... 0.20
Steel – copper alloy	0.07 ... 0.25	/
Steel – aluminium alloy	0.07 ... 0.28	0.05 ... 0.18
Aluminium – aluminium	0.19 ... 0.41	0.07 ... 0.12
Aluminium – aluminium; cleaned/degreased	0.10 ... 0.32	/

Figure 42.2: Interface coefficients of static friction according to [1]

42.2.1.4 Multi-bolted joint with any bolt position

In a multi-bolted joint you can define bolts in any position. They are then affected by shearing force, a bending moment in two directions, and a torsional moment. The values for these bolt positions are in the Position of bolt tab. The load distribution on the bolts is calculated on the assumption that rigid plates are connected by springs at the bolt positions. Forces which do not affect the centroid point must be moved to the centroid point so they can be entered. You can represent different bolt diameters by entering a coefficient.

Enter coefficient = 1 for the bolt for which you want to calculate the proof. The proof is always performed for the bolt specified in the Basic data tab. For all the other bolts in the arrangement, set a coefficient that is either greater than or less than 1. The coefficient represents the diameter of the larger or smaller bolt compared to the bolt for which the proof has been performed. For example, enter coefficient = 2 for a bolt whose diameter is twice the size.

There are the following options for calculating bolts:

- **Validation for all bolts**
  The safeties for all the bolts are displayed either in a table in the results overview or in the report. You can select the most critical bolt and then validate it.
  This option is only useful if the factors = 1 are used in the definition.
- **Validate a single bolt**
  Enter the bolt number to validate a specific bolt.

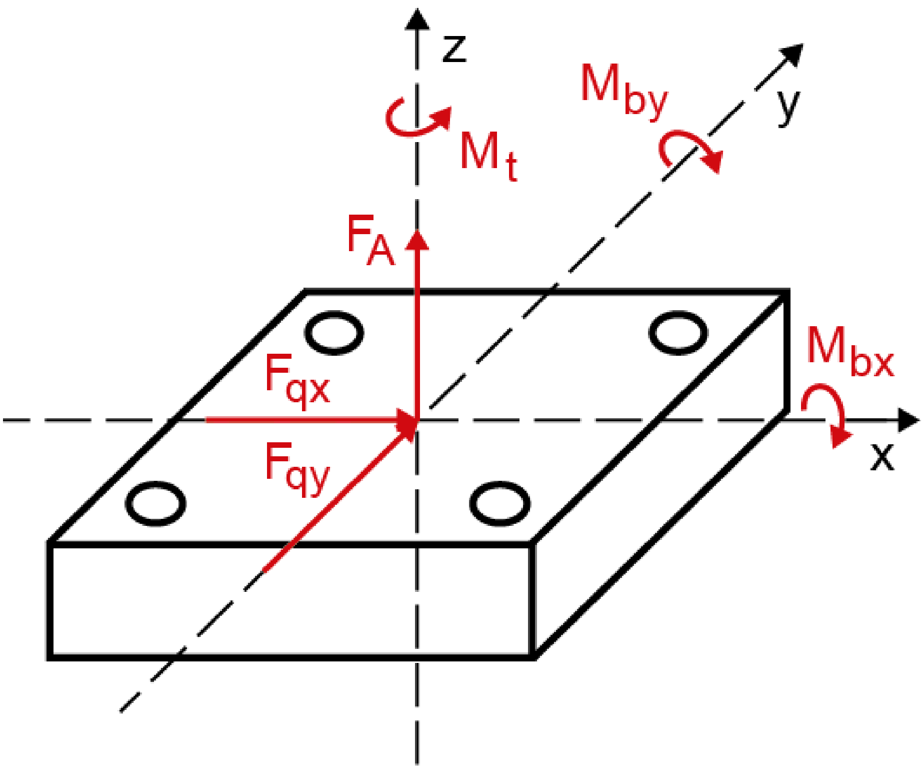


Figure 42.3: Definitions of forces and moments in KISSsoft

Once you have entered the operating data in the Basic data tab, you can define the bolt positions in the Position of bolt tab. You can either enter the bolt positions in a table or import them from a file. The resulting axial forces, and also the clamp loads required to transmit shearing force, are also displayed in the table.

You can also define an additional Factor for thrust bolts, in which it is assumed that compression is transmitted directly via the plates. The neutral axis can be moved by the Factor for thrust bolts. In [80], under the keyword "Multi-bolted Plate Joint", for example, an average pressure point of 1/4 plate height is assumed.

If the Use maximum required clamp load option is not selected, the required clamp load on the particular bolt is used for the calculation.

When you calculate the required clamp load, you can also take into account the algebraic prefix operator set for the shearing forces. Shearing forces caused by torsion and shearing force are then added at specific points and subtracted at other points. You should only include the prefix operator if you know the direction of the shearing force and if this force is constant. The default setting is that all the shearing force values are added together, no matter which direction of force is involved.

Click the Sizing button in the table (above, on the right) in the Position of bolt tab to display this window, in which you can enter different configurations. The selected number of bolts is then automatically inserted at regular intervals in the selected configuration.

The possible configurations are:

- line (values for: starting point, end point, number of bolts)
- circle (values for: center point, radius, number of bolts)
- circle segment (values for: radius, starting angle, end angle, number of bolts)

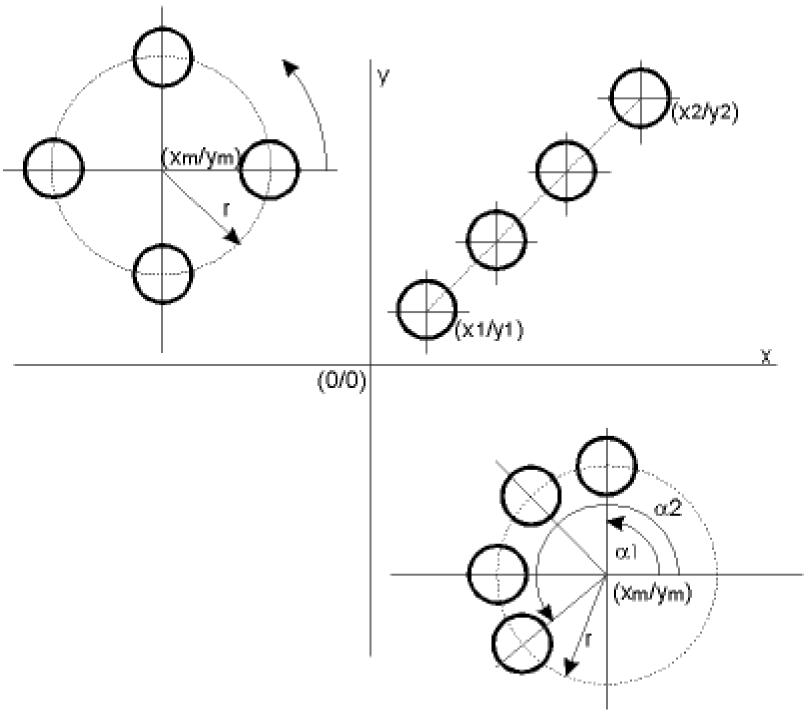


Figure 42.4: Position sizing options

42.2.1.5 Proof for bolts with FEM results

The following behavior can be taken into account in an FEM calculation (instead of a calculation according to VDI 2230 Sheet 1):

- Non-linear material behavior (for example, plasticity or yield)
- Non-linear boundary conditions (for example, load dependency of the contact surfaces),
- Geometric non-linear behavior (for example, significant deformation occurs when the tightening process is simulated)

As specified in VDI 2230 Sheet 2, Multi bolted joints, the FEM modeling is subdivided into 4 model classes:

Model class I:

The bore is not represented. The interface is either completely integrated or connected by a rigid coupling and a reference node for every bolt.

Model class II:

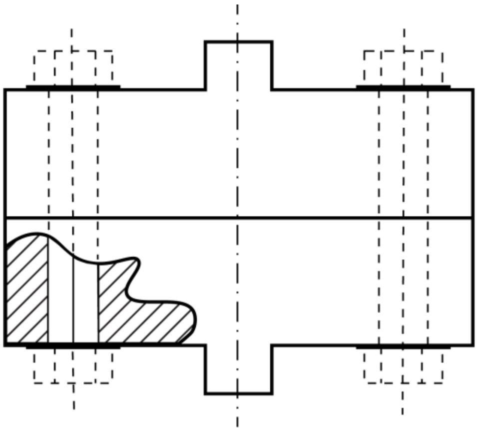
The bolt is represented as a beam or a spring element (with the ideal translational and rotational degrees of freedom for each section). Interface with contact definition.

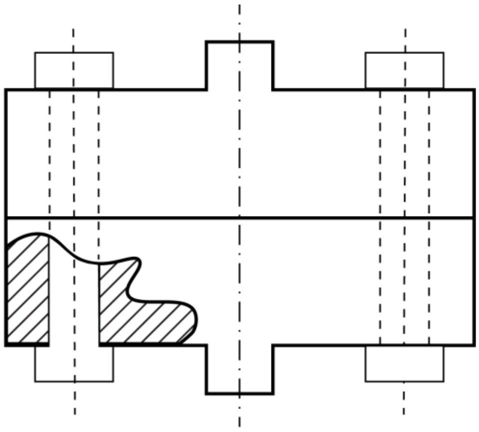
Model class III:

The bolt is shown in 3D as a cylinder with large cylinders at its ends. The interface and support areas are assigned contact definitions in the FEM calculation. The cylinder has the same core diameter as the thread.

Model class IV:

The bolt is modeled with the exact thread geometry. The assembly and working states are treated separately.





A proof according to VDI 2230 Sheet 1 with FEM results is really only a sensible idea for model classes II and III. Model class I does not supply informative enough results to create a proof according to VDI 2230 Sheet 1. If you are using model class IV, you already have all the necessary results and should use a different method from the one in VDI 2230 Sheet 1 for the proof.

You can select these model classes directly from the list in the Results from FEM calculation tab.

List items model class II and model class III:

The main difference in the modeling is that bolt resilience can only be defined in model class III.

In these list items, the "Bolted joint under axial and shearing force" configuration is required for calculating the amount of embedding.

In KISSsoft, the FEM results are embedded in the calculation as follows, according to VDI 2230 Sheet 1:

R0, R1	according to VDI 2230 Sheet 1
R2	You can input the clamp load for the sealing function FKP directly in the basic data, if it has already been determined in the FEM calculation. The lifting force FKab can be determined directly in the FEM calculation and entered here. If the results are not defined, they can be determined according to VDI 2230 Sheet 1. The axial load transmission FKQ is determined as specified in VDI 2230 Sheet 1.
R3	In model classes II and III, the resilience of the plates is taken from the results of the FEM calculation. In model class III, the bolt resilience is also taken from the results of the FEM calculation. In model class II, bolt resilience is derived as specified in VDI 2230 Sheet 1. The amounts of embedding are estimated according to VDI 2230 Sheet 1.
R4	Amounts of embedding are defined according to VDI 2230 Sheet 1, Fv'th is input directly from the FEM calculation, as the result.

R5	Fv'th can be input directly as a result from FEM, and Fkerf, φen, Famax, Fz can be taken from VDI 2230 Sheet 1 > determination of FMmin.
R6	Fmmax = αA*FMmin.
R7	In KISSsoft, assembly stress is calculated according to VDI 2230 Sheet 1. This would take an exceptional amount of time and effort if an FEM calculation were used instead.
R8	FV'th is included in the calculation of Fsmax (total bolt load). If FV'th > 0 it is set to 0, as specified in R8/1, VDI 2230 Sheet 1. All other σz, τmax, σredB are calculated according to VDI 2230 Sheet 1.
R9	FSA, MSA as input from the FEM results. You can specify the upper and lower limit. Proof of dynamic strength according to VDI 2230 Sheet 1 (R9/2). The permitted values are defined according to VDI 2230 Sheet 1. The effects of temperature are included directly in the FEM results.
R10	pBmax can be derived from the FEM results if model class III is being used, otherwise it is calculated according to VDI 2230 Sheet 1. You should only calculate the values for permissible surface pressure pGmax directly in FEM if these values are not already available.
R11	The minimum length of engagement cannot be represented realistically in model classes I to III. It would take a great deal of time and effort to model this in model class IV. The calculation is performed according to VDI 2230 Sheet 1.
R12	The calculation In KISSsoft is performed according to VDI 2230 Sheet 1. The values you need to input here are determined from the FEM calculation.
R13	Not applied.

List item model class III (only forces and moments, without resiliences):

The main difference between this and the other methods is that the calculation is performed without defining the resiliences.

When this list item is used, the load application factor is permanently set to n=1.

The FEM results needed to create a proof according to VDI 2230 Sheet 1 are grouped in the summary of the calculation steps below:

R0	The geometry must be defined according to VDI 2230 in FEM. In KISSsoft, set this to "Plates" for clamped parts.
R1	Tightening factor according to VDI 2230 Sheet 1

R2 to R6	Determined in the FEM model. The results are: FM, FM/α (may be different, because this is a real value), FKP, FSmax, FKR. FKQ which is determined with the values FQ, Mt and ra, is used to calculate safety against sliding.
R7	Mounting conditions: FMzul = FMtab at 90% load, otherwise σredMzul =v*Rp0.2. With FM value for calculating σredM from FE.
R8	Working conditions: FSmax is the result in the FEM calculation. σz, τmax, σredB are calculated according to VDI 2230 Sheet 1. SF = Rp0.2/σzmax
R9	FSA and MSA values from the FEM results. You can specify the upper and lower limit. Proof of dynamic strength according to VDI 2230 Sheet 1 (R9/2). The permitted values are defined according to VDI 2230 Sheet 1. The effects of temperature are included directly in the FEM results. σAbo = FSA,o/As + MSA,o/Ws; σAbu = FSA,u/As + MSA,u/Ws SD = σAS/σab σAS calculated according to the formulae in VDI 2230 Sheet 1.
R10	Input values: permissible surface pressure pGzul (under head and nut). Mounting: pMmax = FMzul/Apmin Operation: pBmax/Apmin SP = pGzul/PM,Bmax
R11	Minimum length of engagement is not calculated.
R12	FKR is an input value. FKerf is calculated according to VDI 2230 Sheet 1. FKQ interim results are calculated from input values FQ, Mt and ra. SG = FKR/FKerf
R13	Not applied.

42.2.2 Distances for eccentric load/clamping

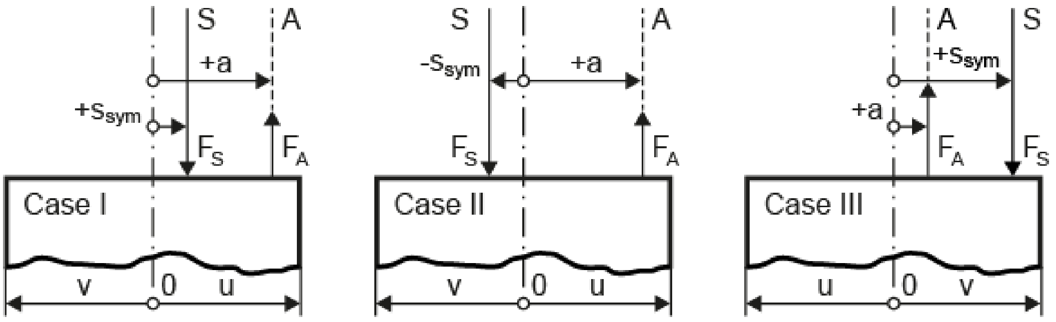


Figure 42.5: Possible load cases in the case of eccentric clamping

As you can see in the figure, the axis of the center of gravity of the clamping solid 0 - 0 determines the zero-point of the distances. The distance between load line of action A - A and the centroid axis 0 - 0 is always positive. The distance s_{sym} between the bolt axis S - S and the axis of the center of gravity 0 - 0 is set as positive if the bolt axis S - S and the load line of action A - A lie on the same side of the axis of the center of gravity 0 - 0. If not, this value is negative.

The dimension u defines the distance from the centroid axis 0 - 0 to the point at which gaping first occurs. This is the distance to the right-hand side in cases I and III, but the distance to the left-hand side in case II. In cases I and II, u must be positive, and in case II it must be negative. The guidelines governing the use of signs specified in VDI 2230 Sheet 1 are applied here.

42.2.3 Load application

The VDI guideline defines equations for calculating the load application factor. Select a configuration as shown in the figure 5ba0bcd87d0c9. The interface must lie within the range shown in gray. The length of the clamped parts h, the distance to the connection piece $a_k$and the length of the connected solid l_A (see Figure 42.7) define the position of the application of load point and therefore also the load application factor.

In single-bolted (tapped thread) connections, only configurations SV1, SV2 and SV4 are available.

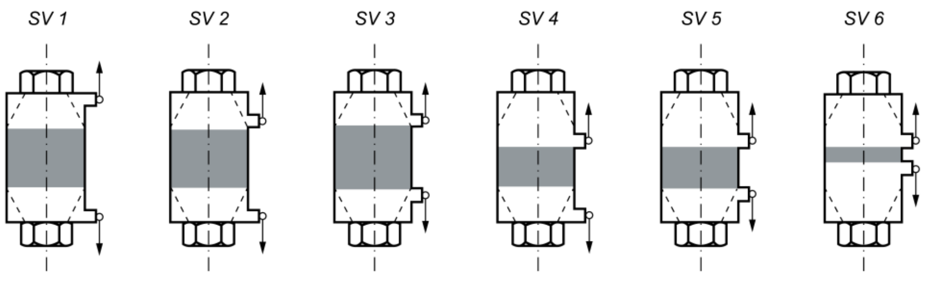


Figure 42.6: Configurations for defining the load application factor as shown in VDI 2230 (2015 edition).

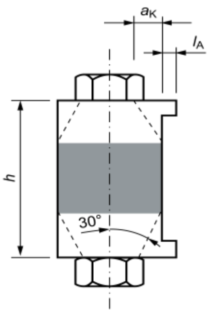


Figure 42.7: Inputs for defining the load introduction factor as shown in VDI 2230 (2015 edition).

42.3 Values input in the "Bolt/Nut" tab

42.3.1 Bolt data

A bolt's type, geometry, surface roughnesses and strength class can all be defined as bolt data.

Bolt type: You can use the following standard bolt descriptions from the database to define the bolt type:

DIN EN ISO 4762	Hexagon socket head cap screw
DIN 7984	Hexagon socket head cap screw with low head
	Standard thread M3.0 to M24.0
DIN EN ISO 4014	Hexagon headed screw with shank (formerly DIN 931 T1)
DIN EN ISO 4017	Hexagon headed screw without shank (formerly DIN 933)
DIN EN ISO 1207	Slotted cheese head screw
DIN EN ISO 8765	Hexagon headed screw with shank, metric fine thread (A B)
	Fine thread M8.0 to M64
DIN EN ISO 8676	Hexagon headed screw without shank, metric fine thread (A B)
	Fine thread M8.0 to M64
DIN EN 1662	Hexagon headed screw with flange, light series form F
	Standard thread M5.0 to M16
DIN EN 1662	Hexagon headed screw with flange, light series form U
	Standard thread M5.0 to M16
DIN EN 1665	Hexagon headed screw with flange, heavy series form F

	Standard thread M5.0 to M20
DIN EN 1665	Hexagon headed screw with flange, heavy series form U
	Standard thread M5.0 to M20
ASME B18.2.1	Square bolts, UNC thread, 0.25 to 1.5in
ASME B18.2.1	Hex bolts, UNC thread, 0.25 to 4in
ASME B18.2.1	Heavy hex bolts, UNC thread, 0.5 to 3in
ASME B18.2.1	Hex cap screws, UNC thread, 0.25 to 3in
ASME B18.2.1	Heavy hex screws, UNC thread, 0.5 to 3in

Reference diameter: You can either input your own value for the nominal diameter or click the Sizing button to calculate an approximate value after you input the operating data. This sizing function usually leads to bolt diameters that are too large. We therefore recommend you input a value that is 1 or 2 standard sizes less than the system's proposed value.

Bolt length: You can input any bolt length if you are inputting your own bolt geometry. Otherwise, after you input the bolt length, the system sets it to the next standard length.

Surface roughness of thread/head bearing area: The surface roughnesses influence the amount of embedding and therefore also the preload loss of the bolted joint.

Strength class: After the entry for standard strength grades you can click the Define... button to define your own strength values. The shearing strength ratios are set according to Table 5.5/2 in VDI 2230 Sheet 1 (2015), according to strength class. The values can be overwritten.

Own definition of bolt geometry: To define your own bolt geometry, you must set the Bolt type selection list to Own input. This activates the Plus button, which you can click to input your own bolt geometry values.

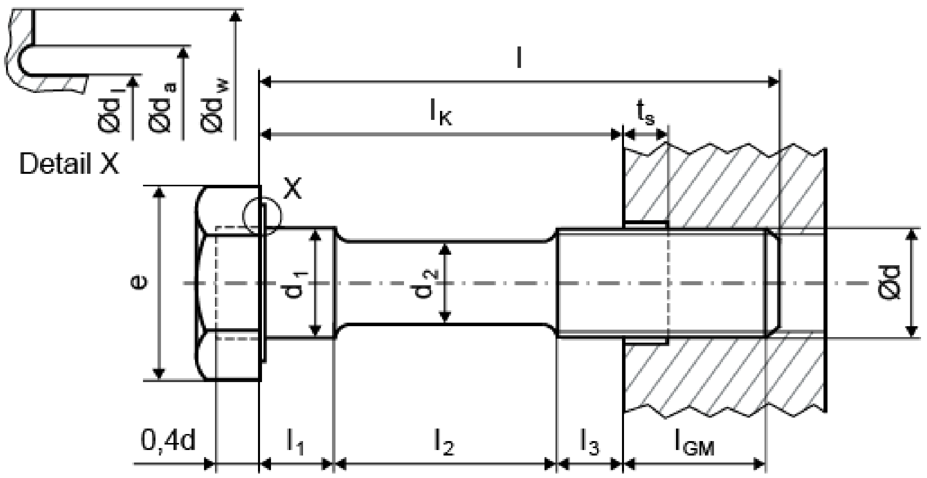


Figure 42.8: Bolt geometry

General tab: If you are using a bolt with a bore, input the bolt head dimensions as well as the bore diameter.

Thread tab: Data from the standard, the size of the thread, the pitch, and the thread length. This is where you define the factors used to calculate the flank diameter d_2 and the core diameter d_3 (d_2 = d - $d_2$factor*P; d_3 = d - $d_3$factor*P).

Bolt shank tab: Data about individual bolt cross sections. Click on the Plus button to add a new cross section. Click the — button to remove the selected cross section. Click on the ✕ button to delete all the cross sections.

42.3.2 Type of bolted joint

To define the type of bolting, select either Nut or Blind hole. This corresponds to the difference between through-bolt and single-bolted tapped thread connections as defined in VDI. Click on the appropriate Define... button to display a dialog in which you input additional data about the nut or blind hole.

Blind hole

The counter bore depth t_s describes a milled groove without thread, which is primarily used to elongate the clamp length (see Figure 42.8).

Nut

In the nut definition screen, you can either select a standard for the geometry or define the dimensions yourself by selecting Own Input.

For example, when calculating the length of engagement, you can either define the hardness from the strength class (as specified in DIN EN ISO 898-2) or define the shearing strength directly from the material.

The Own Input option is also available in both variants. However, when you input the strength class, you must also define the ratio of the shearing strength to tensile strength (τBM/Rm).

The system then converts the hardness value you enter here into tensile strength as part of the hardness conversion process. The tensile strength Rm is then multiplied with the ratio (τBM/Rm) to calculate the shearing strength (τBM).

The minimum hardness value for nuts with a standard thread (including UNC) is taken from the strength class in Table 6 in DIN EN ISO 898-2. The minimum hardness value for fine threaded nuts (including UNF) is taken from the strength class in Table 7 of the same standard.

► **Note**

If the dimension of the interface area DA is only slightly larger than the bearing area diameter of the bolt head dw, it must be calculated as a through-bolt connection (note the deformation cone).

(DA to ~1.4*dw)

42.3.3 Washers

Click on the appropriate checkbox to insert a washer between the nut and the component or the head and the component.

Select Calculation > Settings to display the Settings tab. In it, if you select Determine specific thermal expansion of washers, you can also define the thermal expansion coefficients that are used to calculate the difference in pretension force. You will find a more detailed description in the "Settings" section.

42.3.4 Extension sleeves without external forces

You can specify the length of an individual extension sleeve in the extension sleeves dialog. In a single-bolted (tapped thread) connection, a extension sleeve can be used under the bolt head. In a through-bolt connection, an extension sleeve can be used under the bolt head and under the nut. No external forces are to act on the extension sleeves defined here. These sleeves are to be used to create the space between the bolt head or nut and the part.

The extension sleeves are taken into account when sizing the length, calculating the resiliences, and in the length expansion at operating temperature.

42.3.5 Length of engagement

Select Length of engagement to calculate the length of engagement and check the stripping strength of the thread according to VDI 2230, section 5.5.5.

Click the Plus button to display a screen in which you can enter the data for the length of engagement. Use the Sizing buttons to set the individual defaults which were calculated from the entries in the main window.

The length of engagement meffmin is calculated from the (theoretical) tensile strength R_m of the bolt material, the length of engagement m_{effmax} is calculated for the bolt and internal thread (with R_{mmax}, d_{min} or d_{2min} and D_{2max} or D_{1max} according to VDI 2230 Sheet 1, Equation 210/213). The more critical case is then displayed in the results.

The default value for the R_{mmax}/R_m coefficient is 1.2. This is also given as a value based on practical experience in VDI 2230.
To change the R_{mmax}/R_m coefficient, select Calculation > Settings.

To calculate the worst case (VDI 2230, formula 210), the thread tolerance must also be taken into account.
For tolerances according to ISO 965-1, the required tolerance can be selected in a selection list, for the minimum external bolt diameter d_{min}, the maximum flank diameter of the internal thread D_{2max}, the minimum flank diameter of the bolt thread d_{2min} and the maximum minor diameter of the internal thread D_{1max}. If the tolerances are to be defined on the basis of a different standard, Own input must be selected in the selection list and the appropriate values must be entered.

When the Sizing button is clicked, the nominal value without tolerance is inserted in the relevant field.

The main report lists the stresses, the minimum length of engagement, and the safety against shearing under load, with the maximum pretension force for the connection.

42.4 Values input in the "Assembly" tab

42.4.1 Conditions

In this calculation, you can define the yield point utilization, the maximum assembly pretension force or both tightening torques as constraints. If you define the maximum and minimum tightening torque as constraints, the tightening factor is then calculated from this torque variation and the friction coefficient variation. You can also enter values for the number of load cycles, amount of embedding, preload loss and temperatures for the bolted joint in this window.

Use of the yield point
Usually, when sizing bolts, the bolt is tightened to 90% of its yield point to calculate the pretension force. However, if you use yield point or angle of rotation-controlled tightening, you can increase this utilization value up to 100%.

Number of load cycles
If this number of load cycles is $N_D >= 2*10^6$, the fatigue life for final heat treated and final rolled bolts are calculated according to the formulae specified in VDI 2230, 5.5/20 and 5.5/21. If smaller values are involved ($N_D < 2*10^6$), limited life sizing is performed for the connection (5.5/22 and 5.5/23).

Amount of embedding

The amount of embedding is calculated according to which calculation method you use. You can also input an extra amount of embedding value for flat seals. In addition, you can overwrite the calculated amount of embedding with your own value or input the preload loss directly. If you input your own preload loss, the amount of embedding is no longer taken into account.

Assembly and operating temperature
Now KISSsoft's bolt calculation function has been extended, it can be used for the calculation guideline specified in VDI 2230, which also calculates bolted joints for operating temperatures between -200 and +1000 degrees Celsius. You can specify different temperatures for the bolt and the clamped parts (connecting solids). You can also take into account the temperature-dependent changes to the Young's modulus, the thermal expansion coefficients, the yield point and the pressures permitted for the materials. You can either use empirical formulae to calculate these temperature-dependent values, or specify your own values. Since the empirical formulae for steel have already been determined, you should check the values for high-temperature changes or, even better, enter your own values here.

All the criteria for the bolted joint are checked in the assembled state, and also in a stationary or non-stationary state at operating temperature (according to VDI 2230: pretension force, bolt load, fatigue life, and surface pressure).

KISSsoft automatically performs the calculation for assembly and the operating temperature at the same time. This calculation should also be performed for a higher temperature difference between the bolt and the parts. The minimum temperature difference between the parts or the bolt and the assembly temperature must be at least 30° C, so that results are displayed in the report.

42.4.2 Technical Explanations

The critical influences of temperature on the operating properties of bolts are:

- Change in pretension force due to thermal expansion
- Change in pretension force due to relaxation (at high temperatures)
- Brittleness (at high and low temperatures)

The lack of sufficient general data for the materials for bolts and clamped parts means the number of calculation options is also limited. The change in pretension force due to thermal expansion can be calculated very accurately because, as a first approximation, changes to the thermal expansion value can be viewed as linear, in line with the temperature, at least for the temperature range -100 to +500°C. The other effects (relaxation and brittleness) can be minimized by selecting appropriate materials and taking precautionary measures (see the relevant literature).

The calculation of the change in pretension force due to thermal expansion is performed with temperature-dependent thermal expansion value and Young's modulus, as specified by H. Wiegand in Schraubenverbindungen, 4th Edition 1988, section 7.1.3.1. All other calculations are based on the equations in VDI 2230 with the appropriate values at operating temperature.

KISSsoft suggests sensible values for much of the data you can input (Young's modulus, thermal expansion value, yield point at operating temperature), which are based on current technical literature. This consists of DIN standards or technical documentation from the company Bosshard, in Zug, Switzerland. These suggestions are based on the Young's modulus for ambient temperature and, of course, also on the operating temperature. When calculating the suggestion for the permissible pressure at operating temperature, the proportional change to the yield point was assumed. The suggestions are average values for "commonly used steels". They do not refer to one specific material and must therefore be checked carefully in critical situations because the influence of temperature also varies according to the type of material involved. If you want to calculate material data automatically, using empirical formulae, simply click on the Calculation > Settings.

42.4.3 Tightening technique

Uncertainties such as the scatter of coefficients of friction, differently accurate tightening techniques, and instrument, operating and reading errors, result in the scatter of the assembly pretension force achieved. For this reason, oversizing the bolt is necessary, and is expressed by the tightening factor $\alpha_A = F_{Mmax}/F_{Mmin}$. If the required minimum pretension force F_{Mmin} remains constant, an increasing tightening factor α_A means that the bolt must be sized for a larger maximum required assembly pretension force F_{Mmax} (due to the greater scatter). Tightening technique and associated tightening factors:

Tightening factor α_A	**Tightening technique**	**Adjusting technique**
1.1 to 1.2	Tightening with elongation control	Sound travel time
1.1 to 1.3	Mechanical elongation due to thrust bolts in nut or head	Specify the bolt elongation and the forcing torque of the thrust bolts
1.2 to 1.5	Mechanical elongation due to multipartite nut	Torque
1.1 to 1.5	Tightening with mechanical elongation measurement	Mechanic elongation measurement
1.2 to 1.6	Hydraulic tightening (VDI 2230:1988)	Compression or length measurement
1.1 to 1.4	Hydraulic non-frictional and torsion-free tightening	Compression or length measurement or additional rotational angle of nut
1.0 (*)	Yield point-determined mechanical or manual tightening	Preset torque-angle of rotation coefficient
1.0 (*)	Rotation angle-controlled mechanical or manual tightening	Experimental determination of the preload moment and angle of rotation
1.4 to 1.6	Torque-controlled tightening with hydraulic tool	Pressure measurement
1.4 to 1.6	Torque wrench (with experimental load)	Experimental determination of the required tightening torques on the original bolting part, e.g. by measuring the length of the bolt.

1.6 to 2.0	Torque wrench (by estimating the coefficient of friction, class B)	Determination of the nominal tightening torque by estimating the coefficient of friction, value (surface and lubrication ratios)
1.7 to 2.5	Torque wrench (by estimating the coefficient of friction, class A)	Determination of the nominal tightening torque by estimating the coefficient of friction, value (surface and lubrication ratios)
1.6 to 1.8	Torque wrench (by estimating the coefficient of friction) (VDI 2230:1988)	Determination of the nominal tightening torque by estimating the coefficient of friction, value (surface and lubrication ratios)
2.5 to 4.0	Pulse-controlled tightening with an impact wrench	Torque wrench adjustment with tightening-up moment

(*) FMmax/FMmin is greater than 1. Despite this, αA = 1 is used for the dimensioning formula. See VDI 2230:1988, section 5.4.3.2.

42.4.4 Coefficients of friction

In KISSsoft, you can specify an interval for coefficients of friction. The minimum value is used for calculation with F_M and F_{Mmax} and the maximum value is used for calculation with F_{Mmin} and F_M/α_A. The scattering of the coefficient of friction value therefore affects the scatter of the tightening torques.

Friction coefficient μ_G according to Wiegand, Schraubenverbindungen

thread	material	surface	production	lubrication	outside thread										
				material	steel									A2	
				surface	black finish or phosphate treated				electro galvanised (Zn6)		cadmium galvanised (Cd6)		adhesive		
				production	rolled			cut	cut or rolled						
				lubrication	dry	oiled	MoS_2	oiled	dry	oiled	dry	oiled	dry	dry	oiled
inside thread	steel	bare	cut	dry	0.12 - 0.18	0.10 - 0.16	0.08 - 0.12	0.10 - 0.16	/	0.10 - 0.18	/	0.08 - 0.14	0.16 - 0.25	/	/
inside thread	steel	electro galvanised	cut	dry	0.10 - 0.16	/	/	/	0.12 - 0.20	0.10 - 0.18	/	/	0.14 - 0.25	/	/
inside thread	steel	cadmium galvanised	cut	dry	0.08 - 0.14	/	/	/	/	/	0.12 - 0.16	0.12 - 0.14	/	/	/
inside thread	GG/GT	bare	cut	dry	/	0.10 - 0.18	/	0.10 - 0.18	/	0.10 - 0.16	/	0.08 - 0.16	/	/	/
inside thread	AlMg	bare	cut	dry	/	0.08 - 0.20	/	/	/	/	/	/	/	0.32 - 0.43	0.28 - 0.35
inside thread	A2		cut	dry	/	/	/	/	/	/	/	/	/	0.26 - 0.50	0.12 - 0.23

Figure 42.9: Coefficients of friction in the thread.

Friction coefficient μ_K and μ_M according to Wiegand, Schraubenverbindungen

<table>
<tr><th colspan="5">support area</th><th colspan="12">screw head/nut</th></tr>
<tr><th colspan="5">material</th><th colspan="10">Steel</th><th colspan="2">A2</th></tr>
<tr><th colspan="5">surface</th><th colspan="6">black finish or phosphate treated</th><th colspan="2">electro galvanised (Zn6)</th><th colspan="2">camium galvanised (Cd6)</th><th></th><th></th></tr>
<tr><th colspan="5">produc-tion</th><th colspan="3">pressed</th><th colspan="2">twisted</th><th>grinded</th><th colspan="4">pressed</th><th></th><th></th></tr>
<tr><th>support area</th><th>material</th><th>surface</th><th>produc-tion</th><th>lubri-cation</th><th>dry</th><th>oiled</th><th>MoS2</th><th>oiled</th><th>MoS2</th><th>oiled</th><th>dry</th><th>oiled</th><th>dry</th><th>oiled</th><th>dry</th><th>oiled</th></tr>
<tr><td rowspan="8">counter surface</td><td rowspan="4">steel</td><td rowspan="2">bare</td><td>grinded</td><td rowspan="8">dry</td><td>/</td><td>0.16 - 0.22</td><td>/</td><td>0.10 - 0.18</td><td>/</td><td>0.16 - 0.22</td><td>0.10 - 0.18</td><td>/</td><td>0.08 - 0.16</td><td>/</td><td>/</td><td>/</td></tr>
<tr><td rowspan="3">machined</td><td>0.12 - 0.18</td><td>0.10 - 0.18</td><td>0.08 - 0.12</td><td>0.10 - 0.18</td><td>0.08 - 0.12</td><td>/</td><td>0.10 - 0.18</td><td>0.10 - 0.18</td><td>0.08 - 0.16</td><td>0.08 - 0.14</td><td>/</td><td>/</td></tr>
<tr><td>electro galvanised</td><td colspan="2">0.10 - 0.16</td><td>/</td><td>0.10 - 0.16</td><td>/</td><td>0.10 - 0.18</td><td>0.16 - 0.20</td><td>0.10 - 0.18</td><td>/</td><td>/</td><td>/</td><td>/</td></tr>
<tr><td>cadmium galvanised</td><td colspan="6">0.08 - 0.16</td><td>/</td><td>/</td><td>0.12 - 0.20</td><td>0.12 - 0.14</td><td>/</td><td>/</td></tr>
<tr><td rowspan="2">GG/GTS</td><td rowspan="3">bare</td><td>grinded</td><td>/</td><td>0.10 - 0.18</td><td>/</td><td>/</td><td>/</td><td colspan="3">0.10 - 0.18</td><td>0.08 - 0.16</td><td>/</td><td>/</td><td>/</td></tr>
<tr><td rowspan="3">machined</td><td>/</td><td>0.14 - 0.20</td><td>/</td><td>0.10 - 0.18</td><td>/</td><td>0.14 - 0.22</td><td>0.10 - 0.18</td><td>0.10 - 0.16</td><td>0.08 - 0.16</td><td>/</td><td>/</td><td>/</td></tr>
<tr><td>AlMg</td><td>/</td><td colspan="3">0.08 - 0.20</td><td></td><td>/</td><td>/</td><td>/</td><td>/</td><td>/</td><td>0.35 - 0.50</td><td>0.08 - 0.11</td></tr>
<tr><td>A2</td><td></td><td>/</td><td>/</td><td>/</td><td>/</td><td>/</td><td>/</td><td>/</td><td>/</td><td>/</td><td>/</td><td>0.08 - 0.11</td><td>0.08 - 0.12</td></tr>
</table>

Figure 42.10: Coefficients of friction in the head bearing area and nut bearing area.

You can also use the sizing according to friction classes A to E as specified in VDI 2230 Sheet 1, Annex to Table A5 to define the values for the coefficients of friction. The minimum and maximum coefficients of friction for the thread, the head bearing area and the nut support are then imported into KISSsoft.

42.4.5 Rotation-angle controlled tightening

The angle of rotation is primarily calculated on the basis of the mean value of the achieved preload $(F_M + F_M/\alpha_A)/2$ and the associated elastic deformation of the bolt. The following boundary conditions, which are relevant to assembly, can also be predefined. This data does not directly affect the main calculation, such as the achieved pretension forces, it is derived passively from the results at the end of the bolt calculation instead.

Number of steps

The calculated tightening angle can be divided into a specific number of partial steps. The angles for each partial step are then output separately in the report.

Pretension

Before angle-controlled tightening is performed on a bolt, it is tightened to a threshold torque. This is entered as a proportional percentage of the mean value of the torque achieved during assembly (MA_(FMm)). This pretension point is the starting angle of the tightening angle identified by the letter (b) in the report. The tightening angle, identified by the letter (a) in the report, includes the angle up to when the threshold torque is achieved. However, due to its increasing lack of precision, this is only displayed for small pretension values.

Plastic deformation

If the yield point is exceeded on a cross section, you can specify the proportional percentage by which the length of this segment is to be stretched plastically, here. This plastic change in length is then converted into an angle of rotation, based on the thread lead. This angle is then added to the tightening angle.

Maximum yield point

If a part of the bolt used here is expected to have a yield point that is higher than the nominal yield point, this maximum yield point can be set relative to the nominal yield point. Alternatively, you can enter the maximum yield point as an absolute value in the additional data for the bolt strength class (in the bolt data).

42.5 Values input in the "Clamped parts" tab

The Clamped parts tab contains the data for the materials and geometry of the clamped parts, the distances involved for eccentric loading/clamping and data about the load application factor.

42.5.1 Geometry of clamped parts (connecting solids)

There are several basic types of clamped parts (connecting solids):

- Plates
- Cylinder
- Prismatic body
- Annulus segment

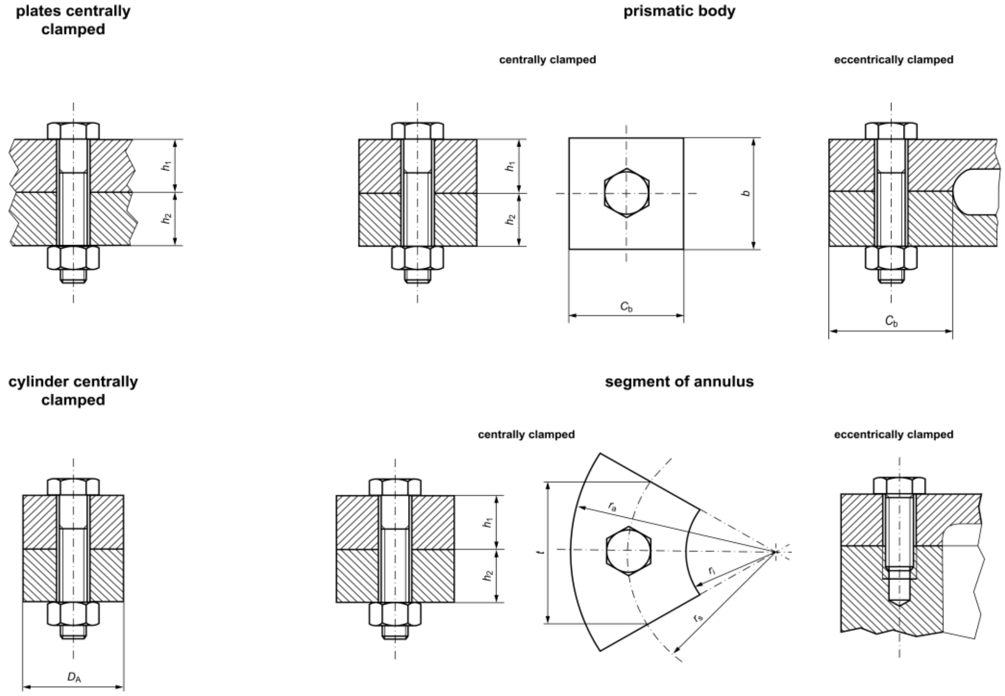


Figure 42.11: Clamped parts (connecting solids)

If you select Plates, it is assumed that the clamping deformation cone will be able to expand freely sideways. For all the other selection options, click the Geometry button to enter the relevant dimensions of the clamped parts you want to use in the calculation.

Click the Bore... button to define the through-bore in the clamped component. You can also define chamfers under the head or nut here. These chamfers are then included when the bearing areas are calculated. The chamfer reduces the external radius of the bearing area, therefore increasing the surface pressure.

You simply enter the different material situations in the list. The upper values for the permissible pressure, Young's modulus and thermal expansion are material values that apply at room temperature and, unless they are values you have entered, are always shown with a gray background. If Calculate temperature dependent material data automatically with estimation formulae is selected in Module specific settings , the values for the operating temperature are calculated empirically and displayed in the lower half of the particular material. You cannot edit these values. If this option is not selected, you must enter your own values here. Click the Sizing buttons to call the different empirical formulae so they can be applied in the calculation. Click the Plus button to add a material, click the — button to delete the selected element and click the X button to delete all the positions. The calculated clamp length is displayed in the lk field.

42.6 Settings

Select Calculations > Settings to enter additional values:

- **Don't abort calculation when error messages (permissible pressure, permissible pretension force) occur.**
  If error messages about exceeding the yield point or the permissible pressure are displayed, the program continues the calculation if this option is selected.
- **Calculate minimum pretension force F_M/α_A**
  If this option is selected, the load case F_M/α_A is also calculated. The pretension force F_M/α_A is the minimum pretension force that must be achieved. α_A is the tightening factor. It describes the pretension force scatter. If this option is selected, the results for this pretension force are displayed in the report.
- **Do not increase required clamping force for eccentric clamping**
  KISSsoft increases the required clamp load to prevent gaping for eccentric clamping. You can switch off this function here. Take care when using this option. The calculation assumes that gaping does not occur!
- **Ignore inside diameter of head bearing area d_a when calculating surface pressure**
  If this option is selected, d_a is not taken into account when the surface pressure is calculated. Consequently, the supporting area is not determined using the greater diameter (inside diameter of head bearing area d_a or bore diameter d_h), but always the bore diameter d_h instead. The supporting area increases to the bore diameter d_h if the area limited by d_a begins to yield.
  This option is not VDI 2230 compliant, as, in it, the greater diameter is always used.
- **Use the resiliences of a through-bolt joint for a tapped thread joint**
  If the interface area of the clamped part is relatively small compared to the bolt head bearing area, the tapped thread joint can be treated like a through-bolt joint when the resiliences are calculated.
- **Operating force only at operating temperature**
  Normally, KISSsoft calculates the minimum pretension force based on the required clamp load and loading at ambient and operating temperatures. This option can be selected if the operating force only occurs at operating temperature. In this case, the minimum pretension force is then only calculated at operating temperature.
- **Calculate temperature dependent material data automatically with estimation formulae**
  KISSsoft can automatically calculate material data at operating temperature by using empirical formulae. These empirical formulae do not take into account the material data you entered: they use an average dependency for "commonly used steels"! If this option is not selected, you can enter the material data at operating temperature manually.
- **Determine specific thermal expansion of washers**
  This displays the input field for thermal expansion values in the sub-window for washers. If this option is not selected, the difference in pretension force is calculated using the average thermal expansion of the plates. In other words, the washer has the same thermal expansion as the plates. This is why you have the option of inputting this value. If you do so, the difference in pretension force is calculated using the value you

specified, but the resilience of the plates is still used in this calculation. VDI 2230 does not specify that a special thermal expansion calculation is to be used for washers.

- **Calculate mounting and operating stress without torsion**
  You can select this option if the connection is tightened using a process in which torsional stresses do not occur in the bolt. As a result, only the tensile stress in the bolt is taken into account for calculating the required safety against yield.
- **Reduction coefficient**
  The reduction coefficient is used to calculate equivalent stress when the machinery is in its working state. In many cases, the torsional stresses in elastically preloaded connections reduce by 50%. For this reason, VDI 2230 recommends the value 0.5 is used here.
- **Exceeding the yield point**
  Three selection options are available here: yield point cannot be exceeded, yield point can only be exceeded during operation, or yield point can be exceeded during operation and mounting. This enables you to select your preferred calculation variant.
- **Hardening coefficient**
  An additional hardening factor, k_v is used when calculating whether the yield point has been exceeded during mounting and during operation. The default value for the hardening coefficient is 1.15. VDI specifies that it should between 1.1 and 1.2.
- **Additional torsional moment during operation**
  An additional torsional moment can be defined when calculating working stress. This torsional moment is then used in the shear stress calculation. This applies both in cases where the yield point cannot be exceeded and in cases in which the yield point can be exceeded.
- **Additional transverse force during operation**
  An additional transverse force can be defined when calculating working stress. This force is then used in the shear stress calculation. This applies in cases where the yield point is exceeded.
- **Infinite life strength**
  Selection list used select the type of bolt for which the infinite life strength calculation is to be performed. In the case of high-strength friction-grip fasteners, the sustainable fatigue life is reduced by 10% because of special geometrical features. In the case of hot-galvanized high-strength friction-grip fasteners, the sustainable fatigue life is reduced by 30%. (Comment in VDI 2230, chapter on alternating stress)
- **Tensile strength of bolt coefficient**
  This coefficient is used to calculate the minimum length of engagement required to achieve a practical value for R_m, as in VDI 2230 (see chapter 42.3.5, Length of engagement).
- **Utilization of yield point (reference)**
  A reference utilization can also be displayed for the Pretension forces and tightening

torques graphic. If you overwrite the value, any reference utilization can be defined and then inserted in the graphic. If you do not overwrite the value, the default value for yield point and rotation angle-controlled tightening with 100% utilization is set. For all other tightening techniques, 90% utilization is applied.

43 Snap Rings

This module is used to perform calculations for shaft or hub snap rings. The calculations are performed according to the manual published by the company Seeger.

43.1 Basic data

Input the following data in the Basic data tab:

- "Geometry" group
  - Shaft/bore ring: specifies whether the calculation is to be performed for a shaft or for a bore ring
  - Retaining ring/circlip: specifies whether the calculation is to be performed for a retaining ring or a circlip
  - d1: nominal length, the shaft diameter for a shaft ring, or the bore diameter for a bore ring
  - d2: groove diameter
  - d3: inside diameter of the snap ring for shafts or external diameter of the snap ring for bores in the unstressed state
  - b: the maximum radial width of the snap ring
  - Dimension l: (see Figure 43.1)
  - s: the thickness of the ring
  - ψ: permissible dishing angle of the snap ring (see Figure 43.2)
  - g: angled or corner distance/radius

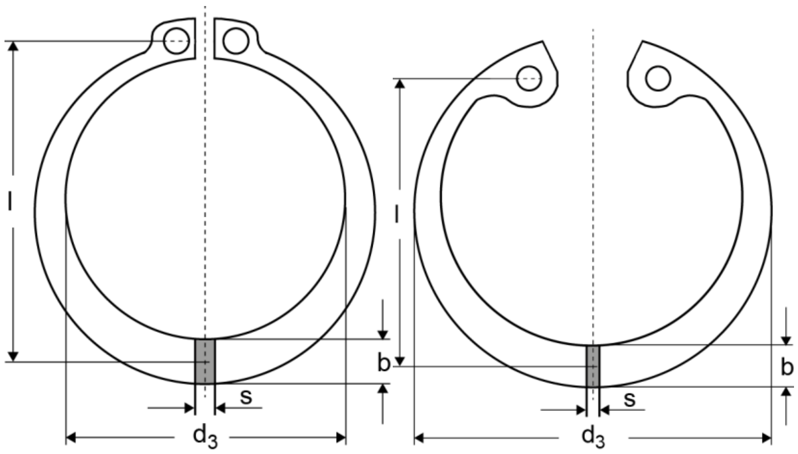


Figure 43.1: Geometry of shaft ring (a) and bore ring (b)

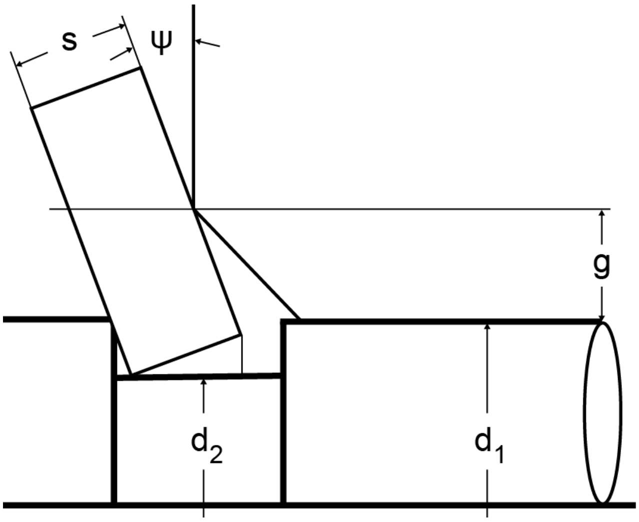


Figure 43.2: Definition of geometry terms s, ψ, g.

