MESYS Calculation Software
USER MANUAL

3.   MESYS Rolling Bearing Calculation   145

General

1   General

This chapter gives some general information about the MESYS Calculation Software. Installation, updates and general usage are covered.

Details about the different calculation programs are provided in the following chapters. Available programs are:

- Shaft Calculation 54
- Rolling Bearing Calculation 146
- Ball Screw Calculation 242
- Hertzian Stress Calculation 264
- Cylindrical Gear Pair Calculation 300
- Axial-Radial-Roller-Bearing Calculation 274
- Gear Positions 308

1.1   Installation

When running the installer, the installation directory can be selected. The default location is "\Program Files\MESYS 06/2024". All files are installed into that directory. Also an entry in the start menu is created.

The uninstaller can be called from the start menu. This deletes the installation directory and the entries in the start menu.

Without a license file the software runs as demo version. In the demo version it is not possible to save or load files, a Demo message is shown on each calculation and some functionality is disabled. The demo version may only be used for evaluation of the software.

The license file 'license.dat' has to be placed in the installation directory (in the same directory as MesysShaft.exe). The name of the license file may not be changed since it will not be found by the software otherwise.

1.1.1   Configuration with INI-File

Some configuration of the software can be done using 'mesys.ini' in the installation folder.

Currently the settings are used for database access and formatting of the report.

Section	Value	Description
General	recentfilenumber	Number of recent files shown in the file menu of the software

Section	Value	Description
	floatinglicense	Path to the licensing file for floating license. It will be written by the software but might be copied to other installations. Format: \\\\Server\\Share\\path\license.lic Note: Each '\' has to be doubled.
	floatingtimeout	Time in seconds after a floating license is released if the program is not used. Default is 1800.
	licenselogfile	For floating licenses a logfile can be defined using this entry. Some information about software usage is reported in this file. For path separators either use '/' or '\\' but not '\'.
	fontsize	Fontsize in points, set 0 for default, dependent on operating system.
	style	Either windows, fusion, windowsvista.
	listseparator	Character used as separator for table export. If not set the default setting in operating is used.
	systemlocale	Set to true for decimal point of operating system or false for '.' as decimal separator.
	usecalculatethread	If set to true calculation is done in a separate thread, else set to false.
	ffmpegpath	Path to the program ffmpeg to create videos from animations. It can found at www.ffmpeg.org For path separators either use '/' or '\\' but not '\'.
	helpFormat	Use "CHM" to use "mesys.chm" to display the help information. This might not work over the network. Then "WEB" can be used to show the help in a web browser. Using "EXE" a custom viewer "MesysManual" is used which also works on network shares. Another option is "HTML" which would look for the help in subfolder "help", please contact support@mesys.ch in this case as the folder is not provided in the default installation.
	allownesteddockwindows	If set to true, docked windows can be set in several columns. The default is false.
	disablewheelforlists	If set to true, the wheel on a mouse does not change selection inputs.
	showcalculationtime	Shows the time needed for a calculation in the status bar if set to true.
database	path	The path to the database file can be defined. The database can be copied onto a server, so all software users share the same database. If the filename is given without path, it is opened from the installation directory.

Section	Value	Description
		For path separators either use '/' or '\\' but not '\'.
	iswritable	Set it to true if the database may be changed. If set to false no changes are made to the database by the software.
	usecache	If set to true the database is read to memory. This speeds up the program in case the database is on a network drive. Default is false.
importdatabases	path_1	A path to additional databases can be defined. These additional databases are read only and optionally encrypted. If the filename is given without path, it is opened from the installation directory. For path separators either use '/' or '\\' but not '\'.
	password_1	The password for the encrypted database
shaft	defaultinputs	Path to a xml-file with default settings For path separators either use '/' or '\\' but not '\'.
	preloadcadlibrary	If set to true, the cad library is loaded on program startup. Then importing a STEP file is faster afterward's.
	calculateonfileload	If set to true the calculation is run when a file is loaded. Default is true.
	acceptfiledrop	If set to true a *.xml input file can be loaded by dropping it on the main window
	logo	A logo for use on the system page can be defined here. The format has to be PNG.
	backupinterval	Interval in seconds to save backup files in TEMP directory
	backupstates	Number of backup states to keep. Use zero to disable automatic backup files. The backup files are deleted after file open, file save or file new.
	numberofthreads	The number of threads that should be used in FEA calculations. The default is the number of physical cores.
	feamaxmemory	The maximum amount of memory in MB used by the fea solver. If more memory is required it will be swapped to disk.
	feaswappath	Path of a directory for the swapfile for FEA calculation in case not enough memory is available. Default is the temp directory of the operating system. Using this setting a fast drive with enough space can be used instead.
	feamaxtotalmemory	The maximum amount of memory in MB used by the fea solver. If more memory would be required by the solver, the calculation is canceled.

Section	Value	Description
		Using this setting instead of 'feamaxmemory' avoids swapping to disk and therefore much longer calculation times. On error the mesh sizes would have to be changed.
	externalcondensationcommand	A path to an executable can be specified to be run for condensation of 3D-elastic parts. This can start MesysShaft of a remote server to run a condensation faster. For path separators either use '/' or '\\' but not '\'.
	externalcondensationparameters	The parameters for the external program above can be specified here. A placeholder %1 will be replaced by the filename of a temporary file.
rbc	defaultinputs	Path to a xml-file with default settings For path separators either use '/' or '\\' but not '\'.
	calculateonfileload	If set to true the calculation is run when a file is loaded. Default is true.
	acceptfiledrop	If set to true a *.xml input file can be loaded by dropping it on the main window.
	logo	A logo for use on the system page can be defined here. The format has to be PNG.
	backupinterval	Interval in seconds to save backup files in TEMP directory
	backupstates	Number of backup states to keep. Use zero to disable automatic backup files. The backup files are deleted after file open, file save or file new.
	allowcontactangleoverride	If set to true the contact angle for angular contact bearings from the database can be overwritten
ballscrew	defaultinputs	Path to a xml-file with default settings For path separators either use '/' or '\\' but not '\'.
	calculateonfileload	If set to true the calculation is run when a file is loaded. Default is true.
	acceptfiledrop	If set to true a *.xml input file can be loaded by dropping it on the main window.
	logo	A logo for use on the system page can be defined here. The format has to be PNG.
	backupinterval	Interval in seconds to save backup files in TEMP directory
	backupstates	Number of backup states to keep. Use zero to disable automatic backup files. The backup files are deleted after file open, file save or file new.
hertz	defaultinputs	Path to a xml-file with default settings For path separators either use '/' or '\\' but not '\'.

Section	Value	Description
	calculateonfileload	If set to true the calculation is run when a file is loaded. Default is true.
	acceptfiledrop	If set to true a *.xml input file can be loaded by dropping it on the main window
	logo	A logo for use on the system page can be defined here. The format has to be PNG.
	backupinterval	Interval in seconds to save backup files in TEMP directory
	backupstates	Number of backup states to keep. Use zero to disable automatic backup files. The backup files are deleted after file open, file save or file new.
axrarbc	numberofthreads	The number of threads that should be used in FEA calculations. The default is 2. More threads have only a small impact on performance, but the calculation time is still reduced.
	backupinterval	Interval in seconds to save backup files in TEMP directory
	backupstates	Number of backup states to keep. Use zero to disable automatic backup files. The backup files are deleted after file open, file save or file new.
report	format	The outputted report file can have different formats. By default, this value is equal to "INTERNALPDFA", but it can be set to "INTERNALPDF", "DOCX", "DOC", ODT" or "PDF" (without quotes).
	topmargin	The top margin for the report in mm
	bottommargin	The bottom margin for the report in mm
	leftmargin	The left margin for the report in mm
	rightmargin	The right margin for the report in mm
	papersize	The size of the paper for the report. Available values are A4 and Letter
	template	Path to the created template file used for the report creation. Supported file formats: "DOCX", "DOC" or "ODT"
	logo	A different logo can be defined, which will be used in the report. The format had to be PNG.
	marginbox1\active	The marginbox is used if set to true, else set it to false.
	marginbox1\rect	The size of the marginbox is defined with values in mm. The format is @Rect(x1 y1 width height). The parameters x1 and y1 describe the upper left corner of the box. Positive values are measured from the top/left negative values from the bottom/right. For example @Rect(-35 -20 30 20) is a rectangle at the right bottom.
	marginbox1\text	The text for the margin box. It has to be set in quotations marks (like in "Text"). Either normal

Section	Value	Description
		text or HTML can be used. Some placeholders are defined: #page, #pageCount, #data, #datetime
	marginbox1\angle	A rotation angle of the margin box can be set in degrees. The orientation of the angle is clockwise if positive.
	marginbox1\isHtml	Either set it to true or false dependent on the type of text.
	marginbox1\drawBox	If set to true a rectangle is drawn indicating the size of the marginbox. Else set it to false.
	marginbox2\...	Like for margin box 1 additional boxes can be defined with increasing numbers.