- "Operating data" group
  - q: the load factor, taking into consideration the effect of the shoulder length ratio
  - μ: the coefficient of friction between the ring surface and the shaft/bore surface
- "Materials" group
  - In this group you can define the material of the ring and shaft/bore. The functionality is similar to the rest of the KISSsoft modules in the "Connections" module group.

43.2 Automatic calculation of load factor q

Click on the Plus button next to q to display a window in which you can calculate q, based on the ratio of the shoulder length n to the groove depth t. The groove depth is defined as:

- $t = (d_1 - d_2)/2$ for shaft rings
- $t = (d_2 - d_1)/2$ for bore rings

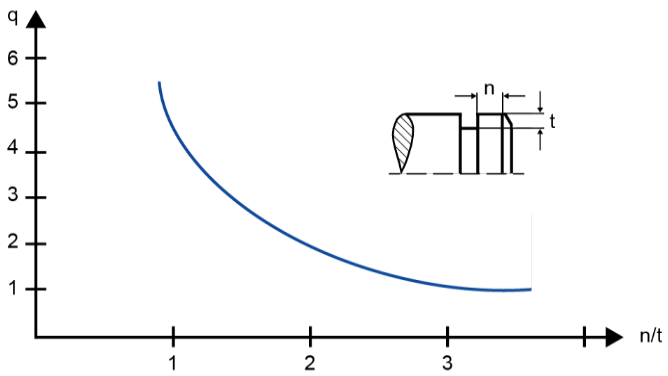


Figure 43.3: (a) Definition of load factor q, shoulder length n and groove depth t.

43.3 Automatic calculation of the dishing angle ψ

The figure below (see Figure 43.4) is used to automatically calculate ψ.

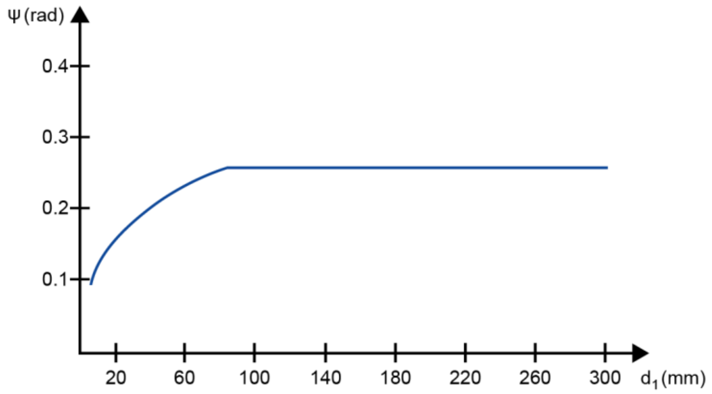


Figure 43.4: Calculation of ψ, based on d1.

43.4 Module specific settings

Click on Module specific settings to enter the minimum required safety value S directly.

44 Hirth coupling

This module performs calculations for Hirth couplings. The calculations are derived and extended from the manual issued by Voith. To start the calculation module, double-click on the "Hirth" module in "Shaft-hub connections" in the module tree.

44.1 Basic data

Enter these values in the "Basic data" tab:

- In Geometry
  Voith profile selection: Specify a Voith profile or select Own Input to enter your own profile.
- The Own input section only appears if you selected Own input for entering the profile
  z: number of teeth
  β: tooth profile
  D: external ring diameter
  d: internal ring diameter
  r: root radius
  s: tip clearance
  n: number of fixing holes in serration surface
  d_L: average bore diameter
- "Materials"
  Work piece 1: Hirth Material 1
  $\eta_{z,1}$:load-bearing percentage, Hirth 1. Voith recommends 0.65 if milled and 0.75 if ground.
  Work piece 2: Hirth Material 2
  $\eta_{z,2}$:load-bearing percentage, Hirth 2. Voith recommends 0.65 if milled and 0.75 if ground.
- "Operating data"
  T: torque
  v: pretension force safety factor. Factor for adjusting the required pretension force. Voith recommends 1.8 to 3.0.

44.2 Module specific settings

The required safety factor SF_{min} is defined here.

45 Answers to Frequently Asked Questions

45.1 Adding new bolt types to the database

The KISSsoft database includes the following bolt types:

- Hexagon socket head cap screw DIN EN ISO 4762
- Hexagon socket head cap screw with low head DIN 7984
- Hexagon headed screw with shank (A B) DIN EN ISO 4014
- Hexagon headed screw without shank (A B) DIN EN ISO 4017
- Slotted cheese head screw DIN EN ISO 1207
- Hexagon headed screw with shank, metric fine thread (A B) DIN EN ISO 8765
- Hexagon headed screw without shank, metric fine thread (A B) DIN EN ISO 8676
- Hexagon headed screws with flange, light series, shape F DIN EN 1662
- Hexagon headed screws with flange, light series, shape U DIN EN 1662
- Hexagon headed screws with flange, heavy series, shape F DIN EN 1665
- Hexagon headed screws with flange, heavy series, shape U DIN EN 1665
- Square bolts ASME B18.2.1
- Hex bolts ASME B18.2.1
- Heavy hex bolts ASME B18.2.1
- Hex cap screws ASME B18.2.1
- Heavy hex screws ASME B18.2.1
- Own definition of bolt geometry

For each of these bolt types, a number of tables list the various bolts sizes (= bolts series). You will find the name of the file that contains the bolts series data in the database (see chapter 9, Database Tool and External Tables).

You enter a new size within an existing bolt type i.e. extend the bolt series (see chapter 45.1.1, Extending an existing bolt series) or input a new bolt type (see chapter 45.1.2, Creating a new bolt type).

45.1.1 Extending an existing bolt series

Example: Enter the data for an M8 bolt with a length of 100 mm in the "hexagon socket head cap bolts EN ISO 4762" series.

Then start the database tool. Display the Screw Type M000 M040Typ table. Select the Hexagon cap screw EN ISO 4762 dataset in that table. In the File name field, you will see the name of the file which includes the bolt series data table. Click the Edit button at the end of the input line to open the file in the editor:

To enter a new bolt:

- Look for a similar bolt (M8, length 80mm).
- Copy this line. Keep the lines in the same sequence.
- Change the data according to Table 1 in EN ISO 4762 (length 100 instead of 80, length l1 72 instead of 52).
- Save the file in the "KISSsoft 20xx/ext/dat" installation directory.
- Document any changes for other users.

45.1.2 Creating a new bolt type

You must already be familiar with the table structure before you can add a new bolt type. You must know which value goes in which column (use the variable names from the descriptions in the table header).

Then, proceed as follows:

- In the database, open the dataset that is most similar to the new bolt type.
- Copy this dataset by clicking on the (+) button under the name of the new bolt type.
- Then, click the **Edit** button to open the new dataset.
- Click the Edit button at the end of the file name input line. This opens a file which still contains the "old" values.
- Overwrite these values with the new values. Note the variables structure (i.e. a specific variable is assigned to a number, depending on where the number appears) and the sequence of the lines.
- Save the updated file in the "KISSsoft 20xx/ext/dat" directory, with a new name, and close the Editor.
- Add the new file name to the new dataset in the database.
- Then, save the new dataset.

VI

Springs

Chapter 46 - 50

46 Compression Springs

Compression springs include cylindrical compression springs and conical compression springs. In both cases, wire is wound or coiled to create a spring. The calculation of compression springs is based on DIN EN 13906-1 (2013) [94]. Conical compression springs are calculated as described in the "Metallfedern" literature [95]. Calculations for individual forces, travel and number of coils are described there. Relaxation is calculated according to DIN EN 13906-1, as are the values for tolerances and materials. The conical compression springs that can be calculated with KISSsoft have a constant wire diameter.

Operating data

When you specify a load, you can use your own value as the spring force or travel. You can also specify whether the spring is to be subject to static, quasi-static or dynamic force.

Geometry

Special features for conical compression springs:

You can select the geometry data according to DIN 2098 Part 1 directly from this table.
If you select Own Input, you can either take selected values from the list or enter your own values. You can also input the spring length and the spring diameter directly in the main screen.
Instead of the spring length in its non-stressed state, L_0, you can also specify a spring length in its stressed state L_1 or L_2.
The selection you make for End of spring and Manufacturing affects the calculation of block length L_c.
Click the Update button to calculate the block lengths and the resulting values for the current situation, for individual springs, and display them in a table.

Special points to note for conical compression springs:

Both the smaller and larger cone diameter can be defined. The spring lengths can be entered in either their stressed or non-stressed state.

If the spring ends have been ground even, the block length is calculated as specified in the literature. Otherwise, it is assumed that the block length is $L_c=n_t \cdot d_{max}$.

46.1 Strength values

The material strengths are stored in different files, depending on diameter. The transverse strength is either saved in the tables, as in DIN EN 13906-1 for thermo-formed springs, or calculated from the predefined tensile strength as $t_{czul} = 0.56 \cdot R_m$.

To calculate the endurance limit, use either the Goodman diagram as defined in DIN EN 13906-1 or an approximation. The approximation assumes a dynamic strength of 0.25·R_m and a pitch of 0.75 for the graph of the upper stress in the Goodman diagram. For shot peened materials, the dynamic strength is increased by 20%. These values roughly correspond to the diagrams in the DIN EN 13906-1 standard, but you should regard the safeties more conservatively.

46.2 Shear stress values

The calculation of the highest shear stress also calculates the axial and shear spring travel.

In the case of conical compression springs, this calculation does not include shear spring travel. Only the proportion of the axial spring travel with the particular diameter is determined.

$$\tau_{\max} = \frac{8}{\pi \cdot d^3} \cdot \left[F\left(D + s_Q\right) + F_Q \cdot (L - d)\right] \quad (47.1)$$

τ_{max}: highest shear stress [N/mm2]
d: wire diameter [mm]
F: spring force [N]
D: coil diameter [mm]
s_Q: shear spring travel [mm]
F_Q: shear spring force [N]
L: spring length [mm]

The highest corrected shear stress is calculated by:

$$\tau k_{\max} = k \cdot \tau_{\max} \quad (47.2)$$

τk_{max}: highest corrected shear stress [N/mm2]
τ_{max}: highest shear stress [N/mm2]
k: stress correction factor

46.3 Bearings coefficient

This coefficient is used for cylindrical compression springs.

The value of the support coefficient v varies according to the support you select, as shown in Figure (see Figure 46.1).

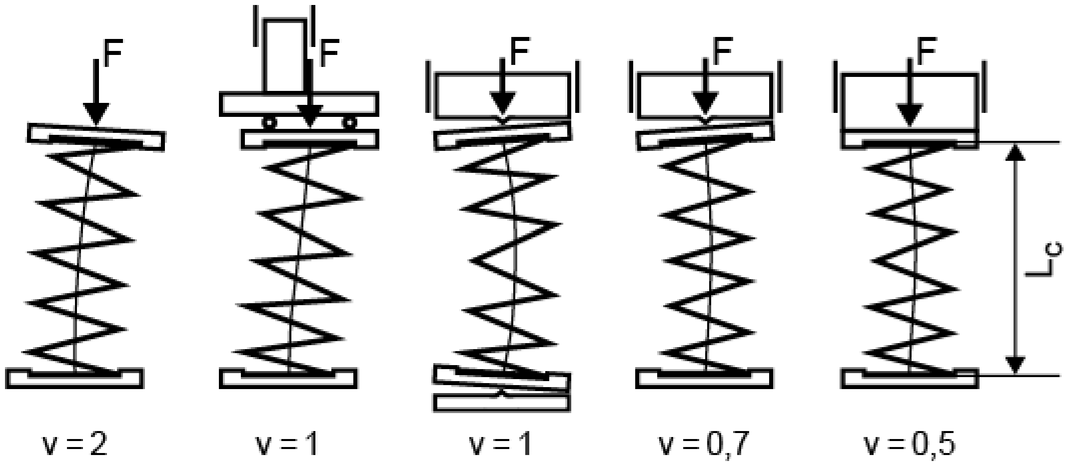


Figure 46.1: Supports with associated support coefficients for axially stressed compression springs

The support coefficient ν is used for calculating the buckling spring travel s_K. If the buckling safety factor is not reached then the spring must be guided, otherwise it will buckle.

46.4 Materials

The selection list includes materials from the DIN EN 10089, DIN 17223-1 (withdrawn), DIN EN 10270-1 (2017), DIN EN 10270-2 (2012), DIN EN 10270-3 (2012, withdrawn) and DIN EN ISO 6931-1 (2017).

Click on the Plus button next to the selection list to display the material values in greater detail in a table. If you selected Own Input, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that these can also be used in subsequent calculations.

46.5 Tolerances

When you select a spring from the table (according to DIN 2098 Part 1, withdrawn), the diameter's tolerance specified in DIN 2076 C is used. To change the diameter tolerance, click on Own Input to display the input fields, which you can then use. Click the Plus button next to the wire diameter field to open another window.

In the Tolerances window, you can select wire diameters according to DIN 2076 (1984, withdrawn), DIN 2077 (1979, withdrawn), DIN EN 10270-1 (2012), DIN EN 10270-2 (2012), DIN DIN EN 10270-3 (2012, withdrawn), DIN EN ISO 6931-1 (2017), DIN EN 10218 (2012) or select Own Input to enter your own value.

If you select a wire diameter tolerance according to the standard, the tolerance will be inserted directly. If you select Own Input, you can define the value yourself.

Other tolerances are listed in accordance with the quality standard. In the Geometry Tolerances and Force Tolerances list in the basic data, you can choose one of the quality standards according to DIN EN 15800 (2009) [96] or DIN 2096 Part 1 (1981) [97]. In the case of conical compression springs, the average diameter is used to define the coil ratio (w=Dm/d).

46.6 Relaxation

The existing spring force can be located after a specific period of time by calculating the relaxation. The compression spring settles to a particular value. Relaxation is also known as creep. The relaxation values are listed in the DIN EN 13906-1 standard, and shown in diagrams. The diagrams show curves at specific diameters and temperatures, which are then recorded in a relaxation-stress diagram. By entering the data from 2 different wire diameters temperatures, you can then infer or extrapolate the relaxation value for a specified level of stress at operating temperature and for a specific wire diameter.

In KISSsoft, the relaxation diagram for 48h can be displayed in relation to diameter, temperature and stress. Other graphics that show the progress of relaxation over time and the spring force are also available. The results for the specified conditions are then displayed in the relaxation report for 48 h. The value of the spring force after 48 h is also calculated.

To extend the data for the materials relaxation curves, or add new data, add this new information to the .dat file for the appropriate spring material.

The relaxation curves in this file can be defined with 2 or 3 predefined measurement points. The curves are then calculated from these points.

46.7 Drawing data

To access the spring data required to create a drawing, click Drawing data. Use the F10SPRING?.RPT file (for compression springs), or F20SPRING?.RPT file (for tension springs), etc. (? = d/e/a/f/i/s/r/p/c for the required language) to modify the template to your own requirements.

46.8 Sizing

If you are working with cylindrical compression springs, and select Own Input in the list under standard, you now see input fields here instead of a table showing the values defined in the standard.

In addition to calculating the wire diameter and the effective coils for cylindrical and conical compression springs, you can click the Sizing button to perform another sizing calculation.

Using the predefined spring rate $R = \Delta F/\Delta s$, the number of turns n can also be calculated if the wire diameter has been predefined. The number of turns is defined by this value, but the strength and the geometric boundary conditions are not checked. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the material's strength.

47 Tension springs

The tension spring calculation is described in the DIN EN 13906-2 (2013) [98] standard.

Operating data

When you specify a load, you can use your own value as the spring force or travel.
The Initial tension F0 is the force required to open the coils which lie one on top of the other. This force is only present if the spring is pretensioned.
If Inner preload is not selected, you can influence the number of effective coils.
You can also specify whether the spring is to be subject to static, quasi-static or dynamic stress.

Geometry

You can also input the spring length and the spring diameter directly in the main screen.
Instead of the spring length in its non-stressed state, L_0, you can also specify a spring length in its stressed state L_1 or L_2.
For the wire diameter, you can either select the diameter values as defined in DIN 2098 Supplement 1 from the list or enter your own value directly in the list.

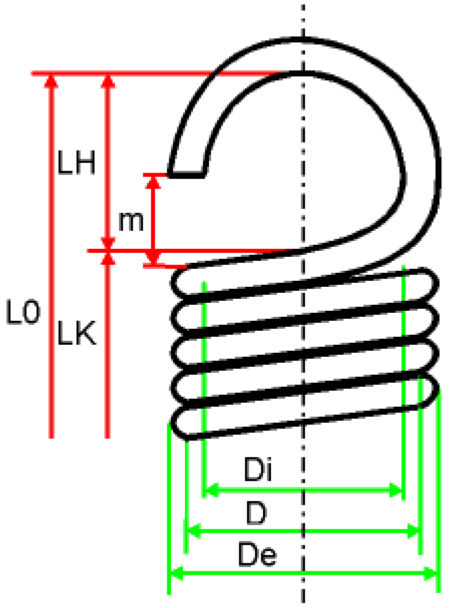


Figure 47.1: Definitions used for tension springs

47.1 Strength values

The Admissible shear stress is calculated from the tensile strength of cold formed tension springs. The tensile strength values depend on the diameter and are stored in files. The shear stress is calculated using the formula $\tau_{zul} = 0.45 \cdot R_m$. Thermo-formed tension springs should not exceed the permissible shear stress of τ_{zul} = 600N/mm^2. These values apply to static or quasistatic cases. Tension springs as defined in DIN 2097 should not be subjected to dynamic stress if at all possible. Shear stress is distributed very unevenly over the cross section of the wire or pin of a tension spring. You can use a stress coefficient k to approximate the highest arithmetical stress. Additional stresses are present at the transitions to the eyes when they are placed under load. They can be well above the permissible shear stresses. As they may be well above the permissible shear stress, no generally applicable fatigue strength values can be given.

47.2 Shear stress values

The shear stress τ is calculated for the sizing of springs that are subject to static and quasistatic stress:

$\tau = \dfrac{8 \cdot D \cdot F}{\pi \cdot d^3}$	(48.1)

τ: Shear stress [N/mm^2]
D: medium coil diameter [mm]
F: spring force [N]
d: wire diameter [mm]

Calculating shear stress for springs subjected to dynamic stress:

$\tau_k = k \cdot \tau$	(48.2)

τ_k: corrected shear stress [N/mm^2]
τ: shear stress [N/mm^2]
k: Stress correction factor

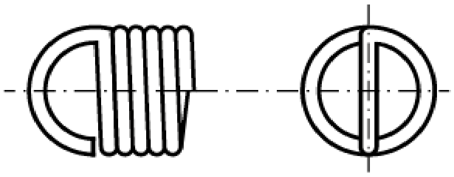
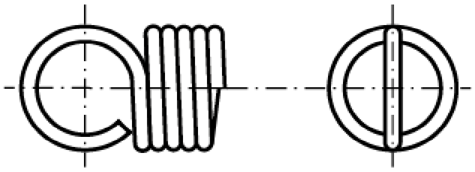
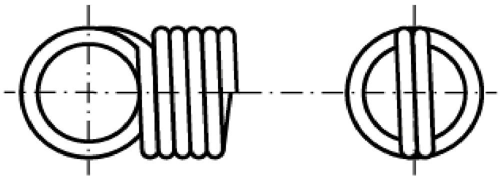
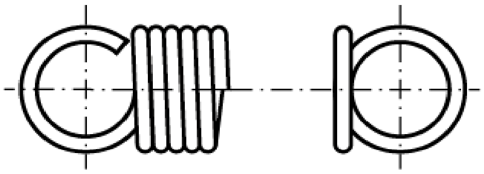
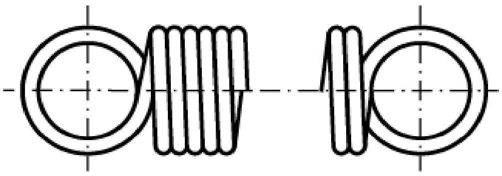
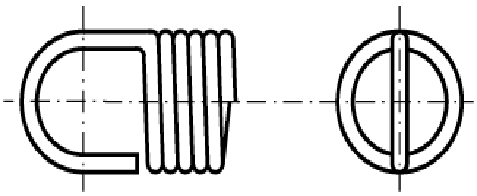
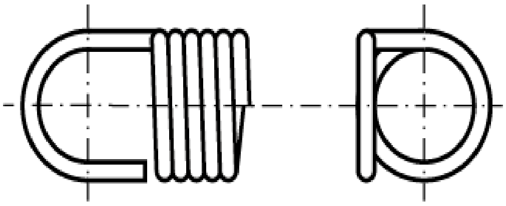
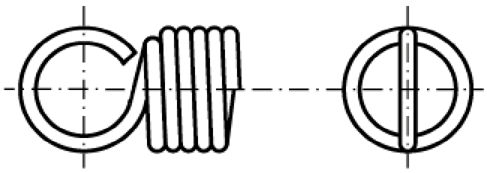
47.3 Manufacturing type

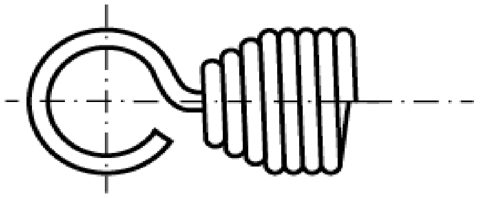
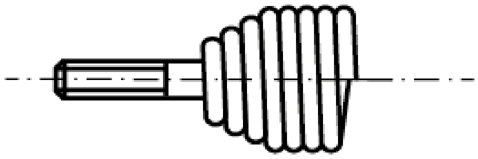
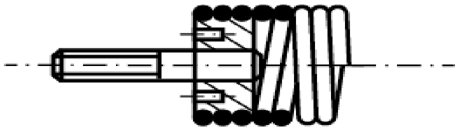
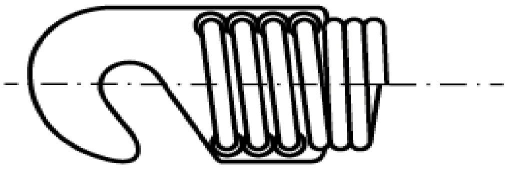
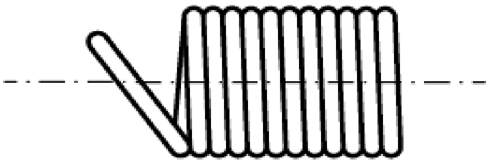
Thermo-formed tension springs cannot be produced with "initial tension" because the heat treatment creates an air gap between the coils. Cold-shaped tension springs can be manufactured in two ways, either by winding on a coiling bench or by winding on a spring winding machine. As defined in DIN EN 13906-2, a formula is specified for each manufacturing method which gives the permissible inner shear stress τ_0.

47.4 Eyes screen

Using the definitions for the Length of eye L_H in this screen, you can determine the total length of the spring. In contrast, the Hook opening m is a reported value that is not used in this calculation.

DIN 2097 defines 13 different eye shapes for tension springs. The program suggests different eye lengths depending on the shape of the eye. The position of the two eye openings is also handled separately in this DIN standard.

 $L_H = 0{,}55\ D_i - 0{,}8\ D_i$ 1/2 German loop	 $L_H = 0{,}8\ D_i - 1{,}1\ D_i$ 1/1 German loop
 $L_H = 0{,}8\ D_i - 1{,}1\ D_i$ 2/1 German loop	 $L_H \approx D_i$ 1/1 German loop at side
 $L_H \approx D_i$ 2/1 German loop at side	 Hook
 Extended side hook	 $L_H \approx 1{,}1\ D_i$ English loop

 Coiled-in hook	 Screwed plug
 $n_t - n = 2 - 4$ Screwed-in screw cap	 $n_t - n = 2 - 4$ Screwed-in shackle
 1/1 German loop inclined	

47.5 Materials

The selection list includes materials from the DIN EN 10089, DIN 17223-1 (withdrawn), DIN EN 10270-1 (2017), DIN EN 10270-2 (2012), DIN EN 10270-3 (2012, withdrawn) and DIN EN ISO 6931-1 (2017).

Click on the Plus button next to the selection list to display the material values in greater detail in a table. If you selected Own Input, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that these can also be used in subsequent calculations.

47.6 Settings

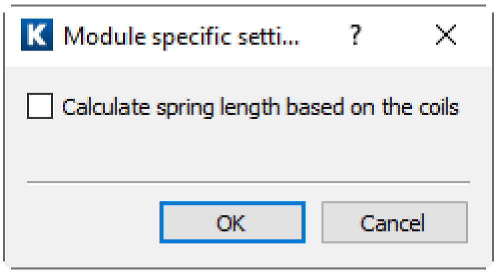


Figure 47.2: Settings: tension springs

If Calculate spring length based on the coils is selected, and the spring is prestressed (Initial tension is selected), the length of the spring is calculated from the number of coils. You can no longer input the lengths in the dialog.

47.7 Tolerances

In the Tolerances window, you can select wire diameters according to DIN 2076 (1984, withdrawn), DIN 2077 (1979, withdrawn), DIN EN 10270-1 (2017), DIN EN 10270-2 (2012), DIN DIN EN 10270-3 (2012, withdrawn), DIN EN ISO 6931-1 (2017), DIN EN 10218 (2012) or select Own Input to enter your own value.
If you select a wire diameter tolerance according to the standard, the tolerance will be inserted directly. If you select Own Input, you can define the value yourself.

Other tolerances are listed in accordance with the quality standard. In the Tolerances list in the basic data, you can choose one of the quality standards according to DIN 2097 [99] or DIN 2096 Part 1 (1981) [97].

47.8 Relaxation

The existing spring force can be located after a specific period of time by calculating the relaxation. The compression spring settles to a particular value. Relaxation is also known as creep. The relaxation values are listed in the DIN EN 13906-1 standard (Compression Springs), and shown in diagrams. It is assumed that the same relaxation values can be used for both tension springs and compression springs. The diagrams show curves at specific diameters and temperatures, which are then recorded in a relaxation-stress diagram. By entering the data from 2 different wire diameters temperatures, you can then infer or extrapolate the relaxation value for a specified level of stress at operating temperature and for a specific wire diameter.

In KISSsoft, the relaxation diagram for 48h can be displayed in relation to diameter, temperature and stress. Other graphics that show the progress of relaxation over time and the spring force are also available. The results for the specified conditions are then displayed in the relaxation report for 48 h. The value of the spring force after 48 h is also calculated.

To extend the data for the materials relaxation curves, or add new data, add this new information to the .dat file for the appropriate spring material.

The relaxation curves in this file can be defined with 2 or 3 predefined measurement points. The curves are then calculated from these points.

47.9 Drawing data

To access the spring data required to create a drawing, click Drawing data. Use the F10SPRING?.RPT file (for compression springs), or the F20SPRING?.RPT file (for tension springs), etc. (? = d/e/a/f/i/s/r/p/c for the required language) to modify the template to your own requirements.

47.10 Sizing

Click the Sizing button to calculate the number of turns n, the Wire diameter and the Effective coils for a predefined wire diameter, using the spring rate $r = \Delta F/\Delta s$. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the material's strength.

48 Leg Springs

The calculation used for leg springs is defined in DIN EN 13906-3 (2014) [100].

Operating data

When you define a load you can either enter a value for the spring force, spring angle, or spring torque. To do this, you must first specify the torsion arm (R_1,R_2) on which the force is applied to the spring.
The value α_0 is used to identify the starting angle. This is used together with the direction of load (sense of winding) to calculate the maximum angle of the spring. Depending on which value you select in the Guiding of spring list, the report will also include a reference value for the diameter of the working mandrel or working sleeve. You can also specify whether the spring is to be subject to static, quasi-static or dynamic force.

Geometry

You can select the geometry data according to DIN 2098 Part 1 directly from this table. If you select Own Input, you can either take selected values from the list or enter your own values. You can select and enter the spring diameter directly. The winding clearance is the distance between the coils.

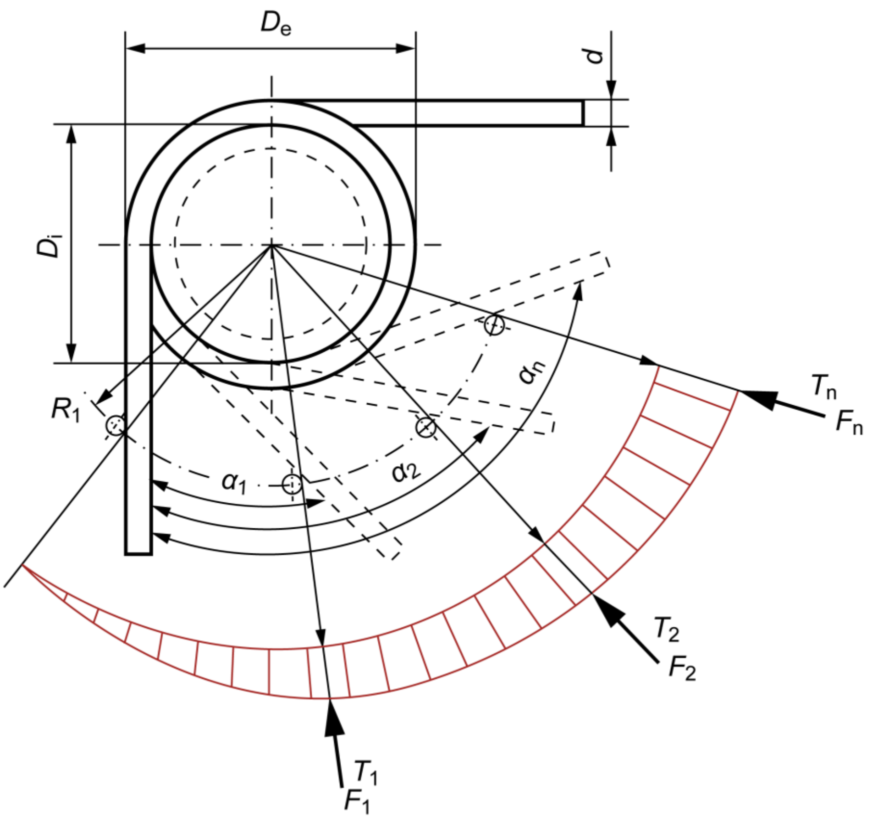


Figure 48.1: Definitions used for leg springs

48.1 Strength values

The admissible bending stress for cold formed leg springs is calculated from the tensile strength. The tensile strength values depend on the diameter and are stored in files. The bending stress is calculated using the formula $\sigma_{zul} = 0.7 \cdot R_m$. These values apply to static or quasistatic cases. The bending of the wire or pin axle due to the load causes the bending stresses to be distributed asymmetrically. To approximate the arithmetical stress (dynamic case), you can use the stress coefficient *q* in the calculation.

48.2 Bending stress values

The bending stress σ is calculated for the sizing of springs that are subject to static and quasistatic stress:

$\sigma = \dfrac{32 \cdot T}{\pi \cdot d^3}$	(49.1)

σ: shear stress [N/mm^2]
T: spring torque [Nm]
d: wire diameter [mm]

Calculating the bending shear stress for springs subject to dynamic stress:

$\sigma_q = q \cdot \sigma$	(49.2)

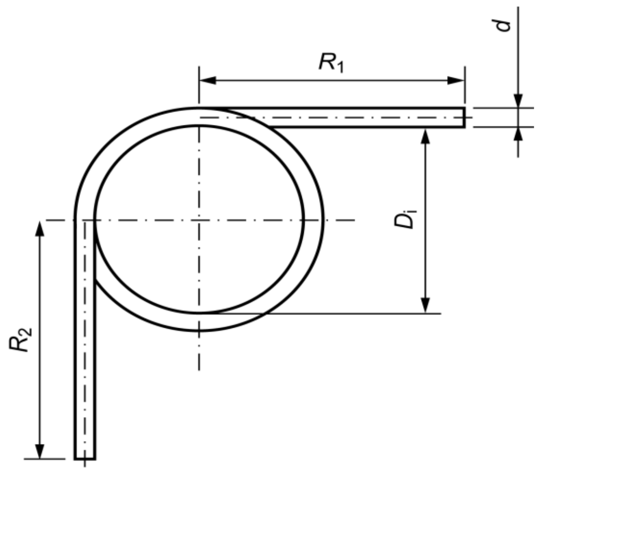
σ_q: corrected bending shear stress [N/mm^2]
σ: bending shear stress [N/mm^2]
q: stress correction factor
(dependent on the ratio D/d)

48.3 Spring design

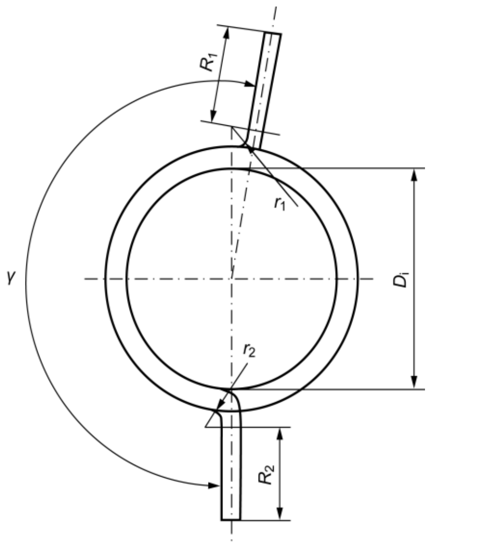
To prevent friction, the coils should either not touch each other or be under only slight stress. The following applies, for the greatest achievable winding clearance:

$$a_{\max} = (0.24 \cdot w - 0.64) \cdot d^{0.83}$$

Generally, leg springs are wound. There are two options for the leg design: they can be either bent with offset (the radius must be specified) or tangential.



with tangential legs



with offset legs

48.4 Assumptions made for the calculation

The calculations apply only to leg springs with fixed or circular guided spring ends. If the leg is not fixed, the spring must be guided by means of a pin or sleeve.

48.5 Materials

The selection list includes materials from the DIN EN 10089, DIN 17223-1 (withdrawn), DIN EN 10270-1 (2017), DIN EN 10270-2 (2012), DIN EN 10270-3 (2012, withdrawn) and DIN EN ISO 6931-1 (2017).

Click on the Plus button next to the selection list to display the material values in greater detail in a table. If you selected Own Input, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that these can also be used in subsequent calculations.

48.6 Tolerances

In the Tolerances window, you can select wire diameters according to DIN 2076 (1984, withdrawn), DIN 2077 (1979, withdrawn), DIN EN 10270-1 (2017), DIN EN 10270-2 (2012), DIN DIN EN 10270-3 (2012, withdrawn), DIN EN ISO 6931-1 (2017), DIN EN 10218 (2012) or select Own Input to enter your own value.
If you select a wire diameter tolerance according to the standard, the tolerance will be inserted directly. If you select Own Input, you can define the value yourself.

In the Tolerances list in the basic data, you can choose one of the quality standards according to DIN 2194 (2002) [101].

48.7 Drawing data

To access the spring data required to create a drawing, click Drawing data. Use the F10SPRING?.RPT file (for compression springs), or F20SPRING?.RPT file (for tension springs), etc. (? = d/e/a/f/i/s/r/p/c for the required language) to modify the template to your own requirements.

48.8 Sizing

Click the Sizing button to calculate the number of turns n, the Wire diameter and the Effective coils for a predefined wire diameter, using the spring momentum rate $RMR = \Delta M/\Delta\alpha$. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the material's strength.

49 Disc Springs

The calculation for disc springs is described in DIN EN 16984 (2017) [102]. The mass and quality requirements are handled according to DIN EN 16983 (2017) [103].

Operating data

When you specify a load, you can use your own value as the spring force or travel. You can also specify whether the spring is to be subject to static, quasi-static or dynamic stress.
The calculations specified in DIN EN 16984 are for disc springs with or without bearing areas for the ratios 16 < De/t < 40 and 1.8 < De/di <2.5 and materials specified in DIN EN 16983.

Geometry

As specified in DIN EN 16983, disc springs are divided into 3 groups and 3 sequences. Groups 1 and 2 contain the springs without a bearing area, whereas group 3 has the springs with a bearing area. The disc thickness for group 1 is less than 1.25 mm, in group 2 it is between 1.25 and 6 mm, and in group 3 it lies between 6 and 14 mm. These groups differ in the hardness of the springs in them. Series A includes hard springs, i.e. they can withstand greater forces in a shorter spring travel. This is followed by series B and series C which can withstand the least force in a larger travel of spring.

You can also input disc springs with inside or outside slits. In addition, you can either set the increased inside diameter or the reduced external diameter. This calculation has been created according to Niemann, Volume 1, 5th Edition (2019) [104]. The ratio (Da-Di')/(Da-Di) is used as an approximation for coefficient K4 for inside-slotted disc springs, and the ratio (Da'-Di)/(Da-Di) is used for outside-slotted disc springs. If the disc spring is designed correctly, the force increases constantly over a specific spring travel.
If you select Own input, the input fields for geometry data become active, and you can enter your own values in them. This type of calculation only applies for springs without a support area, because the ratio of the thicknesses t'/t is not known, but it is still required for the calculation.

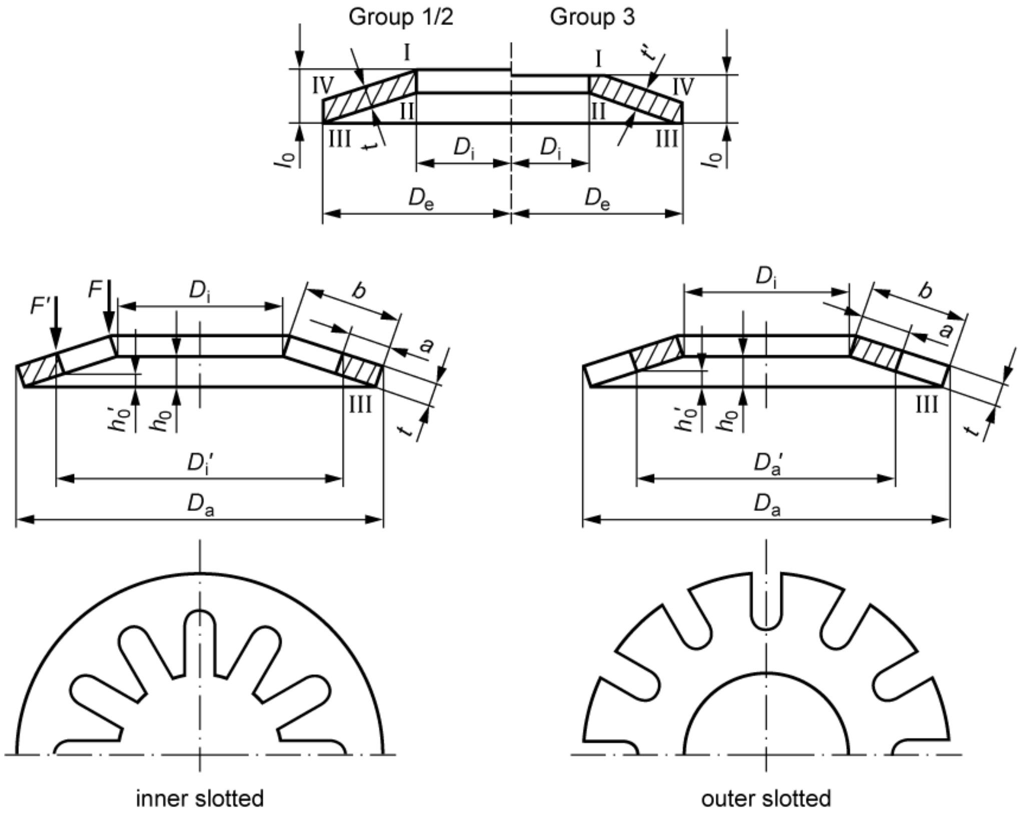


Figure 49.1: Dimensions of the disc springs

49.1 Strength values

In the case of springs that are subject to static or quasi-static load ($N <= 10^4$), the maximum force on the spring is calculated. The formula is predefined in DIN EN 16984. This force then is compared to the spring's effective force F_n (at $s = 0.75 \cdot h_0$) and the spring force utilization is calculated. If the required force is greater than the spring's effective force, F_n, the deviation in the calculation is too large. The DIN formula for calculating force only applies where the spring's travel is $s = 0.8 \cdot h_0$. The springs can be used in packages or columns to handle larger forces. The total calculated system force is then displayed in a force travel diagram. The calculation for disc springs is performed without taking friction into account.
The stresses are also calculated at edge points I through IV. Points I and IV are subjected to compression, and points II and III are put under tension. Under dynamic load, the stress range is calculated using the maximum stress (either at point II or III) with the associated lower stress level. The permissible permanent stress range is defined using a Goodman diagram. These values are then compared. The result is the number of load cycles that the springs should be able to withstand under load. DIN EN 16983 includes Goodman diagrams which are only valid for materials specified in DIN 17221 and 17222. If any other materials are involved, you must contact the spring manufacturer.

49.2 Stress values

Stresses are calculated for edge points I to IV.

Point I:

$\sigma_I = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^2} \cdot K_4 \cdot \frac{s}{t} \cdot \left[K_4 \cdot K_2 \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) + K_3 \right]$	(50.1)

Point II:

$\sigma_{II} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^2} \cdot K_4 \cdot \frac{s}{t} \cdot \left[K_4 \cdot K_2 \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) - K_3 \right]$	(50.2)

Point III:

$\sigma_{III} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^2} \cdot K_4 \cdot \frac{1}{\delta} \cdot \frac{s}{t} \cdot \left[K_4 \cdot (K_2 - 2 \cdot K_3) \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) - K_3 \right]$	(50.3)

Point IV:

$\sigma_{IV} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^2} \cdot K_4 \cdot \frac{1}{\delta} \cdot \frac{s}{t} \cdot \left[K_4 \cdot (K_2 - 2 \cdot K_3) \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) + K_3 \right]$	(50.4)

σ_I - σ_{IV}: stress at points I-IV [N/mm^2]
E: Young's modulus [N/mm^2]
μ: Poisson's ratio [-]
D_e: External diameter [mm]
s: Spring travel of an individual disc [mm]
t: Thickness of an individual disc [mm]
h_0: Travel of spring until flat [mm]
δ: Diameter ratio (D_e/D_i)
K_1 - K_4: Variables calculated from formulae (DIN EN 16984)

49.3 Materials

In the selection list, you can select materials according to the standard.

Click on the Plus button next to the selection list to display the material values in greater detail in a table. If you selected Own Input, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that these can also be used in subsequent calculations.

49.4 Layout number

To estimate the number of discs or columns required, click the Sizing button next to the fields for number of springs per package or number of packages per column. You can define the maximum force and the maximum displacement in this screen. These values are then used to calculate and display the number of springs per package or the number of packages per column.

49.5 Limit dimensions

For all disc springs, the external diameter D_e must lie in the h12 tolerance field and the inner diameter D_i must lie in the H12 tolerance field.

50 Torsion-Bar Springs

The calculation for torsion bar springs is defined in DIN 2091 (1981) [105].

Operating data

When you specify the load, you can enter a value for either an angle of rotation or a torsional moment.
If a default is set for the torsion bar (Torsional bar preplaced is selected), the permitted shear stress on the torsion bar, τ_{zul}, increases.

Geometry

Enter specific parameters to define the geometry of the spring.
For the toothed head form, you must also specify the number of teeth, although this is purely for documentation and is not used in the calculation. The standard assumes shearing modulus G as a constant. However, the calculation is still permitted even if this value is slightly different.

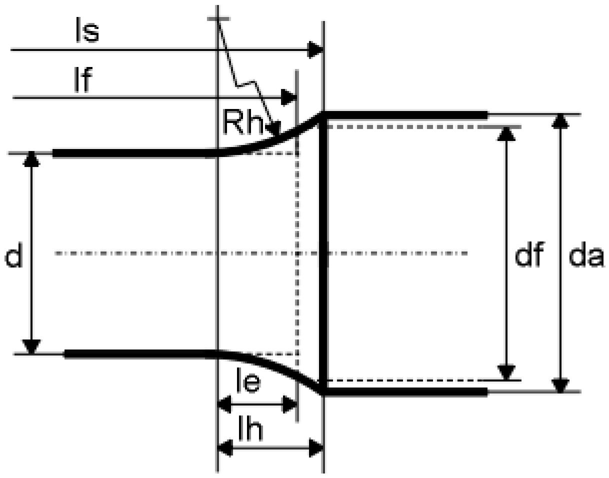


Figure 50.1: Defining a torsion-bar

50.1 Tip forms

Torsion-bar springs as defined in DIN 2091 can have one of three different tip forms: rectangular, hexagonal and toothed. Toothed torsion-bar tips are usually manufactured according to DIN 5481 Part 1 or SAE J 498 b. However, they can also be manufactured with special toothings. The body forms of the tips apply only to bars that are loaded in one shaft rotation direction. Alternating loads require special design measures.

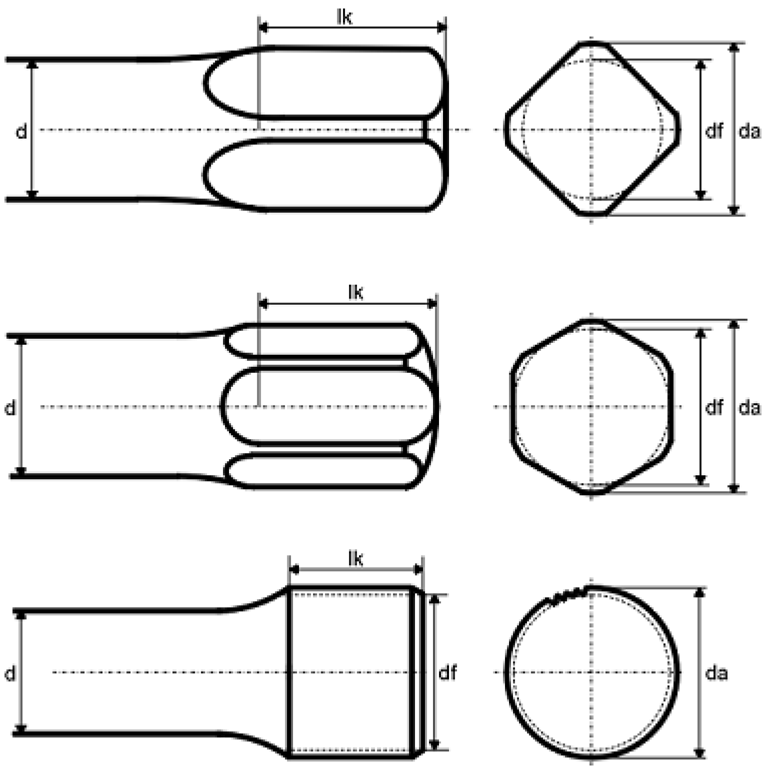


Figure 50.2: Shapes of torsion-bar tips (rectangular, hexagonal, toothed)

50.2 Strength values

Definitions in DIN 2091:

DIN 2091 only applies for the materials defined in DIN 17221 (withdrawn), new DIN EN 10089 (2003).
The permitted shear stresses:
For non-preloaded bars: τ_{zul} = 700 N/mm^2
For preloaded bars: τ_{zul} = 1020 N/mm^2
The heat treatment strength for these values is: R_m = 1600 - 1800 N/mm^2
For the shearing modulus, G = 78500 N/mm^2 is used as the default. Due to preloading (above the yield point, deformed in the direction of operation) after the torsion bar springs have been heat treated, there will be a more favorable distribution of the working stress, and a relief in the boundary zone will be achieved.

50.3 Shear stress

Calculating shear stress τ:

$\tau = \frac{16 \cdot T}{\pi \cdot d^3}$	(51.1)

τ: shear stress [N/mm²]
T: torsional moment [Nm]
d: wire diameter [mm]

50.4 Limiting values

The following limit ratios for torsion bar tips apply to torsion bar springs: rectangular, toothed: d_f/d > 1.3; hexagonal d_f/d > 1.25
The strength values from the DIN standard for bar diameters of 10-60mm apply.
The guide value is a head length between 0.5·and 1.5 *d*.
The ratio R_h/d should lie between 1 and 50.

50.5 Sizing

Enter the torsional moment and angle of rotation and then click the Sizing button to calculate the Wire diameter, the Shank length and the torsional spring rate, from which you can size the bar diameter d or the shank length ls. In this case, the permitted shear stress value is firstly used to calculate d and then the bar diameter is used to calculate the shank length ls. Various values are assumed so that you can size the dimensions. (rectangular, toothed: d_f = 1.35·d; hexagonal: d_f = 1.3·d; d_a = d_f + d_f/7; R_h = (d_a-d)·1.2). These values are not transferred to the main screen.

VII

Belts and chain drives

Chapter 51 - 53

51 V-belts

Preamble:
Follow the manufacturer’s instructions when sizing and verifying V-belt drives. Most catalogs detail the entire calculation method. As the amount of power V-belts can transmit improves because of better materials and flank shapes, manufacturers' data provides the only really reliable values.

Fully automated calculation of V-belts including standard V-belt lengths and standard effective diameters. The module determines the transmittable power per belt while taking into account the speed, effective diameter, transmission ratio and belt length. All the belt data is taken from the tables in manufacturers' catalogs (for example, ContiTech). The belt initial tension is calculated according to the information provided in the catalogs and the belt friction equation. The calculation outputs values for end of rope force in no load/load and the axle load at standstill and in operation. The calculation also outputs the critical speed and tension distance values. The values calculated for the belt-bending test are essential results for belt inspections.

As a variant, the calculation can also be performed with a third pulley (tensioning pulley). You define the X and Y coordinates of the tensioning pulley in the **V-belt** tab. If you open the **Configuration** tab, you can use the mouse to move the tensioning pulley. In this case, the particular x and y value is displayed in the status row. This pulley can be positioned outside or inside as required.

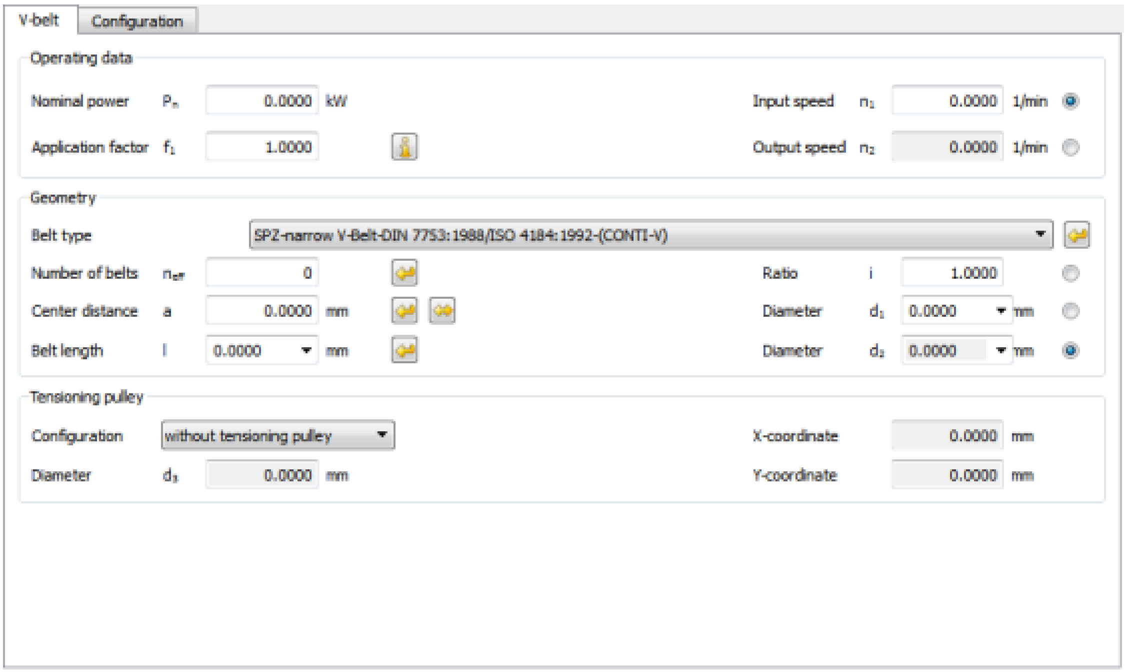


Figure 51.1: Basic data: V-belt calculation

51.1 V-belts data

KISSsoft stores the tabular values (catalog data) in files which you can then edit. You will find these file names in the KISSsoft database tool for the relevant belt types (for example, Z090-015.dat for XPA narrow V-belts).

51.2 V-belt standards

The following belt types are available:

- XPA-High-performance V-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- XPB High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- XPC-High-performance V-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- XPZ-High-performance V-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- 5/- -High-performance v-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- 6/Y-High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- 8/- -High-performance v-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- SPZ-Narrow V-belts-DIN7753/ISO4184-(CONTI-V)
- SPA-Narrow V-belts-DIN7753/ISO4184-(CONTI-V)
- SPB-Narrow V-belts-DIN7753/ISO4184-(CONTI-V)
- SPC-Narrow V-belts-DIN7753/ISO4184-(CONTI-V)
- 8/- -Multiflex-V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 10/Z-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 13/A-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 17/B-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 20/- -Multiflex-V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 22/C-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 25/- -Multiflex-V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 32/D-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 40/E Multiflex V-belts DIN7753/ISO4184 (CONTI-V STANDARD)
- 3V-9J-Force-belts
- 5V-15J-Force-belts
- 8V-25J-Force-belts
- 3V-9N-Narrow-V-Belts-USA-Standard
- 5V-15N-Narrow-V-Belts-USA-Standard
- 8V-25N-Narrow-V-Belts-USA-Standard

51.3 Configuring tensioning pulleys

Here, you can select either:

- Without tensioning pulley
- Tensioning pulley inside
- Tensioning pulley outside

If you selected an inside/outside tensioning pulley, you can input the conical spring washer diameter and the position (x/y) of the tensioning pulley. In the Configuration tab, you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status bar).

51.4 Application factor F1

You can enter this application factor in the basic data screen. If you selected a configuration with a tensioning pulley, you should increase application factor f1 by 0.1. The table shown below is used to define the f1 application factor (refer to the catalogs for more information):

	Drive motor with normal starting torque, combustion engines and turbines with *n* over 600 min^{-1} For daily operation hours:			**Drive motor with high starting torque, combustion engines and turbines with *n* over 600 min^{-1} For daily operation hours:**		
	up to 10 h	from 10 h to 16 h	over 16 h	up to 10 h	from 10 h to 16 h	over 16 h
light drives	1.0	1.1	1.2	1.1	1.2	1.3
medium-heavy drives	1.1	1.2	1.3	1.2	1.3	1.4
heavy drives	1.2	1.3	1.4	1.4	1.5	1.6
very heavy drives	1.3	1.4	1.5	1.5	1.6	1.8

Figure 51.2: V-belt application factor

51.5 Center distance

The minimum center distance is calculated from the two belt sheave diameters. You cannot enter a smaller value here. The sheaves must not touch each other during operation.

The conversion process uses a suggestion for new designs according to the ContiTech catalog. Here:

0.7*(d1+d2) <= a <= 2.0*(d1+d2)

In KISSsoft, the average coefficient for the suggested value is used in the conversion.

a = 1.35*(d1+d2)

51.6 Belt length

You need to know the belt length before you can calculate a V-belt. If you have not specified a length, or if you change to a configuration that involves a tensioning pulley, you must recalculate the belt length.

51.7 Effective number of V-belts

The effective number of V-belts is calculated from the theoretical number by rounding this value up to the next highest whole number.

51.8 Tensioning pulley diameter

The tensioning pulley diameter should be at least as big as the smallest belt sheave. If at all possible, you should not use tensioning pulleys, in particular outside tensioning pulleys. However, if you have to use a tensioning pulley, its diameter should be at least 1.33·d if it is an outside pulley or 1.0·d if it is an inside pulley (d: diameter of the smaller sheave).

Every manufacturer provides slightly different information about tensioning pulleys.

51.9 Position of tensioning pulley (x/y)

When you configure the tensioning pulley, you can enter the position of the pulley (in X/Y-coordinates). Here, the axis of the small sheave is the origin of the coordinates system. If you use the mouse to change the position in the Configuration tab, you can only move the tensioning pulley within the valid area.

51.10 Inspecting V-belts

(belt bending test)

The actual axial stress of V-belt drives is calculated from data provided by the belt bending test. Enthusiastic mechanics have a tendency to over-tighten belt drives, and therefore subject them to loads that are too high for their capabilities.

52 Toothed Belt

Use this method to calculate and size all aspects of toothed belt drives, including the tooth number and belt length, while taking into account standard numbers of teeth. When you enter the required nominal ratio and/or the nominal axis distance, the program calculates the best possible positions. You can also calculate the required belt width, taking into account the correction factors, the minimum number of teeth, and the number of meshing teeth. You can also print out assembly details (belt bending test). The data for each type of belt is saved to text files, whose names indicate their purpose, which can be edited as required.
You can also perform calculations for special stress-resistant toothed belts with integrated steel ropes (e.g. AT5mm).

As a variant, the calculation can also be performed with a third pulley (tensioning pulley). You define the X and Y coordinates of the tensioning pulley in the **Toothed belt** tab. If you open the **Configuration** tab, you can use the mouse to move the tensioning pulley. In this case, the particular x and y value is displayed in the status bar. This pulley can be positioned outside or inside as required.

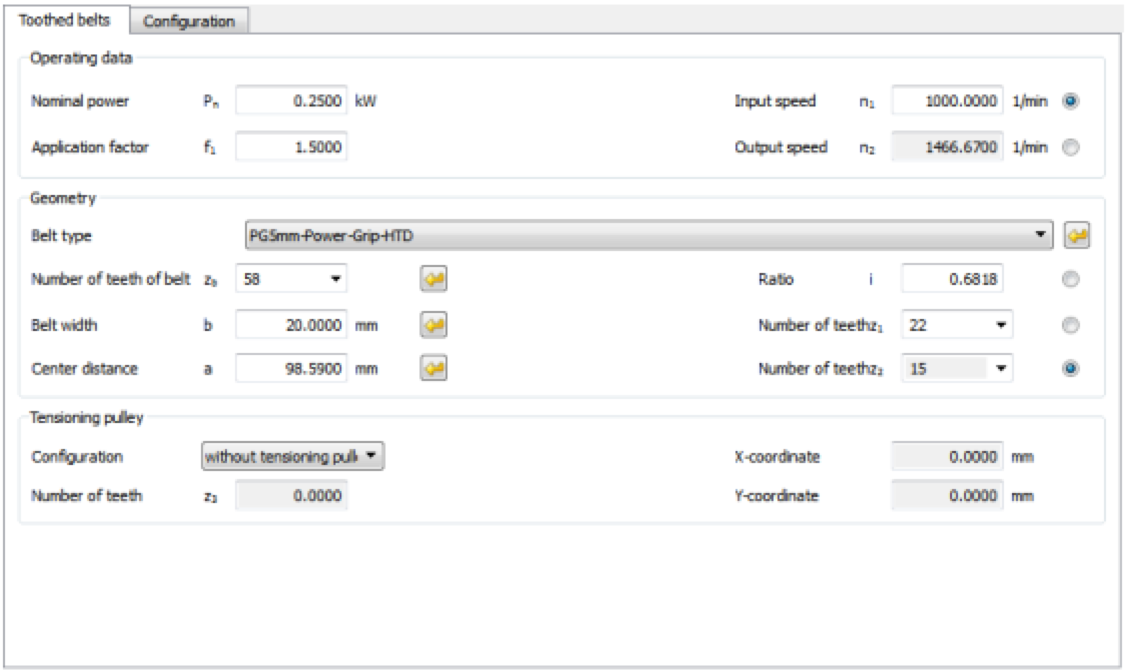


Figure 52.1: Basic data: Toothed belt calculation

52.1 Technical notes (toothed belts)

Preamble:
Follow the manufacturer’s instructions to achieve the best results when sizing and verifying toothed belt drives. Most catalogs detail the entire calculation method. As the amount of power V-belts can

transmit improves because of better materials and flank shapes, manufacturers' data provides the only really reliable values.

Elasticity:
As the manufacturers' catalogs provide very little data on this subject, you must treat the belt elasticity constraint values with caution. The elasticity (in N) is the force required to lengthen the belt by 100%.

Weight:
As the details provided in manufacturers catalogs about this subject are not complete, you must treat these values with caution.

Pretensioning the belt:
As the manufacturers' catalogs provide very little data on this subject, you must treat the constraint values with caution. The calculation method and the factors it uses are stored in the Z091-0??.dat files where they can be changed if required.
You can use one of the following procedures to calculate the required pretensioning values for various types of belts. The data here is taken from the catalogs):

Belt type:	**Pretension:**	
Breco AT5, AT10, AT20	0.5	* Circumferential force
Synchroflex AT3, AT3 GIII, AT5 GIII, AT10 GIII	0.5	* Circumferential force
Isoran XL, L, H, 8, 14	0.625	* Circumferential force
HTD 3, 5, 8, 14	0.25	* max. permitted circumferential force
8MGT, 14MGT Poly Chain GT2	0.5	* Circumferential force
RPP-HPR 8, 14	0.5	* Circumferential force

52.1 table: Pretension

Forces in no load/load are calculated in accordance with [12], equation 27/23.

$$F_V = \frac{F_1 + F_2}{2}$$	(53.1)
$$F_1 = F_V + \frac{F_t}{2}$$	(53.2)
$$F_2 = F_V - \frac{F_t}{2}$$	(53.3)

52.2 Toothed belt standard

You can select one of these standards:

- XL-ISORAN RPP (FENNER)
- L-ISORAN RPP (FENNER)
- H-ISORAN RPP (FENNER)
- 8mm ISORAN RPP (FENNER)
- 14mm ISORAN RPP (FENNER)
- 8mm-ISORAN-RPP-GOLD (Megadyne)
- 14mm-ISORAN-RPP-GOLD (Megadyne)
- 8mm-ISORAN-RPP-SILVER (Megadyne)
- 14mm-ISORAN-RPP-SILVER (Megadyne)
- RP8mm-Pirelli RPP-HPR
- RP14mm-Pirelli RPP-HPR
- PG3mm-Power Grip-HTD
- PG5mm-Power Grip-HTD
- PG8mm-Power Grip-HTD
- PG14mm-Power Grip-HTD
- 8mm-DAYCO-RPP-(Panther)
- 8MGT-Poly-Chain-GT-Carbon (Gates)
- 14MGT-Poly-Chain-GT-Carbon (Gates)
- 8mm MGT-Poly Chain-GT2
- 14mm MGT-Poly Chain-GT2
- AT3mm SYNCHROFLEX
- AT3mm GEN III-SYNCHROFLEX
- AT5mm GEN III-SYNCHROFLEX
- AT10mm GEN III-SYNCHROFLEX
- AT5mm-BRECOflex
- AT10mm-BRECOflex
- AT20mm-BRECOflex

Additional standards are available on request.

52.3 Possible sizings/suggestions

The following sizings are possible if you select the different Sizing buttons:

Variable	Influencing/necessary variables
Belt profile	Power
	Speed (small pulley)
	Operating factor
Number of teeth on belt	Center distance
	Number of teeth on sheave
Center distance	Number of teeth on belt
	Number of teeth on sheave (all)
Ratio	Center distance
	Nominal ratio
	Speed (small pulley)
Number of teeth on tensioning pulley	Number of teeth (small pulley)

52.2 table: Possible Sizings

52.4 Configuring tensioning pulleys

Here, you can select either:

- Without tensioning pulley
- Tensioning pulley inside
- Tensioning pulley outside

If you selected an inside/outside tensioning pulley, you can input the conical spring washer diameter and the position (x/y) of the tensioning pulley. In the Configuration tab, you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status bar).

52.5 Application factor and summand for operational behavior

You can either enter the application factor manually in the load factor interface, or have the program define it from the operating parameters. If you selected a configuration with a tensioning pulley, you must increase the operating factor by 0.1. Use the data in this table to define the factor (refer to the catalogs for more information):

Driven machine	Operating hours per day		
	0-10	**10-16**	**16-24**
Light drive	1.2	1.3	1.4
Medium-light drive	1.4	1.5	1.6
Medium-heavy drive	1.5	1.6	1.7
Heavy drive	1.7	1.8	1.9
Heavyweight drive	1.8	1.9	2.0

52.3 table: Application factors

Summand for operational behavior
(This summand is added to the operating factor in the calculation)

Operational behavior:	Summand
continuous, 0-10 hr/day	0
continuous, 10-16 hr/day	+0.1
continuous, 16-24 hr/day	+0.2
intermittent or with alternating load	-0.1

52.4 table: Summand

52.6 Center distance

The minimum center distance is calculated from the two belt sheave diameters. You cannot enter a smaller value here. The sheaves must not touch each other during operation.

52.7 Belt length and number of teeth on belt

In toothed belt drives, the number of teeth on the belt is used to define the belt length. You need this value when you perform the calculation for the belt. If you did not specify the number of teeth on the belt or, if you switched to configuration with a tensioning pulley, you must ensure that the program recalculates the value for the number of teeth on the belt.

52.8 Effective belt width

The theoretical belt width (minimum width required to transmit the torque) can be calculated from the data in the manufacturer catalogs. The effective belt width is then taken as the next largest standard belt width.

As a general rule, the belt width should not be larger than 5*pitch. A warning message is displayed if you select a belt that is either too wide or too narrow. Although the calculation continues, you use the data it provides at your own risk.

Defining the effective belt width/factor for the belt width:

To define the belt width, you will need the belt width factor (f_b). Use this formula to calculate this factor:

$$f\_b = \frac{Betriebsleistung\ /\ operating\ power}{Nennleistung\ nach\ Katalog\ /\ Nominal\ power\ according\ to\ catalog\ e} \quad (53.4)$$

The nominal power as specified in the catalog is a tabular value taken from the manufacturers' catalogs and is dependent on the speed and number of teeth on the smaller belt sheave.

With the calculated coefficient *f_b* you can then define the effective belt width from a (catalog) table. However, if *f_b* does not match a standard belt width, the next biggest width will be used.

Remarks:
The theoretical belt width in the KISSsoft calculation reports corresponds to an interpolated value, according to calculated factor *f_b*.

KISSsoft stores the tabular values (catalog data) in files which you can then edit. Use the KISSsoft database tool to find the exact file name for a specific belt type (e.g. Z091-001.dat for XL-Isoran).

52.9 Tensioning pulley tooth number

The number of teeth on the tensioning pulley should be at least as large as the diameter of the smallest pulley.
Tensioning pulleys with teeth (toothed belt pulleys) are only used on the inside. Smooth sheaves are used for tensioning pulleys that are used on the outside. The number of teeth can be defined for tensioning pulleys inside and the sheave diameter for tensioning pulleys outside. The diameter of the tensioning pulley should be at least 1.2 *d if positioned outside, or 1.0 *d if positioned inside (d: diameter of the smaller sheave). Every belt manufacturer provides very different data about tensioning pulleys.

For Poly Chain GT:

An outside tensioning pulley reduces service life and should be avoided if possible.

For AT-belts:

AT5mm	with tensioning pulley inside:	25 mm (z > 5)

	with tensioning pulley outside:	50 mm (z > 10)
AT10mm	with tensioning pulley inside:	50 mm (z > 5)
	with tensioning pulley outside:	120 mm (z > 12)
AT20mm	with tensioning pulley inside:	120 mm (z > 6)
	with tensioning pulley outside:	180 mm (z > 9)

52.10 Position of the tensioning pulley x/y

You must enter this value when you configure a tensioning pulley. Here, the axis of the small sheave is the origin of the coordinates system. If you use the mouse to change the position in the Configuration tab, you can only move the tensioning pulley within the valid area.

53 Chain Drives

Use this module to calculate chain drives with roller chains as defined in DIN ISO 606 (with standardized roller chain values taken from a database). The chain geometry (center distance, number of chain elements) for simple and multiple chains and the transmittable power, radial forces, and variation in speed, are calculated by the polygon effect, etc. Basis: DIN ISO 10823, [26] and [8]. During this calculation, the program checks the highest permitted speed and shows a suggested value for the required lubrication.

As a variant, the calculation can also be performed with a third pulley (tensioning pulley). The X and Y coordinates of the tensioning pulley can be defined in the **Chain drives** tab. If you open the **Configuration** tab, you can use the mouse to move the tensioning pulley. In this case, the particular x and y value is displayed in the status row. This pulley can be positioned outside or inside as required.

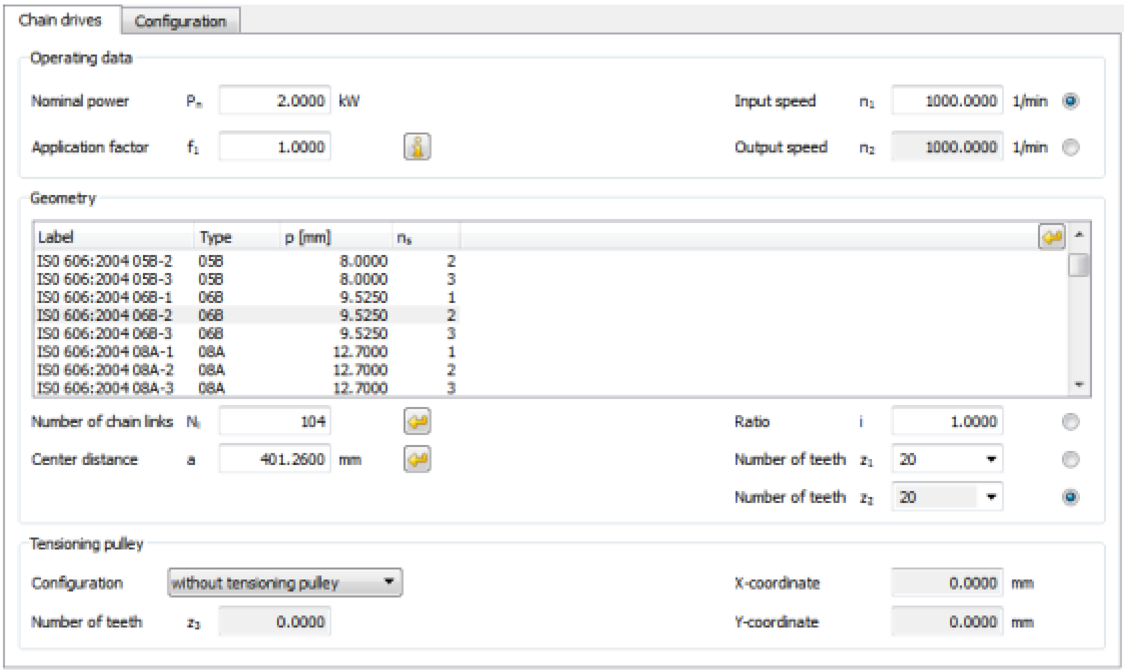


Figure 53.1: Basic data: Chain calculation

53.1 Sizings

- Using the drive data as a starting point, the program displays a list of suggested values for suitable chain drives.
- Calculating the center distance from the chain length.
- Calculating the chain length from the center distance.

53.2 Tensioning pulleys

You require tensioning pulleys if you need to limit the chain deflection or keep to a minimum wrap angle. You must arrange the tensioning pulleys under no load. They must have at least three teeth.

53.3 Standard

Chain profile standard:

- Roller chain DIN ISO 606

The roller chain standard, DIN ISO 606, includes chains as defined in the DIN 8154, 8187 and DIN 8188 standards. Roller chains are the most frequently used type of chain because lubricated rollers reduce noise and wear. The chains defined in DIN 8187 correspond to the European type, those defined in DIN 8188 correspond to the American type. You should only install bush chains as defined in DIN 8154 in a closed housing with sufficient lubrication.

53.4 Chain type

The data shown below depends on the type of chain:

- Chain pitch.
- Maximum permitted speed of the small gear.
- Nominal power at maximum permitted speed.

Tables in DIN ISO 606 pages 8 to 10.

53.5 Number of strands

You can achieve high levels of power by using multiple chains. Chains are often arranged in two or three strands (Duplex, Triplex). The values for duplex and triplex chains are also given in the same standard.

53.6 Application factor

Guide values according to DIN ISO 10823, Table 2:

Working characteristic of the driven machine	Working characteristic of the driving machine		
	uniform	light shocks	moderate shocks
uniform	1.00	1.10	1.30
moderate shocks	1.40	1.50	1.70
heavy shocks	1.80	1.90	2.10

Figure 53.2: Application factor for chain calculation

53.7 Speed/number of teeth/ratio

Range of ratio:

favorable	i = 1. . . 5,
good	i = 1. . . 7,
unfavorable	i > 10.

Number of teeth:
Due to the polygon effect, we recommend a minimum number of teeth of 17 . . . 25. A "Number of teeth" that is fewer than 17 should only be used to produce low levels of power. The preferred numbers of teeth for use in chain gears, as stated in DIN ISO 606, are: 17, 19, 21, 23, 25, 38, 57, 76, 95, 114.

You should use at least three teeth for tensioning pulleys.

53.8 Configuration

You can select one of these configurations:

- without tensioning pulley
- with tensioning pulley inside
- with tensioning pulley outside

In a configuration involving tensioning pulleys, you must specify the number of teeth and the position of the tensioning pulley (x/y). In the Configuration tab, you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status bar).

53.9 Center distance

Recommended center distance:	$a = 30 \cdot p \ldots 50 \cdot p$ (p: pitch)
You should avoid:	$a < 20 \cdot p$ and $a > 80 \cdot p$

Click the Sizing button to calculate the center distance from the number of chain links.

53.10 Polygon effect

When calculating chains, you must take the polygon effect into account both for the reference circle and the center distance.

- Formula for the reference circle:

$d = \dfrac{Teilung}{\sin\left(\dfrac{\pi}{z}\right)}$	(54.1)

- (see also [12], equations 26/46)
- Formula for the center distance:
  The length of the loop on the sprocket differs as follows from the formula used for V-belts/toothed belts:

$l_{UK} = l_{UR} \cdot \dfrac{\sin\left(\dfrac{\pi}{z}\right)}{\dfrac{\pi}{z}}$	(54.2)

- l_{UK}: Length of loop for chains
  l_{UR}: Length of loop for V-belts

53.11 Number of links

The number of links should usually be an even number.

Click the Sizing button to calculate the number of links from the center distance.

53.12 Sprocket geometry

In KISSsoft, you can display and print out the sprocket geometry as a graphic as defined in DIN ISO 606. The graphics are created with the mean allowances.

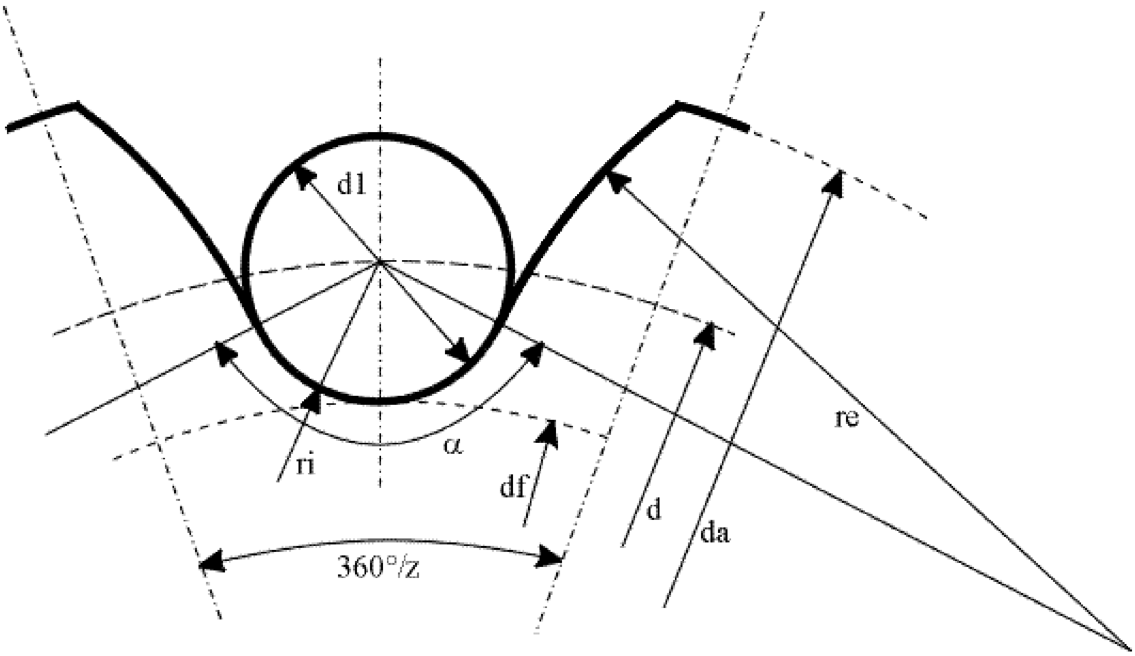


Figure 53.3: Geometry of chain sprocket

You can also output other values for a sprocket in a report. The figures in this section show how specific information is represented in this report.

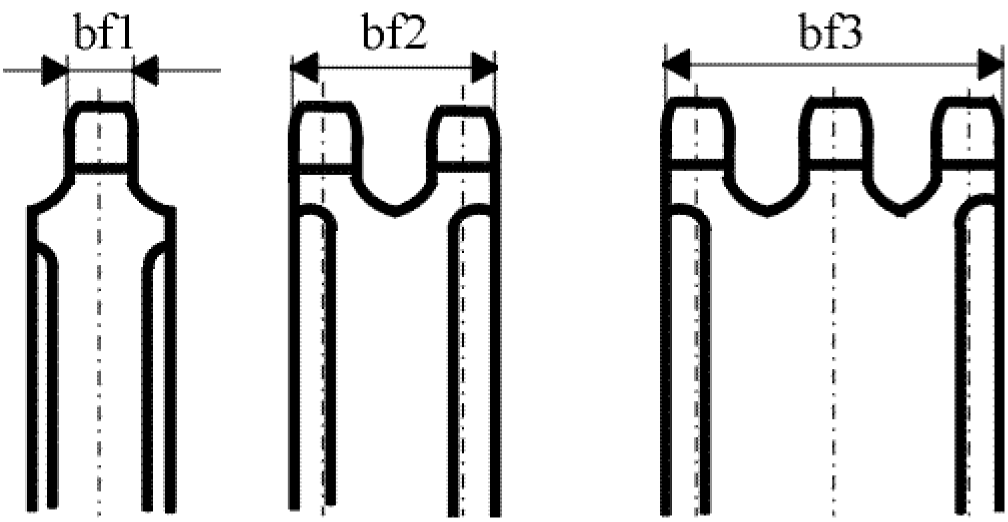


Figure 53.4: Chain sprocket widths

VIII

Automotive

Chapter 54 - 55

54 Synchronization

Use this module to calculate the gear synchronization time and total time, based on the specified geometry, forces and application data. Some additional calculations for heat development, frictional power, and wear resistance, are also performed. Calculations can be performed for common types of synchronizations for a given number of cones (single, double or triple cone).

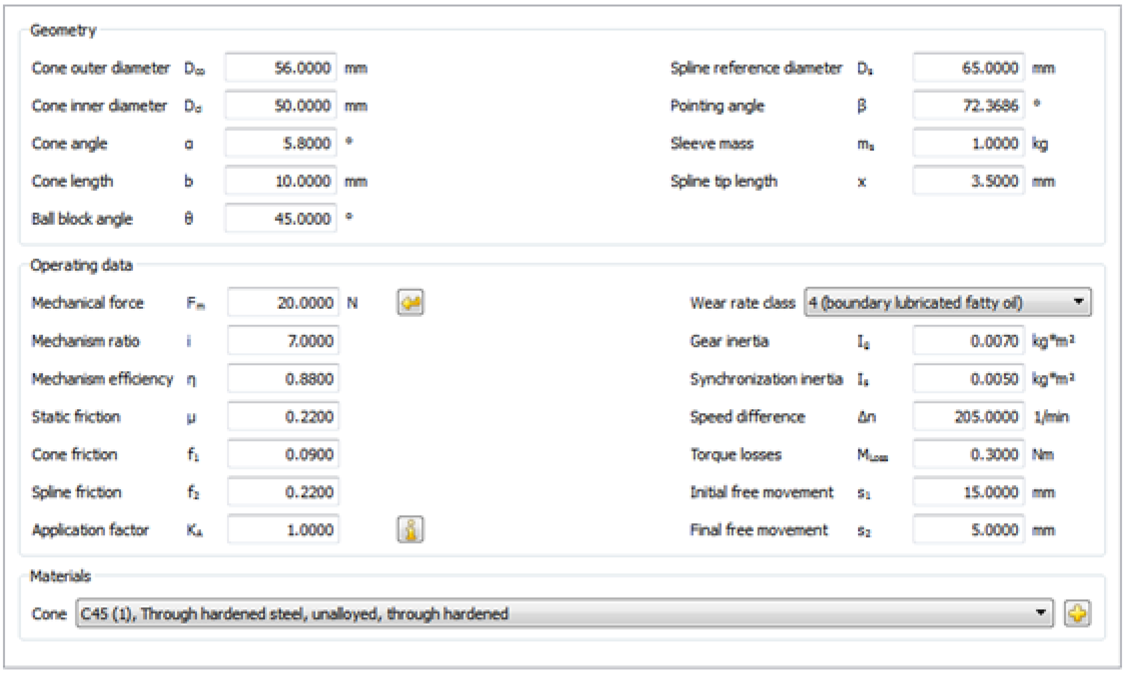


Figure 54.1: The synchronizer module tab

54.1 Geometry

Geometry data is needed for the synchronization ring, also called the cone. Additional data is needed for the spline tip definition (the indexing) and ball block angle. This is the external ball angle which holds the synchronizer in its position (engaged or disengaged). Specific limiting values have been defined for the angle input to ensure synchronization can be guaranteed.

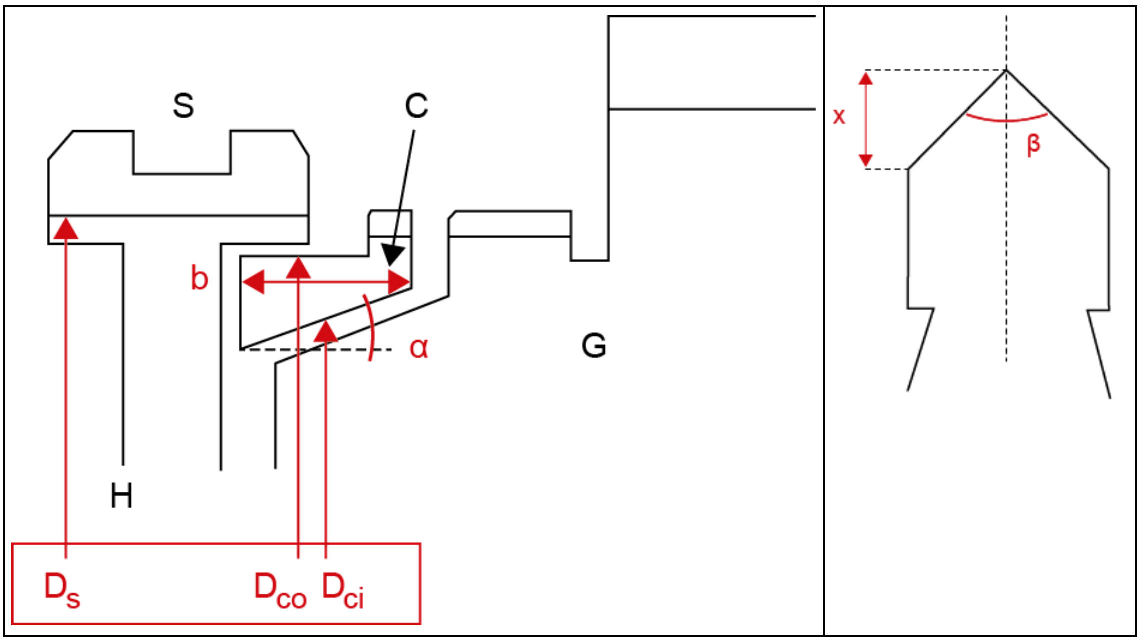


Table 54.1: Figure: (a) Description of main geometry: S = Sleeve, C = Ring/Cone, H = Hub, G = Gear, (b) Spline tip geometry

54.2 Operation data

The mechanical force is the force applied to the shifting handle. This force is multiplied by the mechanical ratio and applied to the sleeve. The coefficient of friction at the beginning of the synchronization can be defined, for the cone and the sleeve.

The gear inertia and the speed difference are required entries. Torque losses due to mechanical friction, oil splashing, and other sources, can be defined. The defined losses during shifting will either help or hinder the process depending on the shifting direction. If there is a small amount of clearance on the synchronizer sleeve before and after the actual synchronization, the distances can be entered here to enable the total time (from another gear to the final end position of the sleeve) to be calculated correctly.

55 Friction couplings

This module is used to calculate friction clutches and brakes in accordance with VDI 2241 [106]. The results of this calculation can then be used to select a suitable clutch or brake. The couplings are operated either mechanically, electromagnetically, or by pressure (e.g. hydraulically), thereby either generating or removing pressing force. The couplings can be designed to run either dry or with lubrication. This has a significant effect on the coefficient of sliding friction and the coefficient of static friction.

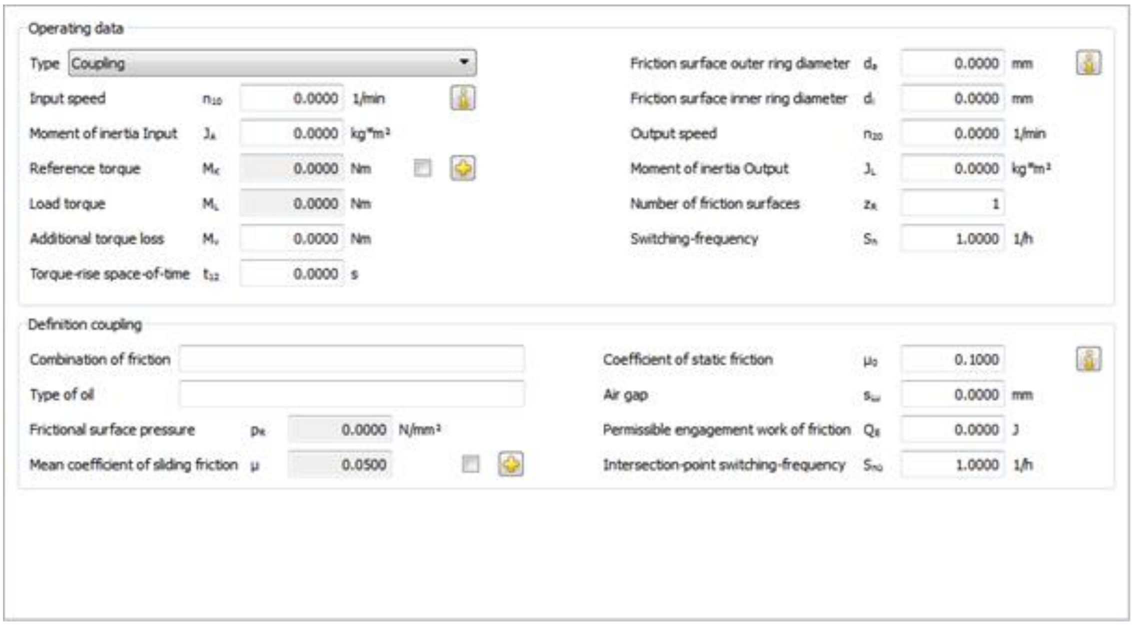


Figure 55.1: Basic data: Friction couplings

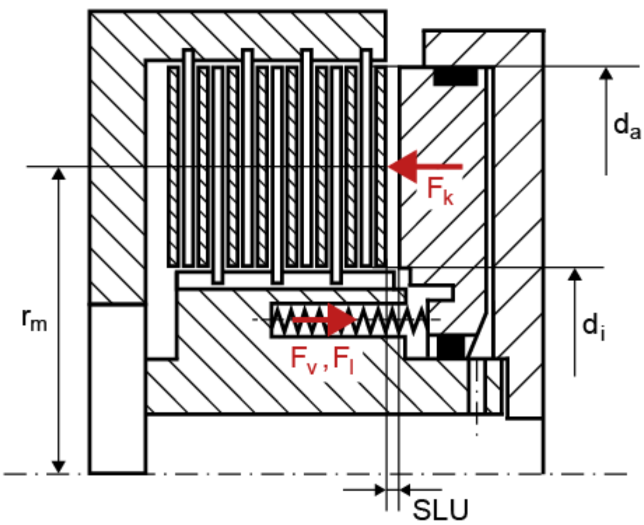


Figure 55.2: View of a closing system

Force is stored in a spring. When the spring is released, the force returns the coupling to its open state, or vice versa. Compression or disc springs are usually used here. Both types of spring are pretensioned in their open state. In this example of a closing system, the compression is created hydraulically, and therefore affects the piston. This additional definition of force storage is not included in the VDI guideline. The guideline assumes that frictional surface pressure is applied directly to the plate. As the dynamic characteristics of the springs can also be non-linear, the force generated by the contact with the first plate is used in the calculation.

In KISSsoft, you can either define the spring forces, or input the reference torque M_K and the load torque M_L directly. As specified in VDI 2241, the engagement work of friction and the switching capacity are defined using an average sliding velocity and an average coefficient of sliding friction. You can also specify the coefficient of sliding friction as a dependency of 5 sliding velocities, because this coefficient can vary greatly depending on which sliding velocity is present. However, this does not take into account the aging of the oil, which would reduce the coefficient of sliding friction.

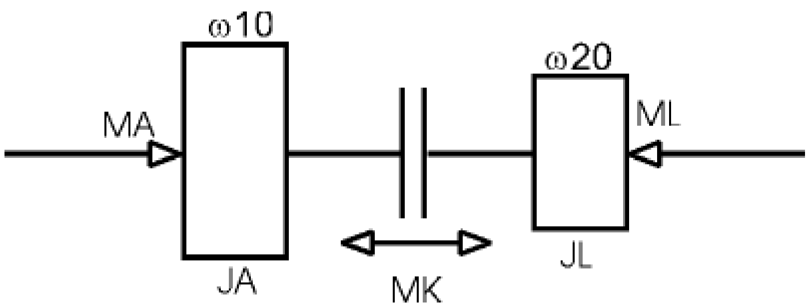


Figure 55.3: Schematic display of a coupling

The dynamic moment of inertia J_L can also be made up of a number of different parameters. If a mass m is present at the distance r from the rotational axis, its moment of inertia can be calculated with the formula $J_{L2} = m*r^2$.

This can then be added to the existing moment of inertia J_L.

$J_L = J_{L1} + J_{L2}$.

Ratios can then reduce the moment of inertia on the clutch shaft $J_{2red} = J_2*(n_2/n_1)2$. This reduced moment of inertia can then be added to the clutch shaft's moment of inertia.

$J_L = J_{L1} + J_{2red}$

55.1 Calculation

Inputting the spring forces/defining the reference torque

If you decide to input the spring forces (Reference torque check is selected), the reference torque is calculated as follows:

$$F_{stat} = Fl$$

$$F_{dyn} = Fk - Fv$$

F_k (N): Piston force ($F_K = p_K/A_K$)

p_K (N/mm^2): Compression on the piston

A_K (mm^2): Piston surface area

F_l (N): Spring force to plates contact

F_v (N): Pretension for spring force

F (N): Resulting force on the first plate (N)

The accelerating torque or the holding torque is then determined from this. Using the different coefficients of sliding friction, if these have been defined, otherwise using the average coefficient of sliding friction:

$$M_A = F_{dyn} \bullet \mu \bullet r_m \bullet z_R$$

$$M_L = F_{stat} \bullet \mu 0 \bullet r_m \bullet z_R$$

M_A (Nm): Accelerating torque

F_{stat}, F_{dyn} (N): Resulting force on the first plate

μ (-): Coefficient of sliding friction

r_m (mm): average friction radius

z_R (-): Number of friction surfaces (plates)

M_L (Nm): Load torque

μ_0 (-): Coefficient of static friction

The reference torque is then calculated from M_A+M_L. You can also define a torque loss, which has a negative sign for a coupling and a positive sign for a brake.

However, if you define the reference torque directly, you cannot also define a torque loss. This must then be taken into account with the reference torque.

The formulae specified in VDI 2241 [106] are used to define the sliding time t_3.

For a coupling:

$M_K=M_A$, where M_K is specified, with the influence of t (in this sequence):

$$t_3 = J_L \bullet (\omega_{10} - \omega_{20})/(M_K - M_L)$$

$$t_3 = (J_A \bullet J_L)/(J_A \bullet (M_K - M_L) + J_L \bullet (M_K - M_A)) \bullet (\omega_{10} - \omega_{20})$$

$$t_3 = J_L \bullet (\omega_{10} - \omega_{20})/(M_K - M_L) + t_{12}/2 \bullet (M_L/M_K + 1)$$

For a brake:

where M_K is specified, with the influence of t_{12} (in this sequence):

$$t_3 = J_L \bullet (\omega_{10} - \omega_{20})/(M_K - M_L)$$

$$t_3 = J_L \bullet (\omega_{10} - \omega_{20})/(M_K - M_L) + t_{12} / 2 \bullet (M_L/M_K + 1)$$

The engagement work of friction Q is then calculated with, or without, taking the "torque-rise space-of-time" t_{12} into account, depending on whether this value has been defined. The switching capacity on the total friction surface and the maximum switching capacity are also calculated.

If you input curve points for the coefficient of sliding friction, the area below the calculated curve in the torque diagram is calculated as the engagement work of friction. The switching capacity is then derived from the time-based conclusion of this calculation.

Each of these values must be input as specific values for the friction surfaces because these are provided by the manufacturers in the relevant catalogs.

Furthermore, when you input the switching frequencies and the permitted engagement work of friction (one-time switching) the program calculates a utilization to show whether the selected coupling will be adequate.

$$Q_{zul} = Q_E \cdot (1 - e^{-Sh\ddot{u}/Sh})$$

Q_{zul} (kJ): permissible engagement work of friction

Q_E (kJ): permissible engagement work of friction (one-time switching)

$S_{h\ddot{u}}$ (1/h): Intersection-point switching frequency

S_h (1/h): Switching frequency per time unit

The utilization A_Q is then determined from this permitted value and the calculated engagement work of friction:

$$A_Q = Q_{zul} / Q$$

When you select a coupling, you must take into account the reference torque, and most importantly, the permissible engagement work of friction Q_E (one-time switching) and the calculated permissible engagement work of friction (for a higher switching frequency).

55.2 Definition of spring forces

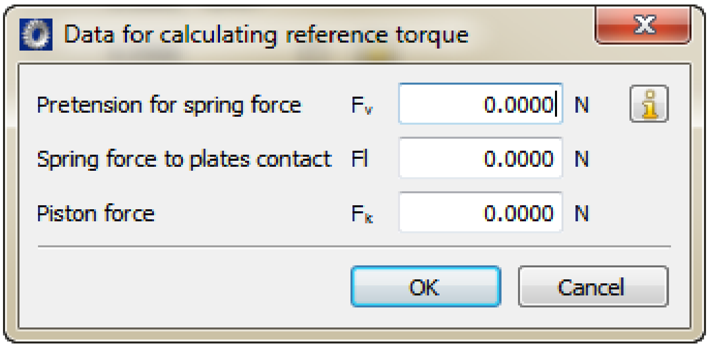


Figure 55.4: Definition of spring forces

These additional inputs, Pretension for spring force Fv and Piston force Fk, are used to determine the characteristic values required to calculate the resulting spring force. The coefficient of sliding friction and the average radius rm and the number of plates are then applied to determine the accelerating torque.

The coefficient of static friction from the Spring force to plates contact Fl is then used to define the holding torque.

The spring force pretension is taken into account as a positive or negative value, depending on what kind of system has been selected (closing or opening).

55.3 Definition of the coefficients of sliding friction and velocities

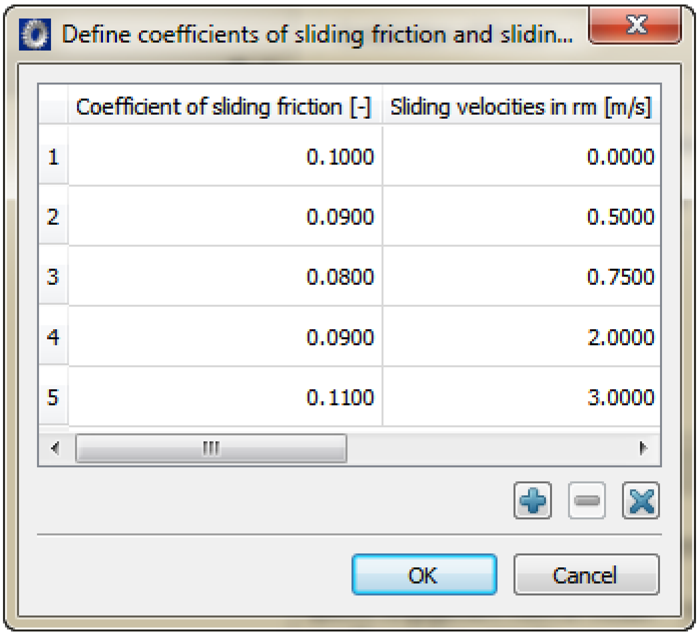


Figure 55.5: Definition of the coefficients of sliding friction and velocities

The coefficients of sliding friction are specified by the manufacturers in accordance with the sliding velocities. The VDI 2241 standard assumes that a constant value is used. However, this may result in a large deviation in results. By inputting a maximum of 5 points you can create a poly line that connects these points. From this line the program can then derive 10 values for the coefficients of friction in the sliding velocity areas at the start and at the end. The 10 different accelerating torques derived from this can then be used later on in the calculation.

55.4 Graphics

The graphics show the speed curve over sliding time t_3, the torque diagram over sliding time t_3 , and the coefficient of sliding friction curve for the sliding velocity, of which a maximum of 5 points have been entered (if defined by the user).

55.5 Settings

If the Use radius to plates gravity center for the calculation check is selected, the radius at the plate center of gravity is used in the calculations instead of the plate mean radius rm. This radius is calculated with:

$$rs = \frac{2}{3} \cdot \frac{ra^3 - ri^3}{ra^2 - ri^3}$$

rs (mm): radius at the center of gravity of the plate

ra (mm): external radius of the plate
ri (mm): internal radius of the plate

IX

Various

Chapter 56 - 61

56 Tolerance Calculation

In this module, you enter the nominal lengths and their corresponding allowances for various elements. These values are then used to calculate an overall tolerance. This calculation uses a constant distribution (arithmetical sum) and the square root of the tolerance squares (normal distribution) to define the maximum and minimum size of the measurement chains. You can also use the appropriate allowances to calculate the nominal length/expected value of the measurement chain. The Tolerance field according ISO is defined according to ISO 286 in which the tolerances are defined up to a size of <= 3,150 mm. In KISSsoft, for fit (tolerance) classes H, h. JS and js, the values used in the standard are extrapolated up to a value of 10,000 mm.

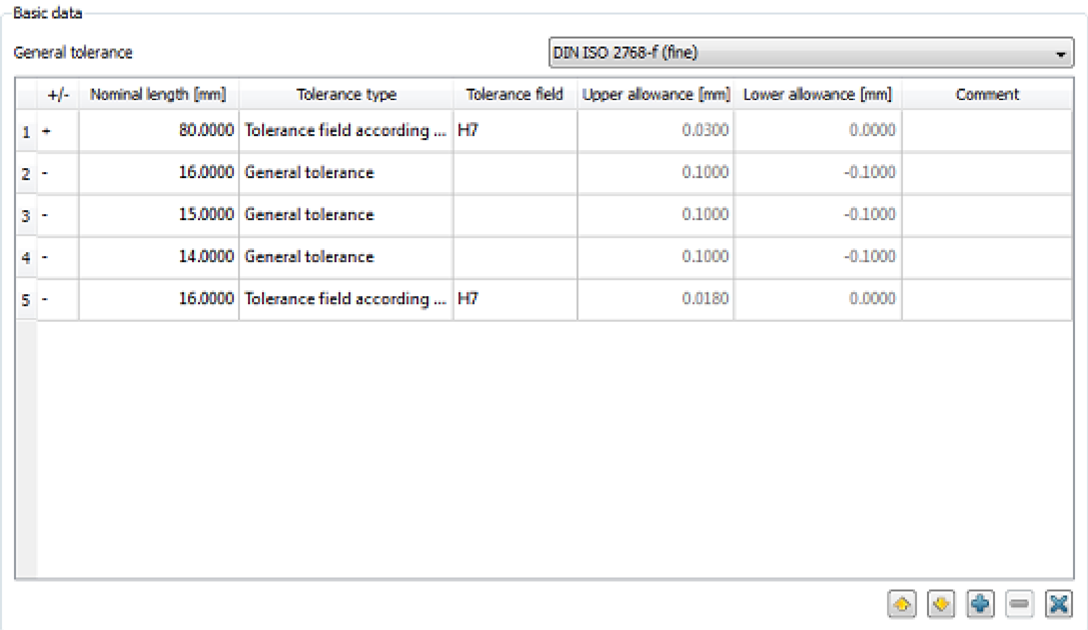


Figure 56.1: Basic data

57 Proof of strength with local stresses

57.1 General

You can start this calculation in the Various section of the modules tree.

57.1.1 Software functionality

The calculation program supplies a complete, written proof of integrity for static and fatigue strength at the proof point W.

The strength verification is supplied according to the local stress concept as described in the FKM "Rechnerischer Festigkeitsnachweis für Maschinenbauteile" Guideline. The idea behind the local stress concept is to estimate the service life on the basis of the elastic-plastic local stress at the critical point on the part compared to the S-N curve (Woehler lines) elongation derived from an unnotched test bar. The local concept is implemented as a stress-based variant within the framework of the FKM Guideline. Therefore, before it can be used, the material must be in a linear elastic state. In this context, the concept used is not really a local concept like the elastic-plastic notch root strain concept, but a concept close to the nominal stress concept, except that only the theoretical stress concentration factor stands on the other side of the equations. It is a useful tool for calculating a static proof and proof of fatigue strength in the high cycle range (N$N > 10^4$).

Input: You can enter the stresses at a proof point W and at a neighboring point B. Alternatively, the stresses at the proof point and the stiffening are estimated mathematically. You must also specify parameters such as surface roughness, part size, etc., before you can calculate the design coefficients. Additional load data, such as number of cycles, spectrum, temperature etc. are also predefined.

Output: The calculation calculates the utilization factors for static cases and fatigue. It creates a complete set of documentation for this.