Please note that the listed options for margin boxes (marginbox1\...) are only valid if format is equal to "INTERNALPDF".

1.1.1.1 Exporting videos

Animations can be exported as videos. To enable this feature install "FFmpeg" from www.ffmpeg.org and set "ffmpegpath" to the executable.

The video export can be started using the context menu in graphic windows.

1.1.1.2 Automatic saving of backup files

If the settings "backupstates" and "backupinterval" are set to values larger of zero, automatic backups are saved into the system temp directory. "backupstates" defines how many files are kept and "backupinterval" the time difference in seconds.

The automatically saved files are removed on "File"->"New" ,"File"->"Open", "File"->"Save" or on selecting "No" regarding saving file question on closing the program. Therefore these backup files are only available in case of a software crash or in case of changes that should be undone as long the file was not saved in between.

If a backup file for a currently active calculation should be loaded, this should be done by starting another instance of the software as the backup files are deleted on "File"->"Open".

1.1.2 Update

If the software is updated with a new version the database 'mesys.db' should not be overwritten. Either the new installation is done in a new directory or the database file is copied to a different location.

After the new version is started the database can be updated by choosing menu 'Extras'->'Database'->'Import from old database'. All custom entries will be updated. Changes to default data will be lost as only custom data will be copied.

For updates of a floating license in addition to the 'license.dat' the setting 'floatinglicense=' in 'mesys.ini' needs to be copied into the new version.

1.2   Requirements

The software is available as 64bit windows program running on Windows 10 or 11. The minimum required processor is Intel Pentium 4 or above.

About 1.6 GB of hard disk space is required for the full installation. All dependencies of the software are available in the installation directory. Therefore, it can just be copied to other machines or started from network or removable disks.

For the floating license a directory on a network share with write permissions is required.

1.3   General usage

To run a calculation first the data on all pages is introduced. Then press the button or F5 to run the calculation. After all data is defined the calculation can be run from each page. So it is easy to make parameter variations.

A symbol is shown the the right of the status bar at the bottom of the program. This shows the current status of the results:

Symbol	Explanation
	The calculation was run and the results are valid.
	Some inputs were changed after the calculation was run. The results might be invalid, dependent on the change of inputs.
	There was an error on running the calculation. The results are invalid.

Next to this symbol in the status bar a small button shows "M" or "A". If the button is clicked it toggles between "M" and "A". With "M" the calculate-button or F5 needs to be clicked to run the calculation. If "A" is shown, the calculation is run automatically after each input. This option should not be used in case of large calculation time for example with large load spectra.

There are some special buttons used in the user interface, which are explained in the following table:

Button	Explanation

✚	This plus button shows a dialog with additional inputs. Some of these inputs need to be defined, some are just optional.
═	This conversion button allows the conversion from other types of input. For example, the radial clearance can be converted from an axial clearance.
★	This proposal button provides a suggestion for an input by the software.

The unit system for the input and output can be selected on the menu 'Extras->Unit system' either as metric or US units.

Using the context menu for the units of input fields, the current unit can be changed. This setting is saved for future restarts of the software.

For tables, the unit of columns can be changed using the context menu (right mouse button) on the header row. In addition to the unit, the number of visible digits can be changed.
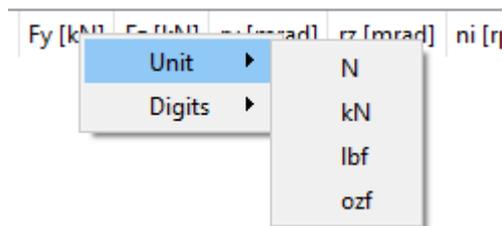
Pressing the right mouse button on an input field, a window for an input of a formula is shown. This can be used for quick calculations.
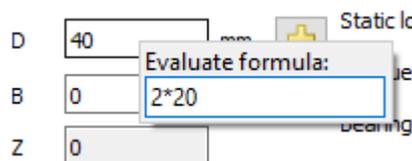
The software is available in English, German, French, Spanish, Turkish, Chinese, Korean and Japanese language. The language can also be selected in menu 'Extras'.