57.1.2 Areas of application for the FKM Guideline

The software is based on the FKM "Rechnerischer Festigkeitsnachweis für Maschinenbauteile" Guideline, Chapters 3 and 4. The guideline applies to mechanical engineering and its associated industrial sectors. In real life scenarios, contractual partners must agree how this guideline is to be implemented. For parts that are subject to mechanical stress, this guideline can be used to calculate the static and fatigue strength either for a finite or infinite working life. However, this guideline does not cover other mathematical proofs such as brittle fracture stability, stability or deformation under load, or experimental strength verifications. Before the guideline can be applied, it is assumed that the parts have been manufactured so that all aspects of their design, material and operation are technically free of error and fit for purpose. The guideline is applicable for parts made of iron and aluminum alloys that are manufactured either by machining or welding, even at elevated temperatures, and in particular for

- parts with geometric notches
- parts with welded joints
- static stress
- fatigue loads ranging from approximately ($N > 10^4$) cycles as an individual or collective load
- rolled and forged steel, including stainless mix cast iron alloys and also forged and cast aluminum alloys
- different temperatures
- a non-corrosive ambient medium.

Supplementary agreements must be drawn up if this guideline is to be used outside the specified area of application. The guideline does not apply if a strength verification is required using other standards, codes or guidelines, or if specific calculation data, such as VDI 2230 for bolted joints, is applicable.

For simple rod-shaped and planiform parts, we recommend you use a calculation method that involves nominal stresses. The calculation using local stresses is to be used for volumetric parts or, in general, where stress is to be calculated using the finite elements method or the boundary element method, if no specifically defined cross sections or simple cross section forms are present or if the theoretical stress concentration factors or notch effect values are unknown.

57.2 Background

57.2.1 The FKM Guideline: Rechnerischer Festigkeitsnachweis für Maschinenbauteile

The idea for this guideline was proposed at a meeting of the DVM in Berlin, Germany, in May 1990, when experts from the then Federal Republic of Germany met together with experts from the then German Democratic Republic. The objective was to combine the standards from what were then two separate guidelines (VDI in the West and TGL in the East), to create one new strength assessment guideline. The new guideline was to be based, in particular, on the former TGL standards for strength calculation, VDI guideline 2226, DIN 18800, Eurocode 3 and the recommendations of the IIW. The FKM Guideline was also updated to take into account the latest discoveries from research into the fatigue strength of metallic parts. The FKM Guideline is designed for use in mechanical engineering and associated industrial sectors. The first edition of the FKM Guideline, "Rechnerischer Festigkeitsnachweis für Maschinenbauteile" (strength verification for machine components), was released in 1994. It was followed in 1998 by a third, completely reworked and extended edition (notable for its much more practical updates and a more user-friendly structure). A fourth edition, which was even more comprehensive, was published in 2002. The main innovation of this edition was the inclusion of aluminum materials. An English translation of this guideline appeared as the fifth edition. The sixth edition of the guideline (2012) has, once again, been completely revised, and now includes the results of new research, such as the data for the proposed "Statischer Festigkeitsnachweis" and "Verbessertes Berechnungskonzept FKM-Richtlinie" tests. In the seventh edition of the Guideline, (2020), more new features and results from new research have been added, to improve the Guideline even more. In the meantime, the FKM Guideline has become widely accepted and is regarded as the best reflection of the current state of technology.

57.2.2 Usefulness of the service life calculation

It is a well-known and proven fact that the service life calculation is not sufficiently accurate. In other words, coefficients in the range from 0.1 to 10, and in some cases even greater, may occur between the calculation and the test. However, a basic, if somewhat simplified, statement about the difficulties in achieving a reliable service life calculation has been made: In this case, the strength verification is based on a comparison of the stress values and the stress itself. In a static strength verification, the occurring force can be compared with the sustainable force. The characteristic functions, i.e. the stress spectrum and the S-N curve (Woehler lines) are compared for a proof of service strength. If the total damage, which is of core significance to the service life calculation, is then understood as a quotient of the characteristic functions for stress and sustainable stress, it is clear that this quotient is very sensitive to changes in these critical values. This means errors in determining the characteristic functions will have a significant effect on the result. In addition, by influencing the critical values, for example, by implementing specific measures when selecting materials and at the production stage, the long-term sustainable service life can be increased.

Three different concepts can be used to calculate the service life of parts that are subjected to cyclical stress. These are: the nominal stress concept, the local concept and the fracture mechanics concept. These concepts have specific application areas. For many decades, the technical set of rules was based solely on the nominal stress concept. However, nowadays the local concept and the fracture mechanics concept are being used more and more frequently in this set of rules. In the nominal stress concept, the complex transfer function between load and service life contained in the total stress-strain event in critical material volumes (notch bottom area) is given directly with the component S-N curve (Woehler lines) for nominal stresses. By contrast, in the local concept, this must be represented mathematically by a number of relatively complex modules. This may be the reason for results according to previous experience not being any more accurate than those achieved with the nominal stress concept.

Possible sources of errors in calculating the local concept:

Load assumptions

It must be emphasized that the load assumption must be as precise as possible to ensure an accurate calculation of component service life. Any errors in load assumption can have significant effects on the service life calculation results. The effect may even be greater than those due to insufficient accuracy of the different methods used for service life estimations. We recommend you check the load assumptions carefully and test them if necessary. In this way, any uncertainties in the load assumptions can be resolved by actual measurements performed at a later date. That is particularly because this type of measurement can be performed non-destructively, and can usually provide important information for subsequent designs.

Local stress

Local stresses can be determined either mathematically or by measurement. It is essential that the part's geometry is entered accurately, especially the splines and wall thicknesses. A convergence check must also be performed to ensure the effective stresses are not underestimated. However, a problem still to be resolved in productive operation is how to calculate the effective level of internal stresses in a part cross-section, or in a surface layer, so that this can be evaluated when subjected to load stresses in a service life calculation.

Combined stress

In the case of combined stress, a strength calculation should fulfill the instance of the invariant (results independent of the selected coordinates system). However, as different S-N curves (Woehler lines) are used for normal and shear stresses, the resulting calculated service life/damage is no longer separate from, and independent of, the selected coordinates system.

Material characteristics

Material characteristics cannot usually be determined by simply measuring the finished part. We recommend you refer to standardized or, at least, well-documented values. It is acknowledged that these values may be not be easy to obtain and not always relevant. It is also not possible to determine reliable endurance limit values from tensile strength Rm alone. The fatigue limit can be estimated using the proof stress Rp0.2. The FKM Guideline defines the values from Rm and also for the material type.

Cyclical deformation characteristic

A check to see whether cyclical compaction or loss of cohesion is present must be performed to see whether or not the sequence of load cycles plays a significant role. This effect is not considered in the calculation program.

Stiffening

A number of different models can be used to ascertain the stiffening. As many comparisons between calculated results and test results have shown, a mathematical estimate of stiffening is fraught with uncertainties.

Production processes

When a local concept is applied, it is assumed that the volume element displays cyclical material behavior. Influences encountered during the production process, in particular surface layer characteristics, surface roughness, material state and internal stresses must be taken into consideration. Currently used calculation methods also have their limitations here.

Damage parameters

A number of damage parameters have been proposed to help determine the influence of mean stress and the influence of multiple shafts. P_{SWT}, the most well-known damage parameter, corresponds to a mean stress sensitivity of M=0.41, which is present in this order of magnitude for through-hardening steel, but assumes entirely different values for low-strength steels or wrought aluminum alloys. The use of P_{SWT} should be seen as a major source of errors. The extent to which the influence of internal stresses can be determined is also in question. In the latter case, this is only known for exceptional cases in practice. Damage parameters are still widely used by researchers to determine multi-shaft behavior, excluding proportional stress. The influence of multi-shaft stress states on service life depends greatly on which material is being used. This is because the material's resilience determines which different damage mechanisms are present.

Damage accumulation

In practice, damage accumulation occurs almost exclusively in accordance with the Palmgren-Miner linear hypothesis. Although the shortcomings of this hypothesis were recognized early on, no significant advances that would lead to tolerable errors in the service life calculation have been made in this area despite decades of intensive international research. The only progress is that, by summarizing the amplitudes below the endurance limit, various modifications have been proposed which achieve much better results than the original Palmgren-Miner rule, and in which no damage is caused to amplitudes below the endurance limit.

Even if the service life calculation methods for evaluating variants and analyzing weak points are implemented correctly, it is not certain that the current level of knowledge can achieve a reliable service life calculation for new parts. This requires the use of strategies where calculations are validated and calibrated by specific experimental analyses. At the current level of knowledge, it is only possible to make relative forecasts about service life on a purely mathematical basis.

57.3 Implementation in KISSsoft

57.3.1 Main screen

57.3.1.1 Selection of the part form

Selection of the part form: you can choose parts that are rod-shaped, shell-shaped or block-shaped. They each have different stress components or stress types, and different indexing. If the local concept is applied, block-shaped parts are usually present. The selected part form influences the data input for the stress components.

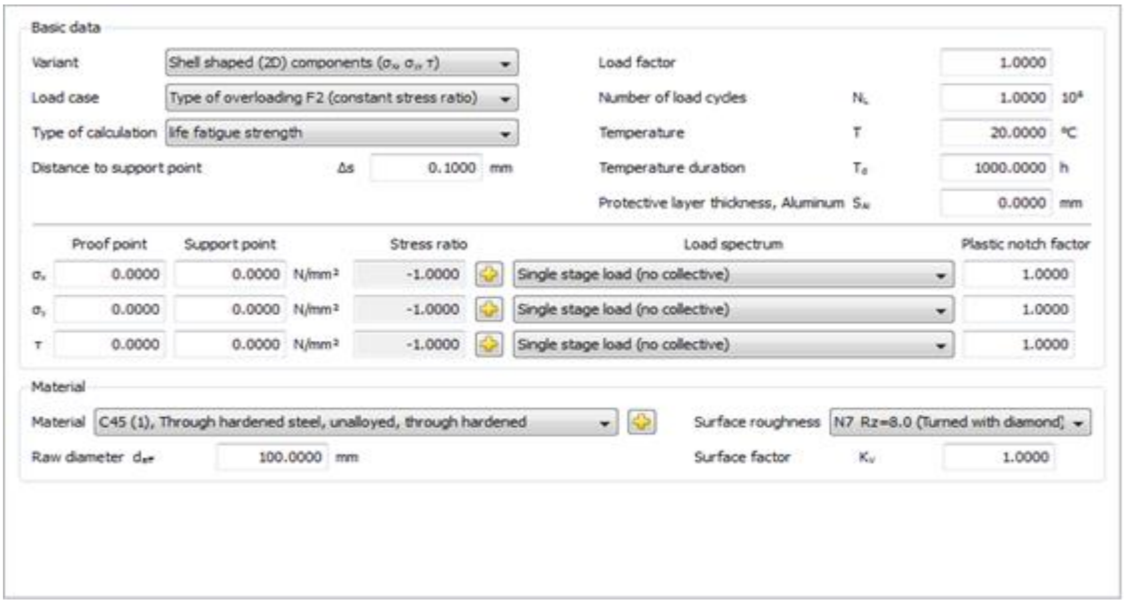


Figure 57.1: Main screen for the proof with local stresses

Rod-shaped parts: the following part-related coordinates system applies, for rod-shaped parts, i.e. rods, beams and shafts: The x-axis lies in the rod axis, and the y- and z-axes are the main axes of the cross section, and need to be specified in such a way that Iy > Iz applies for the moment of inertia.

For planiform (flat) parts, i.e. discs, plates and shells, the following part-related coordinates system should apply in the proof point: the X- and Y-axes lie in the plane, and the Z-axis is vertical to it in the direction of thickness. The normal stress and the shear stresses in the direction of Z should be negligible.

Block-shaped parts: volume-related coordinates systems apply. The primary stresses σ1, σ2 and σ3 need to be calculated. In the proof point W on the free surface of a 3D part, the primary stresses σ1 and σ2 should act in the direction of the surface and the primary stress σ3 points into the interior of the part, vertically to them. Generally, there is one stress gradient that runs vertically to the surface, and two stress gradients that run in the direction of the surface, for all stresses. However, only the stress gradients for σ1 and σ2, running vertically to the surface, can be taken into account in the calculation, and not the stress gradients for σ1 and σ2 in both directions on the interface and none of the stress gradients for σ3.

57.3.1.2 Inputting the stress values on the proof point and on the neighboring point

If the support factor is determined according to the stress state on the neighboring point, then the stresses on the proof point W and on the neighboring point B, and also the distance from point B to point W, will be entered. (Enter compressive stresses as negative values.):

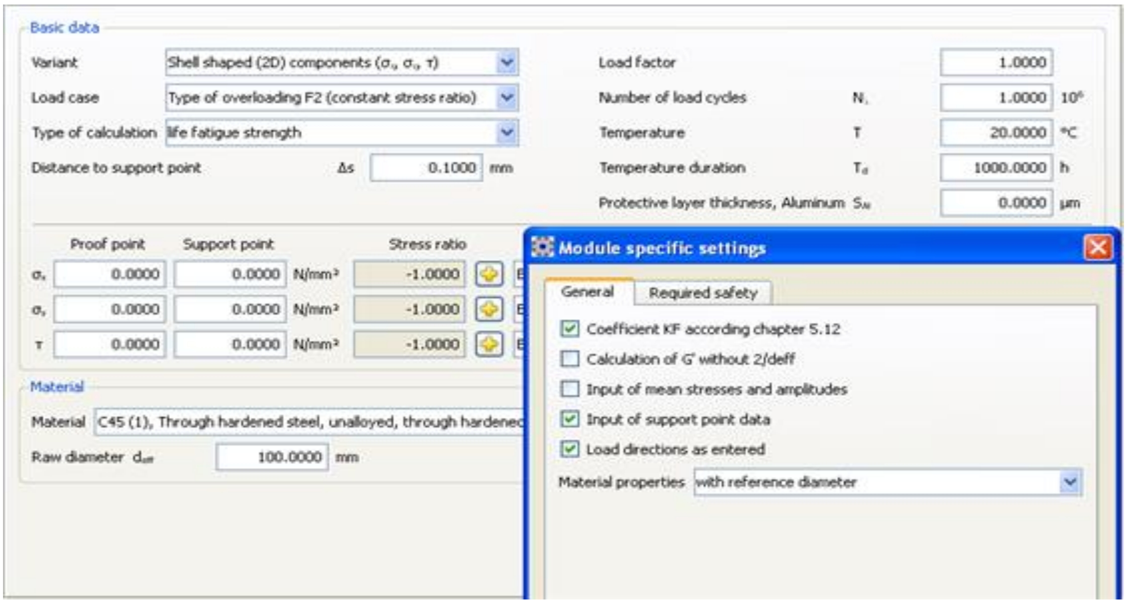


Figure 57.2: Inputting the stress values on the proof point and on the neighboring point. Inputting the neighboring point distance.

57.3.2 Stress cases

In the endurance limit diagram, different assumptions are used to determine different levels for the maximum stress amplitude S_{AK}. Assumptions where sm=const. result in a larger S_{AK} value than for R=const. This is because the limit lines in the Smith diagram rise by an angle < 45° (mean stress sensitivity). The most suitable assumption depends on the expected change in stresses in the part when it is subjected to permitted operational fatigue load. The overload case can therefore be a decisive factor in whether or not a part is overloaded [80].

Stress case

- Type of overloading F1 (constant mean stress): at a constant mean stress, the stress amplitude increases as the decisive operating force increases.
- Type of overloading F2 (constant stress ratio): when the operating force increases, the ratio between the maximum stress and minimum stress remains the same. This overload case usually returns conservative results (compared to other overload cases) and should therefore be used in case of doubt.
- Type of overloading F3 (constant minimum stress): when the operating force increases, the minimum load remains the same.
- Type of overloading F4 (constant maximum stress): when the operating force increases, the maximum load remains the same.

57.3.3 S-N curve (Woehler lines)

Miner elementary, section 4.4.3.5.2 of the FKM Guideline

If a stress collective is present instead of individual stress, the calculation should usually be performed using the Miner elementary process.

Miner consistent, section 4.4.3.5.2 of the FKM Guideline

The Miner consistent process (derived from Haibach, see [18]) takes into consideration the fact that the part infinite life strength will reduce as damage increases. The reduction applies from $K_{D,}\sigma$ =1*10.^{6}.

57.3.4 Number of load cycles

Number of load cycles. If calculation according to Miner elementary is selected, then inputs greater than ND result in constant use.

57.3.5 Temperature

Inputting the temperature in degrees Celsius. The area of application of the FKM Guideline is limited according to material, see section 1.2.1.7. The temperature factor $K_{T,D}$ is then defined, depending on the temperature and material type.

57.3.6 Temperature duration

Time period during which the part is subjected to the temperature.

57.3.7 Protective layer thickness, section 4.3.4

The protective layer factor Ks, which is defined via the protective layer thickness, takes into account the influence of the protective layer on the fatigue strength of a part made of steel, cast iron or aluminum.

In the case of steel or cast iron materials, the protective layer is created by galvanizing, hot-dip galvanizing or zinc-flake coating. In the case of aluminum materials, the protective layer is created by an aluminum oxide layer.

57.3.8 Stress ratios

The mean stress is recorded in the R-value. In comparison to the mean stress-free case (cyclic loading, R=-1), the S-N curve (Woehler lines) is moved to higher sustainable stress amplitudes in trials with mean compression stresses. In trials with mean tensile stresses, the S-N curve (Woehler lines) is moved to lower sustainable stress amplitudes. The sustainable stress amplitude's dependency on the mean stress is material-specific, and is called the influence of the mean stress. This usually increases along with the material's tensile strength.

Here R is defined from -1 up to +1

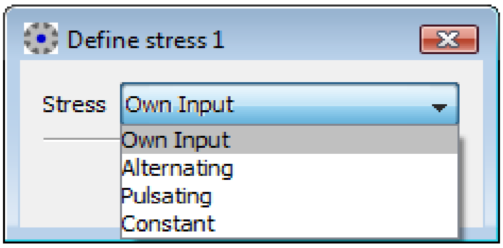


Figure 57.3: Inputting the specific R-value

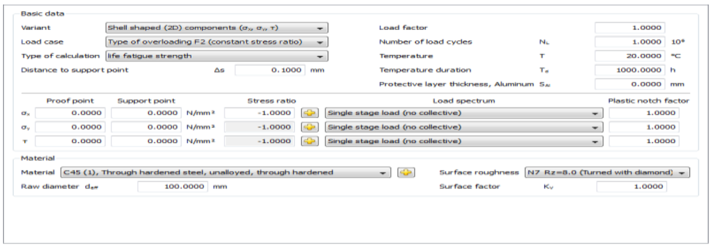


Figure 57.4: Inputting your own R-value.

As the surface roughness increases, the S-N curve (Woehler lines) moves to lower stress amplitudes, but the surface roughness alone is not the cause for this. The strength is much more affected by the detailed properties of the surface. In addition, despite similar surface characteristics and the same surface roughness, different processing procedures can cause different material internal stress states, resulting in S-N curves (Woehler lines) that differ from each other greatly.

57.3.9 Spectra

You can select existing load spectra directly.

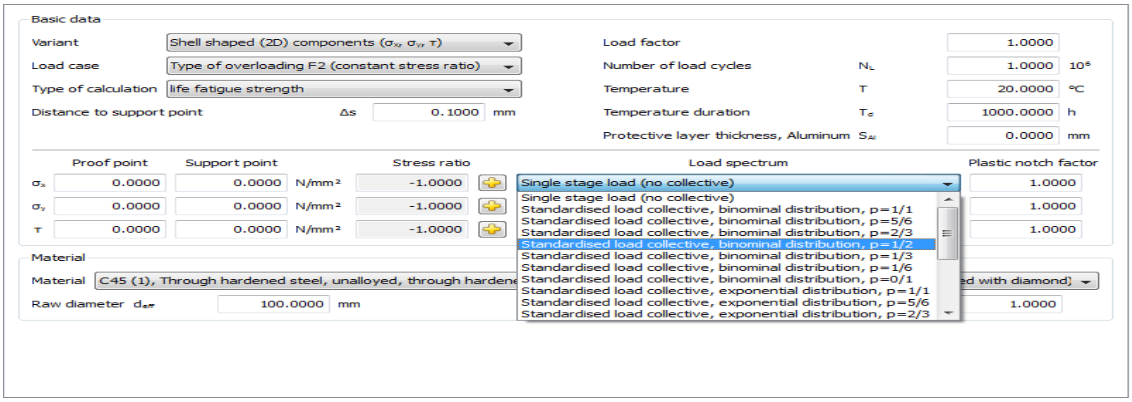


Figure 57.5: Selecting load spectra

You can create a new load spectrum in the database tool (see chapter 15.2.8, Define load spectrum).

57.3.10 Surface factor KV, section 4.3.3, Table 4.3.7

Case factor K_V takes into account the influence of surface treatment on the fatigue strength.

57.4 Materials

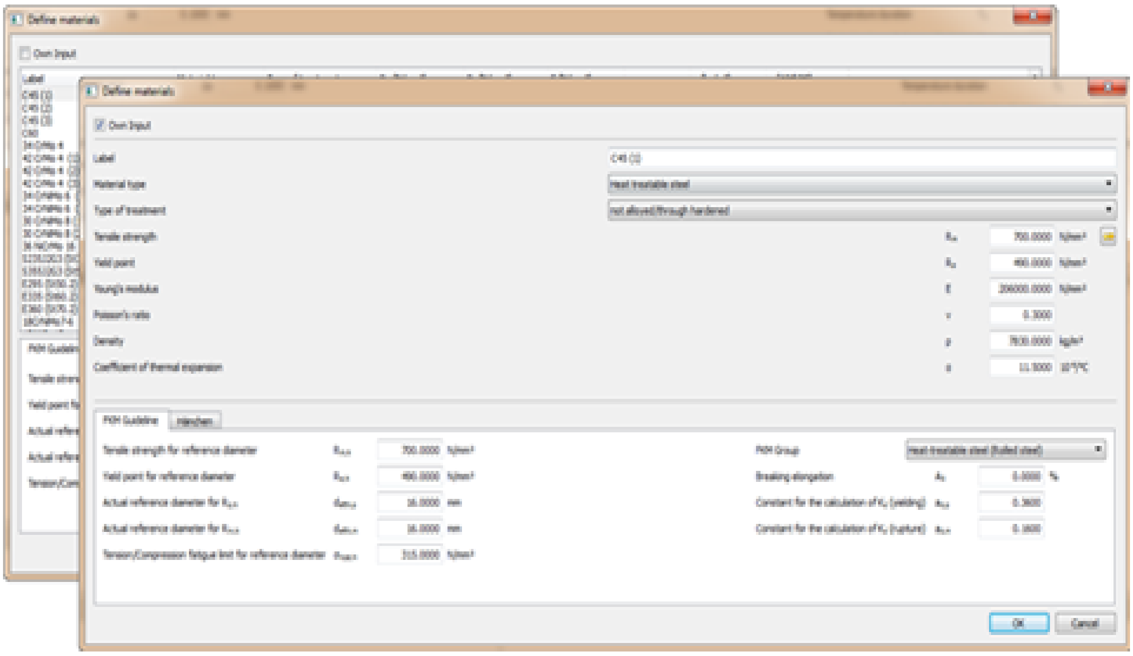


Figure 57.6: Materials screen: Proof of strength with local stresses

The selection list contains materials from the FKM Guideline.
If you have set the Own Input flag, a new dialog is displayed here. This displays the material data

used in the calculation, which you can specify to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that these can also be used in subsequent calculations.

57.4.1 Surface roughness

The roughness factor takes into account the influence of the surface roughness on the part's fatigue strength. Experiments are performed to derive it from the infinite life strengths of unnotched test rods with and without surface roughness, and shown in dependency on the material's total height Rz and tensile strength Rm. For polished surfaces, it has the value 1.0. The average roughness Rz=200µm applies for rolled, forged and gray cast scale. The average roughness can also be defined as your Own Input.

57.5 Settings

57.5.1 General settings

The references are to sections in the FKM guideline.

- **Coefficient KF according to equation 4.3.2 and 4.3.3, described in section 4.3.1.2.**

Notch effect coefficient as an estimated value to enable the effect of the roughness coefficient to be determined, according to the nominal stress concept, when the local stress concept is in use.

Checkbox selected: Kf is defined according to formulae 4.3.2 and 4.3.3, described in section 4.3.1.2.

Checkbox not selected: The KF coefficient is set as shown in Table 4.3.1.

- **Calculating G without 2/deff, section 4.3.1.3.3**

If the Input of neighboring point data checkbox is not selected in the "General data" tab, an approximation of the related stress gradient is calculated, using the calculation based on the equations in section 4.3.18. This contains terms for tension/compression, torsion, and bending. If no bending is present, it is questionable whether the second term (2/d) in the formulae makes any sense. The option programmed here is not provided in the FKM Guideline!

If the checkbox is selected, the stress gradient is determined without applying the second term in formula 4.3.18.

If the checkbox is not selected, the stress gradient is determined while also applying the second term in formula 4.3.18.

- **Input of mean stresses and amplitudes**

If the checkbox is selected then the stresses are input in the main screen, via the medium and amplitude stress.

- **Input of mean stresses and amplitudes**
    - **Material values at reference diameter:**
      values are taken from database (at reference diameter) and multiplied by K1.
    - **Rm, Rp depending on value from database, sigW at reference diameter:**
      Rm, Rp are imported from the database according to size (excluding K1). The fatigue strength is determined for the reference diameter entered in the database, and then multiplied by K1.
    - **Rm, Rp depending on value from database, sigW constant:**
      Fatigue strength not multiplied by K1, correct value must be in database
    - **Rm, Rp depending on value from database, sigW calculated from Rm:**
      Fatigue strength is calculated from Rm. Rm is stored in the database according to size, conversion according to FKM.
- **Input of neighboring point data, section 4.3.1.3.3, Formula 4.3.17**

Checkbox selected: Notch sensitivity factor-related stress slope is defined in the neighboring point via the stress state. To do this, the stress values and the distance between the proof point and neighboring point must be entered in the main screen.

Checkbox not selected: Notch sensitivity factor-related stress gradient is not determined from the values at a neighboring point. The related stress gradient at the point of maximum stress is estimated using formula 4.3.18. To do this, two radii (Radius 1 and Radius 2) must be defined (for the two directions on the surface), and also a typical part dimension d. See also: module specific setting, Calculation of G without 2/deff, above.

- **Direction of load as specified in sections 4.1.0 and 4.6.2.**

Checkbox selected: the calculation is carried out for synchronous stresses.

Checkbox not selected: the calculation is performed for asynchronous stresses (4.6.2.2). It can safely be assumed that this method of approach is a cautious one.

- **Use the mechanical material support, see sections 4.3.1 and 4.3.1.3.2**

If this checkbox is selected, the mechanical material notch factor is used for the calculation. Otherwise, the Stieler notch factor is used. If sharp notches are present, the mechanical material support coefficient takes into account the strength reserves and contains the static size coefficient. The mechanical material support number (nwm) is made up of three parts: the static support number (nst), the mechanical deformation support number (nvm) and fracture mechanical support number (nbm). Assumption: nst = 1 is applied to the "Smooth shaft" and "Own Input" notch types.

- **Selecting materials data, section 3.2.1**

The part standard values Rm and Rp must be calculated from the semi-finished product or test piece standard values $R_{m,N}$ and $R_{p,N}$ or from the part drawing value $R_{m,Z}$. In exceptional situations, the part actual values $R_{m,I}$ and $R_{p,I}$ can be applied.
See section 3.2.1.2. for more information.

- **Calculating the endurance limit for surface-treated parts (section 5.5)**

If this checkbox is enabled, and you have selected a rolled steel and a surface treatment method, the calculation method specified in the FKM Guideline, 7th Edition, section 5.5, is performed.

- **Surface factor**

This selection is only used for calculating the endurance limit of surface-treated parts.

- **Hardening depth**

You can use this entry to define the location of the transition point between the hard surface layer and the core.

- **Determine core hardness from tensile strength Rm**

If this checkbox is selected, the core hardness is estimated from the tensile strength instead of using the core hardness entered when the material was specified.

57.5.2 Required safeties

The FKM Guideline is one of the few calculation guidelines that lists the required safeties according to the consequences of failure etc. When used together with safe load assumptions and an average reliability of the strength variables Pü=97.5%, they apply for both welded and non welded parts. Safety factors are defined as the results of inspections and tests, on the basis of the selected material, the defined consequences of failure and the probability that the specific load will occur. This guideline differentiates between five different classes: steel, cast iron (ductile or non ductile), and aluminum (ductile or non ductile). You can also specify the safety factors manually.

jmt	Safety factor against creep strength depending on time
jp	Safety factor against yield point
jpt	Safety factor against creep limit
jF*jG	Safety against endurance limit

57.6 Estimation of the endurance limit for surface-treated parts (section 5.5)

This calculation should only be used for surface-treated rolled steel. The surface treatments include the following treatment methods applied to the materials:

- case-hardened
- nitrided, gas-nitrided, nitro-carburated
- induction hardened
- rolling
- shot peening

You can define these treatment types either when you specify the material or when you input the surface factor (Settings).

This process is based on the concept of a local endurance limit. Two points on the part are observed. The first point is on the part's surface, and the second point is at the transition point between the hard surface layer and the core. This is calculated with the greatest principal stress σ1. If no principal stresses have been entered, they are calculated first.

You can also input a hardening depth for this calculation in the Settings. The hardening depth is then used to calculate the distance from the component's surface to the transition point between the hard surface layer and the core.

The constant Kf can either be calculated in accordance with formulae 4.3.2 and 4.3.3 or taken from Table 4.3.1. You also have the option of inputting the core hardness in the Settings when you specify the material. Alternatively, this can also be estimated from the tensile strength.

This approach is used to calculate the internal stress, which is included in the calculation of mean stress sensitivity. In this case, the degree of utilization for the point on the component's surface is calculated first, followed by the degree of utilization at the transition point between the hard surface layer and the core. The greater of the two degrees of utilization is then used for the proof. Both degrees of utilization should be < 1.

The results are only displayed in the report if this calculation method has been selected for rolled steel with the predefined treatment types.

58 Hertzian Pressure

Use this module to calculate the Hertzian pressure between two bodies. In the case of a load on a rolling pair that is applied vertically to the contact surface, elliptical flattening occurs for point contact, and rectangular flattening occurs in the case of linear contact.

For the case of general contact, the theory presented in reference [107] is used. Based on this reference, a grid of triangular pressure distribution elements is used to calculate the line contact between bodies with an arbitrary profile, while applying a given vertical line load. The bodies are assumed to extend infinitely, vertical to their profile. The profiles of the two bodies are taken from two data files. The files have three columns that contain the index, the position and the profile value.

```
:TABLE LIST Profile
  INPUT index TREAT NEXT_BIGGER
  OUTPUT Position, ProfileValue

DATA
1       -5      0.635083269
2       -4.95   0.622242132
3       -4.9    0.609538427
4       -4.85   0.596971886
5       -4.8    0.584542241
6       -4.75   0.57224923
7       -4.7    0.560092593
8       -4.65   0.548072075
9       -4.6    0.536187424
10      -4.55   0.524438391
11      -4.5    0.51282473
12      -4.45   0.5013462
13      -4.4    0.490002563
14      -4.35   0.478793582
15      -4.3    0.467719027
16      -4.25   0.456778669
17      -4.2    0.445972282
END
```

Figure 58.1: Format of the body profile data file

All values in this file must be entered in millimeters.

The profile file for each body can have different discretization and position values. However, we recommend you use the same discretization and position for both bodies. The accuracy of the results will depend on the density of the profile points (discretization length). Edge contact effects are also considered. The contacting surfaces are assumed to be frictionless. This general contact option enables the user to generate a 2D plot of the subsurface stresses and a plot of the contact pressure along the profiles. You will find more detailed information about the calculations and equations used in the reference document mentioned above.

The Hertzian equations are used to help calculate the maximum pressure (Hertzian pressure) and also the proximity of the two bodies (ball, cylinder, ellipsoid, plane; convex or concave). The calculation formulae have been taken from "Advanced Mechanics of Materials, 6th Edition [108]. The underlying principle for calculating point contact is that the diameter of the bodies is defined on two main planes, from which an equivalent diameter is then defined. The calculation is performed in one main plane for linear contact, so there is only one equivalent diameter. The location and value of the maximum principal shear stress in the interior of the body are also determined.

An approximation of the cylinder/cylinder configuration has been calculated using the dissertation from Weber/Banaschek [21]. The formula (55) from Norden's book [15] is used to calculate the approximation of the cylinder area.

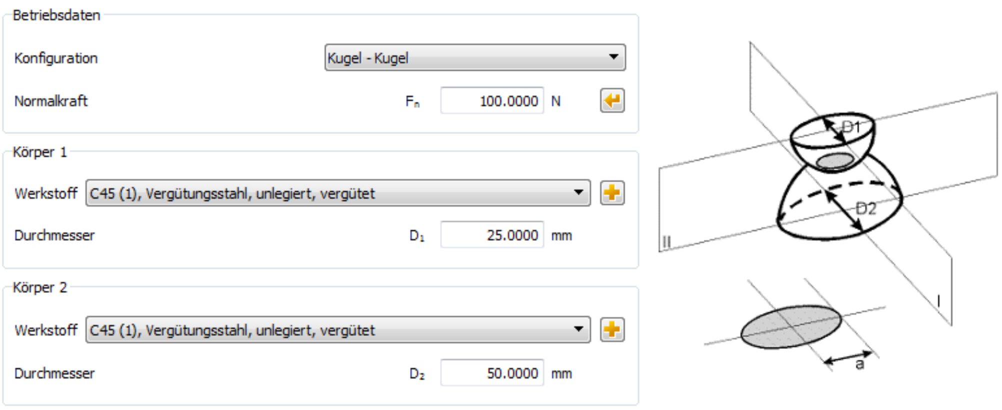


Figure 58.2: Main window for Hertzian pressure

In the main screen for Hertzian pressure (see Figure 58.2) you can define the normal force, the configuration, and also the diameter (and, in the case of linear contact, the supporting length leff) and the materials of the bodies:
You can select one of these configurations:

- Ball - ball
- Ball - cylinder
- Ball - ellipsoid
- Ball - plane
- Ellipsoid - ellipsoid
- Ellipsoid - cylinder
- Ellipsoid - plane
- Cylinder - cylinder
- Cylinder - plane
- Any contact

On the right, in the main screen, an image of the current configuration is displayed to help you to understand the input values better.
For normal force there is also a sizing option. If you click the sizing buttons next to the normal force, you can enter the required Hertzian pressure, and the system will then calculate the normal force from it.

If the support area has a concave bend then you must enter the diameter as a negative value. Negative diameters are only possible in the case of Body 2.

58.1 Settings

Use the Depth display factor to define the depth display in the graphic. The depth of the point τmax is multiplied by this coefficient. The resulting depth is then displayed in the graphic. The default setting of this coefficient is 6.

59 Hardness Conversion

You access the hardness conversion module in the Extras > Hardness conversion menu. Hardness conversion is also present in the materials screens as a sizing function. You can therefore use a hardness value to define the tensile strength in these screens.

This module includes the hardness conversion calculation as specified in DIN EN ISO 18265, Edition 2-2014. During the conversion, select Extras > Hardness conversion to display a selection list in which you can select the required material. The other conversions (for the materials) use the table for unalloyed and low-alloy steels and steel casting. The stored tables can be used to convert the value of the tensile strength into Vickers, Brinell or Rockwell hardness, as required. Due to possible variations, the received values should only be used if the default testing process cannot be applied. The interim values of the value conversion table are interpolated from the neighboring values.

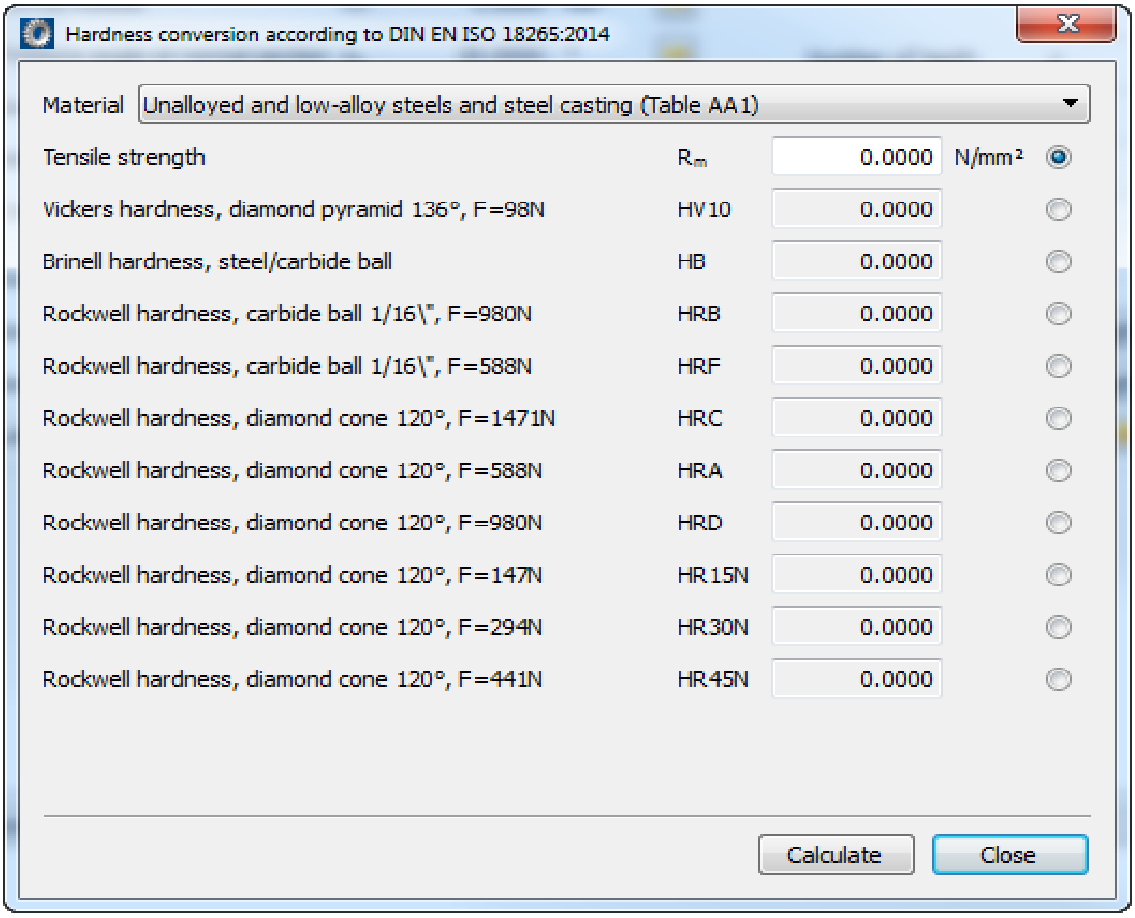


Figure 59.1: Hardness conversion input screen

Integrated conversions of the steels and steel groups according to DIN EN ISO 18625:

- Unalloyed and low-alloy steels and steel casting (Table A.1)

- Through hardening steel in heat treated state (Table B.2)
- Through hardening steel in untreated, soft-annealed or normally annealed state (Table B.3)
- Through hardening steel in hardened state (Table B.4)
- Cold-working steels (Table C.2)
- High-speed steels (X80WMo6.5, X82WMo6.5, X90WMo6.5, X97WMo3.3, X100WMo6.5, X85WMoCo6.5.5, X105WMoCo6.5.5 and X79WCo18.5) (Table D.2)

The range of validity for unalloyed and low-alloy steels and cast steel (with the conversion in the material screens applied) is limited as follows:

- Tensile strength Rm: 255 to 2180 N/mm2
- Vickers hardness HV: 80 to 940 HV
- Brinell hardness HB: 76 to 618 HB
- Rockwell hardness HRB: 41 to 105 HRB
- Rockwell hardness HRF: 82.6 to 115.1 HRF
- Rockwell hardness HRC: 20.3 to 68 HRC
- Rockwell hardness HRA: 60.7 to 85.6 HRA
- Rockwell hardness HRD: 40.3 to 76.9 HRD
- Rockwell hardness HR 15N: 69.6 to 93.2 HR 15N
- Rockwell hardness HR 30N: 41.7 to 84.4 HR 30N

Rockwell hardness HR 45N: 19.9 to 75.4 HR 45N

60 Linear Drive Train

Use this calculation module to calculate drive screws. Drive screws are used to convert rotational movement into longitudinal movement or to generate great forces.

Although trapezoidal screws are almost exclusively used as drive screws, some rough operations also use buttress threads.

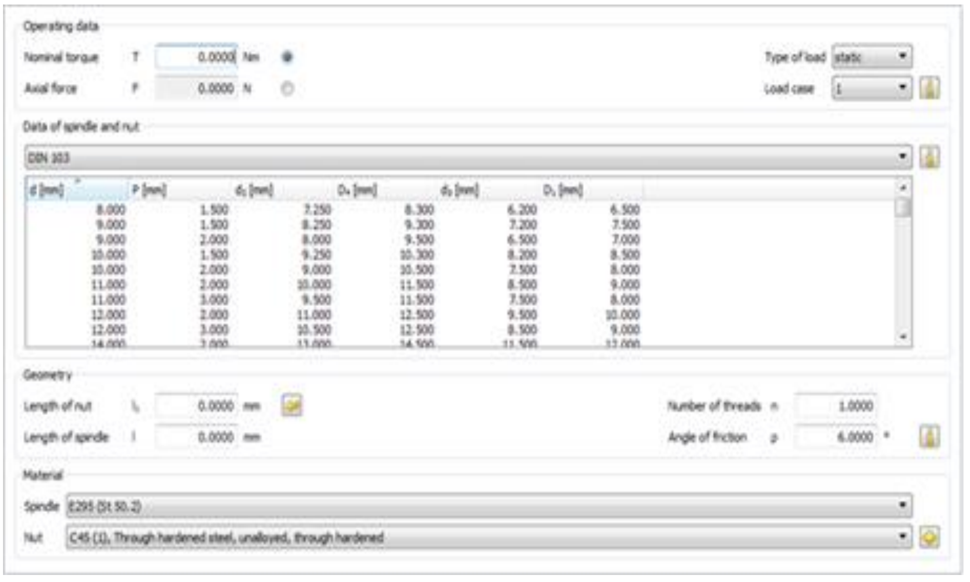


Figure 60.1: Linear drive train basic data

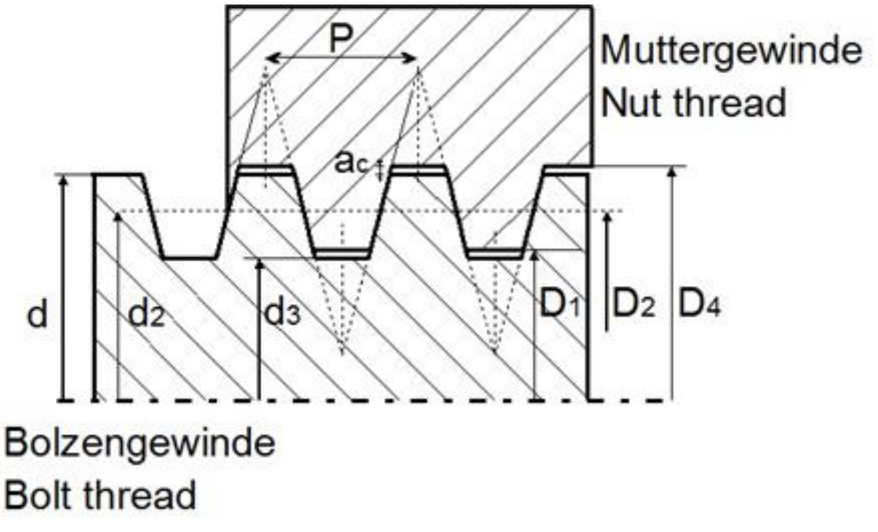


Figure 60.2: Dimensions of trapezoidal screws

Two different linear drive train configurations can be calculated:

- Load case 1
  Stress on the spindle in a spindle press
- Load case 2
  Stress on the spindle in a gate valve

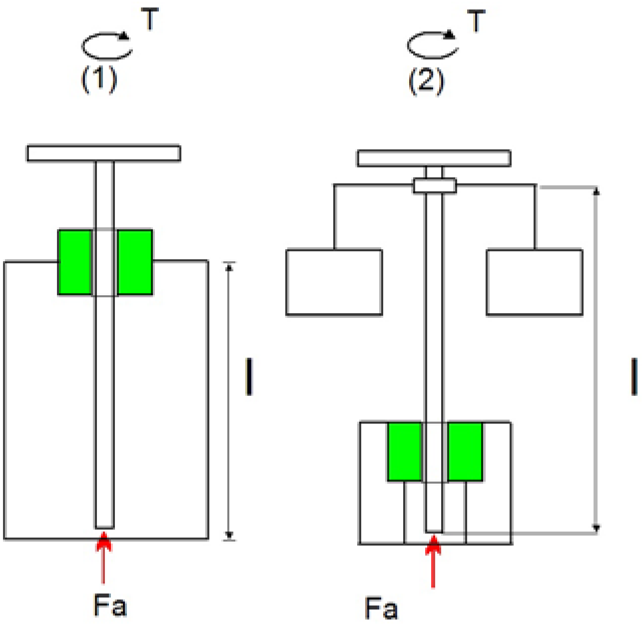


Figure 60.3: Linear drive train stress cases

The information provided in Roloff Matek [80] is used to calculate linear drives (drive screws).

60.1 Calculation

Short and long linear drive trains subjected to pressure are handled separately in the calculation process.

Short pressure stressed drive screws

Short pressure stressed drive screws are not at risk of buckling and therefore are not tested for this.

The required cross section of the thread can therefore be defined using the formula:

$$A_3 \geq \frac{F}{\sigma_{d(z)zul}}$$

σd(z)zul: under static load: Rp/1.5; under pulsating load σzdSch/2.0; under alternating load: σzdW/2.0;

Long pressure stressed drive screws

The formula for calculating the necessary core diameter of the thread is taken from the Euler equation:

$$d_3 = \sqrt[4]{\frac{64 \cdot F \cdot S \cdot lk^2}{\pi^3 \cdot E}}$$

S: Safety (S≈6 to 8)

lk: mathematical buckling length, lk≈0.7*l (Euler buckling case 3 is used for general guided spindles)

Calculation of the strength:

Stress case 1:

The upper part of this configuration is subject to torsion and the lower part is subject to compression and therefore buckling.

Torsional stress:

$$\tau_t = \frac{T}{Wp} \leq \tau_{tzul}$$

Wp: polar moment of resistance Wp»0.2*d3^3

τtzul: permissible torsional stress; Static load τtF/1.5; Pulsating load τtsch/2.0; Alternating load τtW/2.0;

Compressive (tensile) stress:

$$\sigma_{d(z)} = \frac{F}{A_3} \leq \sigma_{d(z)zul}$$

A3: Minor diameter cross section thread

σd(z)zul: permissible compressive (tensile) stress

Stress case 2:

The upper part of this configuration is subject to torsion and the lower part is subject to compression, infrequent tension and torque.

Formula for the part to be checked:

$$\sigma_v = \sqrt{\sigma_{d(z)}{}^2 + 3 \cdot (1 \cdot \tau_t)^2} \leq \sigma_{d(z)zul}$$

The required torque corresponds to the thread torque, if not subject to any moments of friction.

$$T = F \cdot d_2 / 2 \cdot \tan(\varphi \pm \rho')$$

d2: Flank diameter of the thread

φ: Lead angle of the thread (for single thread trapezoidal screws φ≈3°...5.5°)

ϱ': Thread friction angle

Material		Lubricant	Friction angle ρ' [°]
Screw	**Nut**		
Steel	Cast iron	Dry	12
Steel	CuZn – CuSn alloys	Dry	10
Steel	Special plastics	Dry	6
Steel	Cast iron	Lubricated	6
Steel	CuZn – CuSn alloys	Lubricated	6
Steel	Special plastics	Lubricated	2.5

Figure 60.4: Values for the friction angle

The + in the formula stands for "tightening the spindle", and - stands for "loosening the spindle". The KISSsoft calculation calculates both situations and outputs the results in a report.

Calculation for buckling (only for long spindles):

First of all, calculate the slenderness ratio.

$$\lambda = \frac{lk}{i} = \frac{lk}{\sqrt{I / A_3}} = \frac{lk}{\sqrt{\frac{\pi \cdot d_3{}^4 \cdot 4}{64 \cdot d_3{}^2 \cdot \pi}}} = \frac{lk \cdot 4}{d_3}$$

λ: Slenderness ratio of the spindle

lk: mathematical buckling length

i: Gyration radius

Only 3 different materials can be used for the spindle so that the slenderness ratio can be defined correctly.

Elastic buckling is present if λ>=λ0 = 105 for S235, or λ>=89 for E295 and E335.

$$\sigma_K = \frac{E \cdot \pi^2}{\lambda^2}$$

The non-elastic area as defined by Tetmajer and λ <105 for S235.

$$\sigma_K = 310 - 1.14 \cdot \lambda$$

For λ<89 and for E295 and E335:

$$\sigma_K = 335 - 0.62 \cdot \lambda$$

The Johnson parabola equation can also be used for the calculation for a non-elastic case. (also for other materials)

$$\sigma_K = \sigma_{dS} - (\sigma_{dS} - \sigma_{dP}) \cdot \left(\frac{\lambda}{\lambda 0} \right)^2$$

The safety can then be calculated as follows:

$$S = \frac{\sigma_K}{\sigma_{vorh}} \geq S_{erf}$$

The required safety for elastic buckling is Serf≈3 to 6. For non-elastic buckling, it is Serf≈4 to 2.

Buckling no longer needs to be calculated for a slenderness ratio < 20.

Calculation of the nut:

The surface pressure of the nut is calculated from the nut length:

$$p = \frac{F \cdot P}{l_1 \cdot d_2 \cdot \pi \cdot H_1} \leq p_{zul}$$

P: Pitch of thread

l1: Length of the nut thread

d2: Flank diameter of the thread

H1: Flank engagement of the thread

pzul: permissible surface pressure

Due to the uneven distribution of surface pressure, the nut length should be no greater than 2.5*d. During sizing, the length is limited to 2.5*d even if a longer one is input.

Efficiency and self-locking:

The efficiency of the conversion from rotational movement into longitudinal movement:

$$\eta \approx \frac{\tan\varphi}{\tan(\varphi + \rho')}$$

The conversion of movement is only possible for non-self-locking threads, because the limiting value in this case is that, $\varphi=\varrho'$, the efficiency is 0.5.

If $\varphi>\varrho'$ the thread is no longer self-locking.

Each permissible value is taken from the Roloff Matek tables.

60.2 Sizings

When you select Own Input, this calculation module can calculate the core diameter d3 of a long spindle that is subject to pressure.

It can also define the nut length on the basis of permissible surface pressure and the required safety.

60.3 Settings

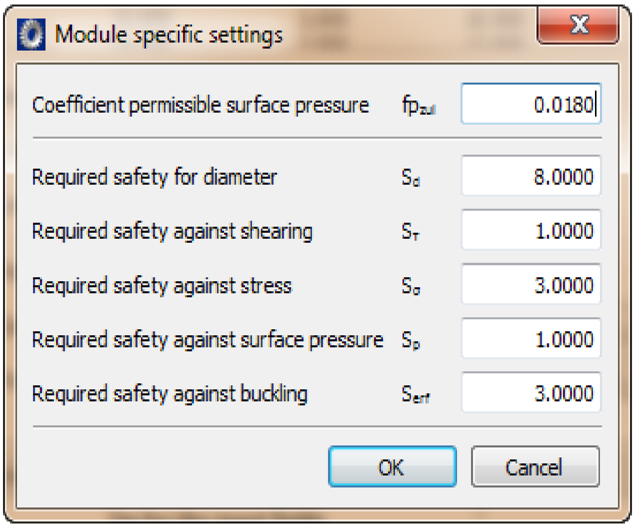


Figure 60.5: Settings input window

- Coefficient permissible surface pressure: this factor is used to define the ratio to Rm, in other words pzul = fpzul*Rm
- Required safeties for diameter, shearing, stress, surface pressure and buckling: for the calculation and the sizings

60.4 Materials

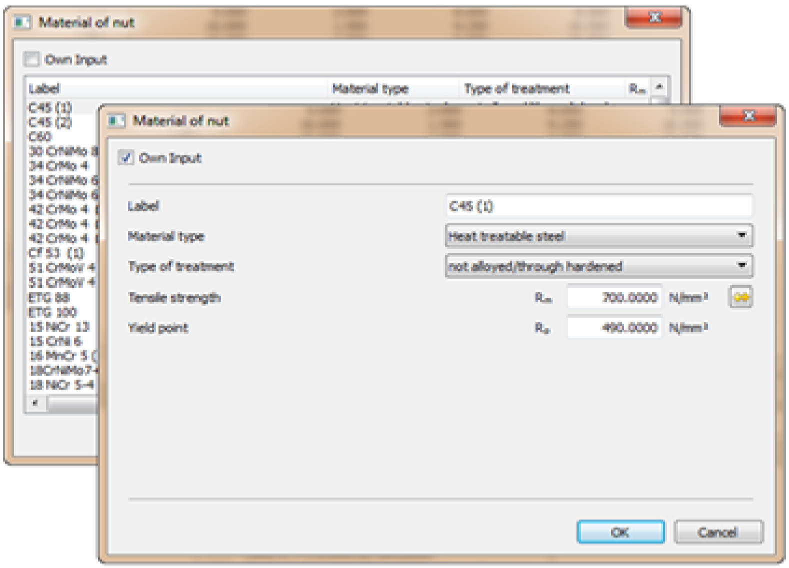


Figure 60.6: Nut materials input window.

The selection list contains materials from the standard.
If you have set the Own Input flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see chapter 9, Database Tool and External Tables), so that these can also be used in subsequent calculations.

You can only select these different materials for nuts. For the spindle material you can choose E295 (St 50.2), E335 (St 60.2) and S235 (37.3) materials, because the calculation of buckling is only designed for use with these materials.

The strength values for the 3 materials have been fixed:

- E295 (St 50.2): Rp02 = 295 N/mm2; σzdSch = 295 N/mm2; σzdW = 195 N/mm2; λ0 = 89; τtSch = 205 N/mm2; τtW = 145 N/mm2
- E335 (St 60.2): Rp02 = 335 N/mm2; σzdSch = 335 N/mm2; σzdW = 235 N/mm2; λ0 = 89; τtSch = 230 N/mm2; τtW = 180 N/mm2
- S235 (St 37.3): Rp02 = 235 N/mm2; σzdSch = 225 N/mm2; σzdW = 140 N/mm2; λ0 = 105; τtSch = 160 N/mm2; τtW = 105 N/mm2

61 Plastics Manager

This module can be used to add new plastic materials to the KISSsoft material database, if the material characteristics are available. When a material is added to the database, KISSsoft generates its associated .dat material file.

Currently, KISSsoft has over 67 plastic materials in its database. However, although a large number of different plastics are available for gear calculations, you may often need to enter your own materials in the KISSsoft database. Before you can add a new material to the KISSsoft database, you need to know at least the material properties displayed below:

- Poisson's ratio
- Coefficient of friction for oil, grease and dry run
- Temperature-dependent Young's modulus

If other material properties (tensile strength and yield point, temperature-dependent wear coefficients, thermal expansion coefficient, permissible root/flank stresses, etc.) are also known, they can also be added to the KISSsoft database. The Young's modulus, tensile strength and yield point can also be entered for dry material properties. The more material characteristics this database contains, the more accurately plastic gear calculations can be performed.

If gear test results are available, the permissible root or flank stresses can also be calculated, and the results entered in a .dat file for the material. This data can then be used to calculate safety factors over the entire rating life of the gears.

In the Basic data tab, you can input general material characteristics (density, material type, absorption of water etc.) and also tribological characteristics (coefficient of friction and wear coefficient). Specific heat capacity and conductivity are used purely for information purposes in the material database.

Tribological properties can be entered individually for oil, grease and dry runs. If the data is available, you can enter temperature-dependent wear coefficients for all lubrication conditions in the appropriate temperature-dependent wear factor table. Click the Plus button next to the Material name input field to import material data from the M-Base database (if available). If a material dataset is not available, please contact M-Base for more information.

If the wear coefficient's temperature dependence is not known, KISSsoft assumes the same temperature range for the wear coefficient as was used for the Young's modulus data. Additional comments are also displayed in an appropriate .dat file.

You can also input temperature-dependent material data (Young's modulus, tensile strength and yield point) in the Test data tab.

If results from gear testing are available, it is possible to calculate the permissible fatigue strength for root and/or flank. Enter the test data in the "Test gear measurements" table in the Test data tab.

61.1 Gear test results

There are two different methods for using test results to calculate the permitted root and/or flank stress. In the first result type (Case I), all measurements on the test rig are performed with an unchanged cylindrical gear pair configuration (calculation file with file extension "*.Z12"). The geometry of the test gears does not change between the measurement results. In the second result type (Case II), you can use different cylindrical gear geometries for each measurement (calculation file with extension for 2 gears ("*.Z12"), 3 gears ("*.Z15"), 4 gears ("*.Z16") and planetary gear units ("*.Z14") with different lubrication conditions. This means you can assign different geometries for each measurement result. However, you cannot enter and calculate both datasets at the same time. To switch between the two cases, go to Module specific settings > Test gear geometry based on one file.

61.1.1 Case I, test results for unchanged test gears

Such test results are usually measured on test rigs. Geometry of the test gears must be defined in a separate KISSsoft .Z12 file, which can be selected on the user interface (Test data tab). In this case, only 1 lubrication regime is possible and must be defined in test file.

Before test result data is entered in the table, a file must be created and selected for the test gear. The table in the user interface is adjusted to suit the test, using the settings in the gear file. The material characteristics of the failed gear in the test file are not important, as they are overwritten with user-defined values set in the Basic data and Test data tabs. The selected material of the failed gear must be a plastic (for example, PA66 (VDI 2736)). The correct material characteristics must be set for the material from which the counter gear that is to be mated is made.

A failed gear must be specified for the selected test file. If both materials from the selected test file are a type of plastic, the user can select either gear 1 or gear 2. In other cases, the failed gear is set automatically. It is only possible to have failure on either gear 1 or gear 2, not on both gears simultaniously.

The following limitations apply for the test files:

- Calculation method for plastic gears must be selected
- The Calculate temperatures root/flank option must be deselected (for grease and dry runs)
- Calculation without load spectrum
- At least 1 gear must be made of plastic material
- The Rating tab must be visible.

To calculate the permissible root and/or flank strength, the following test results must be available:

- Torque, speed and cycles to failure of the failed gear. According to VDI 2736, all the tests should run until failure
- Environmental, root and flank temperature (dependent on the "failure cause") or oil temperature (for oil lubrication) of the failed gear.
- If the mating gear is plastic, and the failure cause is set to flank, the mating gear's flank temperature must also be set.
- The type of "failure cause" can be set to no failure, root, flank or wear.

The test gear allowances table can also be imported from a file. However, you must use a suitable file structure (see example file K17_testResults.dat).

Additional settings that control the calculation of permissible root and/or flank strength are available in the Test data tab in Module specific settings. Sections 1.2 and 1.3 have more detailed explanations of the individual parameters.

61.1.2 Case II. Test results with different gear geometry

To enable this option, deactivatet the "Test gear geometry based on one file" option in Module specific settings. The required input fields in the "Test gear measurements" table in the Test data tab are modified to suit the selection you make.

In the "Test gear measurements" table, select the failed gear, "failure cause" and mating gear for every test file. The gear numbers for the .Z12 file are [1, 2], for "*.Z15" they are [1, 2, 3], for "*.Z16" they are [1, 2, 3, 4] and for "*.Z14" they are [1=sun, 2=planets, 3=internal gear]. Enter values for torque, speed, cycles to failure and temperatures in the individual gear files in the test.

Additional settings that control the calculation of permissible root and/or flank strength are available in the Test data tab and in Module specific settings. Sections 1.2 and 1.3 have more detailed explanations of the individual parameters.

61.2 Additional settings in the "Test data" tab

Damage probability: Calculated average cycles to failure (50% damage probability) are recalculated to a selected damage probability according to the selected statistical method.

Merge temperature differences $\Delta\theta_{merge}$: As specified in VDI 2736-4 [14], at least 3 tests should be performed for every test condition (torque and speed). In theory, gears running under the same test conditions should have identical root and flank temperatures. However, this is often not the case in reality. These temperature deviations can be taken into account with $\Delta\theta_{merge}$.

	Torque	Speed	Root temperature	NL
1	1.00 Nm	750 rpm	100 °C	$1.01 \cdot 10^6$

2	1.05 Nm	750 min	105 °C	$1.13 \cdot 10^6$
3	1.00 Nm	750 rpm	120° C	$0.82 \cdot 10^6$
4	1.00 Nm	750 rpm	98 °C	$1.05 \cdot 10^6$

Table 61.1: Merging test results

If $\Delta\theta_{merge}$= 8°C, then tests 1, 2 and 4 from Table 1 will be merged together and used for further evaluation. Test 3 will be ignored.

Torque merge deviation: If the torque (or stress) deviation is lower than the set torque merge deviation (in %), then these measurements are used for further evaluation. Example from Table 1: Tests 1, 2 and 4 are combined if the merge deviation is 5%. If the Merge deviation is smaller, then just tests 1 and 4 are merged together (test 3 is not taken into consideration because of the $\Delta\theta_{merge}$ condition).

Temperature difference for grouping $\Delta\theta_{group}$: After merging, the results must also be grouped according to the root or flank temperature so they can be written to a material .dat file. With $\Delta\theta_{group}$, it is possible to group results even though the measured temperatures are not identical. If $\Delta\theta_{group}$ = 5°C, then tests 1, 2 and 3 from Table 2 will be grouped together and written at the same temperature of 20°C (the minimum temperature is used always as a reference temperature).

	Stress	Root temperature	NL
1	100 MPa	20°C	$1.01 \cdot 10^6$
2	80 MPa	22°C	$2.13 \cdot 10^6$
3	60 MPa	25°C	$3.82 \cdot 10^6$
4	40 MPa	98°C	$5.14 \cdot 10^5$
5	30 MPa	102°C	$9.02 \cdot 10^5$

Table 61.2: Grouping results in a .dat file

The following **statistical methods** are available for calculating the cycles to failure (the calculation is based on the user-defined damage probability):

- Statistical method acc. to VDI 2736-4
- Normal distribution
- Weibull distribution (2 parameter)

Statistical method acc. to VDI 2736-4
The procedure specified in VDI 2736-4 [14] (load cycle) to failure is used for the statistical calculation of the cycles. The only difference is that a default damage probability of 10% is used in the VDI standard. In contrast, in KISSsoft, the cycles to failure can be calculated with a user-defined damage probability.

Normal distribution
A normal (Gaussian) distribution can be used to calculate the number of cycles to failure. You will find more detailed information about this method in the technical literature [109].

Weibull distribution (2 parameters)
A 2-parameter Weibull distribution can be used to calculate the number of cycles to failure. You will find more detailed information about the method applied here in [110]. The relative least squares method is used to calculate the Weibull parameter.

Gear data measured for dry materials

Click on the checkbox to specify whether test gears are to be measured in their dry state. The calculation will then take into account the relevant material properties for calculating tooth flank stress.

61.3 Module specific settings

Minimum number of data points for merging:
The user can limit the minimum number of tests needed for merging. The minimum possible number is 2, however VDI 2736 requires at least 3 measurements.

Minimum number of data points for grouping
Use this setting to specify the minimum number of merged points needed to group results in a .dat material file. The minimum possible number is 2.

Permitted deviation of cycles to failure:
For each testing condition (torque and temperature), at least 3 tests should be performed to get a statistical distribution of cycles to failure. A warning message is displayed if the standard deviation of cycles to failure, divided by the mean value of cycles to failure, exceeds the set value.

Permitted speed deviation:
A warning message is displayed if the standard deviation for speed divided by the mean value for speed exceeds the set value.
Stress correction coefficient of reference test gear (Y_{ST}):
Calculated tooth root stresses are normalized by the stress correction factor and then recorded in a .dat material file.

$$\sigma_{F,DAT} = \frac{\sigma_{F,CALC}}{Y_{ST} \cdot S_F}$$

Root safety (S_{FMIN}):
Calculated tooth root stresses are divided by the root safety factor and then written to a .dat file.

Flank safety (S_{HMIN}):
Calculated tooth flank stresses are divided by the flank safety factor and then written to a .dat file.

Power-on time, Housing heat-transfer resistance, Housing heat-dissipating surface:
The set values are used to calculate gear heat transfer coefficients based on the measured root and flank temperatures. For more information see VDI 2736-2 [14].

Pulsator test results

Select this option to evaluate tooth root endurance data measured by the pulsator machine.

Test gear geometry based on one file

Here, you can select either one of two different test result types (see chapter 61.1, Gear test results).

Calculate wear factors from measured gear wear

If you select this option, the wear coefficients are calculated on the basis of the average wear as defined in the Test data tab.

Display permissible root/flank stresses in LOG scale

The tooth root/flank stress can be displayed either in a LOG-LOG scaling or in a LOG-LIN scaling.

61.4 Extrapolation of the calculated permissible root and/or flank stresses

The calculated results can be extrapolated if root and/or flank fatigue results are available. Do this in the Data extrapolation tab. In the first step, you can extrapolate data up to the user-defined number of cycles (Extrapolate to cycles). This extrapolation can be performed with an average pitch or 0.5·average pitch. In steps 2 and 3, you can define which step is to be performed according to the user-defined load cycles. These options are available in the first step:

- Set permissible stress to 0
- Set the number of cycles to infinite (for a specific permitted stress)
- Extrapolate load cycles with varying pitch.

Individual extrapolation options are available for root, flank oil, flank grease and flank dry results. The effect of data extrapolation can be seen in Graphics under S-N curve (Woehler line).

Select the Extend temperature range option to extend the calculated results to lower temperatures. Set the parameters for the minimum temperature (must be lower than the lowest calculated temperature in the "Test gear measurements" table) and the factor for increasing permissible stress (the factor must be ≥ 1). The results are displayed in the S-N curve (Woehler line) graphics or in the generated .dat file.

61.5 Other calculation options

If test results are available, the wear factor k_w can be calculated from the worn tooth geometry as defined in VDI 2736-2. This option must be enabled in Module specific settings (select Calculate wear factors from measured gear wear). An additional field, the "Average wear" field, is then displayed in the measurement table. To calculate the coefficient k_w, select "wear" as the cause of failure. The calculated wear coefficients are displayed in the report. The wear coefficients are not written to the .dat file.

If lubrication is either grease or dry running, the heat transfer coefficients k_θ for the root and flank of the failed gear are automatically calculated according to VDI 2736-2. The calculated coefficients are also shown in the report.

61.6 Writing material data to the KISSsoft database

In the Material DAT file tab, you can generate and open a .dat data file, and automatically save material data to a KISSsoft database. The permissible root or flank stresses are calculated automatically according to VDI 2545 (YF, methods B and C) and according to VDI 2736 (YF, methods B and C). All permissible stresses are automatically written to the .DAT file.

61.7 Graphics

The following graphics are available, based on the material inputs and calculated results.

- Young's modulus versus temperature
- Tensile strength and/or yield point versus temperature
- Wear coefficient versus temperature
- S-N curves (Woehler lines) for root and flank (all lubrication types)

61.8 Importing material files from M-Base

Material datasets from the M-Base Material Data Center can also be processed. If a dataset is available, click on Calculation > Import material data to import it into KISSsoft and process and convert it. Material datasets from the M-Base Material Data Center contain general information about the relevant plastics (Young's modulus, tensile stresses etc.). However, these datasets do not include information about infinite life strengths.

X

KISSsys

62 KISSsys: Calculation Systems

62.1 General

KISSsys is an extension to the KISSsoft calculation program. With KISSsoft, you can size, optimize and recalculate individual shafts, gears or shaft-hub connections. In contrast, KISSsys is suitable for administering machine element systems.

Some special links between different calculations are already present in KISSsoft. For example, bearing forces can be transferred from the shaft calculation and gears can be placed on a shaft. However, in the case of larger systems, such as a multi-stage gearbox with several shafts and gears, power and speed data must be entered separately for each stage. If several loads are to be calculated, the load must be updated in each calculation.

In contrast to KISSsoft, where the individual calculations take center stage, KISSsys provides a way of observing a system as a whole. KISSsys has not been designed to replace KISSsoft. Instead, it is an extension that uses the tried and tested calculation modules that are already present in KISSsoft. Fundamentally: KISSsys administers the relationships between individual elements in a model but leaves the calculation of the individual elements to KISSsoft.

62.1.1 Structure of KISSsys

KISSsys is based on an object management system called Classcad. Classcad administers KISSsys elements, evaluates the expressions for variables and provides an interpreter with which the user can also generate functions for special purposes.

This forms the basis for a user interface and a link to KISSsoft. The user interface functionality is different depending on the selected mode. In administrator mode, the user can generate new systems or change system structures. In user mode, it is only possible to change data, perform calculations and check results in an already existing structure. It takes more effort, and a better understanding of the program structure, to generate new systems than to use an existing system, which is easy to do.

62.1.2 Ways in which KISSsys can be used

At the most basic level, you can use KISSsys to group calculations. All the calculations in a system can be called up from one interface. You can also display an overview of the most important results of all calculations. This makes it immediately obvious which particular gear pair or shaft is critical.

Even just this view of all the calculations that are of interest makes work considerably easier.

Using KISSsys, you can then to specify relationships between variables. For example, you can calculate the speeds in a gear unit from the initial speeds and the transmission ratios. KISSsys can also describe the power flow. In KISSsys, you therefore only need to enter the load for the calculations in a few places. This enables you to quickly recalculate a complex system for varying load cases.

KISSsys enables you to store tables for loading cases or even variants. This means you do not have to constantly reenter the load data. KISSsys can also store the data for variants of a construction. You can then perform all the calculations for a selected load or variant with one click of a button.

For example, imagine a shaft with a radial force of unknown direction (e.g. via a belt drive/ belt force, whose direction is only determined when the equipment is installed). If it is necessary to define the worst case scenario, you could use KISSsys to rotate this force in steps of up to 360°.

KISSsys is not only extremely useful in the design stage, but is also useful in the sales environment. With KISSsys, you can, for example, store a standard gear unit in your computer. If the client later requests results for different loads on a similar gear unit, KISSsys can be used to quickly check whether the gear unit will meet the new load requirements.

Different example applications are available on the KISSsoft CD or on the website.

62.1.3 The user interface

The user interface provides several views of the administered data. There are table views, which are primarily designed to provide you with a good overview of the calculations. Another view shows the hierarchy for a tree structure. The additional two-dimensional power flow diagram is primarily designed to display the system's kinematic coupling. You can also create a three-dimensional display of the entire system or of particular subsystems.

This section details the functionality available to a user, in KISSsys, if they do not have administrator rights.

62.1.3.1 Tree view

The tree view lists all the elements present in the system, hierarchically. Use this option to display an assembly structure. A bitmap image is displayed next to each element's name. This image identifies the element type. Bitmaps with a blue background are KISSsoft calculations, and bitmaps with a gray background represent machine elements or connections. Right-click to open a separate context menu which has functions for an element.

Each element has a Properties dialog which you can display at this point. The Properties dialog has an overview of the available data elements or variables. However, these can only be changed by the administrator.

In the KISSsoft calculations, you can select kSoftInterface in the context menu to start the appropriate KISSsoft module. The operating data can then be changed or evaluated in the relevant KISSsoft module. Select kSoftReport to display the calculation report. Select Calculate to perform the calculation in the background without opening the KISSsoft interface. Data is only exchanged with KISSsoft via the blue calculation elements in KISSsys.

62.1.3.2 Diagram view

Diagram view shows the kinematic coupling of the elements. To start with, the element structure has nothing to do with the calculations. The calculations only use the data that relates to the shafts, gears and connections. These can be added or deleted as required.

The structure consists of shafts and their sub-elements: gears, forces, couplings and bearings. The kinematic coupling and the power flow between the shafts is achieved via connections. The connection has the calculation standard for transferring the speed and the torque to the next element. The torque can also display a loss of efficiency.

The externally supplied torque and speed are defined with the boundary condition elements. In each case, you can specify whether the speed or torque are known, or whether they should be calculated by KISSsys. The number of predefined values must correspond to the number of degrees of freedom.

Use the left-hand mouse button to move elements within the diagram view. Right-click to display a context menu, like the one in the tree view. You can change the zoom factor by clicking on the '+' or '-' buttons, or in the context menu which you access with a right-click.

62.1.3.3 Table view

To display the tables, select the Show function in the context menu in tree view. The contents of the tables are defined when the system is set up. Although the values displayed in black cannot be changed, you can edit numbers or texts shown in red. A special table for user interfaces has fields with a gray background. Start these functions by double-clicking on them with the left-hand mouse button.

You can print the contents of the table, or press Ctrl-C to copy it and then, for example, paste it into a spreadsheet.

62.1.3.4 3D view

To display the windows for the 3D view, select the Show function in the context menu in tree view. Use the left-hand mouse button to rotate the view, use the right-hand mouse button to move the view, and use the center mouse button to enlarge or reduce the size of the view. One of the main views can be selected from the menu or from the tool bars.

In the 3D view, you can use the context menu to export the 3D geometry to the CAD system. If a 3D kernel is present and you want to generate solid elements, a STEP file will be generated directly.

62.1.3.5 Message output

There is an output window for messages in the lower part of the program window. Error messages and warnings from KISSsoft calculations are displayed under Messages. Calls by KISSsoft are reported under KISSsoft, so this view is usually not required.

In the lower program window, as in KISSsoft, you will see an information tab. If information about a particular function is present, it is displayed in this tab.

62.1.3.6 Menus, context menus and the tool bar

Use the File main menu to open, store or save models. Also use it to open and close projects, or close and exit KISSsys. You can also open or close KISSsys templates.

Each individual docking window in the user interface can be hidden or displayed by selecting the appropriate option in the **View** main menu. You can also refresh all views.

System main menu options enable you to generate the KISSsoft report and run kinematics and KISSsoft calculations. You can also select a load spectrum for the model from the KISSsoft database or define your own spectra in KISSsys. Use these functions to call the properties of the elements and variables overviews.

Use Insert menu options to call an Assistant, which will guide you through the steps required to create a simple gear unit or a single stage planetary gear unit. In the same way as with the Assistant, you can use the Elements box to add elements to a model, but without any restrictions. The Group box contains predefined gear stages which can also be used to create a model. Default templates are used to add existing calculation templates to a finished model.

In the Extras menu, you will find the administrator settings, the license tool, the configuration tool and the language setting. Select Extras -> Settings to change general program settings such as the names of individual elements or table settings.

In accordance with Windows conventions, you will find the **Help** icon at the end of the menu bar. You can use it to navigate in the KISSsoft manual and the KISSsys programming help. Select Help -> Info to display information about the program version and the support provided by KISSsoft.

In the Main menu window, you will find options for organizing the opened sub-windows, such as tables and 3D views. The print option is only enabled if a table is open.

In addition to the main menu, KISSsoft uses context menus in many locations. Use context menus to access actions for a particular area or model element. Normally, you click the right-hand mouse button to display context menus.

The tool bar gives you faster access to actions from the menus that are used especially frequently. You should also read the tool tips: they display information about the actions in the tool bar and also the more detailed explanations in the status bar.

62.2 Creating Models in KISSsys

This chapter is intended for KISSsys users.

You can create new models in KISSsys in four different ways. These are described in the next four sections.

62.2.1 Classic method

If you select KISSsys Extras -> Administrator, and enable the Administrator, the system displays all the elements required to create a model, in the Template tab. To create a model, copy the elements you want from the template and insert them in the navigation tree. You can also create elements by moving them from the template to the navigation tree. All possible models can be created using this classical method.

62.2.2 Element Assistant

The Elements box, which you call by selecting Insert -> Elements, is based on the classical method. If you use this assistant, you no longer have to drag and drop or cut and paste data. To insert an element, simply click on it. The system then automatically inserts it at the previously selected level in the navigation tree. You can use the Elements box to create all possible kinds of variants.

62.2.3 System Assistant

When you create gear units, you can select Insert -> Assistant in KISSsys, to display an Assistant for creating the model. This Assistant leads you through the model step by step when you are creating it. You use it in the same way as the Elements Box.

Using the **Parallel Shafts** Assistant, you can create the following gear unit combinations in KISSsys:

- Cylindrical gear
- Bevel gear
- Worm wheel
- Face gear

Use the **Planetary Gear Stage** Assistant to create a single stage planetary gear unit.

62.2.4 Modeling using icons

The default setting is that all the elements are listed on the left-hand and right side of the screen as icons, so you can build up a model from the very beginning.

62.2.5 Create and modify tables

You can use a predefined table, called **UserInterface**, to create your own tables in a KISSsys Model. This table is stored in **Tables**, in the template. Using this "UserInterface" table, you can add all parameters from the elements and your own texts. You can change the table's name to suit your needs.

For each cell, no matter what it contains, you can right-click with the mouse to select Format in the context menu. There, you can modify the font, color, background color, and text position in that cell.

62.2.6 Adding variables to tables

One way of inserting variables in the UserInterface table is to select a variable from the Properties dialog and then click on the appropriate icon in the Toolbar menu. You can then insert the variables either as text (Name), as a reference, or as an expression.

Variable as text

You can either enter texts directly in the cell or click on the Text icon to insert them. To do so, you must first select the variable and preselect the cell in the table in an element's Properties window. Then click on the icon to transfer the value you want into the cell. You can also hold down the left-hand mouse button and drag and drop the required parameter, to insert it directly into the cell. The default setting is that the value set in the variable is inserted as an expression. Click on Extras -> Settings and then the Tables tab to define the default setting as a text, reference or expression.

Variable value as a reference

Referenced data is displayed in red. These values reference another variable of the same element, or a different element, in the Properties window. You can either change this value in the table or in the window.

You can add referenced values to cells in the same way as you add texts. To do so, you can select the parameter you want to insert, in an element's property. After this, the cell should be preselected in the table. Finally, click on the Reference icon. This transfers the required value to the cell. Alternatively, you can hold down the right-hand mouse button and insert the parameter in the appropriate cell. A selection window is displayed, in which you can select a text, reference or expression, as required.

Variable value as an expression

The expression is merely shown as a value and cannot be modified in the table. To insert an expression in the cell, use the same procedure as described in the previous method, **Variable value as a reference**.

62.2.7 Individual names for elements

Individual names can be used for all KISSsys elements. The individual name is assigned to the element automatically when it is inserted in the model. This behavior can set individually for each element (Extras -> Settings -> Elements).

Use the **<autoInc>** and **<localInc>** tags to add an index to the individual name at the insertion position. The first of these tags increments the index globally. This means that no other element in the model can have the same name. The second of these tags increments the index locally (in the same folder). Use the **<parentName>** tag to add the name of the hierarchically superior element.

The Automatically/Ask option is set to suppress or display the dialog that prompts the user to define the name for the new element for the model. Click on Reset to select the KISSsys default name. You can use the Ask all, Automatic all and Reset all functions to modify all the elements at the click of a button.

62.3 Extended functionality for developers

In addition to the functionality already described, more functions are available for developers.

- To open a template file, click on File -> Open Templates. Alternatively, click on Insert -> Default templates to load the template file. It is displayed as a tree under Templates.
- To add new elements in tree view, you can Copy and Paste them. The new elements are added as copies from a template file.
- You can use context menu functions to rename and delete elements.
- The data in the Properties dialog can be edited. New variables can be added and deleted
- Hidden variables will be displayed and all functions can be performed.
- Hide messages by selecting Extras -> Suppress messages.

62.3.1 The Properties dialog

In tree view, or in the diagram for an element, you can open the KISSsys Properties dialog via the context menu. There, it is possible to add new variables or change existing ones. Only one Properties dialog is available. A second one is not displayed.

The following fields are available for the variables:

- **Type:** Display the variable type (see chapter 62.4.1, Variables).
- **Name:** The name of the variable. You can change the name here. However, if a variable has to be used in formulae or references, you must also change the name there, as otherwise the variable cannot be found.
- **Reference:** Enter the reference target here for reference elements. A name must be entered in quotation marks. As an alternative, you could use the name of a string variable (see chapter 62.4.1.1, References). For variants (see chapter 62.4.1.2, Variants), enter the index in an array. The system marks an invalid reference in red.
- **Value:** The current value of the variable.
- **Expression:** An expression used for calculating the variable. If an expression is present, the value is calculated on the basis of that expression.
- **KISSsoft** → KISSsys option: The variable can be transferred from KISSsoft to KISSsys.
- **KISSsys** → KISSsoft option: The variable can be transferred from KISSsys to KISSsoft.

You can convert the variable into a reference or variant variable, or vice versa, by clicking on the Reference or Variant button.

62.3.2 Table view

The table format is defined in the hidden **definition** variable. There are different types of tables:

- **Table for calculations:** This table is best suited for displaying the data for several elements of the same type. The definition format is:

  [[typ,rows,columns],['variable1','variable2',..],
  [element1,element2,..]]

  In the case of type 1, you can edit each displayed value. In the case of type 2, you can edit all values that have no expression, and in the case of type 3 you can edit all values for which the KISSsys → KISSsoft flag has been set. The Number of Rows or Columns is not used.
- **Table for arrays or variants**: In this table, the arrays or variant variables are each displayed in a separate column. The definition format is:

  [[typ,rows,columns],['variable1','variable2',..]]

  In the case of type 21, you can edit each displayed value. In the case of type 22, you can edit all values that have no expression, and in the case of type 23, you can edit all values for which the KISSsys → KISSsoft option is activated. The Number of Columns is not used.

- **Table for user interface:** You can configure this table to suit your needs. The definition is [[type,rows,columns],[[A1,B1],[A2,B2]]]. The contents can be inserted via a context menu in the table, and should not be changed in the definition. Since the definition is changed interactively, you must not set an expression here. The number of rows or columns should also only be changed in a dialog. Otherwise information about reference elements will be lost.

62.4 The following elements are available

62.4.1 Variables

The following variables can be used:

- **Real**: A numerical value.
- **String**: A character string. Input in quotation marks e.g. "Text".
- **Point**: A coordinate or vector with 3 components. Input in {1,2,3} format.
- **Array**: A one-dimensional or multidimensional field. Input as e.g ["text",1.23,{1,2,3},[1,2]].
- **Function**: An executable function. Input best entered via the special input screen.
- **Element ID**: The ID of a Classcad object. Output as $31, input as name of the object with no quotation marks.
- **List**: Displayed as a selection list and acts as a number in the Interpreter (list index, beginning with 0). The selection list is defined as an array via the Edit list menu option, e.g. ["one","two","three"].
- **Database List**: The name from the KISSsoft database is displayed in a selection list. In the Interpreter, this type also acts as a number according to the database ID. To define the database assignment as an array, select the Edit the list menu option: ["database","table"]

Each of the variables has a name, a value, an expression and different flags. If an expression is present, the value of the variables is defined via this expression. The expression is therefore particularly well suited for inputting formulae. If, in contrast, a formula is entered in place of the value, this formula will be evaluated and the result will be assigned. The actual formula will be lost. The KISSsoft -> KISSsys and KISSsys-> KISSsoft options determine how data is exchanged between the two programs. Only variables with the appropriate option activated are exchanged.

There are additional reference elements and variant elements for the Real, String, Point, List and Database List data types.

62.4.1.1 References

A reference element behaves like any other variable, with the difference that another variable fetches the data. A valid variable name must be entered as the target for the reference element. The reference target must be entered as a character string. This will be either an actual name in quotation marks or an expression resulting in a character string, e.g. a concatenation of character strings (e.g. gear1+'.z' with the string variables gear1 or 'gearwheel1.z'). The system marks an invalid reference in red.

62.4.1.2 Variants

Internally, the variant elements administer a field of variables, whereas externally they behave like a normal variable. The variant is assigned an index variable as additional data. This index variable is used to index the field. The index variable must be entered as an array of variables (e.g.[system.index]). You can use these data types to store load spectra or system variants. The results can then be displayed in tables.

62.4.2 Calculation elements

All the elements for KISSsoft calculations are derived from classes which begin with the name kSoft. They have a blue background in the tree view.

The calculation elements have a range of functions:

- **Calculate:** performs a KISSsoft calculation in the background.
- **kSoftInterface:** starts KISSsoft interactively.
- **kSoftReport:** runs the calculation and displays the report.
- **SetFlags**: sets the flags for data exchange between KISSsoft and KISSsys to suit the required storage location.
  - KISSsoft -> KISSsys and KISSsys -> KISSsoft: the data is transferred in both directions.
  - Only KISSsoft -> KISSsys: data with a stored expression is transferred from KISSsys to KISSsoft. All other data is only transferred in the other direction.

This function sets the flags only once, when it is selected. It therefore has no effect on later changes.

- **kSoftModul**: This hidden function displays the KISSsoft module name
- **getTranslationTable**: this hidden function shows the translation table for variable names from KISSsys to KISSsoft. In the calculation element, the translation table can be extended via the TranslationTable array: For example, the entry [['eps_a_min','ZP[0].Eps.aEffI'],['eps_a_max','ZP[0].Eps.aEffE'] adds a link between the

variables eps_a_min and eps_a_max and the relevant KISSsoft variables. Up to now, the names of the KISSsoft variables could only be taken from the report templates, which are .rpt files.

- **getUtilization**: this function returns the utilization, and the required safety/safety ratio.

In the fileName variable, you can specify a KISSsoft calculation file which will automatically be loaded at the start of the calculation, before any other variables are transferred. You can use the savingMode variables to specify whether this KISSsoft calculation file should be saved automatically:

- **Don't ask and don't save:** When KISSsoft is shut down, you will not be prompted to save the file if changes have been made to it.
- **Ask for saving:** When KISSsoft is closed, you will be prompted to save the file.
- **Save automatically:** When KISSsoft is closed, the calculation file is saved automatically without a user confirmation prompt.
- **Save file in KISSsys:** No file name will be entered in the fileName variable. Instead, the entire calculation file will be saved in the KISSsys element.

The shaft calculation includes the special **UpdateShaftElements** method. Use this method if a force element has to be added/deleted on a shaft. It evaluates the number of force elements on the shaft, and their type, and transfers them into the Forces array in the shaft calculation.

62.4.2.1 Relationship of calculations with elements

Templates which automatically link the calculation with the shafts and gears are provided. The Dialog function is designed for this purpose. If you want to make a fundamental change, i.e. if more elements of forces are added to the shaft, you must call this dialog again to update the relationships.

62.4.2.2 Importing existing KISSsoft calculations

If KISSsoft calculations are already present for the elements in a new KISSsys system, you can simply load the files in the KISSsoft window. Click on File -> Open to do this. However, you should note a few points:

- The file name stored in the fileName variable, in the KISSsys calculation element, will be changed. The name must either be deleted or modified.
- The elements of forces and the bearings are overwritten during the shaft calculation. For this reason, you need to call the Dialog or UpdateShaftElements function after importing the calculation. You cannot import force elements and bearings that are not present in the KISSsys model tree structure. Also note that the names of the force elements and bearings in the loaded shaft calculation must be identical to the names used in the model.

- In the case of gears, you must ensure that the sequence of the gears (gear 1 and gear 2) matches the definition in the model.

62.4.3 Elements for shafts

Different elements can be placed onto shafts. They will also be transferred to the KISSsoft shaft calculation. The position on the shaft is defined with the position variable.

- **kSysHelicalGear**: A cylindrical gear.
- **kSysBevelGear**: A bevel gear. The position of the peak is defined by the direction variable.
- **kSysWorm**: A worm.
- **kSysWormGear**: A worm wheel.
- **kSysCoupling**: A coupling. Diameter d and Width b can be entered for the 3D display.
- **kSysBearing**: A normal type of bearing. Losses can be recorded in Tloss. The direction of the loss torque should be defined with a -sign(speed) in the expression.
- **kSysRollerBearing**: A rolling bearing. The bearing geometry is imported from the KISSsoft bearings database every time you refresh the calculation. Losses can be recorded in Tloss. The direction of the loss torque should be defined with a -sign(speed) in the expression.
- **kSysCentricalLoad**: A centrical load.
- **kSysMass**: An additional mass on the shaft.
- **kSysRopeSheave**: A pulley (rope sheave). The belt force is calculated via the connection.
- **kSysFaceGear:** A face gear.

62.4.4 Connection elements

- **kSysGearPairConstraint**: a connection between two cylindrical or bevel gears.
- **kSysPlanetaryGearPairConstraint**: a connection between a gear and a planet. You can select the type of pairing: sun-planet, planet-internal gear or planet-planet. Both gears must also be entered in this sequence. A planet carrier must also be selected. The number of planets must be defined either in the same dialog or in the planet carrier coupling NofPlanets variable.
- **kSysPlanetaryBevelGearConstraint**: a connection between a bevel gear and a rotating bevel gear for bevel gear differentials. As in the case of the planetary connection, the sequence of the bevel gears and the number of planets must be defined. You can specify the efficiency of this stage in the same dialog.
- **kSysWormGearConstraint**: a connection between a worm and worm wheel.

- **kSysCouplingConstraint**: A connection with ratio 1 between two couplings. The coupling's kinematic force can be selected or deselected. You can also specify a value for slip, e.g. for multi disc couplings or synchronizations. The torque in the connection is usually calculated, but can also be specified.
- **kSysBeltConstraint**: A connection between belt sheaves. The transmission ratio is calculated from the diameter ratio. You can also enter a value for slip.
- **kSysConnectionBearing:** A connection between two shafts.
- **kSysConnectionRollerBearing:** A connecting rolling bearing between two shafts.

Use the setConfig(slipConstraint_r/[slipConstraint_r, slip_r], torqueConstraint_r/[torqueConstraint_r, torque_r]) function to activate or deactivate the connection:

- 1. Closed, without slip: setConfig([TRUE, 0], FALSE),
- 2. Open, without torque: setConfig(FALSE, FALSE),
- 3. Open, with torque: setConfig(FALSE, [TRUE, 20]),
- **kSysSpeedOrForce**: An element for specifying speed or torque. Both values can either be specified or calculated. You can also preset the power as an alternative to the torque.

Use the setConfig(speedConstraint_r, torqueConstraint_r/[torqueConstraint_r, type_r, torque_r]) function to change the preset values. If you specify a load type, the values shown below have these meanings: 0. Torque with prefix operator, 1. Torque, driving, 2. Torque, driven, 3. Power, driving, 4. Power, driven. Examples:

- 1. Speed and torque specified: setConfig(TRUE, TRUE),
- 2. Speed and torque with value specified: setConfig(TRUE, [TRUE, 0, 20]),
- 3. Only power driving predefined: setConfig(FALSE, [TRUE, 3, 20])

62.4.5 Displaying elements in a 3D graphic

Each element has an OnRefresh3DView function which generates the 3D display. If necessary, this function can be overwritten. You can set the color of an element in the range from 0 to 255, with the kSys_3DColor variable, and set the transparency with the kSys_3DTransparency variable. These two variables must be created if necessary.

62.4.6 System settings

You can use any of the setting options in the System element:

- **kSoftAcceptChanges:** default setting yes: the changes will be applied from KISSsoft. If the setting is no, nothing will be applied. If the setting is asked, you will be prompted to confirm whether the changes should be applied when KISSsoft is closed.
- **kSysKinematicFunc:** you can call up the OnCalcTorque function while the kinematics is being calculated. The standard implementation of this function calls up the calculation of the bearing actions for all shafts.
- **kSysKinematicMode:** kinematics can either be calculated iteratively or non-iteratively. Iterations for the torque must be selected if you want to include the efficiencies. Iterations for speeds are only necessary if formulae for speeds have been entered.
- **kSys3DElements:** you can display either graphical elements or solid elements (3D kernel required). Graphical elements are generated more quickly, and solid elements are more detailed. You can also display an imported housing in this module.
- **project_name:** the project name is displayed in the KISSsoft calculation reports.
- **project_contract:** the commission number is displayed in the KISSsoft calculation reports.

62.5 Programming in the Interpreter

The expressions used in variables and in functions have programming options.

62.5.1 Expressions in variables

The programming options in expressions are restricted. No local variables can be used.

The following operators are defined between the data types, and a variety of different mathematical functions are also available:

Data type	Operations	Description
Real	+,-	Addition and subtraction
	*,/	Multiplication and division
	<,>=,=,!=,>=,>	Relational operators
	!,AND,OR	Logical operators
String	+,LEN	Concatenation and length operators
	<,>=,=,!=,>=,>,!	Relational operators
Point	+,-	Addition and subtraction
	*,**	Scalar and vector multiplication

	:x,:y,:z	Access to components
	LEN	Vector length
Array	[],+,LEN	Indexing, concatenation and length operator

Table 62.1: Permitted operators for data types

abs(x)	Supplies the value of x
sign(x)	Supplies the prefix operator of x (+1, -1 or 0 if x=0)
min(a,b,...)	Supplies the smallest value of the arguments
max(a,b,...)	System supplies the largest value of the arguments
a_r(x)	System converts from degrees to radian measure
r_a(x)	System converts from curve to degrees
sin(x)	System calculates sin of x in the radian measure
sinh(x)	System calculates sinh of x in the radian measure
asin(x)	System calculates arcsin of x in the radian measure
cos(x)	System calculates cos of x in the radian measure
cosh(x)	System calculates cosh of x in the radian measure
acos(x)	System calculates arccos of x in the radian measure
tan(x)	System calculates tan of x in the radian measure
tanh(x)	System calculates tanh of x in the radian measure
atan(x)	System calculates arctan of x in the radian measure
atan(y,x)	System calculates arctan of x in the radian measure
exp(x)	System calculates e to the power of x
ln(x)	System calculates the natural logarithm of x
log(x)	System calculates the decadic logarithm of x calculates
sqrt(x)	System calculates square root of x
pow(x,y)	System calculates x to the power of y
fmod(x,y)	System calculates x modulo y

Table 62.2: Predefined mathematical functions

A variable's expression can include the specified operations and any function calls. If limited expressions are to be used, the expression must begin with # and the result must be returned with RETURN:

```
#
IF a>b THEN
RETURN a;
ELSE
RETURN b;
ENDIF
```

62.5.2 Functions

The different options for programming in functions are best described with the help of examples. A function's header looks like this:

```
//Übergebene Variablen vom aufrufenden Programm

PAR Parameter1,Parameter2;

//Deklaration von Konstanten

CONST PI=3.1415926,E=2.71828;

//Deklaration von lokalen Variablen

VAR a,b,c,d;
```

Here, the lines that begin with // are comments. Each of these three lines must only occur once, and the declared variables must be separated with a comma. A non-initialized parameter or variable is VOID. This can be checked with ISVOID(variable).

Limited statements have two variants: IF or SWITCH statements:

```
//IF-Anweisung mit optionalem ELSIF und ELSE Block

IF Parameter1>5 THEN

a=sin(PI*Parameter1);

ELSIF Parameter1<0 THEN

a=Parameter1; ELSE

a=0;

ENDIF

//SWITCH Anweisung mit Auswahl über Zahlen oder Texte

SWITCH Parameter2
```

```
CASE 'Null':b=0;

CASE 'Eins':b=1;

DEFAULT :b=5;

ENDSWITCH
```

There are four program variants for loops:

```
//FOR Schleife mit optionaler Schrittweite

FOR a=1 TO 8 STEP 2 DO

b=b+a;

IF b>100 THEN

BREAK;//beendet die Schleife

ENDIF

NEXT

//WHILE Schleife

WHILE b<100 DO

b=b*10;

WHEND

//DO Schleife

DO

b=b*10;

UNTIL b>100;

//FORALL Schleife wird für alle Elemente eines Arrays ausgeführt

c=[1,2,3,4,5,6,7,8,9];

a=0;

FORALL c d DO//d bekommt jeweils den Wert eines Elementes von c

a=a+d;
```

```
NEXT
```

There is a special syntax for calling up functions that belong to objects. The standard method is to specify the object name followed by a point and the name of the function. However, the name of an object can also be contained in a local variable. This enables you to change the object for the function call at runtime.

```
//Die Funktion OBJ_GetMember wird für Objekt1 aufgerufen.

Objekt1.OBJ_GetMember('variablenname');

//a ist lokale Variable vom Typ String mit dem Namen eines Objektes

a='Objekt1';

//Es wird eine Servicefunktion für das Objekt mit Namen a aufgerufen

b=a.OBJ_GetMember('variablenname');

//ruft vom Benutzer angelegte Funktion für Objekt1 auf.

a.Benutzerfunktion();

//die vom Benutzer angelegte Funktion wird für das

//aktuelle Objekt aufgerufen.

Benutzerfunktion();

//die vom Benutzer angelegte Funktion wird für das

//übergeordnete Objekt aufgerufen.

^.Benutzerfunktion();
```

The system searches for variable names relative to the current object. If object.z is used in an expression, the system will first attempt to find this variable below the current object. If it is not present, the search will continue in the hierarchically superior object (in accordance with ^.object.z) and so on.

62.5.3 Important service functions

OBJ_GetChildren()	Supplies an array with all child objects.
OBJ_GetName()	Supplies the name of the object.
OBJ_GetId()	Supplies the ID of the object.
OBJ_HasMember()	Tests whether a variable is present.

OBJ_GetMember()	Supplies the variable of the current object.
OBJ_FindMember()	Supplies the variable of the current or hierarchically superior object.

Table 62.3: Important service functions

62.5.4 Variable dialogs

In interpreter functions, variable dialogs can be generated for the input of variables. The call is:

res = CADH_VarDialog(["Title", Width, Height, Pitch], [dialog element 1], [dialog element 2], etc.);

The title is displayed in the dialog's title line. The Width and Height values are the dimensions of the dialog in pixels. The division (a value between 0 and 1) describes the ratio between the width of the field description and the dialog width (default value 0.4). This definition of the dialog size can be followed by any number of arrays that each define the individual dialog elements.

The return value is an array whose first value is res[0]=1 if the dialog was closed by selecting OK. Otherwise, the first value is zero. The other elements of the returned array supply the results of the input fields.

The following convention is used to define the type of a variable: _str=String, _n=Int, _r=Real, _b=Bool. For example, in the case of Caption_str, this means that the Beschr variable is of the type String.

62.5.4.1 Dialog elements for the variable dialog

The following dialog elements are available for the variable dialogs:

Horizontal grouping:

The horizontal grouping provides a framework in which the individual dialog elements are lined up beside each other. Their position must always be defined by a vertical group, which means that all dialog elements in a horizontal grouping must be defined in a vertical group. A horizontal group is defined as follows:

[C:VDLG_HORZ, Caption_str, DistAbove_n, DistAfter_n, [Dialogelem]]

- C: VDLG_HORZ: type definition for horizontal grouping.
- Caption: the horizontal grouping's caption. If "Caption" is not an empty string, a frame will be drawn around the horizontal group.
- DistAbove: distance from the next dialog element, above the horizontal group.

- DistAfter: distance from the next dialog element, below the horizontal group. "DistAfter" and "DistAbove" are specified in pixels.
- [Dialogelem]: element array used to define the dialog elements located in the horizontal grouping. This array must only contain elements of the type VDLG_Vert.

Vertical grouping:

The vertical grouping provides a framework in which the individual dialog elements are lined up below each other. The width of the dialog elements is defined by the vertical group. A vertical group is defined as follows:

[C:VDLG_Vert,Caption_str,[XStart_r,XEnd_r],XPart_r,[Diag],Marg_n]

- C:VDLG_Vert: type definition for vertical grouping.
- Caption: the vertical grouping's caption. The vertical grouping always has a frame drawn around it.
- [XStart,XEnd]: XStart and XEnd define a factor (between 0 and 1) for the width of the vertical group relative to the width of the hierarchically superior dialog. They also define the X-position of the vertical group.
- XPart: a factor between 0 and 1 that defines the ratio between the prompted value and the input value for the dialog fields (the text assigned to an input field is called the "prompt"). If XPart=-1, the prompt is positioned above the dialog element.
- [Diag]: element array used to define the dialog elements in the vertical grouping.
- Marg: an optional parameter that defines the displacement of the dialog elements relative to the edge of the vertical group. This means that the dialog elements contain the "Marg" (margin) distance from both the left-hand and right-hand edge of the vertical group.

RealEditFeld:

Provides an edit box in which you can enter a floating comma number.

[C:VDLG_Real,Prompt_str,Preset_r,res,res,Places_n]

- C:VDLG_Real: RealEdit field type definition.
- Prompt: text assigned to the input field.
- Preset: Preset value.
- res: a space (place) has been reserved here for two optional parameters which are not currently in use. However, these places must not be left empty in the definition (e.g. [C:VDLG_Real,Prompt,Preset,0,0,Places] would be a correct solution but not [C:VDLG_Real,Prompt,Preset,,,Places]).

- Point: an optional parameter that defines the number of decimal places in the input field.
- Return value: The return value is the input string.

IntEditFeld:

Provides an edit box in which you can enter a whole number.

[C:VDLG_Int,Promt_str,Preset_n]

- C:VDLG_Int: IntEdit field type definition.
- Prompt: text assigned to the input field.
- Preset: Preset value.
- Return value: The return value is the input string.

StringEditFeld:

Provides an edit box in which you can enter text.

[C:VDLG_Str,Promt_str,Preset_str]

- C:VDLG_Str: StringEdit field type definition.
- Prompt: text assigned to the input field.
- Preset: Preset text.
- Return value: The return value is the input string.

Text display:

The system generates a text display. If an empty string is entered instead of text, the text field can also be used to define a space.

[C:VDLG_Prompt,Prompt_str,Fieldheight_n]

- C:VDLG_Prompt: Text display type definition.
- Prompt: Field text.
- Field height: height at which the text is displayed.

IntComboBox:

Provides a combo box in which you can enter a whole number.

[C:VDLG_IntCom,Prompt_str,[Entr_n],Sign_n/[Ind_n],0,0,AsVal_b]

- C:VDLG_IntCom: IntComboBox type definition.
- Prompt: text assigned to the combo box.
- [Eintr]: element array with the available list items (in the case of an IntComboBox, the components must be whole numbers).
- Sign/[Ind]: here, you have the option of using "Sign" to either set a constraint value, which is contained in the list, directly, or using "Ind" to select a value in a particular list position as a constraint value (the first element in the list is located at position 0)."Sign" and "[Ind]" are optional parameters.
- AsVal: if the optional "AsVal" parameter has been set, and does not equal 0, the return value becomes the input. Otherwise, the return value is the index of the selected entry.

IntEditComboBox:

Provides a editable combo box in which you can enter a whole number. Please ensure that the values entered here are whole numbers.

[C:VDLG_IntComE,Prompt_str,[Entr_n],Sign_n/[Ind_n]]

- see IntComboBox
- Return value: The return value is the input string.

RealComboBox:

Provides a combo box in which you can enter a floating comma number.

[C:VDLG_RealCom,Prompt_str,[Entr_r],Sign_r/[Ind_n],0,0,AsVal_b]

- see IntComboBox

RealEditComboBox:

Provides a editable combo box in which you can enter a floating comma number.

[C:VDLG_RealComE,Prompt_str,[Entr_r],Sign_r/[Ind_n]]

- see IntComboBox
- Return value: The return value is the input string.

StringComboBox:

Provides a combo box in which you can input a string.

[C:VDLG_StrCom,Prompt_str,[Entr_str],Sign_str/[Ind_n],AsPos_n]

- see IntComboBox
- AsPos: In contrast to the IntComboBox, the return value here represents the index of the selected field, if the optional parameter "AsPos" has been set and does not equal 0. Otherwise, the return value is the input.

StringEditComboBox:

Provides a editable combo box in which you can enter a string.

[C:VDLG_StrCom,Prompt_str,[Entr_str],Sign_str/[Ind_n]]

- see IntComboBox
- Return value: The return value is the input string.

62.5.4.2 Example application of a variable dialog

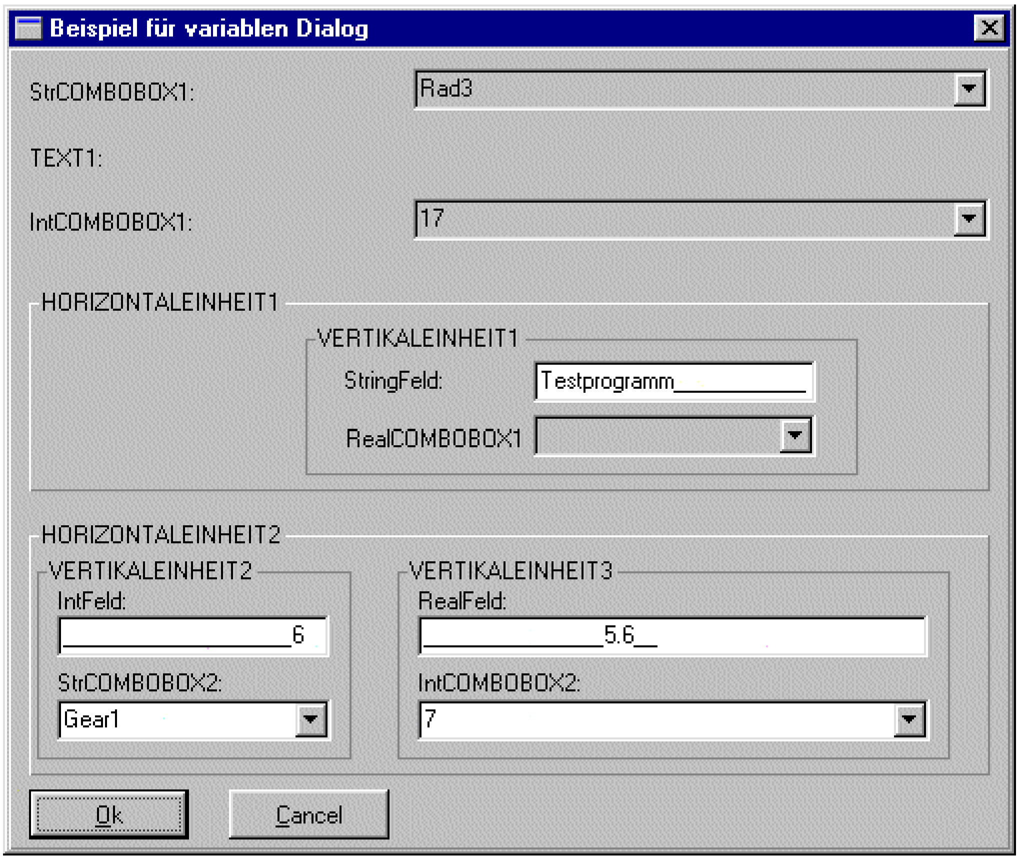


Figure 62.1: Example of a variable dialog

The program code for the variable dialog is shown in the example below. As many elements as possible have been used in it:

```
// VARIABLES DECLARATION
VAR res,result1,result2,result3,result4,result5,fullResult;
// DIALOG AND INPUT DATA
res = CADH_VarDialog(["Example of Variable Dialog",500,400,0.4],
[C:VDLG_StrCom,"StrCOMBOBOX1:",["Gear1","Gear2","Gear3"],[2],0],
[C:VDLG_Prompt,"TEXT1:",30],
[C:VDLG_IntCom,"IntCOMBOBOX1:",[12,17,19],17,0,0,1],
// HORIZONTAL GROUP WITH ONE VERTICALGROUP
[C:VDLG_HORZ,"HORIZONTAL UNIT1",20,10,
[ // Warning: remember brackets!
[C:VDLG_VERT,"VERTICAL UNIT1",[0.3,0.9],-1,
[
[C:VDLG_Str,"StringFld:","Test Program"],
[C:VDLG_RealComE,"RealCOMBOBOX1",[5.3,7.1,9.1],[2]]
],
20
]
] // Warning: remember brackets!
],
// HORIZONTAL GROUP WITH TWO VERTICAL UNITS GROUPS
[C:VDLG_HORZ,"HORIZONTAL UNIT2",10,10,
[
[C:VDLG_VERT,"VERTICAL UNIT2",[0.01,0.35],-1,
[
[C:VDLG_Int,"IntFld:",6],
```

```
[C:VDLG_StrComE,"StrCOMBOBOX2:",["Gear1","Gear2"],[0]]

],

10

],

[C:VDLG_VERT,"VERTICAL UNIT3",[0.4,1],-1,

[

[C:VDLG_Real,"RealFld:",5.6,0,0,3,3],

[C:VDLG_IntComE,"IntCOMBOBOX2:",[5,7,9],7]

]

]

]

]

);

// res [0] contains 1 if OK was pressed , or else

IF res[0] THEN

// READ RESULTS FROM DIALOG:

result1 = res[1]; //res [1]= Gear3

result2 = res[2]; //res[2]= TEXT1:

result3 = res[3]; //res[3]= 17

result4 = res[4]; //res [4]= [["Test Program",9.1]]

result5 = res[5]; //res[5]= [[6,"Gear1"],[5.6,7]]

fullResult=res; //res=["Gear3","TEXT1:",17,[[''Test
Program",9.1]],[[6,"Gear1"],[5.6,7]]]

CADH_Message(fullResult);

ENDIF
```

62.5.4.3 Interactions with variable dialogs

You can interact with variable dialogs. Changes in input fields, and selections in lists, can trigger callbacks to a user-defined function. Then, you can also change dialog elements from this callback routine.

To do this, set a local function as a callback via the title input in the variable dialog:

res = CADH_VarDialog([["Title",PROC(Callback)], Width, Height, Pitch], [dialog element 1], [dialog element 2], etc.);

The local callback function will now be called if you make changes to the dialog. The function is declared as follows:

```
PARres;

PROCCallback

PARhandle,elemNo,event,eventPar;

IFTYP(elemNo)=STRINGTHEN

IFelemNo='@combo'ANDevent=C:CBN_SELCHANGETHEN

IFeventPar=0THEN//owninput,enableinput

CADH_VarDialogAccess(handle,[['@input1',C:VDLG_ENABLE,TRUE]]);

ELSE//disableinput,setvaluetozero

CADH_VarDialogAccess(handle,[['@input1',C:VDLG_ENABLE,FALSE],

['@input1',c:VDLG_ASSIGN,0]]);

ENDIF

ENDIF

ENDIF

ENDPROC

res=CADH_VarDialog([['Titel',PROC(Callback)],400,4000.4],

[[C:VDLG_Real,'@input1'],'Input1:',2],

[[C:VDLG_StrCom,'@combo'],'Selection:',

['owninput','calculate'],[0],TRUE]);
```

A handle is transferred to the dialog as a code parameter, along with an element identifier, the event, and additional parameters. The possible events are:

Element type	**Event**	**Parameter**
Dialog	Initialization	none
	WM_INITDIALOG	
Combobox	Selection	Current value
	CBN_SELCHANGE	
Input field	Leave field	Current value
	WM_KILLFOCUS	
Button	selected	none
	BN_CLICKED	

Either the element number according to the index in the results array is transferred as the element number, or the element name is transferred. As in the example, a name can be defined by transferring an array, with a type and name in its first element, for the dialog element.

Use this function to access the dialog from the callback routine:

CADH_VarDialogAccess(handle, [[elemNo, action, param],[elemNo, action, param],...]

The following actions are permitted here:

Action	**Description**	**Parameter**
VDLG_ASSIGN	Assignment to input field	New value
VDLG_SELECT	Selection in combo box	[position]/value
VDLG_ENABLE	Select or deselect	TRUE/FALSE
VDLG_SETFOCUS	Focus on new element	Element's ID

If no action is specified, the value in the input field is returned. The data is returned as an array, with as many elements as code parameters.

62.5.5 Defining 2D graphics

In KISSsys, you can generate two-dimensional graphics for displaying results which are present in arrays. You can store a graphic definition in the data variable expression, in the kSys2DPlot graphical element. Bar and line graphics can be displayed in at the same time. The definition of the graphic consists of three parts:

- Axis system (1- or 2-axis systems can be defined)
- XY line graphic
- Bar chart

Each of these parts is described in more detail below.

62.5.5.1 Axis system definition

At least one axis system must be defined. The second one is optional. The definition for the axis system is as follows:

```
[ | Xaxisname_str , | min_x_r , | max_x_r ] , [ | Yaxisname_str , |
min_y_r , | max_y_r ] , [ axiscolour_str/array , | axiscross_x_r ,
axiscross_y_r ] , [ | scaleinterval_x_r , | scaleinterval_y_r , [ |
exponential_x_n , | exponential_y_n ]
```

where:

- **XAxisname**: X-axis name.
- **YAxisname**: Y-axis name.
- **min**: minimum value of the axis (optional).
- **max**: maximum value of the axis (optional).
- **axiscolour**: the axis color is defined in a string (red, green, blue, yellow, white, gray, cyan, brown, magenta, purple, black) or as an array [r_n, g_n, b_n] (where "r", "g" and "b" represent the red, green and blue color values from 0 to 255 (optional).
- **axiscross**: the intersection point of the axes (optional).
- **scaleinterval**: increment of the axis scaling.
- **exponential**: if 1 is input, the axis will be logarithmically subdivided.

62.5.5.2 XY line graphic definition

The following information is required for an XY-line graphic:

```
grouptype_n , [ dataarray_x_r ] , [ dataarray_y_r ] , [ | linename_str , |
|linecolour_str/array , | linestyle_n ] , | assignaxis_n
```

where:

- **grouptype**: = 1 (for line graphic).
- **dataarray**: contains the data's X- or Y-coordinates.
- **linename**: name of the element.
- **linecolour**: line color.

- **linestyle**: line type (0: continuous, 1: interrupted, 2: dashed, 3: semicolon, 4: dash dot dot).
- **assignaxis**: number 1 or 2 of the coordinates system.

62.5.5.3 Bar chart definition

A group of data is defined as follows for a bar chart:

```
grouptype_n , [ dataarray_1_r , ... , |dataarray_n_r ] , [
barcolour_str/array ] , | bargroupname_1_str , [ | barelementlabel_1_str ,
... , barelementlabel_1_str ] , | barclass_n
```

where:

- **grouptype**: grouptype: = 2 (for bar chart).
- **dataarray**: contains the data for the group.
- **barcolour**: color of the group's bars.
- **bargroupname**: name of the group.
- **barelementlabel**: names for individual elements.
- **barclass**: display as group (=0) or sorted by elements (=1).

62.5.5.4 Entire definition

The entire definition must begin with the definition of the axis system. After this, you can list any number of definitions for line and bar charts. Each part definition must be enclosed in square brackets, just like the entire definition:

```
[ [af_1] , | [ af_2] , | [dg_l_1] , ..., | [ dg_l_ n1 ] , | [ dg_b_ 1 ] ,
..., [ dg_b_ n2 ]]
```

If lines and bars are used simultaneously, a second coordinates system will automatically be applied. This can, however, be changed by the definition of a second coordinates system. An example of the options is shown below:

```
[

[['x-ACHSE'],['y-Achse',0],[[40,250,150],[-1000,-10]],[30,20,0,0]],

[['x-ACHSE2'],['y-Achse2',0],['blue',[0,0]],[30,20,0,0]],

[1,[-1000,-500,0,500,1000],[5,20,40,55,71],['LINE1','red',0]],

[1,[-1000,-500,0,500,1000],[2,20,46,60,83],['LINE2',[200,5,150],3]],
```

```
[2,[5,25,16,10,4],['red',3],'group1'],

[2,[40,35,25,20,12],['red',3],'group2']

]
```

The example shows two lines and two groups of bars in two separate coordinates systems.

62.5.5.5 Displaying graphics

After you have defined the graphic in the data variable, you can use the graphical element's Show function to display the graphic. You can update it later by selecting the Refresh function in the menu or the graphics window.

62.6 Specific functions

Some specific calculations are integrated in KISSsys.

62.6.1 Load spectrum calculation

With KISSsys the user can generate an entire gearbox or drive train in a single file. Calculations can be then performed for the whole system using KISSsoft modules. If several stress cases are present (load spectrum or load cycle), you can use this load spectrum in KISSsys to analyze an entire system.

The user is then able to define load cases for the whole system and perform the calculation for all the components. Service life calculations with a load spectrum can be used with the same components as in KISSsoft.

The safety factors and service lives of different gears and bearings can be calculated. This load spectrum can also be used to calculate shaft fatigue and static safeties.

You can use the load spectrum function to perform these calculations:

- Calculation of a gear unit with a user-defined load.
- Calculation of a gear unit with a single load stage taken from the load spectrum.
- Calculation of a gear unit with a predefined load spectrum (similar to the ones defined in KISSsoft).

It is also possible to extend the load spectrum by adding extra additional values or settings, for example, to consider different speeds for each load step. A frequency, torque or power, and a speed is defined for each load step

The following KISSsoft modules are also required to perform at least one strength calculation for the elements that need to be calculated:

- Gear service life and safety factors with load spectrum:
  ZZ1 (load spectra) including modules Z16, Z16a, Z18 and Z18a.
- Shaft calculation with load spectrum:
  WA8 (load spectra) including modules W01s and W06s.
- KISSsys full version for implementing templates and setup:
  SYS module K11c.

62.6.2 Efficiency Calculation

You can use the efficiency calculation in KISSsys to calculate the heat level in a particular gear unit. Several different methods have been implemented to enable you to select how the calculation is to be performed, according to the ISO/TR 14179 standard, Part 1 and Part 2.

Thermal analysis can be defined in two sections: power loss and heat dissipation. An external cooler can also be taken into account. Power loss and heat dissipation can be split up into several sections to enable the effect of all the individual gear unit components to be taken into consideration.

There are two main types of power loss: load-dependent and non-load-dependent. Both types of loss are usually present when a gear unit is running. Power loss can also be subdivided into gear unit elements, such as gears, bearings and seals. Meshing and churning losses are taken into account for gears, whereas rolling and sliding friction are taken into account for bearings, and seal friction is taken into account for seals. In some cases, the results must be treated with caution, because the calculation methods used may not fully support the geometry type.

Heat dissipation can be categorized as heat dissipation through the housing, foundation and rotating parts (input/output shafts and couplings) and cooling oil flow.

You can then simply calculate a gear unit's total efficiency and total heat dissipation capacity for a given lubricant temperature, cooler power and input power. You can also specify two of these three entries and calculate the optimum value for the third parameter, which is the value with which you achieve the best heat level for the gear unit. In other words, this is the value at which the dissipated heat equals the heat generated through the power loss

.The difference between Part 1 and Part 2 of the standard is the way in which the different values are entered for the calculations. The main benefit of Part 1 is that it enables you to enter your own heat transfer coefficient for heat dissipation through the housing (if it has a very specific shape), whereas, in Part 2, this coefficient is calculated using an approximation of the shape of the housing. The main benefit of this part is that it also takes fins, foundations and rotating parts into consideration when calculating heat dissipation.

62.6.3 Housing deformation in static calculations

The inclusion of housing deformation in KISSsys static calculations is based on the use of a reduced stiffness matrix for the housing, as calculated by the Finite Element Method (FEM). This reduced stiffness matrix should include the nodes that refer to the center point of the bearings that connect the gear unit shafts to the housing.

62.6.3.1 Main calculation steps

The calculation steps for performing this kind of analysis are summarized below. The actual process used to generate the reduced stiffness matrix is not described, because it is different for each FEM computer program. Please refer to your FEM program manuals for more information.

Step 1: Importing the stiffness matrix and the FEM nodes coordinates

The first step is to read the stiffness matrix and the FEM node coordinates. To do this, calling the relevant function in the housing element, in the KISSsys model (right-click, select ImportStiffnessMatrix). Both the stiffness matrix and node coordinates should be positioned in the same file, together with information about the system of units used, in the header. An example of this type of file (which can now be handled by KISSsys) is shown below:

```
UNIT SYSTEM        (1 = SI, 2 = CGS, 3 = BFT, 4 = BIN, 5 = MKS, 6 = MPA, 7 = uMKS)
Active Unit System = 1
-----------------------------------------------------------------------------------------
MASTER NODE POSITION
              Number           x-coord          y-coord          z-coord
              1                *******          *******          *******
              2                *******          *******          *******

STIFFNESS MATRIX
```

****

You will find more detailed information about the FE programs supported by the software, and the file format requirements, in the relevant instructions (available on request).

Step 2: Positioning the housing correctly in the KISSsys model

Since the FEM model and the KISSsys model may not have the same coordinates system, you should then position the housing correctly in the KISSsys model. To do this, right-click on the housing element again, and select the ResetPosition function. In the next dialog, you can either input the

origin and alignment of the housing CS directly, or use the ThreePointPositioning function. To use this function, select three points (e.g. bearing) in the KISSsys model from a drop-down list, and then enter the coordinates of the same three points in the housing coordinates system. Make sure that these three points are not collinear. This procedure returns the housing CS with respect to the KISSsys CS. You can also perform a visual check to see that the positioning is correct by importing a simplified step file of the housing (e.g. only wireframe) and displaying it in the KISSsys 3D viewer. To import a step file into the housing, right mouse-click on it and select Dialog. We recommend you use a simplified version to avoid overloading the KISSsys model. Finally, you can also decide how the FEM nodes are displayed on the KISSsys model, by right-clicking on the housing element and selecting ShowNodes in the context menu. Every time the orientation of the housing is changed, you must select ShowNodes again, to update the nodes view in KISSsys. If you click on ShowNodes, you also see the IDs of the displayed FEM nodes, which makes it easier to validate the positioning. Note here that you do not need to add a cutting model to the housing element. However, the simplified CAD model is useful as an additional aid for validating the correct positioning.
Note that when a step model is used for the geometrical representation of the housing, it should be in the same coordinate system as the reduced FEM stiffness matrix to ensure the model is displayed correctly after the positioning.

Step 3: Performing the analysis

Click on the housing calculation button to start the analysis. The first step in the calculation is to map the FEM nodes on KISSsys bearings. The program reports all the nodes and displays their distance to the closest bearing. At this point, you need to know if the positions of the specified nodes actually correspond to the bearings. You can then decide whether to continue with the calculation or cancel it. One possible reason for specific bearings not corresponding to nodes is that the housing is positioned incorrectly in the KISSsys CS. If this is the case, then the previous step must be repeated. If this is not the case, and the difference between FEM nodes and bearings (as reported in the mapping message) is not too great, you can change the tolerance used in the mapping process. This may happen, for example, if the FEM node is positioned at the edges of a bearing instead of in the middle. The tolerance used in the mapping can be changed in the tolerance variable, in the housing properties in KISSsys (right-click on the housing element and open the Properties window). There, the tolerance is given in milimeters.

If you continue the calculation, the program reduces the stiffness matrix for the part that corresponds to the mapped nodes, and therefore ignores all nodes that have not been mapped to bearings. The calculation also ignores any predefined offsets and tilting values previously specified in the bearings and sets them to zero. The algorithm runs all the KISSsoft calculations and derives the forces on the bearings from their results. The program then uses these forces to calculate the offsets and tilting on the bearings (using the FEM stiffness matrix). The KISSsoft calculations are then run again with the resulting offsets, which may result in new bearing forces and offset values. This procedure is continued iteratively until there is convergence between successive forces and offset calculations. During the calculation process, it might sometimes happen that the maximum permitted number of iterations is reached due to housings with low stiffness. In this case, the system will display a message about the percentage difference between the last two iterations and apply the results from the last iteration. You can set the "Maximum no. of iterations" in the housing element's properties (right-click on the housing element, and then select Properties). The relevant property is called "maxNumberOfIterations". You should input a number that is greater than 4, to ensure the algorithm finds a useful solution. After the calculation is finished, you can perform further investigations. For example, you can perform contact analysis on gears to see the effect of housing stiffness on the gear unit design parameters.

You can also use several housings, each with a different stiffness matrix, in the KISSsys model. In this case, the program prompts you to select the housing it should use, before starting the calculation. This can be very useful if you want to compare the effect of different housing designs on the gear unit design. The results for each housing calculated using this method are then stored in the housing element. These results can then be viewed again by clicking on the housing element's RestoreOffsetResults function (right-click on the housing).

The following functions for handling displacement are also available (the tolerances remain the same).

1. ResetBearingOffsets: reset all bearing offset values to zero.
2. SaveBearingOffsets: save the current displacement values.
3. RestoreBearingOffsets: recover the saved displacement values.

62.6.4 Modal analysis of shaft systems

The modal analysis of shaft systems function in KISSsys is used to calculate the eigenfrequencies and eigenmodes of an entire shaft system, including the effect of the gear connection between shafts. Performing a modal analysis for individual shafts is not realistic. This analysis must be performed for the entire shaft system structure. The necessary calculation steps are described below, together with important restrictions.

62.6.4.1 Calculation procedure

To calculate the system dynamics, first import a KISSsys kSoftSystem calculation into the model. Right-click and then select "Modal analysis" in the context menu, to display a dialog in which you can set various parameters for the calculation. You must define the number of eigenfrequencies to be calculated, specify whether only torsional or all vibration types are to be included, and whether gyroscopic effects are to be taken into account (does not apply to torsional vibrations), and select which calculation method is to be used to calculate tooth contact stiffness. The following selections are available for this last option:

- Tooth contact stiffness as defined in ISO 6336
- Using the KISSsoft Contact Analysis (CA) algorithm, where a full contact analysis is performed in the gear connections. If KISSsoft does not have a contact analysis calculation for a particular gear pair type, or if the gear pair does not transfer power, the ISO 6336 process is used for that specific pair.
- Infinite: the tooth contact stiffness is assumed to be infinite. Select this option if you want to check limiting conditions.

- Ignore: the tooth contact stiffness is assumed to be zero, and there is therefore no connection between the vibrating shafts (each shaft is vibrating independently).

All the dynamic calculation properties mentioned above are also available in the calculation's Properties window (right-click on Calculation and select the Properties window).

62.6.4.2 Results

After the calculation is finished, a new tab opens, in which a 3D animation of the vibrating system can be displayed. There, you can select the eigenfrequency you want to view and also define the animation speed and the scaling of the deformations. The eigenfrequency values and tooth contact stiffnesses used for each gear pair are also displayed in the system dynamics report, together with other useful analysis results and a 2D plot diagram. To display this report, right-click on the ShaftSystem calculation and then select ShowReport. If necessary, click on the SavePlot button to save the 3D plot if it is to be used again (unchanged) in subsequent calculations (so that a new plot is generated each time). After the calculation has finished, the program also generates a table that contains the results for all the modes on all shafts in the system.
Please note that the only gears displayed in the Animation window are those that belong to a shaft calculation file.

Finally, also note that, if a modal analysis is performed for a planetary system, this does not take into account the effect of the turning planets' position on the system bending stiffness. This is similar to the quasi-static calculation procedure usually followed in eigenfrequencies analysis.

62.6.5 Campbell diagram for shaft systems

To investigate the effects of shaft speed on the eigenfrequencies, the system Campbell diagram can be drawn and be used to calculate shaft systems. It determines the critical eigenfrequencies for each speed or multiple of that speed.

62.6.5.1 Calculation procedure

To run a calculation for shaft systems with a Campbell diagram, click on the kSoftSystem calculation element in KISSsys. Right-click on the element, and then select the CampbellDiagram option. The Campbell diagram dialog has all the necessary entries. You can select the reference shaft for the calculation from a list of shafts in the system that include a coupling with a defined boundary condition. In this dialog, you can also select the calculation method for calculating the gears and the speed range of the reference shaft. You can define the various different speeds, together with the number of eigenfrequencies, that are to be taken into account in the Campbell diagram. Finally, you can select the number of resonance curves that are to be drawn in the Campbell diagram. The calculation starts with a kinematic analysis of the system for each speed of the reference shaft. The speeds of all of the shafts are updated and then a modal analysis is performed for each of these speeds.

62.6.5.2 Results

Once the calculation is finished, the user can see the 2D plot for the Campbell diagram directly in KISSsys. A more detailed 2D display, and a number of other useful analysis results, appear in the report, which is generated when you right-click on the kSoftSystem calculation and then select ShowReport. All the calculation data is also available in the results table that is generated in KISSsys. You can also click on the Save plot button to save the 2D plot, if it is to be used (unchanged) in subsequent calculations (so that a new curve is generated each time).

62.6.6 Analysis of unbalance response of shaft systems

The unbalance response analysis functions can be used to calculate the real dynamic behavior of a shaft system that is subjected to dynamic loads (unbalance masses). The calculated behavior includes deformations, rotation, forces and moments. The necessary inputs and the results achieved by the calculation are described below.

62.6.6.1 Calculation procedure

To call the unbalance response analysis, click on a kSoftSystem calculation element in KISSsys. Right-click on the element, and then select the ForcedResponse option. The next dialog contains all the inputs required to perform the calculation. You can select the reference shaft for the calculation from a list of shafts in the system that include a coupling with a defined boundary condition. You can then select the X-axis of the unbalance response diagram for the calculation (and therefore also define the type of calculation to run). Two options are available here:

- **Reference shaft speed:** The reference shaft speed is modified within the range you specified (min/max speed) with the predefined number of steps. A kinematic calculation is performed for each speed in the entire shaft system and the speed of all the shafts is calculated. These speeds are then used to calculate the dynamic loads, which are then applied to the model. The result is the unbalance response in the specified reference position on the shaft.
- **Y-coordinate of the reference shaft:** In this case, the length of the reference shaft is subdivided into the predefined number of sections, and the unbalance response calculation is performed for the specified speed. This results in the exact shape of the reference shaft at this speed.

You can then also select the calculation method you want to use to calculate contact stiffness, which is similar to the modal analysis calculation.

The effect of speed on the stiffness of rolling bearings can also be taken into account (but only for bearings with internal geometry). If this option is selected, a static calculation is performed for each speed, and the bearing stiffness used in the dynamic analysis is adjusted accordingly.

Finally, you can also define the damping for torsion, axial and bending vibration in this dialog. Note that the viscous damping of bearings must be defined separately for each bearing in the shaft calculation (freely definable units) or in the bearing's properties in KISSsys (SI units).

62.6.6.2 Results

Once the calculation is complete, a 2D plot is generated from the data you have entered. More detailed analysis results and other plot data are displayed in the report which is generated when you right-click on the kSoftSystem KISSsys element and then select ShowReport. You can also click on the SavePlot button to save the 2D plot if it is to be used (unchanged) in subsequent calculations (so that a new curve is generated each time). A table with all the data used in the plot is also generated. The unbalance response analysis is primarily performed for the predefined reference position on the reference shaft. However, the analysis results are also calculated for all documentation points that have been defined in the shaft calculations in the system. These documentation points can therefore be used as measuring points for dynamic behavior. The results of the documentation points are displayed both in the report and in the results table.

XI

Bibliography and Variable Directory

Chapter 63 - 64

63 Bibliography

[1] VDI 2230; Systematische Berechnung hochbeanspruchter Schraubenverbindungen, Blatt 1 und Blatt 2; 2015

[2] Schaeffler Technologies AG; Wälzlagerpraxis, Handbuch zur Gestaltung und Berechnung von Wälzlagerungen; Mainz; Vereinigte Fachverlage GmbH; 2015

[3] Roth, K.; Zahnradtechnik - Evolventen-Sonderverzahnungen zur Getriebeverbesserung; Berlin; Springer Verlag; 1998

[4] Burdick, R.; Manufacturing Single-Enveloping Worm Gear Sets; Gear Solutions; 2003

[5] Beermann, S.; Simulation der Schwingungen von Stirnradgetrieben unter Verwendung der Spektralsimulation, PhD Thesis, Karlsruhe, 2000 ; Düsseldorf; Fortschrittberichte VDI, Reihe 11, Schwingungstechnik, Nr. 288, VDI-Verlag; 2000

[6] Ebrahimi,S., Kissling, U.; Contact phase shifting of successive gear pairs for Improvement of NVH characteristics of powertrain systems; https://www.vdiconference.com/news/contact-phase-shifting-of-successive-gear-pairs/; vdiconderence.com; VDI Wissensforum; 2023

[7] Niemann G.; Maschinenelemente, Band 2; ; Berlin; Springer Verlag; 1983

[8] Niemann G.; Maschinenelemente, Band 1; ; Berlin; Springer Verlag; 2005

[9] Plewe, H.-J.; Untersuchung über den Abriebverschleiss von geschmierten, langsam laufenden Zahnrädern, Dissertation; TU München; 1980

[10] Schlecht, B; Maschinenelemente 2, Getriebe - Verzahnungen - Lagerungen; Pearson Studium; 2010

[11] DET NORSKE VERITAS; Calculation of gear rating for marine transmissions; Norway; 2003

[12] Niemann G.; Maschinenelemente, Band 3; Berlin; Springer Verlag; 1985

[13] VDI 2545; Zahnräder aus thermoplastischen Kunststoffen; 1981

[14] VDI 2736; Thermoplastische Zahnräder, Blatt 1-4; 2016

[15] Annast, R.; Kegelrad-Flankenbruch; Technische Universität München; 2002

[16] Witzig, J.; Flankenbruch - Eine Grenze der Zahnradtragfähigkeit, Dissertation; TU München; 2012

[17] Bertsche, B.; Reliability in Automotive and Mechanical Engineering; Berlin; Springer Verlag; 2008

[18] Haibach, E.; Betriebsfestigkeit, Verfahren und Daten zur Bauteilberechnung, 2. Auflage; ; Düsseldorf; VDI Verlag; 1989

[19] ISO 6336; Calculation of load capacity of spur and helical gears, Teil 1, 2, 3, 4, 5, 6; 2006

[20] Obsieger; Zahnformfaktoren von Aussen- und Innenverzahnungen; ; Zeitschrift Konstruktion 32, S.443-447; 1980

[21] Weber C., Banaschek K.; FVA-Bericht 129 und 134, Elastische Formänderung der Zähne und der anschliessenden Teile der Radkörper von Zahnradgetrieben, FVA 1955

[22] Langheinrich, A.; Geometrie, Beanspruchung und Verformung asymmetrischer Stirnradverzahnungen, Dissertation, FZG, München, 2014

[23] F.E.M. 1.001; Rules for the design of hoisting appliances, Booklet 2; Fédération Européenne de la Manutention

[24] Linke, H.; Stirnradverzahnung; München; Carl Hanser Verlag; 1996

[25] Kissling, U.; Sicher dimensioniert; Antriebstechnik 6, S. 64-68; 2007

[26] Dubbel, H.; Taschenbuch für den Maschinenbau, 16. Auflage; Berlin; Springer Verlag; 1987

[27] Bureau Veritas; Rules for The Classification of Naval Ships; FREMM 3.1; 2004

[28] FVA-Informationsblatt Nr. 54/7; Testverfahren zur Untersuchung des Schmierstoffeinflusses auf die Entstehung von Graufleckigkeit bei Zahnrädern; Frankfurt; FVA; 1999

[29] Hellmann.M.,; Berücksichtigung bei Fertigungsabweichungen bei der Auslegung von Zahnflankenmodifikationen für Stirnradverzahnungen. Dissertation.; RWTH Aachen; 2015

[30] Bolz, H., Otto, M., Stiller, S.; Korrektureinfluss Lastverteilung: Berücksichtigung des Einflusses von Flankenkorrekturen auf die Last-, Pressungs- und Zahnfußspannungsverteilung von Stirnradverzahnungen - Berechnungsstudie; Abschlussbericht; Forschungsvorhaben Nr. 609 I

[31] NIHS 20-25; Uhrenindustrie, Schweizer Norm SN 282 025; 1993

[32] Langheinrich, A.; Geometrie, Beanspruchung und Verformung asymmetrischer Stirnradverzahnungen, Dissertation; München; FZG; 2014

[33] Kissling, U., Pogacnik, A.; Design of asymmetric gears - potential and limits.; 978-1-911033-43-1; Lyon; Proceedings International Gear Conference 2018; 2018

[34] KISSsoft; Klassische Anleitungen zu den Berechnungsmodulen: KISSsoftGear Pump Analysis; Hombrechtikon; 2005

[35] Lachenmaier, S.; Auslegung von evolventischen Sonderverzahnungen für schwingungs- und geräuscharmen Lauf von Getrieben, WZL Reihe 11 Nr. 54; ; Düsseldorf; VDI Verlag; 1983

[36] Börner, J.:; Dresdner Berechnungsmethoden für die Zahnkontaktanalyse – Grundlagen und Anwendung auf spezielle Verzahnungen; Dresdner Maschinenelemente Kolloquium; 2005

[37] Winter, H., Podlesnik B.; Zahnfedersteifigkeit von Stirnradgetrieben, Teil 1 - 3; Antriebstechnik 22; 1983

[38] Girdhar, P. & Scheffer, C. ; Practical Machinery Vibration Analysis and Predictive Maintenance.; Elsevier; 2004

[39] Tuma, J. ; Vehicle gearbox noise and vibration: Measurement, signal analysis, signal processing and noise reduction measures ; John Wiley & Sons; 2014

[40] Miao, Q. & Zhou, Q. ; Planetary Gearbox Vibration Signal Characteristics Analysis and Fault Diagnosis. Shock and Vibration; 1-8; 2015

[41] Feng, Z. & Zuo, M. J. ; Vibration signal models for fault diagnosis of planetary gearboxes. Journal of Sound and Vibration; 331(22), 4919–4939; 2012

[42] Winterton, J. G. ; Component identification of gear-generated spectra; Orbit, June, 11–14.; 1991

[43] Fernandez A. ; Troubleshooting of gears; https://power-mi.com/content/troubleshooting-gears; Power-MI; 2020

[44] Fernandez, A. ; Frequencies of a gear assembly; Power-MI, 26.01., https://power-mi.com/content/frequencies-gear-assembly; 2021

[45] ANSI/AGMA 6123-C16; Design Manual for Enclosed Epicyclic Gear Drives; 2016

[46] Glinsky, C.; Geometry / Planetary gears layouts; Drivetrain Hub; 17.02.2021, https://drivetrainhub.com/notebooks/gears/geometry/Chapter%204%20-%20Planetary%20Gears.html#Mesh-Phasing; 2021

[47] DIN 3991; Tragfähigkeitsberechnungen von Kegelrädern; Berlin; Beuth Verlag; 1990

[48] ISO/DIS 10300; Calculation of load capacity of bevel gears, Teil 1, 2, 3, ISO 10300 (2001), ISO/DIS 10300 (2022); 2011

[49] Klingelnberg, J.; Kegelräder Grundlagen, Anwendungen; Berlin; Springer Verlag; 2008

[50] Basstein, G., Sijtstra, A.; Neue Entwicklung bei Auslegungen und Fertigung von Kronenrädern; Antriebstechnik, 32, Nr 11; 1993

[51] Kissling, U., Beermann, S., Hirn, T.; Kronenräder: Geometrie und Festigkeit; Antriebstechnik 10; 2003

[52] Bock G., Nocj R., Steiner O.; Zahndickenmessung an Getriebeschnecken nach der Dreidrahtmethode; Braunschweig; Physikalisch-Technische Bundesanstalt; 1974

[53] Hoechst, High Chem; Technische Kunststoffe - Berechnen, Gestalten, Anwenden, B.2.2; Hoechst AG; 1992

[54] Pech, M.; Tragfähigkeit und Zahnverformung von Schraubenradgetrieben der Werkstoffpaarung Stahl/Kunststoff; Bochum; Lehrstuhl für Maschinenelemente, Getriebe und Kraftfahrzeuge; 2011

[55] Tsai, S.-J., Wu, S.-H.; Geometrical Design of Conical Gear Drives with Profile-shifted Transmission; 12th IFToMM World Congress; 2007

[56] Tsai, S.-J., Wu, S.-H.; Designing Skew Conical Gear Drives in Approximate Line Contact for Power Transmission; Proceedings of MPT2009-Sendai JSME International Conference on Motion and Power Transmission; 2009

[57] AGMA 2001-B88; Fundamental Rating Factors and Calculation Methods for Involute Spur and Helical Gear Teeth; 2351tt; 1988

[58] Akahori, H., Sato, Y., Nishida, T., Kubo, A.; Prove di durata di Face Gear; Organi di trasmissione, Nr.12 oder MTP2001-Fukuoka, The JSME Int. Conference, Japan (2001); 2002

[59] Theissen, J.; Berechnung der Sicherheit gegen Graufleckigkeit von Industriegetrieben auf der Grundlage des neuen Rechenverfahrens nach FVA 259, Dresdner Maschinenkolloquium; TU Dresden, Tagungsband S.195-212; 2003

[60] FVA-Informationsblatt Nr. 54/7; Testverfahren zur Untersuchung des Schmierstoffeinflusses auf die Entstehung von Graufleckigkeit bei Zahnrädern; Frankfurt; FVA; 1999

[61] Feulner, R.; Verschleiss trocken laufender Kunststoffgetriebe; Erlangen; Lehrstuhl Kunststofftechnik; 2008

[62] Pestel, E.C.; Leckie, F.A.; Matrix Methods in Elastomechanics; McGraw-Hill; 1963

[63] DIN 743; Tragfähigkeitsberechnung von Wellen und Achsen; Berlin; Beuth Verlag; 2012

[64] Hänchen, R., Decker, K.H.; Neue Festigkeitslehre für den Maschinenbau, 3. Auflage; München; Carl Hanser Verlag; 1967

[65] DIN 732; Wälzlager - Thermische Bezugsdrehzahl - Berechnung und Beiwerte; Berlin; Beuth Verlag; 2010

[66] DIN ISO 15312; Wälzlager - Thermisch zulässige Betriebsdrehzahl - Berechnung und Beiwerte; Berlin; Beuth Verlag; 2004

[67] Harris, T.; Rolling Bearing Analysis, 4th Edition; John Wiley & Sons Inc.

[68] Harris, T., Rumbarger, J.H., Butterfield, C.P.; Wind Turbine Design Guideline DG03: Yaw and Pitch Rolling Bearing Life; NREL Report No. TP-500-42362, 63 pp

[69] ISO 7902; Hydrodynamic plain journal bearings under steady-state conditions, Teil 1 (2015), Teil2 und 3 (1998); 2020

[70] DIN 31657; Hydrodynamische Radial-Gleitlager im stationären Betrieb, DIN Taschenbuch 198; Berlin; Beuth Verlag; 2015

[71] Lang, O., Steinhilper R.; Gleitlager, Konstruktionsbücher Band 31; Berlin; Springer Verlag; 1978

[72] DIN 31652 1-3; Hydrodynamische Radial-Gleitlager im stationären Bereich; Berlin; Beuth Verlag; 2017

[73] DIN 31653; Hydrodynamische Axial-Gleitlager im stationären Bereich, DIN Taschenbuch 198; Berlin; Beuth Verlag; 1991

[74] DIN 31654; Hydrodynamische Axial-Gleitlager im stationären Bereich, DIN Taschenbuch 198; Berlin; Beuth Verlag; 1991

[75] Massa, E.; Costruzione di macchine; Milano; Editori Masson Italia; 1981

[76] Niemann G.; Maschinenelemente, Band 1; ; Berlin; Springer Verlag ; 1981

[77] Niemann G.; Maschinenelemente, Band 2; Berlin; Springer Verlag ; 1981

[78] Kollmann, F.; Welle-Nabe-Verbindungen; ; Berlin; Springer Verlag; 1984

[79] DIN 7190; Berechnung und Anwendung von Pressverbänden, Teil 1 und 2; Berlin; Beuth Verlag; 2017

[80] Matek W., Muks D., Wittel H., Becker M., Jannasch D.; Roloff/Matek Maschinenelemente, 15. Auflage; Braunschweig; Vieweg Verlag; 2001

[81] Decker, K.-H.; Maschinenelemente, Funktion, Gestaltung und Berechnung; München; Hanser Verlag; 2001

[82] DIN 6885; Passfedern, Blatt 1-3; Berlin; Beuth Verlag; 1968

[83] DIN 6892; Passfedern - Berechnung und Gestaltung; Berlin; Beuth Verlag; 2012

[84] Matek, W., Muks D., Wittel H.; Roloff/Matek Maschinenelemente, 11. Auflage; ; Braunschweig; Vieweg Verlag; 1987

[85] Spinnler, Prof.; Manual de calcul d'organes des machines; EPFL Lausanne; 1990

[86] Decker, K.H.; Maschinenelemente, 10. Auflage; ; München; Carl Hanser Verlag; 1990

[87] Dudley, Darle W.; When Splines need stress control; Product Engineering; 1953

[88] DIN 5480; Zahnwellen-Verbindungen mit Evolventenflanken; Berlin; Beuth Verlag; 2006

[89] DIN 32711 ; Welle-Nabe-Verbindung - Polygonprofil P3G; Berlin; Beuth Verlag; 2009

[90] DIN 32712 ; Welle-Nabe-Verbindung - Polygonprofil P4C; Berlin; Beuth Verlag; 2009

[91] DIN 32711 ; Welle-Nabe-Verbindung - Polygonprofil P3G; Berlin; Beuth Verlag; 2023

[92] DIN 32712 ; Welle-Nabe-Verbindung - Polygonprofil P4C; Berlin; Beuth Verlag; 2012

[93] Matthias, K.; Schraubenkräfte in einer Flanschverbindung; Berlin; Maschinenbau 34, S.517; 1985

[94] DIN EN 13906-1; Druckfedern: Berechnung und Konstruktion, DIN Taschenbuch 29, 12. Auflage; Berlin; Beuth Verlag; 2015

[95] Meissner, M.; Schorcht, H.J.; Metallfedern - Grundlagen, Werkstoffe, Berechnung, Gestaltung und Rechnereinsatz, 2. Auflage; Berlin; Springer-Verlag; 2007

[96] DIN EN 15800; Zylindrische Schraubenfedern aus runden Drähten: Gütevorschrift für kaltgeformte Druckfedern, DIN Taschenbuch 29; Berlin; Beuth Verlag; 2011

[97] DIN 2096; Zylindrische Schraubenfedern aus runden Drähten und Stäben: Gütevorschrift für warmgeformte Druckfedern, DIN Taschenbuch 29; Berlin; Beuth Verlag; 2011

[98] DIN EN 13906-2; Zugfedern: Berechnung und Konstruktion, DIN Taschenbuch 29, 12. Auflage; Berlin; Beuth Verlag; 2015

[99] DIN 2097; Zylindrische Schraubenfedern aus runden Drähten: Gütevorschriften für kaltgeformte Zugfedern, DIN Taschenbuch 29; Berlin; Beuth Verlag; 2011

[100] DIN EN 13906-3; Drehfedern: Berechnung und Konstruktion, DIN Taschenbuch 29, 12. Auflage; Berlin; Beuth Verlag; 2015

[101] DIN 2194; Zylindrische Schraubenfedern aus runden Drähten und Stäben: Kaltgeformte Drehfedern (Schenkelfedern), Gütenorm, DIN Taschenbuch 29; Berlin; Beuth Verlag; 2011

[102] DIN 2092; Tellerfedern: Berechnung, DIN Taschenbuch 29; Berlin; Beuth Verlag; 2011

[103] DIN 2093; Tellerfedern: Masse, Qualitätsanforderungen, DIN Taschenbuch 29; Berlin; Beuth Verlag; 2011

[104] Niemann G.; Maschinenelemente, Band 1; Berlin; Springer Verlag; 2019

[105] DIN 2091; Drehstabfedern mit rundem Querschnitt: Berechnung und Konstruktion, DIN Taschenbuch 29; Berlin; Beuth Verlag; 2011

[106] VDI 2241; Schaltbare fremdbetätigte Reibkupplungen und -bremsen, Blatt 1 (1982), Blatt 2 (1984); 1984

[107] Johnson, K.L.; Contact Mechanics; Cambridge University Press; 1987

[108] Boresi, A.P., Schmidt R.J.; Advanced mechanic of material, 6th Edition; John Wiley and Sons Inc.; 2002

[109] Ribeiro, M.; Gaussian Probability Density Functions: Properties and Error Characterization; 2014

[110] Zaka, A.; Akhter, A.; Methods for Estimating the Parameters of the Power Function Distribution; 2013

64 Variables directory for calculation modules

The variables listed in the following subchapters can be used for COM interface and SKRIPT programming. The respective variable assignments for the calculation modules are shown in tables with 3 columns, consisting of variable name, description and unit (if any).

64.1 Z012 Cylindrical gear pair

ZR[0].z	Number of teeth gear1	
ZR[1].z	Number of teeth gear2	
ZP[0].a	Center distance	mm
ZR[0].x.nul	Addendum modification coefficient gear1	
ZR[1].x.nul	Addendum modification coefficient gear2	
ZR[0].b	Facewidth gear1	mm
ZR[1].b	Facewidth gear2	mm
ZR[0].Tnominal	Nominal torque gear1	Nm
ZR[1].Tnominal	Nominal torque gear2	Nm
ZS.Pnominal	Nominal power	kW
ZR[0].nnominal	Absolute speed gear1	1/min
ZR[1].nnominal	Absolute speed gear2	1/min
ZR[0].Schrage	Helix direction gear1	
ZR[1].Schrage	Helix direction gear1	
ZS.Geo.mn	Normal module	mm
ZS.Geo.alfn	Normal pressure angle	°

ZS.Geo.beta	Helix angle at reference diameter	°
ZR[0].Tool.RefProfile.DBID	Database selection reference profile gear1	
ZR[1].Tool.RefProfile.DBID	Database selection reference profile gear2	
ZR[0].mat.DBID	Database selection material gear1	
ZR[1].mat.DBID	Database selection material gear2	
ZR[0].d	Reference diameter gear1	mm
ZR[1].d	Reference diameter gear2	mm
ZPP[0].dw	Pitch diameter gear1	mm
ZPP[1].dw	Pitch diameter gear2	mm
ZR[0].da.nul	Tip diameter gear1	mm
ZR[1].da.nul	Tip diameter gear2	mm
ZR[0].df.nul	Root diameter gear1	mm
ZR[1].df.nul	Root diameter gear2	mm
ZR[0].Fased	Tip chamfer gear1	mm
ZR[1].Fased	Tip chamfer gear2	mm
ZPP[0].Fuss.SF	Root safety gear1	
ZPP[1].Fuss.SF	Root safety gear2	
ZPP[0].Flanke.SH	Safety factor for contact stress gear1	
ZPP[1].Flanke.SH	Safety factor for contact stress gear2	
ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient gear1	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient gear2	

Zst.KwbVariant	Method alternating bending factor	
ZP[0].FIT.SSint	Safety against scuffing	
ZP[0].FBT.SB	Safety factor for scuffing (flash-temp)	
ZS.KA	Application factor	
ZS.H	Required service life	h
ZS.ED	Power-on time	%
ZS.jt.i	Total torsional angle	°
ZS.jt.E	Total torsional angle	°
ZR[0].Vqual	Accuracy grade	
ZR[1].Vqual	Accuracy grade	
ZS.Oil.SchmierTypID	Database selection lubricant	
ZS.Oil.theOil	Oil temperature	°C
ZS.Oil.SchmierungsArt	Lubrication type	
ZP[0].KHdat.Belastung	Load according to DIN 3990-1:1987 Diagram 6.8	
ZP[0].Impulsnominal	Driving gear	
ZP[0].ImpulsUI	Driving gear	
ZS.ZeigerAufDr	During calculation, the system takes into account the fact that this gear is the reference	
ZR[0].NLFlag	Type consideration of load cycles gear1	
ZR[0].NLVorgabe	Number of load cycles gear1	
ZR[1].NLFlag	Type consideration of load cycles gear2	
ZR[1].NLVorgabe	Number of load cycles gear2	
ZS.theUmg	Ambient temperature	°C

ZP[0].alfwt	Working pressure angle	°
ZR[0].Tool.RefProfile.haP	Addendum coefficient	
ZR[1].Tool.RefProfile.haP	Addendum coefficient	
ZR[0].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[1].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[0].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[1].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[0].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[1].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[0].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[1].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[0].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[1].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[0].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[1].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[0].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[1].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[0].ZDTolID	Database selection tooth thickness tolerance gear1	
ZR[1].ZDTolID	Database selection tooth thickness tolerance gear2	
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[0].di	Inner diameter	mm

ZR[1].di	Inner diameter	mm
ZR[0].Tool.RefProfile.topping	Topping tool gear1	
ZR[1].Tool.RefProfile.topping	Topping tool gear2	
ZR[0].Tool.Aufmass.E	Information on pre-machining	
ZR[0].Tool.q	Final machining stock	
ZR[1].Tool.Aufmass.E	Information on pre-machining	
ZR[1].Tool.q	Input for final machining stock	
ZR[0].Tool.Aufmass.i	Information on pre-machining	
ZR[1].Tool.Aufmass.i	Information on pre-machining	
ZR[0].Ca	Tip relief left/right	µm
ZR[1].Ca	Tip relief left/right	µm
ZR[0].Cf	Root relief, right flank	µm
ZR[1].Cf	Root relief, right flank	µm
ZR[0].kXmn	Tip alteration	mm
ZR[1].kXmn	Tip alteration	mm
ZR[0].KopfKant	Tip chamfer gear1	mm
ZR[1].KopfKant	Tip chamfer gear2	mm
ZR[0].Adf_input	Root diameter allowance gear1	mm
ZR[1].Adf_input	Root diameter allowance gear2	mm
ZR[0].Ada.E	Tip diameter	mm
ZR[1].Ada.E	Tip diameter	mm
ZR[0].Ada.i	Tip diameter	mm

ZR[1].Ada.i	Tip diameter	mm
ZR[0].Adf.E	Root diameter	mm
ZR[1].Adf.E	Root diameter	mm
ZR[0].Adf.i	Root diameter	mm
ZR[1].Adf.i	Root diameter	mm
ZR[0].As.E	Tooth thickness tolerance, normal section	mm
ZR[1].As.E	Tooth thickness tolerance, normal section	mm
ZR[0].As.i	Tooth thickness tolerance, normal section	mm
ZR[1].As.i	Tooth thickness tolerance, normal section	mm
ZR[0].crowning	Flank line crowning gear1	mm
ZR[1].crowning	Flank line crowning gear2	mm
ZR[0].Tool.finishing	Finishing tool gear1	
ZR[1].Tool.finishing	Finishing tool gear2	
ZS.ArtProfKorr	Type of profile modification	
ZP[0].KHdat.FlankenLin	Flank line modification	
ZP[0].Ft	Nominal tangential force at base diameter	
ZPP[0].Fa	Axial force	N
ZPP[0].Fr	Radial force	N
ZPP[0].Fnorm	Normal force	N
Zst.KHbVariant	Method for face load factor	
ZP[0].KHb_nominal	Face load factor KHb	
Zst.KHaFlag	Method for transverse load factor	

ZP[0].KHa	Transverse load factor	
Zst.KVFlag	Method for dynamic factor	
ZP[0].KV.KV	Dynamic factor	
ZS.Geo.mt	Transverse module	mm
ZP[0].AXTolName	Center distance tolerance	
ZP[0].Aa.E	Centre distance allowances	mm
ZP[0].Aa.i	Centre distance allowances	mm
ZR[0].KM.Wk.nul	Base tangent length no backlash	mm
ZR[1].KM.Wk.nul	Base tangent length no backlash	mm
ZR[0].KM.Wk.E	Base tangent length	mm
ZR[1].KM.Wk.E	Base tangent length	mm
ZR[0].KM.Wk.i	Base tangent length	mm
ZR[1].KM.Wk.i	Base tangent length	mm
ZR[0].KM.k	Number of teeth spanned	
ZR[1].KM.k	Number of teeth spanned	
ZP[0].KHdat.l	Bearing distance l of pinion shaft	mm
ZP[0].KHdat.S	Distance s of pinion shaft	mm
ZP[0].KHdat.dsh	Outside diameter of pinion shaft	mm
ZP[0].Eps.a	Transverse contact ratio	
ZP[0].Eps.b	Overlap contact ratio	
ZP[0].Eps.G	Total contact ratio	
ZPP[0].zetaa	Specific sliding at the tip	

ZPP[1].zetaa	Specific sliding at the tip	
ZPP[0].zetaf	Specific sliding at the root	
ZPP[1].zetaf	Specific sliding at the root	
ZP[0].Flanke.sigH	Nominal contact stress	N/mm²
ZR[0].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZR[1].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZPP[0].Flanke.sigHP	Permissible contact stress	N/mm²
ZPP[1].Flanke.sigHP	Permissible contact stress	N/mm²
ZPP[0].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[1].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZR[0].mat.limf	Tooth root strength at NL	N/mm²
ZR[1].mat.limf	Tooth root strength at NL	N/mm²
ZPP[0].Fuss.sigFP	Admissible bending stress	N/mm²
ZPP[1].Fuss.sigFP	Allowable bending stress	N/mm²
ZP[0].KFb	Face load factor tooth root	
ZP[0].KHdat.fsh	from deformation of shaft	µm
RechSt.RechenMethID	Calculation method	
ZS.WirkungsGrad	Meshing efficiency	%
ZS.Woehler00	Modification of Woehler line	
INP03Ein.baMin_ownI	Fine sizing facewidth minimum	mm
INP03Ein.baMax_ownI	Fine sizing facewidth maximum	mm
INP03Ein.MinZaZahl	Fine sizing number of teeth minimum	

INP03Ein.MaxZaZahl	Fine sizing number of teeth maximum	
INP03Ein.bmnMin_ownI	Fine sizing module minimum	mm
INP03Ein.bmnMax_ownI	Fine sizing module maximum	mm
INP03Ein.bd1Min_ownI	Fine sizing diameter gear1 minumum	mm
INP03Ein.bd1Max_ownI	Fine sizing diameter gear1 maximum	mm
ZS.Kgam	Mesh load factor	
ZS.PfeilZahn	Double helical gearing	
ZS.settingsEL.lastfaktor	Partial load for calculation	
ZS.settingsEL.a	Center distance	mm
ZR[0].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZR[1].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZP[0].achsNeigung	Axis alignment, pair 1	µm
ZP[0].achsSchraenkung	Axis alignment, pair 1	µm
ZP[0].MP_ISO.Slam	Safety against micropitting	
ZPP[0].delWnPC	Medium wear removal	mm
ZPP[1].delWnPC	Medium wear removal	mm
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZP[0].Verlust.PVZ	Gear power loss	kW
ZS.Oil.roOil	Density at 15°C	kg/dm³

ZS.Oil.nu40	Nominal viscosity at 40°C	mm²/s
ZS.Oil.nu100	Nominal viscosity at 100°C	mm²/s
ZS.P_Limit	Power	kW
ZS.P_Usage	Stress	
ZPP[0].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[1].flankbreak.SFFB	Safety against tooth flank fracture	
ZR[0].KSt.SW	Safety against wear	
ZR[1].KSt.SW	Safety against wear	
ZS.SSi.Fuss	Required safety for tooth root	
ZS.SSi.Flanke	Required safety for tooth flank	
ZS.SSi.FrInt	Required scuffing safety (integral temperature)	
ZS.SSi.FrBli	Required scuffing safety (flash temperature)	
ZS.SSi.Slam	Required safety for frosting	
ZS.SSi.SFF	Required safety for tooth flank fracture	
ZS.SSi.VerfKSt	Required safety	
ZS.SSi.VerschleissSch	Required safety for wear	
ZS.Hatt	System service life	h
ZP[0].bv	Axial offset	mm
ZR[0].KSt.TempFuss	Tooth root temperature	°C
ZR[0].KSt.TempFlanke	Flank temperature	°C
ZR[1].KSt.TempFuss	Tooth root temperature	°C
ZR[1].KSt.TempFlanke	Flank temperature	°C

ZS.lastKElem	Calculation for load bin no	
ZP[0].MP_ISO.mym	Coefficient of friction	
ZS.BFSpiel.theRef	Reference temperature	°C
ZS.BFSpiel.theGehmin	Housing temperature, min ... max	°C
ZS.BFSpiel.theGehmax	Housing temperature, min ... max	°C
ZS.BFSpiel.theRadmin	Gear body temperature, min ... max	°C
ZS.BFSpiel.theRadmax	Gear body temperature, min ... max	°C
ZS.BFSpiel.theDifmin	Permitted temperature difference	Δ°C
ZS.BFSpiel.theDifmax	Permitted temperature difference	Δ°C
ZP[0].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[0].cmin	Minimum tip clearance	
ZP[0].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[0].jtOP[1].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[1].cmin	Minimum tip clearance	mm
ZP[0].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[1].epsa.E	Max. transverse contact ratio	
ZR[0].RZF	Mean peak-to-valley roughness R_z, root	µm

ZR[1].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[0].RZH	Mean peak-to-valley roughness R_z, Flank	µm
ZR[1].RZH	Mean peak-to-valley roughness R_z, Flank	µm
ZR[0].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[1].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[0].RAH	Average surface roughness for Lx, pinion	µm
ZR[1].RAH	Average surface roughness for Lx, pinion	µm
ZS.ReqSysReliability	Required reliability	
RechSt.asymmetric	Asymmetric gears consideration	
ZS.Planet.nStegnominal	Speed planet carrier	1/min
ZS.Planet.ActivatePairCalculation	Consider planet carrier	
ZS.LKPlanet.ID	Database selection load spectrum for planet	
ZS.LowestPriority	Input of power or torque	
ZR[0].YT	Technology factor gear1	
ZR[1].YT	Technology factor gear2	

64.2 Z014 Cylindrical Gear - Planetary Gear Stage

ZR[0].z	Number of teeth	
ZR[1].z	Number of teeth	
ZR[2].z	Number of teeth	
ZS.Geo.utot	Ratio	
ZP[0].a	Center distance	mm

ZR[0].x.nul	Addendum modification coefficient	
ZR[1].x.nul	Addendum modification coefficient	
ZR[2].x.nul	Addendum modification coefficient	
ZR[0].b	Face width	mm
ZR[1].b	Face width	mm
ZR[2].b	Face width	mm
ZR[0].Tnominal	Torque gear1	Nm
ZR[1].Tnominal	Torque gear2	Nm
ZR[2].Tnominal	Torque gear3	Nm
ZS.Planet.TStegnominal	Torque carrier	Nm
ZS.Pnominal	Power	kW
ZS.AnzahlZwi	Number of bevel gears in the differentials gearing	
ZR[0].nnominal	Absolute speed	1/min
ZR[1].nnominal	Absolute speed	1/min
ZR[2].nnominal	Absolute speed	1/min
ZS.Planet.nStegnominal	Speed planet carrier	1/min
ZR[0].Schrage	Helix direction	
ZR[1].Schrage	Helix direction	
ZR[2].Schrage	Helix direction	
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm

ZS.Geo.alfn	Normal pressure angle	°
ZS.Geo.alfn	Normal pressure angle	°
ZS.Geo.alfn	Normal pressure angle	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZR[0].mat.DBID	Database selection material gear1	
ZR[1].mat.DBID	Database selection material gear2	
ZR[2].mat.DBID	Database selection material gear3	
ZR[0].d	Reference diameter	mm
ZR[1].d	Reference diameter	mm
ZR[2].d	Reference diameter	mm
ZR[0].da.nul	Tip diameter	mm
ZR[1].da.nul	Tip diameter	mm
ZR[2].da.nul	Tip diameter	mm
ZR[0].df.nul	Root diameter	mm
ZR[1].df.nul	Root diameter	mm
ZR[2].df.nul	Root diameter	mm
ZPP[0].dw	Pitch diameter	mm
ZPP[1].dw	Pitch diameter	mm
ZPP[2].dw	Pitch diameter	mm
ZR[0].Fased	Tip chamfer	mm

ZR[1].Fased	Tip chamfer	mm
ZR[2].Fased	Tip chamfer	mm
ZPP[0].Fuss.SF	Root safety	
ZPP[1].Fuss.SF	Root safety	
ZPP[2].Fuss.SF	Root safety	
ZPP[3].Fuss.SF	Root safety	
ZPP[0].Flanke.SH	Safety factor for contact stress on flank	
ZPP[1].Flanke.SH	Safety factor for contact stress on flank	
ZPP[2].Flanke.SH	Safety factor for contact stress on flank	
ZPP[3].Flanke.SH	Safety factor for contact stress on flank	
ZP[0].FIT.SSint	Safety against scuffing	
ZP[1].FIT.SSint	Safety against scuffing	
ZP[0].FBT.SB	Safety factor for scuffing (flash-temp)	
ZP[1].FBT.SB	Safety factor for scuffing (flash-temp)	
ZS.KA	Application factor	
ZS.H	Required service life	h
ZS.ED	Power-on time	%
ZS.jt.i	Total torsional angle	°
ZS.jt.E	Total torsional angle	°
ZR[0].Vqual	Accuracy grade	
ZR[1].Vqual	Accuracy grade	
ZR[2].Vqual	Accuracy grade	

ZS.Oil.SchmierTypID	Database selection lubricant	
ZS.Oil.theOil	Oil temperature	°C
ZS.Oil.SchmierungsArt	Lubrication type	
ZP[0].Impulsnominal	Driving gear	
ZP[0].ImpulsUI	Driving gear	
ZS.ZeigerAufDr	During calculation, the system takes into account the fact that this gear is the reference gear	
ZS.theUmg	Ambient temperature	°C
ZP[0].alfwt	Working pressure angle	°
ZP[1].alfwt	Working pressure angle	°
ZR[0].Tool.RefProfile.haP	Addendum coefficient	
ZR[1].Tool.RefProfile.haP	Addendum coefficient	
ZR[2].Tool.RefProfile.haP	Addendum coefficient	
ZR[0].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[1].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[2].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[0].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[1].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[2].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[0].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[1].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[2].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[0].Tool.RefProfile.alf_prP	Protuberance angle	°

ZR[1].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[2].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[0].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[1].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[2].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[0].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[1].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[2].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[0].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[1].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[2].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[0].di	Inner diameter	mm
ZR[1].di	Inner diameter	mm
ZR[2].di	Inner diameter	mm
ZR[0].Ca	Tip relief left/right	µm
ZR[1].Ca	Tip relief left/right	µm
ZR[2].Ca	Tip relief left/right	µm
ZR[2].Cf	Root relief, right flank	µm
ZR[0].Cf	Root relief, right flank	µm
ZR[1].Cf	Root relief, right flank	µm
ZR[0].KopfKant	Section	
ZR[1].KopfKant	Section	

ZR[2].KopfKant	Section	
ZR[0].Ada.E	Tip diameter	mm
ZR[1].Ada.E	Tip diameter	mm
ZR[0].Ada.i	Tip diameter	mm
ZR[1].Ada.i	Tip diameter	mm
ZR[2].Ada.E	Tip diameter	mm
ZR[2].Ada.i	Tip diameter	mm
ZR[0].Adf.E	Root diameter	mm
ZR[1].Adf.E	Root diameter	mm
ZR[2].Adf.E	Root diameter	mm
ZR[0].Adf.i	Root diameter	mm
ZR[1].Adf.i	Root diameter	mm
ZR[2].Adf.i	Root diameter	mm
ZR[0].As.E	Tooth thickness tolerance, normal section	mm
ZR[1].As.E	Tooth thickness tolerance, normal section	mm
ZR[2].As.E	Tooth thickness tolerance, normal section	mm
ZR[0].As.i	Tooth thickness tolerance, normal section	mm
ZR[1].As.i	Tooth thickness tolerance, normal section	mm
ZR[2].As.i	Tooth thickness tolerance, normal section	mm
ZP[0].Ft	Nominal tangential force at base diameter	
ZP[1].Ft	Nominal tangential force at base diameter	
ZPP[0].Fa	Axial force	N

ZPP[1].Fa	Axial force	N
ZPP[2].Fa	Axial force	N
ZPP[3].Fa	Axial force	N
ZPP[0].Fr	Radial force	N
ZPP[1].Fr	Radial force	N
ZPP[2].Fr	Radial force	N
ZPP[3].Fr	Radial force	N
ZPP[0].Fnorm	Normal force	N
ZPP[1].Fnorm	Normal force	N
ZPP[2].Fnorm	Normal force	N
ZPP[3].Fnorm	Normal force	N
ZP[0].Ft	Nominal tangential force at base diameter	
ZR[0].kXmn	Tip alteration	mm
ZR[1].kXmn	Tip alteration	mm
ZR[2].kXmn	Tip alteration	mm
ZS.Geo.mt	Transverse module	mm
ZP[0].AXTolName	Center distance tolerance	
ZP[1].AXTolName	Center distance tolerance	
ZP[0].Aa.E	Center distance allowance	mm
ZP[0].Aa.i	Center distance allowance	mm
ZP[1].Aa.E	Center distance allowance	mm
ZP[1].Aa.i	Center distance allowance	mm

ZR[0].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[1].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[2].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[0].KM.Wk.E	Base tangent length	mm
ZR[1].KM.Wk.E	Base tangent length	mm
ZR[2].KM.Wk.E	Base tangent length	mm
ZR[0].KM.Wk.i	Base tangent length	mm
ZR[1].KM.Wk.i	Base tangent length	mm
ZR[2].KM.Wk.i	Base tangent length	mm
ZR[0].KM.k	Number of teeth spanned	
ZR[1].KM.k	Number of teeth spanned	
ZR[2].KM.k	Number of teeth spanned	
ZP[0].Eps.a	Transverse contact ratio	
ZP[1].Eps.a	Transverse contact ratio	
ZP[0].Eps.b	Overlap contact ratio	
ZP[1].Eps.b	Overlap contact ratio	
ZP[0].Eps.G	Total contact ratio	
ZP[1].Eps.G	Total contact ratio	
ZPP[0].zetaa	Specific sliding at the tip	
ZPP[1].zetaa	Specific sliding at the tip	
ZPP[2].zetaa	Specific sliding at the tip	
ZPP[3].zetaa	Specific sliding at the tip	

ZPP[0].zetaf	Specific sliding at the root	
ZPP[1].zetaf	Specific sliding at the root	
ZPP[2].zetaf	Specific sliding at the root	
ZPP[3].zetaf	Specific sliding at the root	
ZP[0].Flanke.sigH	Nominal contact stress	N/mm²
ZP[1].Flanke.sigH	Nominal contact stress	N/mm²
ZR[0].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZR[1].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZR[2].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZPP[0].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[1].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[2].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[3].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[0].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[1].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[2].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[3].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZR[0].mat.limf	Tooth root strength at NL	N/mm²
ZR[1].mat.limf	Tooth root strength at NL	N/mm²
ZR[2].mat.limf	Tooth root strength at NL	N/mm²
ZPP[0].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[1].Fuss.sigFP	Allowable bending stress	lb/in²

ZPP[2].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[3].Fuss.sigFP	Allowable bending stress	lb/in²
ZP[0].KHdat.fsh	from deformation of shaft	µm
ZP[1].KHdat.fsh	from deformation of shaft	µm
RechSt.RechenMethID	Calculation method	
ZS.WirkungsGrad	Meshing efficiency	%
ZS.Woehler00	Modification of Woehler line	
ZR[0].Tool.Aufmass.E	Information on pre-machining	
ZR[1].Tool.Aufmass.E	Information on pre-machining	
ZR[2].Tool.Aufmass.E	Information on pre-machining	
ZR[0].Tool.Aufmass.i	Information on pre-machining	
ZR[1].Tool.Aufmass.i	Information on pre-machining	
ZR[2].Tool.Aufmass.i	Information on pre-machining	
ZR[0].Tool.q	Final machining stock	mm
ZR[1].Tool.q	Final machining stock	mm
ZR[2].Tool.q	Final machining stock	mm
Zst.KVFlag	Method for dynamic factor	
ZP[0].KV.KV	Dynamic factor	
ZP[1].KV.KV	Dynamic factor	
ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[2].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	

ZS.settingsEL.lastfaktor	Partial load for calculation	
ZR[0].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZR[2].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZP[0].MP_ISO.Slam	Safety against micropitting	
ZP[1].MP_ISO.Slam	Safety against micropitting	
ZPP[0].delWnPC	Medium wear removal	mm
ZPP[1].delWnPC	Medium wear removal	mm
ZPP[2].delWnPC	Medium wear removal	mm
ZPP[3].delWnPC	Medium wear removal	mm
ZS.Kgam	Mesh load factor	
ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[2].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZP[0].MP_ISO.Slam	Safety against micropitting	
ZP[1].MP_ISO.Slam	Safety against micropitting	
ZP[0].MP_ISO.SlamB	Safety against micropitting, method B	
ZP[1].MP_ISO.SlamB	Safety against micropitting, method B	
ZPP[0].Flanke.SEHT	Safety of the hardened layer	
ZPP[1].Flanke.SEHT	Safety of the hardened layer	
ZPP[2].Flanke.SEHT	Safety of the hardened layer	
ZPP[3].Flanke.SEHT	Safety of the hardened layer	
ZR[0].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	

ZR[2].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZP[0].Verlust.PVZ	Gear power loss	kW
ZP[1].Verlust.PVZ	Gear power loss	kW
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[2].ZchNr	Drawing or article number	
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[2].ZchNr	Drawing or article number	
ZS.Oil.roOil	Density at 15°C	kg/dm³
ZS.Oil.nu40	Nominal viscosity at 40°C	mm²/s
ZS.Oil.nu100	Nominal viscosity at 100°C	mm²/s
ZS.P_Limit	Power	kW
ZS.P_Usage	Stress	%
ZPP[0].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[1].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[2].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[3].flankbreak.SFFB	Safety against tooth flank fracture	
ZR[0].KSt.SW	Safety against wear	
ZR[1].KSt.SW	Safety against wear	
ZR[2].KSt.SW	Safety against wear	
ZS.SSi.Fuss	Required safety for tooth root	

ZS.SSi.Flanke	Required safety for tooth flank	
ZS.SSi.FrInt	Required scuffing safety (integral temperature)	
ZS.SSi.FrBli	Required scuffing safety (flash temperature)	
ZS.SSi.Slam	Required safety for frosting	
ZS.SSi.SFF	Required safety for tooth flank fracture	
ZS.SSi.VerfKSt	Required safety	
ZS.SSi.VerschleissSch	Required safety for wear	
ZS.Hatt	System service life	h
ZP[0].MP_ISO.mym	Coefficient of friction	
ZP[1].MP_ISO.mym	Coefficient of friction	
ZS.lastKElem	Calculation for load bin no.	
ZP[0].bv	Axial offset	mm
ZP[1].bv	Axial offset	mm
ZR[0].KSt.TempFuss	Tooth root temperature	°C
ZR[0].KSt.TempFlanke	Flank temperature	°C
ZR[1].KSt.TempFuss	Tooth root temperature	°C
ZR[1].KSt.TempFlanke	Flank temperature	°C
ZR[2].KSt.TempFuss	Tooth root temperature	°C
ZR[2].KSt.TempFlanke	Flank temperature	°C
ZS.BFSpiel.theRef	Reference temperature	°C
ZP[0].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[0].jt.E	Max. circumferential backlash	mm

ZP[0].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[0].cmin	Minimum tip clearance	mm
ZP[0].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[0].jtOP[1].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[1].cmin	Minimum tip clearance	mm
ZP[0].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[1].epsa.E	Max. transverse contact ratio	
ZP[1].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[1].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[1].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[1].jtOP[0].cmin	Minimum tip clearance	mm
ZP[1].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[1].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[1].jtOP[1].jt.i	Min. circumferential backlash	mm
ZP[1].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[1].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[1].jtOP[1].cmin	Minimum tip clearance	mm
ZP[1].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[1].jtOP[1].epsa.E	Max. transverse contact ratio	

ZR[0].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[1].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[2].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[0].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[1].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[2].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[0].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[1].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[2].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[0].RAH	Average surface roughness for Lx, pinion	µm
ZR[1].RAH	Average surface roughness for Lx, pinion	µm
ZR[2].RAH	Average surface roughness for Lx, pinion	µm
ZS.ReqSysReliability	Required reliability	%
ZR[0].YT	Technology factor	
ZR[1].YT	Technology factor	
ZR[2].YT	Technology factor	

64.3 Z015 Cylindrical Gear - Three Gears Train

ZR[0].z	Number of teeth	
ZR[1].z	Number of teeth	
ZR[2].z	Number of teeth	
ZP[0].a	Center distance	mm

ZP[1].a	Center distance	mm
ZR[0].x.nul	Addendum modification coefficient	
ZR[1].x.nul	Addendum modification coefficient	
ZR[2].x.nul	Addendum modification coefficient	
ZR[0].b	Face width	mm
ZR[1].b	Face width	mm
ZR[2].b	Face width	mm
ZR[0].Tnominal	Torque gear1	Nm
ZR[1].Tnominal	Torque gear2	Nm
ZR[2].Tnominal	Torque gear3	Nm
ZS.Pnominal	Nominal power	kW
ZR[0].nnominal	Absolute speed	1/min
ZR[1].nnominal	Absolute speed	1/min
ZR[2].nnominal	Absolute speed	1/min
ZR[0].Schrage	Helix direction	
ZR[1].Schrage	Helix direction	
ZR[2].Schrage	Helix direction	
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm
ZS.Geo.alfn	Normal pressure angle	°
ZS.Geo.alfn	Normal pressure angle	°

ZS.Geo.alfn	Normal pressure angle	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZR[0].d	Reference diameter	mm
ZR[1].d	Reference diameter	mm
ZR[2].d	Reference diameter	mm
ZPP[0].dw	Wälzkreisdurchmesser	mm
ZPP[1].dw	Pitch diameter	mm
ZPP[2].dw	Pitch diameter	mm
ZPP[3].dw	Pitch diameter	mm
ZR[0].da.nul	Tip diameter	mm
ZR[1].da.nul	Tip diameter	mm
ZR[2].da.nul	Tip diameter	mm
ZR[0].df.nul	Root diameter	mm
ZR[1].df.nul	Root diameter	mm
ZR[2].df.nul	Root diameter	mm
ZR[0].Fased	Tip chamfer	mm
ZR[1].Fased	Tip chamfer	mm
ZR[2].Fased	Tip chamfer	mm
ZPP[0].Fuss.SF	Root safety	
ZPP[1].Fuss.SF	Root safety	

ZPP[2].Fuss.SF	Root safety	
ZPP[3].Fuss.SF	Root safety	
ZPP[0].Flanke.SH	Safety factor for contact stress on flank	
ZPP[1].Flanke.SH	Safety factor for contact stress on flank	
ZPP[2].Flanke.SH	Safety factor for contact stress on flank	
ZPP[3].Flanke.SH	Safety factor for contact stress on flank	
ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[2].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZP[0].FIT.SSint	Safety against scuffing	
ZP[1].FIT.SSint	Safety against scuffing	
ZP[0].FBT.SB	Safety factor for scuffing (flash-temp)	
ZP[1].FBT.SB	Safety factor for scuffing (flash-temp)	
ZS.KA	Application factor	
ZS.H	Required service life	h
ZS.ED	Power-on time	%
ZS.jt.E	Total torsional angle	°
ZS.jt.i	Total torsional angle	°
ZR[0].Vqual	Accuracy grade	
ZR[1].Vqual	Accuracy grade	
ZR[2].Vqual	Accuracy grade	
ZS.Oil.theOil	Oil temperature	°C

ZP[0].KHdat.Belastung	Load according to DIN 3990-1:1987 Diagram 6.8	
ZP[1].KHdat.Belastung	Load according to DIN 3990-1:1987 Diagram 6.8	
ZS.ZeigerAufDr	During calculation, the system takes into account the fact that this gear is the reference gear	
ZS.theUmg	Ambient temperature	°C
ZP[0].alfwt	Working pressure angle	°
ZR[0].Tool.RefProfile.haP	Addendum coefficient	
ZR[1].Tool.RefProfile.haP	Addendum coefficient	
ZR[2].Tool.RefProfile.haP	Addendum coefficient	
ZR[0].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[1].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[2].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[0].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[1].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[2].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[0].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[1].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[2].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[0].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[1].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[2].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[0].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[1].Tool.RefProfile.alf_KP	Ramp angle	°

ZR[2].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[0].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[1].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[2].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[0].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[1].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[2].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[2].ZchNr	Drawing or article number	
ZR[0].di	Inner diameter	mm
ZR[1].di	Inner diameter	mm
ZR[2].di	Inner diameter	mm
ZR[0].Tool.Aufmass.E	Information on pre-machining	
ZR[1].Tool.Aufmass.E	Information on pre-machining	
ZR[2].Tool.Aufmass.E	Information on pre-machining	
ZR[0].Tool.q	Final machining stock	mm
ZR[1].Tool.q	Final machining stock	mm
ZR[2].Tool.q	Final machining stock	mm
ZR[0].Tool.Aufmass.i	Information on pre-machining	
ZR[1].Tool.Aufmass.i	Information on pre-machining	
ZR[2].Tool.Aufmass.i	Information on pre-machining	

ZR[0].Ca	Tip relief left/right	µm
ZR[1].Ca	Tip relief left/right	µm
ZR[2].Ca	Tip relief left/right	µm
ZR[0].Cf	Root relief, right flank	µm
ZR[1].Cf	Root relief, right flank	µm
ZR[2].Cf	Root relief, right flank	µm
ZR[0].kXmn	Tip alteration	mm
ZR[1].kXmn	Tip alteration	mm
ZR[2].kXmn	Tip alteration	mm
ZR[0].KopfKant	Section	
ZR[1].KopfKant	Section	
ZR[2].KopfKant	Section	
ZR[0].Ada.E	Tip diameter	mm
ZR[1].Ada.E	Tip diameter	mm
ZR[2].Ada.E	Tip diameter	mm
ZR[0].Ada.i	Tip diameter	mm
ZR[1].Ada.i	Tip diameter	mm
ZR[2].Ada.i	Tip diameter	mm
ZR[0].Adf.E	Root diameter	mm
ZR[1].Adf.E	Root diameter	mm
ZR[2].Adf.E	Root diameter	mm
ZR[0].Adf.i	Root diameter	mm

ZR[1].Adf.i	Root diameter	mm
ZR[2].Adf.i	Root diameter	mm
ZR[0].As.E	Tooth thickness tolerance, normal section	mm
ZR[1].As.E	Tooth thickness tolerance, normal section	mm
ZR[2].As.E	Tooth thickness tolerance, normal section	mm
ZR[0].As.i	Tooth thickness tolerance, normal section	mm
ZR[1].As.i	Tooth thickness tolerance, normal section	mm
ZR[2].As.i	Tooth thickness tolerance, normal section	mm
ZP[0].Ft	Nominal tangential force at base diameter	
ZP[1].Ft	Nominal tangential force at base diameter	
ZPP[0].Fa	Axial force	N
ZPP[1].Fa	Axial force	N
ZPP[2].Fa	Axial force	N
ZPP[3].Fa	Axial force	N
ZPP[0].Fr	Radial force	N
ZPP[1].Fr	Radial force	N
ZPP[2].Fr	Radial force	N
ZPP[3].Fr	Radial force	N
ZPP[0].Fnorm	Normal force	N
ZPP[1].Fnorm	Normal force	N
ZPP[2].Fnorm	Normal force	N
ZPP[3].Fnorm	Normal force	N

ZP[0].KV.KV	Dynamic factor	
ZP[1].KV.KV	Dynamic factor	
ZS.Geo.mt	Transverse module	
ZP[0].AXTolName	Center distance tolerance	
ZP[1].AXTolName	Center distance tolerance	
ZP[0].Aa.E	Center distance allowance	mm
ZP[1].Aa.E	Center distance allowance	mm
ZP[0].Aa.i	Center distance allowance	mm
ZP[1].Aa.i	Center distance allowance	mm
ZR[0].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[1].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[2].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[0].KM.Wk.E	Base tangent length	mm
ZR[1].KM.Wk.E	Base tangent length	mm
ZR[2].KM.Wk.E	Base tangent length	mm
ZR[0].KM.Wk.i	Base tangent length	mm
ZR[1].KM.Wk.i	Base tangent length	mm
ZR[2].KM.Wk.i	Base tangent length	mm
ZR[0].KM.k	Number of teeth spanned	
ZR[1].KM.k	Number of teeth spanned	
ZR[2].KM.k	Number of teeth spanned	
ZP[0].KHdat.l	Bearing distance l of pinion shaft	mm

ZP[1].KHdat.l	Bearing distance l of pinion shaft	mm
ZP[0].KHdat.S	Distance s of pinion shaft	mm
ZP[1].KHdat.S	Distance s of pinion shaft	mm
ZP[0].KHdat.dsh	Outside diameter of pinion shaft	mm
ZP[1].KHdat.dsh	Outside diameter of pinion shaft	mm
ZP[0].Eps.a	Transverse contact ratio	
ZP[1].Eps.a	Transverse contact ratio	
ZP[0].Eps.b	Overlap contact ratio	
ZP[1].Eps.b	Overlap contact ratio	
ZP[0].Eps.G	Total contact ratio	
ZP[1].Eps.G	Total contact ratio	
ZPP[0].zetaa	Specific sliding at the tip	
ZPP[1].zetaa	Specific sliding at the tip	
ZPP[2].zetaa	Specific sliding at the tip	
ZPP[3].zetaa	Specific sliding at the tip	
ZPP[0].zetaf	Specific sliding at the root	
ZPP[1].zetaf	Specific sliding at the root	
ZPP[2].zetaf	Specific sliding at the root	
ZPP[3].zetaf	Specific sliding at the root	
ZP[0].Flanke.sigH	Nominal contact stress	N/mm²
ZP[1].Flanke.sigH	Nominal contact stress	N/mm²
ZR[0].mat.limh	Strength against Hertzian pressure at NL	N/mm²

ZR[1].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZR[2].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZPP[0].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[1].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[2].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[3].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[0].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[1].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[2].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[3].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZR[0].mat.limf	Tooth root strength at NL	N/mm²
ZR[1].mat.limf	Tooth root strength at NL	N/mm²
ZR[2].mat.limf	Tooth root strength at NL	N/mm²
ZPP[0].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[1].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[2].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[3].Fuss.sigFP	Allowable bending stress	lb/in²
ZP[0].KHdat.fsh	from deformation of shaft	µm
ZP[1].KHdat.fsh	from deformation of shaft	µm
RechSt.RechenMethID	Calculation method	
ZS.WirkungsGrad	Meshing efficiency	%
ZS.Kgam	Mesh load factor	

ZP[0].MP_ISO.Slam	Safety against micropitting	
ZP[1].MP_ISO.Slam	Safety against micropitting	
ZP[0].MP_ISO.SlamB	Safety against micropitting, method B	
ZP[1].MP_ISO.SlamB	Safety against micropitting, method B	
ZPP[0].Flanke.SEHT	Safety of the hardened layer	
ZPP[1].Flanke.SEHT	Safety of the hardened layer	
ZPP[2].Flanke.SEHT	Safety of the hardened layer	
ZPP[3].Flanke.SEHT	Safety of the hardened layer	
ZP[0].Verlust.PVZ	Gear power loss	kW
ZP[1].Verlust.PVZ	Gear power loss	kW
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[2].ZchNr	Drawing or article number	
ZS.Oil.roOil	Density at 15°C	kg/dm³
ZS.Oil.nu40	Nominal viscosity at 40°C	mm²/s
ZS.Oil.nu100	Nominal viscosity at 100°C	mm²/s
ZS.settingsEL.lastfaktor	Partial load for calculation	
ZS.settingsEL.a	Center distance	mm
ZR[0].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZR[1].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZR[2].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZP[0].achsNeigung	Axis alignment, pair 1	µm

ZP[1].achsNeigung	Axis alignment, pair 1	µm
ZP[0].achsSchraenkung	Axis alignment, pair 1	µm
ZP[1].achsSchraenkung	Axis alignment, pair 1	µm
ZP[0].MP_ISO.Slam	Safety against micropitting	
ZP[1].MP_ISO.Slam	Safety against micropitting	
ZPP[0].delWnPC	Medium wear removal	mm
ZPP[1].delWnPC	Medium wear removal	mm
ZPP[2].delWnPC	Medium wear removal	mm
ZPP[3].delWnPC	Medium wear removal	mm
ZS.P_Limit	Power	kW
ZS.P_Usage	Stress	%
ZPP[0].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[1].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[2].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[3].flankbreak.SFFB	Safety against tooth flank fracture	
ZR[0].KSt.SW	Safety against wear	
ZR[1].KSt.SW	Safety against wear	
ZR[2].KSt.SW	Safety against wear	
ZS.SSi.Fuss	Required safety for tooth root	
ZS.SSi.Flanke	Required safety for tooth flank	
ZS.SSi.FrInt	Required scuffing safety (integral temperature)	
ZS.SSi.FrBli	Required scuffing safety (flash temperature)	

ZS.SSi.Slam	Required safety for frosting	
ZS.SSi.SFF	Required safety for tooth flank fracture	
ZS.Hatt	System service life	h
ZP[0].MP_ISO.mym	Coefficient of friction	
ZP[1].MP_ISO.mym	Coefficient of friction	
ZS.lastKElem	Calculation for load bin no.	
ZP[0].bv	Axial offset	mm
ZP[1].bv	Axial offset	mm
ZR[0].KSt.TempFuss	Tooth root temperature	°C
ZR[0].KSt.TempFlanke	Flank temperature	°C
ZR[1].KSt.TempFuss	Tooth root temperature	°C
ZR[1].KSt.TempFlanke	Flank temperature	°C
ZR[2].KSt.TempFuss	Tooth root temperature	°C
ZR[2].KSt.TempFlanke	Flank temperature	°C
ZS.BFSpiel.theRef	Reference temperature	°C
ZP[0].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[0].cmin	Minimum tip clearance	mm
ZP[0].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[0].jtOP[1].jt.i	Min. circumferential backlash	mm

ZP[0].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[1].cmin	Minimum tip clearance	mm
ZP[0].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[1].epsa.E	Max. transverse contact ratio	
ZP[1].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[1].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[1].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[1].jtOP[0].cmin	Minimum tip clearance	mm
ZP[1].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[1].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[1].jtOP[1].jt.i	Min. circumferential backlash	mm
ZP[1].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[1].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[1].jtOP[1].cmin	Minimum tip clearance	mm
ZP[1].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[1].jtOP[1].epsa.E	Max. transverse contact ratio	
ZR[0].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[1].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[2].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[0].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[1].RZH	Mean peak-to-valley roughness R_z, flank	µm

ZR[2].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[0].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[1].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[2].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[0].RAH	Average surface roughness for Lx, pinion	µm
ZR[1].RAH	Average surface roughness for Lx, pinion	µm
ZR[2].RAH	Average surface roughness for Lx, pinion	µm
ZS.ReqSysReliability	Required reliability	%
ZS.LowestPriority	Input power or torque	
ZR[0].YT	Technology factor	
ZR[1].YT	Technology factor	
ZR[2].YT	Technology factor	

64.4 Z016 Cylindrical Gear - Four Gears Train

ZR[0].z	Number of teeth	
ZR[1].z	Number of teeth	
ZR[2].z	Number of teeth	
ZR[3].z	Number of teeth	
ZP[0].a	Center distance	mm
ZP[1].a	Center distance	mm
ZP[2].a	Center distance	mm
ZR[0].x.nul	Addendum modification coefficient	

ZR[1].x.nul	Addendum modification coefficient	
ZR[2].x.nul	Addendum modification coefficient	
ZR[3].x.nul	Addendum modification coefficient	
ZR[0].b	Face width	mm
ZR[1].b	Face width	mm
ZR[2].b	Face width	mm
ZR[3].b	Face width	mm
ZR[0].Tnominal	Torque	Nm
ZR[1].Tnominal	Torque	Nm
ZR[2].Tnominal	Torque	Nm
ZR[3].Tnominal	Torque	Nm
ZS.Pnominal	Power	kW
ZS.KA	Application factor	
ZS.H	Required service life	h
ZS.ED	Power-on time	%
ZR[0].nnominal	Absolute speed	1/min
ZR[1].nnominal	Absolute speed	1/min
ZR[2].nnominal	Absolute speed	1/min
ZR[3].nnominal	Absolute speed	1/min
ZS.Geo.mt	Transverse module	mm
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm

ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	°
ZS.Geo.alfn	Pressure angle	°
ZS.Geo.alfn	Pressure angle	°
ZS.Geo.alfn	Pressure angle	°
ZS.Geo.alfn	Pressure angle	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZR[0].d	Reference diameter	mm
ZR[1].d	Reference diameter	mm
ZR[2].d	Reference diameter	mm
ZR[3].d	Reference diameter	mm
ZPP[0].dw	Pitch diameter	mm
ZPP[1].dw	Pitch diameter	mm
ZPP[2].dw	Pitch diameter	mm
ZPP[3].dw	Pitch diameter	mm
ZPP[4].dw	Pitch diameter	mm
ZPP[5].dw	Pitch diameter	mm
ZR[0].da.nul	Tip diameter	mm
ZR[1].da.nul	Tip diameter	mm

ZR[2].da.nul	Tip diameter	mm
ZR[3].da.nul	Tip diameter	mm
ZR[0].df.nul	Root diameter	mm
ZR[1].df.nul	Root diameter	mm
ZR[2].df.nul	Root diameter	mm
ZR[3].df.nul	Root diameter	mm
ZR[0].Fased	Tip chamfer	mm
ZR[1].Fased	Tip chamfer	mm
ZR[2].Fased	Tip chamfer	mm
ZR[3].Fased	Tip chamfer	mm
ZPP[0].Fuss.SF	Root safety	
ZPP[1].Fuss.SF	Root safety	
ZPP[2].Fuss.SF	Root safety	
ZPP[3].Fuss.SF	Root safety	
ZPP[4].Fuss.SF	Root safety	
ZPP[5].Fuss.SF	Root safety	
ZPP[0].Flanke.SH	Safety factor for contact stress on flank	
ZPP[1].Flanke.SH	Safety factor for contact stress on flank	
ZPP[2].Flanke.SH	Safety factor for contact stress on flank	
ZPP[3].Flanke.SH	Safety factor for contact stress on flank	
ZPP[4].Flanke.SH	Safety factor for contact stress on flank	
ZPP[5].Flanke.SH	Safety factor for contact stress on flank	

ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[2].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[3].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZP[0].FIT.SSint	Safety against scuffing	
ZP[1].FIT.SSint	Safety against scuffing	
ZP[2].FIT.SSint	Safety against scuffing	
ZP[0].FBT.SB	Safety factor for scuffing (flash-temp)	
ZP[1].FBT.SB	Safety factor for scuffing (flash-temp)	
ZP[2].FBT.SB	Safety factor for scuffing (flash-temp)	
ZS.jt.E	Total torsional angle	°
ZS.jt.i	Total torsional angle	°
ZR[0].Vqual	Accuracy grade	
ZR[1].Vqual	Accuracy grade	
ZR[2].Vqual	Accuracy grade	
ZR[3].Vqual	Accuracy grade	
ZS.Oil.theOil	Oil temperature	°C
ZP[0].KHdat.Belastung	Load according to DIN 3990-1:1987 Diagram 6.8	
ZP[1].KHdat.Belastung	Load according to DIN 3990-1:1987 Diagram 6.8	
ZP[2].KHdat.Belastung	Load according to DIN 3990-1:1987 Diagram 6.8	
ZP[0].Impulsnominal	Driving gear	
ZP[1].Impulsnominal	Driving gear	

ZP[2].Impulsnominal	Driving gear	
ZS.ZeigerAufDr	During calculation, the system takes into account the fact that this gear is the reference gear	
ZS.theUmg	Ambient temperature	°C
ZP[0].alfwt	Working pressure angle	°C
ZR[0].Tool.RefProfile.haP	Addendum coefficient	
ZR[1].Tool.RefProfile.haP	Addendum coefficient	
ZR[2].Tool.RefProfile.haP	Addendum coefficient	
ZR[3].Tool.RefProfile.haP	Addendum coefficient	
ZR[0].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[1].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[2].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[3].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[0].Tool.RefProfile.rhoaP	Tip radius coefficient	mm
ZR[1].Tool.RefProfile.rhoaP	Tip radius coefficient	mm
ZR[2].Tool.RefProfile.rhoaP	Tip radius coefficient	mm
ZR[3].Tool.RefProfile.rhoaP	Tip radius coefficient	mm
ZR[0].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[1].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[2].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[3].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[0].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[1].Tool.RefProfile.alf_prP	Protuberance angle	°

ZR[2].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[3].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[0].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[1].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[2].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[3].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[0].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[1].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[2].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[3].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[0].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[1].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[2].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[3].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[2].ZchNr	Drawing or article number	
ZR[3].ZchNr	Drawing or article number	
ZR[0].di	Inner diameter	mm
ZR[1].di	Inner diameter	mm
ZR[2].di	Inner diameter	mm
ZR[3].di	Inner diameter	mm

ZR[0].Tool.Aufmass.E	Information on pre-machining	
ZR[1].Tool.Aufmass.E	Information on pre-machining	
ZR[2].Tool.Aufmass.E	Information on pre-machining	
ZR[3].Tool.Aufmass.E	Information on pre-machining	
ZR[0].Tool.Aufmass.i	Information on pre-machining	
ZR[1].Tool.Aufmass.i	Information on pre-machining	
ZR[2].Tool.Aufmass.i	Information on pre-machining	
ZR[3].Tool.Aufmass.i	Information on pre-machining	
ZR[0].Tool.q	Final machining stock	mm
ZR[1].Tool.q	Final machining stock	mm
ZR[2].Tool.q	Final machining stock	mm
ZR[3].Tool.q	Final machining stock	mm
ZR[0].Ca	Tip relief left/right	µm
ZR[1].Ca	Tip relief left/right	µm
ZR[2].Ca	Tip relief left/right	µm
ZR[3].Ca	Tip relief left/right	µm
ZR[0].Cf	Root relief, right flank	µm
ZR[1].Cf	Root relief, right flank	µm
ZR[2].Cf	Root relief, right flank	µm
ZR[3].Cf	Root relief, right flank	µm
ZR[0].kXmn	Tip alteration	mm
ZR[1].kXmn	Tip alteration	mm

ZR[2].kXmn	Tip alteration	mm
ZR[3].kXmn	Tip alteration	mm
ZR[0].KopfKant	Section	
ZR[1].KopfKant	Section	
ZR[2].KopfKant	Section	
ZR[3].KopfKant	Section	
ZR[0].Ada.E	Tip diameter	mm
ZR[1].Ada.E	Tip diameter	mm
ZR[2].Ada.E	Tip diameter	mm
ZR[3].Ada.E	Tip diameter	mm
ZR[0].Ada.i	Tip diameter	mm
ZR[1].Ada.i	Tip diameter	mm
ZR[2].Ada.i	Tip diameter	mm
ZR[3].Ada.i	Tip diameter	mm
ZR[0].Adf.E	Root diameter	mm
ZR[1].Adf.E	Root diameter	mm
ZR[2].Adf.E	Root diameter	mm
ZR[3].Adf.E	Root diameter	mm
ZR[0].Adf.i	Root diameter	mm
ZR[1].Adf.i	Root diameter	mm
ZR[2].Adf.i	Root diameter	mm
ZR[3].Adf.i	Root diameter	mm

ZR[0].As.E	Tooth thickness tolerance, normal section	mm
ZR[1].As.E	Tooth thickness tolerance, normal section	mm
ZR[2].As.E	Tooth thickness tolerance, normal section	mm
ZR[3].As.E	Tooth thickness tolerance, normal section	mm
ZR[0].As.i	Tooth thickness tolerance, normal section	mm
ZR[1].As.i	Tooth thickness tolerance, normal section	mm
ZR[2].As.i	Tooth thickness tolerance, normal section	mm
ZR[3].As.i	Tooth thickness tolerance, normal section	mm
ZP[0].KHdat.FlankenLin	Flank line modification	
ZP[1].KHdat.FlankenLin	Flank line modification	
ZP[2].KHdat.FlankenLin	Flank line modification	
ZP[0].Ft	Nominal tangential force at base diameter	
ZP[1].Ft	Nominal tangential force at base diameter	
ZP[2].Ft	Nominal tangential force at base diameter	
ZPP[0].Fa	Axial force	N
ZPP[1].Fa	Axial force	N
ZPP[2].Fa	Axial force	N
ZPP[3].Fa	Axial force	N
ZPP[4].Fa	Axial force	N
ZPP[5].Fa	Axial force	N
ZPP[0].Fr	Radial force	N
ZPP[1].Fr	Radial force	N

ZPP[2].Fr	Radial force	N
ZPP[3].Fr	Radial force	N
ZPP[4].Fr	Radial force	N
ZPP[5].Fr	Radial force	N
ZPP[0].Fnorm	Normal force	N
ZPP[1].Fnorm	Normal force	N
ZPP[2].Fnorm	Normal force	N
ZPP[3].Fnorm	Normal force	N
ZPP[4].Fnorm	Normal force	N
ZPP[5].Fnorm	Normal force	N
ZP[0].KV.KV	Dynamic factor	
ZP[1].KV.KV	Dynamic factor	
ZP[2].KV.KV	Dynamic factor	
ZP[0].AXTolName	Center distance tolerance	
ZP[1].AXTolName	Center distance tolerance	
ZP[2].AXTolName	Center distance tolerance	
ZP[0].Aa.E	Center distance allowance	mm
ZP[1].Aa.E	Center distance allowance	mm
ZP[2].Aa.E	Center distance allowance	mm
ZP[0].Aa.i	Center distance allowance	mm
ZP[1].Aa.i	Center distance allowance	mm
ZP[2].Aa.i	Center distance allowance	mm

ZR[0].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[1].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[2].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[3].KM.Wk.nul	Base tangent length, no backlash	mm
ZR[0].KM.Wk.E	Base tangent length	mm
ZR[1].KM.Wk.E	Base tangent length	mm
ZR[2].KM.Wk.E	Base tangent length	mm
ZR[3].KM.Wk.E	Base tangent length	mm
ZR[0].KM.Wk.i	Base tangent length	mm
ZR[1].KM.Wk.i	Base tangent length	mm
ZR[2].KM.Wk.i	Base tangent length	mm
ZR[3].KM.Wk.i	Base tangent length	mm
ZR[0].KM.k	Number of teeth spanned	
ZR[1].KM.k	Number of teeth spanned	
ZR[2].KM.k	Number of teeth spanned	
ZR[3].KM.k	Number of teeth spanned	
ZP[0].KHdat.l	Bearing distance l of pinion shaft	mm
ZP[1].KHdat.l	Bearing distance l of pinion shaft	mm
ZP[2].KHdat.l	Bearing distance l of pinion shaft	mm
ZP[0].KHdat.S	Distance s of pinion shaft	mm
ZP[1].KHdat.S	Distance s of pinion shaft	mm
ZP[2].KHdat.S	Distance s of pinion shaft	mm

ZP[0].KHdat.dsh	Outside diameter of pinion shaft	mm
ZP[1].KHdat.dsh	Outside diameter of pinion shaft	mm
ZP[2].KHdat.dsh	Outside diameter of pinion shaft	mm
ZP[0].Eps.a	Transverse contact ratio	
ZP[1].Eps.a	Transverse contact ratio	
ZP[2].Eps.a	Transverse contact ratio	
ZP[0].Eps.b	Overlap contact ratio	
ZP[1].Eps.b	Overlap contact ratio	
ZP[2].Eps.b	Overlap contact ratio	
ZP[0].Eps.G	Total contact ratio	
ZP[1].Eps.G	Total contact ratio	
ZP[2].Eps.G	Total contact ratio	
ZPP[0].zetaa	Specific sliding at the tip	
ZPP[1].zetaa	Specific sliding at the tip	
ZPP[2].zetaa	Specific sliding at the tip	
ZPP[3].zetaa	Specific sliding at the tip	
ZPP[4].zetaa	Specific sliding at the tip	
ZPP[5].zetaa	Specific sliding at the tip	
ZPP[0].zetaf	Specific sliding at the root	
ZPP[1].zetaf	Specific sliding at the root	
ZPP[2].zetaf	Specific sliding at the root	
ZPP[3].zetaf	Specific sliding at the root	

ZPP[4].zetaf	Specific sliding at the root	
ZPP[5].zetaf	Specific sliding at the root	
ZP[0].Flanke.sigH	Nominal contact stress	N/mm²
ZP[1].Flanke.sigH	Nominal contact stress	N/mm²
ZP[2].Flanke.sigH	Nominal contact stress	N/mm²
ZR[0].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZR[1].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZR[2].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZR[3].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZPP[0].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[1].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[2].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[3].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[4].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[5].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[0].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[1].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[2].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[3].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[4].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[5].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZR[0].mat.limf	Tooth root strength at NL	N/mm²

ZR[1].mat.limf	Tooth root strength at NL	N/mm²
ZR[2].mat.limf	Tooth root strength at NL	N/mm²
ZR[3].mat.limf	Tooth root strength at NL	N/mm²
ZPP[0].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[1].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[2].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[3].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[4].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[5].Fuss.sigFP	Allowable bending stress	lb/in²
ZS.WirkungsGrad	Meshing efficiency	%
ZS.Kgam	Mesh load factor	
ZS.Oil.SchmierTyp	Lubrication type	
ZS.Kgam	Mesh load factor	
ZP[0].MP_ISO.Slam	Safety against micropitting	
ZP[1].MP_ISO.Slam	Safety against micropitting	
ZP[2].MP_ISO.Slam	Safety against micropitting	
ZP[0].MP_ISO.SlamB	Safety against micropitting, method B	
ZP[1].MP_ISO.SlamB	Safety against micropitting, method B	
ZP[2].MP_ISO.SlamB	Safety against micropitting, method B	
ZPP[0].Flanke.SEHT	Safety of the hardened layer	
ZPP[1].Flanke.SEHT	Safety of the hardened layer	
ZPP[2].Flanke.SEHT	Safety of the hardened layer	

ZPP[3].Flanke.SEHT	Safety of the hardened layer	
ZPP[4].Flanke.SEHT	Safety of the hardened layer	
ZPP[5].Flanke.SEHT	Safety of the hardened layer	
ZP[0].Verlust.PVZ	Gear power loss	kW
ZP[1].Verlust.PVZ	Gear power loss	kW
ZP[2].Verlust.PVZ	Gear power loss	kW
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[2].ZchNr	Drawing or article number	
ZR[3].ZchNr	Drawing or article number	
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[2].ZchNr	Drawing or article number	
ZR[3].ZchNr	Drawing or article number	
ZS.Oil.roOil	Density at 15°C	kg/dm³
ZS.Oil.nu40	Nominal viscosity at 40°C	mm²/s
ZS.Oil.nu100	Nominal viscosity at 100°C	mm²/s
ZS.settingsEL.lastfaktor	Partial load for calculation	
ZP[0].achsNeigung	Axis alignment, pair 1	µm
ZP[1].achsNeigung	Axis alignment, pair 1	µm
ZP[2].achsNeigung	Axis alignment, pair 1	µm
ZP[0].achsSchraenkung	Axis alignment, pair 1	µm

ZP[1].achsSchraenkung	Axis alignment, pair 1	µm
ZP[2].achsSchraenkung	Axis alignment, pair 1	µm
ZP[0].MP_ISO.Slam	Safety against micropitting	
ZP[1].MP_ISO.Slam	Safety against micropitting	
ZP[2].MP_ISO.Slam	Safety against micropitting	
ZPP[0].delWnPC	Medium wear removal	mm
ZPP[1].delWnPC	Medium wear removal	mm
ZPP[2].delWnPC	Medium wear removal	mm
ZPP[3].delWnPC	Medium wear removal	mm
ZPP[4].delWnPC	Medium wear removal	mm
ZPP[5].delWnPC	Medium wear removal	mm
ZS.P_Limit	Power	kW
ZS.P_Usage	Stress	%
ZPP[0].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[1].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[2].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[3].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[4].flankbreak.SFFB	Safety against tooth flank fracture	
ZPP[5].flankbreak.SFFB	Safety against tooth flank fracture	
ZR[0].KSt.SW	Safety against wear	
ZR[1].KSt.SW	Safety against wear	
ZR[2].KSt.SW	Safety against wear	

ZR[3].KSt.SW	Safety against wear	
ZS.SSi.Fuss	Required safety for tooth root	
ZS.SSi.Flanke	Required safety for tooth flank	
ZS.SSi.FrInt	Required scuffing safety (integral temperature)	
ZS.SSi.FrBli	Required scuffing safety (flash temperature)	
ZS.SSi.Slam	Required safety for frosting	
ZS.SSi.SFF	Required safety for tooth flank fracture	
ZS.SSi.VerfKSt	Required safety	
ZS.SSi.VerschleissSch	Required safety for wear	
ZP[0].MP_ISO.mym	Coefficient of friction	
ZP[1].MP_ISO.mym	Coefficient of friction	
ZP[2].MP_ISO.mym	Coefficient of friction	
ZS.lastKElem	Calculation for load bin no.	
ZP[0].bv	Axial offset	mm
ZP[1].bv	Axial offset	mm
ZP[2].bv	Axial offset	mm
ZR[0].KSt.TempFuss	Tooth root temperature	°C
ZR[0].KSt.TempFlanke	Flank temperature	°C
ZR[1].KSt.TempFuss	Tooth root temperature	°C
ZR[1].KSt.TempFlanke	Flank temperature	°C
ZR[2].KSt.TempFuss	Tooth root temperature	°C
ZR[2].KSt.TempFlanke	Flank temperature	°C

ZR[3].KSt.TempFuss	Tooth root temperature	°C
ZR[3].KSt.TempFlanke	Flank temperature	°C
ZS.BFSpiel.theRef	Reference temperature	°C
ZP[0].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[0].cmin	Minimum tip clearance	mm
ZP[0].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[0].jtOP[1].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[1].cmin	Minimum tip clearance	mm
ZP[0].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[1].epsa.E	Max. transverse contact ratio	
ZP[1].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[1].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[1].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[1].jtOP[0].cmin	Minimum tip clearance	mm
ZP[1].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[1].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[1].jtOP[1].jt.i	Min. circumferential backlash	mm

ZP[1].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[1].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[1].jtOP[1].cmin	Minimum tip clearance	mm
ZP[1].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[1].jtOP[1].epsa.E	Max. transverse contact ratio	
ZP[2].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[2].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[2].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[2].jtOP[0].cmin	Minimum tip clearance	mm
ZP[2].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[2].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[2].jtOP[1].jt.i	Min. circumferential backlash	mm
ZP[2].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[2].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[2].jtOP[1].cmin	Minimum tip clearance	mm
ZP[2].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[2].jtOP[1].epsa.E	Max. transverse contact ratio	
ZS.ReqSysReliability	Required reliability	%
ZR[0].YT	Technology factor	
ZR[1].YT	Technology factor	
ZR[2].YT	Technology factor	
ZR[3].YT	Technology factor	

64.5 Z070 Bevel and Hypoid Gears

ZkegR[0].z	Number of teeth	
ZkegR[1].z	Number of teeth	
ZkegP[0].a	Hypoid offset	mm
ZR[0].x.nul	Addendum modification coefficient	
ZR[1].x.nul	Addendum modification coefficient	
ZR[0].b	Face width	mm
ZR[1].b	Face width	mm
ZR[0].Tnominal	Torque	Nm
ZR[1].Tnominal	Torque	Nm
ZS.Pnominal	Power	kW
ZkegR[0].nnominal	Absolute speed	1/min
ZkegR[1].nnominal	Absolute speed	1/min
ZR[0].Schrage	Hand of gear	
ZR[1].Schrage	Hand of gear	
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm
ZS.Geo.alfn	Pressure angle	°
ZS.Geo.alfn	Pressure angle	°
ZkegP[0].alfnd.Coast	Nominal pressure angle - coast side	°
ZkegP[0].alfnd.Drive	Nominal pressure angle - drive side	°
ZkegP[0].alfne.Coast	Effective pressure angle - coast side	°

ZkegP[0].alfne.Coast	Effective pressure angle - coast side	°
ZkegP[0].alfne.Coast	Effective pressure angle - coast side	°
ZkegP[0].alfne.Drive	Effective pressure angle - drive side	°
ZkegP[0].alfne.Drive	Effective pressure angle - drive side	°
ZkegP[0].alfne.Drive	Effective pressure angle - drive side	°
ZkegR[0].delta	Pitch angle	°
ZkegR[1].delta	Pitch angle	°
ZkegR[0].WI.betm	Mean spiral angle	°
ZkegR[1].WI.betm	Mean spiral angle	°
ZR[0].Tool.RefProfile.DBID	Reference profile	
ZR[1].Tool.RefProfile.DBID	Reference profile	
ZR[0].mat.DBID	Material	
ZR[1].mat.DBID	Material	
ZkegR[0].dm	Mean pitch diameter	mm
ZkegR[1].dm	Mean pitch diameter	mm
ZkegR[0].dae	External diameter	mm
ZkegR[1].dae	External diameter	mm
ZkegR[0].dai	Mean diameter	mm
ZkegR[1].dai	Mean diameter	mm
ZkegR[0].dam	Mean tip diameter	mm
ZkegR[1].dam	Mean tip diameter	mm
ZkegR[0].dfe	Outer root diameter	mm

ZkegR[1].dfe	Outer root diameter	mm
ZkegR[0].dfi	Inner root diameter	mm
ZkegR[1].dfi	Inner root diameter	mm
ZkegR[0].dfm	Mean root diameter	mm
ZkegR[1].dfm	Mean root diameter	mm
ZR[0].di	Inner diameter	mm
ZR[1].di	Inner diameter	mm
ZPP[0].Fuss.SF	Root safety	
ZPP[1].Fuss.SF	Root safety	
ZPP[0].Flanke.SH	Safety factor for contact stress on flank	
ZPP[1].Flanke.SH	Safety factor for contact stress on flank	
ZP[0].FIT.SSint	Safety against scuffing	
ZS.KA	Application factor	
ZS.H	Required service life	h
ZS.ED	Power-on time	%
ZPP[1].jt	Circumferential backlash, transverse section	
ZPP[0].jt	Circumferential backlash, transverse section	
ZR[0].Vqual	Accuracy grade	
ZR[1].Vqual	Accuracy grade	
ZS.Oil.SchmierTypID	Lubricant	
ZS.Oil.theOil	Oil temperature	°C
ZS.Oil.SchmierungsArt	Lubrication type	

ZP[0].Impulsnominal	Driving gear	
ZP[0].ImpulsUI	Driving gear	
ZS.ZeigerAufDr	During calculation, the system takes into account the fact that this gear is a planet gear	
ZR[0].Tool.RefProfile.haP	Addendum coefficient	
ZR[1].Tool.RefProfile.haP	Addendum coefficient	
ZR[0].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[1].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[0].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[1].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[0].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[1].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[0].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[1].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[0].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[1].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[0].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[1].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[0].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[1].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[0].ZDTolID	Tooth thickness allowance	
ZR[1].ZDTolID	Tooth thickness allowance	
ZR[0].ZchNr	Drawing or article number	

ZR[1].ZchNr	Drawing or article number	
ZR[0].di	Inner diameter	mm
ZR[1].di	Inner diameter	mm
ZR[0].Ca	Tip relief left/right	µm
ZR[1].Ca	Tip relief left/right	µm
ZR[0].Cf	Root relief, right flank	µm
ZR[1].Cf	Root relief, right flank	µm
ZR[0].KopfKant	Section	
ZR[1].KopfKant	Section	
ZR[0].Ada.E	Tip diameter	mm
ZR[1].Ada.E	Tip diameter	mm
ZR[0].Ada.i	Tip diameter	mm
ZR[1].Ada.i	Tip diameter	mm
ZR[0].Adf.E	Root diameter	mm
ZR[1].Adf.E	Root diameter	mm
ZR[0].Adf.i	Root diameter	mm
ZR[1].Adf.i	Root diameter	mm
ZR[0].As.E	Tooth thickness tolerance, normal section	mm
ZR[1].As.E	Tooth thickness tolerance, normal section	mm
ZR[0].As.i	Tooth thickness tolerance, normal section	mm
ZR[1].As.i	Tooth thickness tolerance, normal section	mm
ZR[0].crowning	Crowning	

ZR[1].crowning	Crowning	
ZP[0].Ft	Nominal tangential force at base diameter	
ZPP[0].Fa	Axial force	N
ZPP[1].Fa	Axial force	N
ZPP[0].Fr	Radial force	N
ZPP[1].Fr	Radial force	N
ZPP[0].Fnorm	Normal force	N
ZR[0].kXmn	Tip alteration	mm
ZR[1].kXmn	Tip alteration	mm
ZS.theUmg	Ambient temperature	°C
ZP[0].Flanke.sigH	Nominal contact stress	N/mm²
ZR[0].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZR[1].mat.limh	Strength against Hertzian pressure at NL	N/mm²
ZPP[0].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[1].Flanke.sigHP	Permissible contact stress	lb/in²
ZPP[0].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZPP[1].Fuss.sigF	σF0 without gear rim factor	N/mm²
ZR[0].mat.limf	Tooth root strength at NL	N/mm²
ZR[1].mat.limf	Tooth root strength at NL	N/mm²
ZPP[0].Fuss.sigFP	Allowable bending stress	lb/in²
ZPP[1].Fuss.sigFP	Allowable bending stress	lb/in²
ZP[0].Sigma	Shaft angle	°

RechSt.RechenMethID	Calculation method	
ZS.WirkungsGrad	Meshing efficiency	%
ZR[0].Tool.Aufmass.E	Information on pre-machining	
ZR[1].Tool.Aufmass.E	Information on pre-machining	
ZR[0].Tool.Aufmass.i	Information on pre-machining	
ZR[1].Tool.Aufmass.i	Information on pre-machining	
ZR[0].Tool.q	Final machining stock	mm
ZR[1].Tool.q	Final machining stock	mm
ZS.Oil.theOil	Oil temperature	°C
ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZP[0].KHb_nominal	Face load factor	
ZS.LK.ID	Load spectrum	
ZS.LK.ID	Load spectrum	
ZS.LKflag	Consider load spectrum	
ZP[0].Verlust.PVZ	Gear power loss	kW
ZkegP[0].KHbbe	Load distribution modifier	
ZP[0].FBT.SB	Safety factor for scuffing (flash-temp)	
ZS.P_Limit	Power	kW
ZS.P_Usage	Stress	%
ZPP[0].flankbreak.SFFB	Safety against tooth flank fracture	

ZPP[1].flankbreak.SFFB	Safety against tooth flank fracture	
ZS.SSi.Fuss	Required safety for tooth root	
ZS.SSi.Flanke	Required safety for tooth flank	
ZS.SSi.FrInt	Required scuffing safety (integral temperature)	
ZS.SSi.FrBli	Required scuffing safety (flash temperature)	
ZS.SSi.SFF	Required safety for tooth flank fracture	
ZS.SSi.VerfKSt	Required safety	
ZS.SSi.VerschleissSch	Required safety for wear	
ZS.Hatt	System service life	h
RechSt.BevelMeth_mum	Calculation method for coefficient of friction	
ZP[0].Verlust.mum	Coefficient of friction	
ZR[0].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZR[1].torqueDirection	Torque, 0: -, 1: < I, 2: < II, 3: < from shaft calculation	
ZS.settingsEL.lastfaktor	Partial load for calculation	
ZS.settingsEL.a	Center distance	
caResults.AchsNeigung	Inclination error of axis	
caResults.AchsSchraenkung	Deviation error of axis	
caResults.TransmissionError.delta	Transmission error	
caResults.TangentStiffness.stddeviation	Tangents Stiffness curve	
caResults.ContactTemperature.max	Contact temperature	
caResults.TransverseContactRatio	Transverse contact ratio under load (max)	

caResults.PowerLoss.average	Power loss	
caResults.WearGearA.average	Wear gear 1	
caResults.WearGearB.average	Wear gear 2	
ZS.lastKElem	Calculation for load bin no.	
ZR[0].KSt.TempFuss	Tooth root temperature	°C
ZR[0].KSt.TempFlanke	Flank temperature	°C
ZR[1].KSt.TempFuss	Tooth root temperature	°C
ZR[1].KSt.TempFlanke	Flank temperature	°C
ZR[0].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[1].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[0].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[1].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[0].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[1].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[0].RAH	Average surface roughness for Lx, pinion	µm
ZR[1].RAH	Average surface roughness for Lx, pinion	µm
ZS.ReqSysReliability	Required reliability	%
ZS.Planet.nStegnominal	Speed planet carrier	1/min
ZkegR[0].MountD	Mounting distance	mm
ZkegR[1].MountD	Mounting distance	mm
ZkegP[0].isZerol	Zerol	
ZkegP[0].isZerol	Zerol	

ZR[0].YT	Technology factor	
ZR[1].YT	Technology factor	

64.6 Z060 Face Gears

ZkegR[0].z	Number of teeth	
ZkegR[1].z	Number of teeth	
ZkegP[0].a	Hypoid offset	mm
ZS.Pnominal	Power	kW
ZkegR[0].nnominal	Absolute speed	1/min
ZkegR[1].nnominal	Absolute speed	1/min
ZR[0].Tnominal	Torque	Nm
ZR[1].Tnominal	Torque	Nm
ZS.KA	Application factor	
ZS.H	Required service life	h
ZP[0].Sigma	Shaft angle	°
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm
ZS.Geo.alfn	Pressure angle	°
ZS.Geo.alfn	Pressure angle	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZR[0].Schrage	Hand of gear	

ZR[1].Schrage	Hand of gear	
ZR[0].Tool.RefProfile.DBID	Reference profile	
ZR[1].Tool.RefProfile.DBID	Reference profile	
ZR[0].b	Face width	mm
ZR[1].b	Face width	mm
ZkegR[1].bv	Axial offset	mm
ZR[0].Vqual	Accuracy grade	
ZR[1].Vqual	Accuracy grade	
ZR[0].mat.DBID	Material	
ZR[1].mat.DBID	Material	
ZS.Oil.SchmierTypID	Lubricant	
ZS.Oil.SchmierungsArt	Lubrication type	
ZS.Oil.theOil	Oil temperature	°C
ZS.theUmg	Ambient temperature	°C
ZkegR[1].dae	External diameter	mm
ZkegR[1].dai	Mean diameter	mm
ZR[1].Hz	Heights of face gear	mm
ZR[1].Hf	Height of gear body under tooth root	mm
ZR[0].d	Reference diameter	mm
ZR[0].da.nul	Tip diameter	mm
ZPP[0].dw	Pitch diameter	mm
ZR[0].df.nul	Root diameter	mm

ZP[0].Ft	Nominal tangential force at base diameter	
ZPP[0].Fnorm	Normal force	N
ZPP[0].Fa	Axial force	N
ZPP[1].Fa	Axial force	N
ZPP[0].Fr	Radial force	N
ZPP[1].Fr	Radial force	N
ZP[0].KHb_nominal	Face load factor	
ZP[0].KHa	Transverse load factor	
ZP[0].KV.KV	Dynamic factor	
ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[0].x.nul	Addendum modification coefficient	
ZR[1].x.nul	Addendum modification coefficient	
ZPP[1].jt	Circumferential backlash, transverse section	mm
ZPP[0].jt	Circumferential backlash, transverse section	mm
ZS.ZeigerAufDr	During calculation, the system takes into account the fact that this gear is a planet gear	
ZR[0].Tool.RefProfile.haP	Addendum coefficient	
ZR[1].BP_f.haP	Addendum reference profile, in module	
ZR[0].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[1].BP_f.hfP	Dedendum reference profile, in module	
ZR[0].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[0].Tool.RefProfile.rhoaP	Tip radius coefficient	

ZR[0].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[0].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[0].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[0].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[0].ZDTolID	Tooth thickness allowance	
ZR[1].ZDTolID	Tooth thickness allowance	
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[0].di	Inner diameter	mm
ZR[1].di	Inner diameter	mm
ZR[0].Ada.E	Tip diameter	mm
ZR[1].Ada.E	Tip diameter	mm
ZR[0].Ada.i	Tip diameter	mm
ZR[1].Ada.i	Tip diameter	mm
ZR[0].Adf.E	Root diameter	mm
ZR[1].Adf.E	Root diameter	mm
ZR[0].Adf.i	Root diameter	mm
ZR[1].Adf.i	Root diameter	mm
ZR[0].As.E	Tooth thickness tolerance, normal section	mm
ZR[1].As.E	Tooth thickness tolerance, normal section	mm
ZR[0].As.i	Tooth thickness tolerance, normal section	mm
ZR[1].As.i	Tooth thickness tolerance, normal section	mm

ZR[0].kXmn	Tip alteration	mm
ZR[1].kXmn	Tip alteration	mm
ZPP[0].Fuss.SF	Root safety	
ZPP[1].Fuss.SF	Root safety	
ZPP[0].Flanke.SH	Safety factor for contact stress on flank	
ZPP[1].Flanke.SH	Safety factor for contact stress on flank	
RechSt.RechenMethID	Calculation method	
ZR[0].Ca	Tip relief left/right	µm
ZR[1].Ca	Tip relief left/right	µm
ZR[0].Tool.Aufmass.E	Information on pre-machining	
ZR[0].Tool.Aufmass.i	Information on pre-machining	
ZR[1].Tool.Aufmass.E	Information on pre-machining	
ZR[1].Tool.Aufmass.i	Information on pre-machining	
ZR[0].Tool.q	Final machining stock	mm
ZS.SSi.Fuss	Required safety for tooth root	
ZS.SSi.Flanke	Required safety for tooth flank	
ZS.SSi.FrInt	Required scuffing safety (integral temperature)	
ZS.SSi.FrBli	Required scuffing safety (flash temperature)	
ZS.SSi.Slam	Required safety for frosting	
ZS.SSi.SFF	Required safety for tooth flank fracture	
ZS.SSi.VerfKSt	Required safety	
ZS.SSi.VerschleissSch	Required safety for wear	

ZS.Hatt	System service life	
ZS.lastKElem	Calculation for load bin no.	
ZR[0].KSt.TempFuss	Tooth root temperature	°C
ZR[0].KSt.TempFlanke	Flank temperature	°C
ZR[1].KSt.TempFuss	Tooth root temperature	°C
ZR[1].KSt.TempFlanke	Flank temperature	°C
ZR[0].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[1].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[0].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[1].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[0].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[1].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[0].RAH	Average surface roughness for Lx, pinion	µm
ZR[1].RAH	Average surface roughness for Lx, pinion	µm
ZS.ReqSysReliability	Required reliability	%
ZR[1].hiFG	Gear body height, inside	mm
ZR[0].YT	Technology factor	
ZR[1].YT	Technology factor	

64.7 Z080 Worms with Enveloping Worm Wheels

ZR[0].z	Number of teeth	
ZR[1].z	Number of teeth	

ZS.Pnominal	Nominal power	
ZR[0].Schn.P	Power	kW
ZR[1].Schn.P	Power	kW
ZR[0].nnominal	Absolute speed	1/min
ZR[1].nnominal	Absolute speed	1/min
ZR[0].Tnominal	Nominal torque	Nm
ZR[1].Tnominal	Nominal torque	Nm
ZR[0].b	Face width	mm
ZR[1].b	Face width	mm
ZR[1].Schn.b2R	Worm wheel rim width b2R	mm
ZR[1].Schn.b2H	Worm gear wheel width b2H	mm
ZP[0].a	Center distance	mm
ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZR[0].Schrage	Hand of gear	
ZR[1].Schrage	Hand of gear	
ZR[0].mat.DBID	Material	
ZR[1].mat.DBID	Material	
ZS.Geo.mt	Transverse module	mm
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm
ZS.Oil.SchmierungsArt	Lubrication type	

ZS.Oil.SchmierTypID	Lubrication	
ZS.Geo.alfn	Normal pressure angle	°
ZS.Geo.alfn	Normal pressure angle	°
ZR[0].x.nul	Addendum modification coefficient	
ZR[1].x.nul	Addendum modification coefficient	
ZR[0].Schn.dm	Reference operating diameter	mm
ZR[1].Schn.dm	Reference operating diameter	mm
ZR[1].d	Reference diameter	mm
ZR[0].da.nul	Tip diameter	mm
ZR[1].da.nul	Tip diameter	mm
ZR[0].df.nul	Root diameter	mm
ZR[1].df.nul	Root diameter	mm
ZR[0].Tool.RefProfile.DBID	Reference profile	
ZR[1].Tool.RefProfile.DBID	Reference profile	
ZR[0].Vqual	Accuracy grade	
ZR[1].Vqual	Accuracy grade	
ZP[0].Schn.SW	Safety against wear	
ZPP[1].Flanke.SH	Safety factor for contact stress on flank	
ZP[0].Schn.Sdel	Safety against deflection	
ZPP[1].Fuss.SF	Root safety	
ZS.Schn.ST	Temperature safety	
ZS.KA	Application factor	

ZS.H	Required service life	h
ZS.ED	Power-on time	%
ZS.Oil.theOil	Oil temperature	°C
ZS.theUmg	Ambient temperature	°C
ZP[0].KHdat.l	Bearing distance l of pinion shaft	mm
ZP[0].KHdat.S	Distance s of pinion shaft	mm
ZS.ZeigerAufDr	During calculation, the system takes into account the fact that this gear is a planet gear	
ZR[0].Tool.RefProfile.haP	Addendum coefficient	
ZR[1].Tool.RefProfile.haP	Addendum coefficient	
ZR[0].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[1].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[0].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[1].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[0].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[1].Tool.RefProfile.rhoaP	Tip radius coefficient	
ZR[0].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[1].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[0].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[0].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[1].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[0].Tool.RefProfile.hFaP	Tip form height coefficient	
ZS.Schn.alfa0	Generating angle	°

ZP[0].Schn.q	Diametral factor q	
ZP[0].Schn.etaz	Meshing efficiency	%
ZS.Schn.etaGes	Total efficiency	%
ZPP[1].jt	Circumferential backlash, transverse section	mm
ZPP[0].jt	Circumferential backlash, transverse section	mm
ZR[0].ZDTolID	Tooth thickness allowance	
ZR[1].ZDTolID	Tooth thickness allowance	
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[0].Ada.E	Tip diameter	mm
ZR[0].Ada.i	Tip diameter	mm
ZR[1].Ada.E	Tip diameter	mm
ZR[1].Ada.i	Tip diameter	mm
ZR[0].Adf.E	Root diameter	mm
ZR[1].Adf.E	Root diameter	mm
ZR[0].Adf.i	Root diameter	mm
ZR[1].Adf.i	Root diameter	mm
ZR[0].As.E	Tooth thickness tolerance, normal section	mm
ZR[1].As.E	Tooth thickness tolerance, normal section	mm
ZR[0].As.i	Tooth thickness tolerance, normal section	mm
ZR[1].As.i	Tooth thickness tolerance, normal section	mm
ZPP[0].Schn.Ft	Nominal circumferential force at operating pitch circle	N

ZPP[1].Schn.Ft	Nominal circumferential force at operating pitch circle	N
ZPP[0].Fa	Axial force	N
ZPP[1].Fa	Axial force	N
ZPP[0].Fr	Radial force	N
ZPP[1].Fr	Radial force	N
ZPP[0].Fnorm	Normal force	N
RechSt.RechenMethID	Calculation method	
ZS.Schn.PVZ	Meshing power loss	kW
ZS.SSi.Fuss	Required safety for tooth root	
ZS.SSi.Flanke	Required safety for tooth flank	
ZS.SSi.VerschleissSch	Required safety for wear	
ZS.SSi.DurchbiegeSch	Required safety for deflection	
ZS.SSi.TemperaturSch	Required safety for temperature	
ZS.Hatt	System service life	h
ZS.lastKElem	Calculation for load bin no.	
ZR[0].KSt.TempFuss	Tooth root temperature	°C
ZR[0].KSt.TempFlanke	Flank temperature	°C
ZR[1].KSt.TempFuss	Tooth root temperature	°C
ZR[1].KSt.TempFlanke	Flank temperature	°C
ZS.BFSpiel.theRef	Reference temperature	°C
ZS.BFSpiel.theGehmin	Housing temperature, min ... max	°C
ZS.BFSpiel.theGehmax	Housing temperature, min ... max	°C

ZS.BFSpiel.theRadmin	Gear body temperature, min ... max	°C
ZS.BFSpiel.theRadmax	Gear body temperature, min ... max	°C
ZS.BFSpiel.theDifmin	Permitted temperature difference	Δ°C
ZS.BFSpiel.theDifmax	Permitted temperature difference	Δ°C
ZP[0].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[0].cmin	Minimum tip clearance	mm
ZP[0].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[0].jtOP[1].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[1].cmin	Minimum tip clearance	mm
ZP[0].jtOP[1].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[1].epsa.E	Max. transverse contact ratio	
ZR[0].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[1].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[0].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[1].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[0].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[1].RAF	Arithmetic mean roughness value R_a, root	µm

ZR[0].RAH	Average surface roughness for Lx, pinion	µm
ZR[1].RAH	Average surface roughness for Lx, pinion	µm
ZR[0].di	Inner diameter	mm
ZR[1].di	Inner diameter	mm
ZS.ReqSysReliability	Required reliability	%
ZR[0].YT	Technology factor	
ZR[1].YT	Technology factor	

64.8 Z170 Crossed Helical Gears

ZR[0].z	Number of teeth	
ZR[1].z	Number of teeth	
ZP[0].a	Center distance	mm
ZR[0].x.nul	Addendum modification coefficient	
ZR[1].x.nul	Addendum modification coefficient	
ZR[0].b	Face width	mm
ZR[1].b	Face width	mm
ZR[0].nnominal	Absolute speed	1/min
ZR[1].nnominal	Absolute speed	1/min
ZS.Geo.mn	Normal module	mm
ZS.Geo.mn	Normal module	mm
ZS.Geo.alfn	Normal pressure angle	°
ZS.Geo.alfn	Normal pressure angle	°

ZS.Geo.beta	Helix angle at reference diameter	°
ZS.Geo.beta	Helix angle at reference diameter	°
ZR[1].SR_beta	Helix angle	°
ZR[0].SR_gamma	Lead angle at reference diameter	°
ZR[1].SR_gamma	Lead angle at reference diameter	°
ZR[0].d	Reference diameter	mm
ZR[1].d	Reference diameter	mm
ZPP[0].dw	Pitch diameter	mm
ZPP[1].dw	Pitch diameter	mm
ZR[0].da.nul	Tip diameter	mm
ZR[1].da.nul	Tip diameter	mm
ZR[0].df.nul	Root diameter	mm
ZR[1].df.nul	Root diameter	mm
ZR[0].Fased	Tip chamfer	mm
ZR[1].Fased	Tip chamfer	mm
ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZP[0].FBT.SB	Safety factor for scuffing (flash-temp)	
ZS.KA	Application factor	
ZS.H	Required service life	h
ZS.ED	Power-on time	%
ZR[0].Vqual	Accuracy grade	

ZR[1].Vqual	Accuracy grade	
ZP[0].KHdat.Belastung	Load according to DIN 3990-1:1987 Diagram 6.8	
ZS.ZeigerAufDr	During calculation, the system takes into account the fact that this gear is a planet gear	
ZS.theUmg	Ambient temperature	°C
ZS.Oil.theOil	Oil temperature	°C
ZP[0].alfwt	Working pressure angle	°
ZR[0].Tool.RefProfile.haP	Addendum coefficient	
ZR[1].Tool.RefProfile.haP	Addendum coefficient	
ZR[0].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[1].Tool.RefProfile.hfP	Dedendum coefficient	
ZR[0].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[1].Tool.RefProfile.rhofP	Root radius coefficient	
ZR[0].Tool.RefProfile.rhoaP	Tip radius	mm
ZR[1].Tool.RefProfile.rhoaP	Tip radius	mm
ZR[0].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[1].Tool.RefProfile.alf_prP	Protuberance angle	°
ZR[0].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[1].Tool.RefProfile.alf_KP	Ramp angle	°
ZR[0].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[1].Tool.RefProfile.hprP	Protuberance height coefficient	
ZR[0].Tool.RefProfile.hFaP	Tip form height coefficient	
ZR[1].Tool.RefProfile.hFaP	Tip form height coefficient	

ZP[0].Schn.etaz	Meshing efficiency	%
ZS.Schn.etaGes	Total efficiency	%
ZPP[2].Flanke.SH	Safety factor for contact stress on flank	
ZS.jt.E	Total torsional angle	°
ZS.jt.i	Total torsional angle	°
ZPP[0].Flanke.SH	Safety factor for contact stress on flank	
ZPP[1].Flanke.SH	Safety factor for contact stress on flank	
ZPP[0].Fuss.SF	Root safety	
ZPP[1].Fuss.SF	Root safety	
ZP[0].FIT.SSint	Safety against scuffing	
ZP[0].FIT.SSL	Safety referring to transmittable torque	
ZP[0].Sigma	Shaft angle	°
ZR[0].ZchNr	Drawing or article number	
ZR[1].ZchNr	Drawing or article number	
ZR[0].Ada.E	Tip diameter	mm
ZR[1].Ada.E	Tip diameter	mm
ZR[0].Ada.i	Tip diameter	mm
ZR[1].Ada.i	Tip diameter	mm
ZR[0].Adf.E	Root diameter	mm
ZR[1].Adf.E	Root diameter	mm
ZR[0].Adf.i	Root diameter	mm
ZR[1].Adf.i	Root diameter	mm

ZR[0].As.E	Tooth thickness tolerance, normal section	mm
ZR[1].As.E	Tooth thickness tolerance, normal section	mm
ZR[0].As.i	Tooth thickness tolerance, normal section	mm
ZR[1].As.i	Tooth thickness tolerance, normal section	mm
ZPP[0].Schn.Ft	Nominal circumferential force at operating pitch circle	N
ZPP[1].Schn.Ft	Nominal circumferential force at operating pitch circle	N
ZPP[0].Fa	Axial force	N
ZPP[1].Fa	Axial force	N
ZPP[0].Fr	Radial force	N
ZPP[1].Fr	Radial force	N
ZPP[0].Fnorm	Normal force	N
RechSt.RechenMethID	Calculation method	
ZS.Oil.SchmierTyp	Lubrication type	
ZS.Schn.PVZ	Meshing power loss	kW
ZS.P_Limit	Power	kW
ZS.P_Usage	Stress	%
ZS.SSi.Fuss	Required safety for tooth root	
ZS.SSi.Flanke	Required safety for tooth flank	
ZS.SSi.FrInt	Required scuffing safety (integral temperature)	
ZS.SSi.VerfKSt	Required safety	
ZS.Hatt	System service life	h
ZS.lastKElem	Calculation for load bin no.	

ZR[0].KSt.TempFuss	Tooth root temperature	°C
ZR[0].KSt.TempFlanke	Flank temperature	°C
ZR[1].KSt.TempFuss	Tooth root temperature	°C
ZR[1].KSt.TempFlanke	Flank temperature	°C
ZS.BFSpiel.theRef	Reference temperature	°C
ZS.BFSpiel.theGehmin	Housing temperature, min ... max	°C
ZS.BFSpiel.theGehmax	Housing temperature, min ... max	°C
ZS.BFSpiel.theRadmin	Gear body temperature, min ... max	°C
ZS.BFSpiel.theRadmax	Gear body temperature, min ... max	°C
ZS.BFSpiel.theDifmin	Permitted temperature difference	Δ°C
ZS.BFSpiel.theDifmax	Permitted temperature difference	Δ°C
ZP[0].jtOP[0].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[0].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[0].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[0].cmin	Minimum tip clearance	mm
ZP[0].jtOP[0].epsa.i	Min. transverse contact ratio	
ZP[0].jtOP[0].epsa.E	Max. transverse contact ratio	
ZP[0].jtOP[1].jt.i	Min. circumferential backlash	mm
ZP[0].jtOP[1].jt.E	Max. circumferential backlash	mm
ZP[0].jtOP[1].dfpt	Pitch error due to dilatation	µm
ZP[0].jtOP[1].cmin	Minimum tip clearance	mm
ZP[0].jtOP[1].epsa.i	Min. transverse contact ratio	

ZP[0].jtOP[1].epsa.E	Max. transverse contact ratio	
ZR[0].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[1].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[0].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[1].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[0].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[1].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[0].RAH	Average surface roughness for Lx, pinion	µm
ZR[1].RAH	Average surface roughness for Lx, pinion	µm
ZR[0].di	Inner diameter	mm
ZR[1].di	Inner diameter	mm
ZS.ReqSysReliability	Required reliability	%
ZR[0].YT	Technology factor	
ZR[1].YT	Technology factor	

64.9 W010 Shaft Calculation

Qu[0].sErmuedung	Fatigue safety	
Qu[0].sStatisch	Static safety	
Qu[0].sResErmuedung	Fatigue result	%
Qu[0].sResStatisch	Static result	%
Qu[0].sAnriss	Safety against incipient crack	
WelG.speed	Speed	1/min

WelG.rotateCounterClockwise	Sense of rotation	
WelG.AnzahlDr	Number of eigenfrequencies	
Bieg.X.BiegMin	Minimum deflection x	
Bieg.X.BiegMax	Maximum deflection x	
Bieg.Z.BiegMin	Minimum deflection z	
Bieg.Z.BiegMax	Maximum deflection z	
WelG.GVector[0]	Weight towards x	
WelG.GVector[1]	Weight towards y	
WelG.GVector[2]	Weight towards z	
WelG.LageFlag	Position of shaft axis	
WelG.Oil.theOil	Lubricant temperature	
WelG.Oil.SchmierTypID	Lubricant	
WelG.W050Allg_ErwLebensd	Modified rating life	
WelG.housingTemperature	Housing temperature	°C
WelG.Oil.V_Id	Contamination	
WelG.WLagerBeruecksichtigen	Calculation method bearing	
WelG.W050Allg_SollLeben	Required rating life	h
WelG.Lebensdauer	Required rating life shaft	h
WelG.aktivLK	Consider load spectrum	
WelG.frictionMeth	Friction method bearings	
WelG.oelstand	Oil level	mm
WelG.lubricationType	Lubrication type	

shafts[0].outerGeometry	Outer geometry	
shafts[0].innerGeometry	Inner geometry	
shafts[0].name	Name	
shafts[0].drawingNumber	Drawing	
shafts[0].mass	Mass of shaft, including additional masses	kg
shafts[0].length	Length	mm
shafts[0].temperature	Temperature	°C
shafts[0].material.DBID	Material list selection	
shafts[0].material.bez	Material description	
shafts[0].w060.WerkstArtStr	Material type	
shafts[0].w060.WerkstBehStr	Material treatment	
shafts[0].w060.OberflVerfStr	Surface treatment	
shafts[0].w060.Hatt	Calculated life	h
shafts[0].w060.usage	Maximum utilization	
shafts[0].w060.damage	Damage	
WelG.referenceTemperature	Reference temperature	°C
WelG.housingThermalReference	Thermal housing reference point	mm
WelG.considerGearOffsets	Consider gear offsets	
W060Allg.Haen.sSollStreck	Nominal safety	
W060Allg.Din.sSollStreck	Required safeties	
W060Allg.AGMA.sSafetyStatic	Required safety	
W060Allg.Din.sSollErmuedung	Required safeties	

W060Allg.AGMA.sSafetyFatigue	Required safety against fatigue	
W060Allg.Haen.sSollBruch	Nominal safety	
WelG.flagoelstand	Consider oil level	
WelG.sealCalc	Seals bearing	
WelG.writeResultsToCSV	Export data to CSV file	
WelG.Lage	Position in space	°
WelG.lastKElem	Stress analysis with load bin	
WelG.housingMaterial.DBID	Housing material	
WelG.considerGyroscopicEffect	Consider gyroscopic effect	
WelG.ZahnradBeruecksichtigen	Consider gear	
WelG.GewichtBeruecksichtigen	Consider weight	
WelG.serializationExport.serializedDataForSaving	Save own bearings	
WelG.serializationExport.isSerializationAllowed	Allow export	
WelG.ReqSysReliability	Reliability of the configuration for required service life	
W060Allg.limitedLife	Calculation method shaft	
shafts[0].speed	Speed	1/min
shafts[0].startPosition	Initial position	mm
WelG.referenceTemperature	Reference temperature	°C

64.10 M010 Cylindrical Interference Fit

m01w.di	Inside diameter	mm
m01allg.df	Diameter of joint	mm

m01n.da	Equivalent outside diameter	mm
m01w.mat.DBID	Material shaft	
m01n.mat.DBID	Material hub	
m01M.nenn	Nominal torque	Nm
m01allg.FA	Axial force	N
m01allg.Mb	Bending moment	Nm
m01allg.Fr	Radial force	N
m01allg.n	Speed	1/min
m01allg.l	Length of interference fit	mm
m01w.tol.bez	Tolerance shaft	
m01n.tol.bez	Tolerance hub	
m01r.si[1]	Safety against sliding	
m01w.SiRm[2]	Safety against fracture, shaft	
m01w.SiRe[2]	Safety against yield point, shaft	
m01n.SiRm[2]	Safety against fracture, hub	
m01n.SiRe[2]	Safety against yield point, Hub	
m01allg.cb	Application factor	
m01sollSi.ReN	Required safety against yield point	
m01sollSi.ReN	Required safety against yield point	
m01sollSi.Ru	Required safety against sliding	
m01r.si[0]	Safety against sliding	
m01r.si[1]	Safety against sliding	

m01r.si[2]	Safety against sliding	
m01w.SiRm[0]	Safety against fracture, shaft	
m01w.SiRm[1]	Safety against fracture, shaft	
m01w.SiRm[2]	Safety against fracture, shaft	
m01w.SiRe[0]	Safety against yield point, shaft	
m01w.SiRe[1]	Safety against yield point, shaft	
m01w.SiRe[2]	Safety against yield point, shaft	
m01sollSi.ReN	Required safety against yield point	
m01sollSi.RmW	Required safety against fracture	
m01sollSi.Ru	Required safety against sliding	
m01allg.cb	Application factor	

64.11 M02a Key

m02Aa.Mnenn	Nominal torque	Nm
m02Aa.Mmax	Maximum torque	Nm
m02Aa.TRmin	Minimal frictional torque for interference fit	Nm
m02Aw.ResultW	Safeties	
m02An.ResultN	Safeties	
m02Ak.ResultP	Safeties	
m02Aw.mat.DBID	Material shaft	
m02An.mat.DBID	Material hub	
m02Ak.mat.DBID	Material key	

m02Aw.dWa	Shaft diameter	mm
m02Aw.lW	Supporting key length	mm
m02An.lN	Supporting key length	mm
m02Aa.Methode	Calculation method	
m02Aa.StossFak	Application factor	
m02An.D1	Small outside diameter of hub	mm
m02Aw.s1	Chamfer on shaft	mm
m02An.s2	Chamfer on hub	mm
m02An.c	Width of hub-part with D2	mm
m02An.D2	Big outside diameter of hub	mm
m02Aa.NW	Number of changes of load direction	
m02An.a0	Distance a0 (Figure 2, DIN 6892)	mm
m02Aw.SollS	Required safety shaft	
m02An.SollS	Required safety hub	
m02Ak.SollS	Required safety key	
m02An.lN	Supporting key length	mm
m02Aw.lW	Supporting key length	mm

64.12 Z09a Splines (Strength and Geometry)

ZS.Geo.mn	Normal module	mm
ZS.Geo.beta	Helix angle at reference diameter	°
ZR[0].z	Number of teeth	

ZR[1].z	Number of teeth	
ZS.Geo.alfn	Normal pressure angle	°
ZR[0].b	Face width	mm
ZR[1].b	Face width	mm
ZR[0].x.nul	Addendum modification coefficient	
ZR[1].x.nul	Addendum modification coefficient	
ZR[0].da.nul	Tip diameter	mm
ZR[1].da.nul	Tip diameter	mm
ZR[0].df.nul	Root diameter	mm
ZR[1].df.nul	Root diameter	mm
ZR[0].T	Torque	Nm
ZS.KA	Application factor	
ZPP[0].Flanke.SHw	Safety	
ZR[0].mat.DBID	Material shaft	
ZR[1].mat.DBID	Material hub	
ZR[0].Tool.RefProfile.DBID	Reference profile shaft	
ZR[1].Tool.RefProfile.DBID	Reference profile hub	
ZS.Q5480	(Resulting) Shearing force	N
ZS.Fax5480	Axial force	N
ZS.MbA5480	Bending moment, start of joint	Nm
ZS.MbU5480	Bending moment, in undisturbed range	
ZS.mu5480	Static coefficient of friction	

ZS.G5480	Friction distribution factor	
ZS.kb5480	Face load factor	
ZS.kphi5480	Circumferential factor	
ZR[0].n	Speed	1/min
ZS.R_T5480	Stress ratio R, torque	
ZS.R_Q5480	Stress ratio R, shearing force	
ZS.R_Mb5480	Stress ratio R, bending moment	
ZS.R_Fax5480	Stress ratio R, axial force	
Z5480P[0].L1	Projecting shaft	mm
ZR[0].Tol.Fp	Total cumulative pitch deviation	µm
ZR[0].Tol.Fp	Total cumulative pitch deviation	µm
m02Ca.Mmax	Maximum torque	Nm
m02Ca.NL	Number of load peaks	
m02Ca.NW	Number of changes of load direction	
m02Cn.D1	Small external diameter	mm
m02Cn.D2	Big external diameter	mm
m02Cn.c	Width of center part with D_2	mm
m02Cn.a0	Distance a0	
ZR[0].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZR[1].Fuss.Kwb	Alternating bending factor, mean stress influence coefficient	
ZS.Kgam	Mesh load factor	
ZS.Q5480	(Resulting) Shearing force	N

ZS.SSi.Flanke	Required safety for tooth flank	
ZR[0].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[1].RZF	Mean peak-to-valley roughness R_z, root	µm
ZR[0].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[1].RZH	Mean peak-to-valley roughness R_z, flank	µm
ZR[0].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[1].RAF	Arithmetic mean roughness value R_a, root	µm
ZR[0].RAH	Average surface roughness for Lx, pinion	µm
ZR[1].RAH	Average surface roughness for Lx, pinion	µm

64.13 M02b Straight-Sided Spline

m02Ba.Mnenn	Nominal torque	Nm
m02Ba.Mmax	Maximum torque	Nm
m02Ba.StossFak	Application factor	
m02Bw.ResultW	Minimum safety	
m02Bn.ResultN	Minimum safety	
m02Bw.mat.DBID	Material shaft	
m02Bn.mat.DBID	Material hub	
m02Bk.d1Keil	Inside diameter	mm
m02Bk.d2Keil	Outside diameter	mm
m02Ba.iANZ	Number of keys	
m02Bk.bKeil	Width	mm

m02Bk.hKeil	Height of key	mm
m02Ba.ltr	Width of hub-part with D2	mm
m02Bn.D1	Small outside diameter	mm
m02Bn.D2	Big outside diameter	mm
m02Bn.c	Width of hub-part with D2	mm
m02Ba.NW	Number of changes of load direction	
m02Ba.NL	Number of load peaks	
m02Bn.a0	Distance a0	mm
m02Ba.SollS	Required safety	

64.14 M02d Polygon

m02Da.Mnenn	Nominal torque	Nm
m02Da.Mmax	Maximum torque	Nm
m02Da.StossFak	Application factor	
m02Dw.ResultW	Safety at T	
m02Dn.ResultN	Safety at T	
m02Dw.mat.DBID	Material shaft	
m02Dn.mat.DBID	Material hub	
m02Dk.di	Diameter of inner circle	mm
m02Dk.da	Diameter of outer circle	mm
m02Dk.e	Eccentricity	mm
m02Da.ltr	Supporting length	mm

m02Dk.y	Coefficient hub wall	
m02Dn.D1	Outside diameter of hub	mm
m02Da.NW	Number of changes of load direction	
m02Da.NL	Number of load peaks	
m02Da.SollS	Required safety	
m02da.RechnenMeth	Calculation method	

64.15 M02e Woodruff Key

m02Ea.Mnenn	Nominal torque	Nm
m02Ea.Mmax	Maximum torque	Nm
m02Ea.StossFak	Application factor	
m02Ew.ResultW	Safeties	
m02En.ResultN	Safeties	
m02Ek.ResultP	Safeties	
m02Ew.mat.DBID	Material shaft	
m02En.mat.DBID	Material hub	
m02Ek.mat.DBID	Material key	
m02Ew.dWa	Shaft diameter	mm
m02Ek.bKeil	Width of Woodruff Key	mm
m02Ek.hKeil	Height of Woodruff Key	mm
m02Ek.DKeil	Diameter	mm
m02Ew.lW	Width of hub-part with D2	mm

m02Ek.t1MaxKeil	Groove depth, shaft	mm
m02En.D1	Small outside diameter	mm
m02En.D2	Big outside diameter	mm
m02En.c	Width of hub-part with D2	mm
m02Ea.iANZ	Number of Woodruff Keys	
m02Ea.NW	Number of changes of load direction	
m02Ea.NL	Number of load peaks	
m02En.a0	Distance a0	mm
m02Ea.SollS	Required safety	

64.16 Z090 V-Belts

z090k.DinIdK	Type	
belt.neff	Number of belts	
belt.Lange	Belt length	mm
z090k.i	Ratio	
z090k.a	Center distance	mm
sheave[0].d	Reference diameter	mm
sheave[1].d	Reference diameter	mm
sheave[2].d	Reference diameter	mm
z090k.n1	Speed	1/min
z090k.n2	Speed	1/min
z090k.n3	Speed	1/min

z090k.cB	Operating factor	
z090k.T1	Service torque	Nm
z090k.T2	Service torque	Nm
z090k.PN	Nominal power	kW
belt.Konfig	Configuration	
z090k.Sich	Utilization	%
sheave[0].umschl	Loop	°
sheave[1].umschl	Loop	°
sheave[0].AxKftBet	Radial force	N
sheave[1].AxKftBet	Radial force	N
z090k.LeerTrF	End of rope force in no load/load	N
z090k.LastTrF	End of rope force in no load/load	N

64.17 Z091 Toothed Belts

z091k.DinIdK	Type	
belt.ZahneZ	Number of parts (of chain)	
belt.Lange	Belt length	mm
z091k.i	Ratio	
z091k.a	Center distance	mm
sheave[0].d	Reference diameter	mm
sheave[1].d	Reference diameter	mm
sheave[2].d	Reference diameter	mm

z091k.n1	Speed	1/min
z091k.n2	Speed	1/min
z091k.n3	Speed	1/min
z091k.z1	Number of teeth	
z091k.z2	Number of teeth	
z091k.z3	Number of teeth	
z091k.cB	Operating factor	
z091k.T1	Service torque	Nm
z091k.T2	Service torque	Nm
z091k.PN	Nominal power	kW
belt.beff	Effective belt width	mm
belt.Konfig	Configuration	
z091k.Sich	Utilization	%
sheave[0].umschl	Loop	°
sheave[1].umschl	Loop	°
sheave[0].AxKftBet	Radial force	N
sheave[1].AxKftBet	Radial force	N
z091k.LeerTrF	End of rope force in no load/load	N
z091k.LastTrF	End of rope force in no load/load	N

64.18 Z092 Chain Drives

z092k.TypID	Type	

z092k.Z	Number of strands	
belt.Lange	Belt length	mm
belt.ZahneZ	Number of parts (of chain)	
z092k.a	Center distance	mm
sheave[0].d	Reference diameter	mm
sheave[1].d	Reference diameter	mm
sheave[2].d	Reference diameter	mm
z092k.z1	Number of teeth	
z092k.z2	Number of teeth	
z092k.z3	Number of teeth	
z092k.n1	Speed	1/min
z092k.n2	Speed	1/min
z092k.n3	Speed	1/min
z092k.f1	Application factor	
z092k.T1	Torque	Nm
z092k.T2	Torque	Nm
z092k.PN	Nominal power	kW
z092k.Sich	Utilization	%
z092k.i1	Ratio	
sheave[2].x	X-Koord. Center point	mm
sheave[2].y	Y-Koord. Center point	mm
sheave[0].umschl	Loop	°

sheave[1].umschl	Loop	°
sheave[0].AxKftBet	Radial force	N
sheave[1].AxKftBet	Radial force	N