1.3.1   Graphics 2D and Diagrams

For 2D graphics and diagrams several options are available in the context menu.
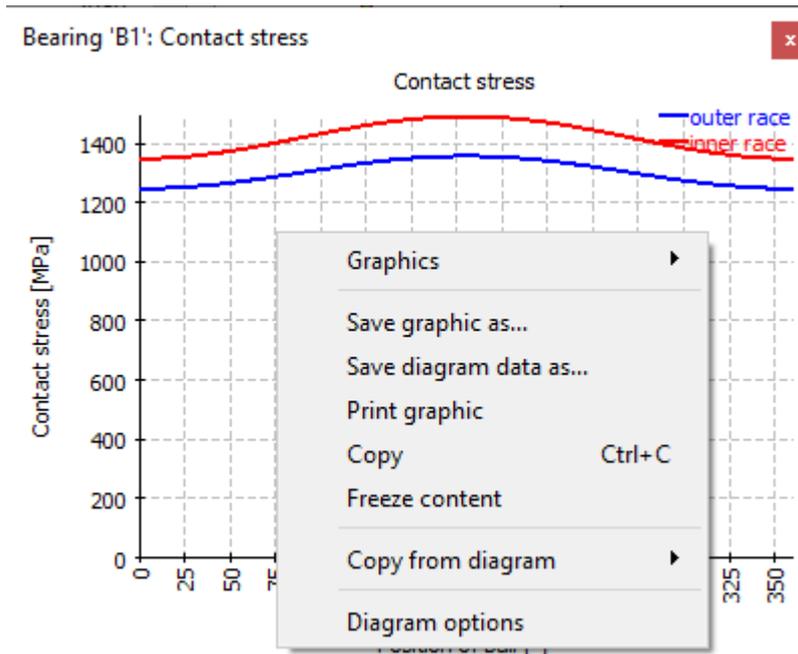
All graphics can be printed or exported in different graphic formats like PNG or JPEG using the context menu (right mouse button) in the graphic window. For the export the size of the graphic can be specified. The option "Copy" copies the image to the clipboard, so it can be pasted into other applications. Selecting "Freeze content" suppresses updates of the graphic on further runs of the calculation, this allows a comparison of graphics between calculations with different input data.

For diagrams "Save diagram data as...", "Copy from diagram" and "Diagram options" are available in addition. "Save diagram data as..." allows to save the numeric values for the diagrams in CSV or XLS formats. Using "Copy from diagram" curves from other open diagrams can be copied into the current diagram if they have the same units. This can be used for comparisons.

The diagram option allow to change the formatting of the diagram
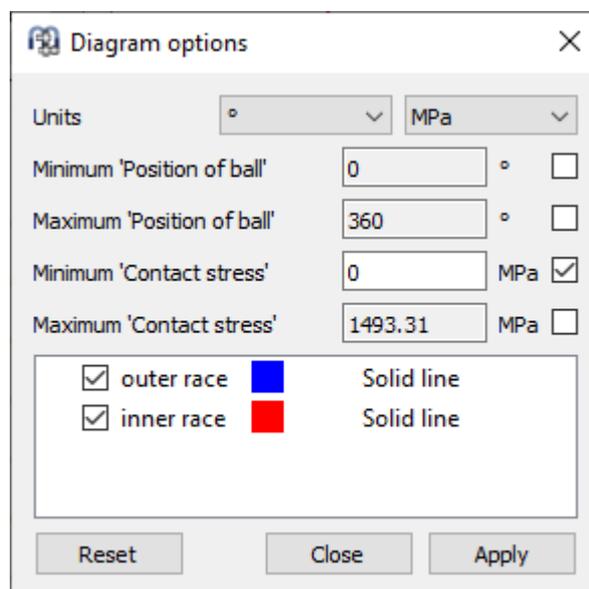
Both the units and bounds for the axes can be set. Moreover, if any of the graphs is of no interest, it can be disabled with the checkboxes. The name, the color and line style of each curve can be changed too. The changes in diagram options are only used for the current diagram, they are not saved and not used for newly created graphic windows.

1.3.2   Graphics 3D

For 3D graphic windows of the software, there are different buttons for the view manipulation in 3D, such as the zoom-in, zoom-out and fit-to-window functions and also it is possible to select the point of view from different planes. The 3D-model can be dragged with the mouse by holding down Shift-key as well as zoomed in and out by holding down the Ctrl-key. A 3D-mouse is supported in 3D-graphics. Animated 3D-graphics can be saved as video in case a path to ffmpeg is provided in mesys.ini [16].
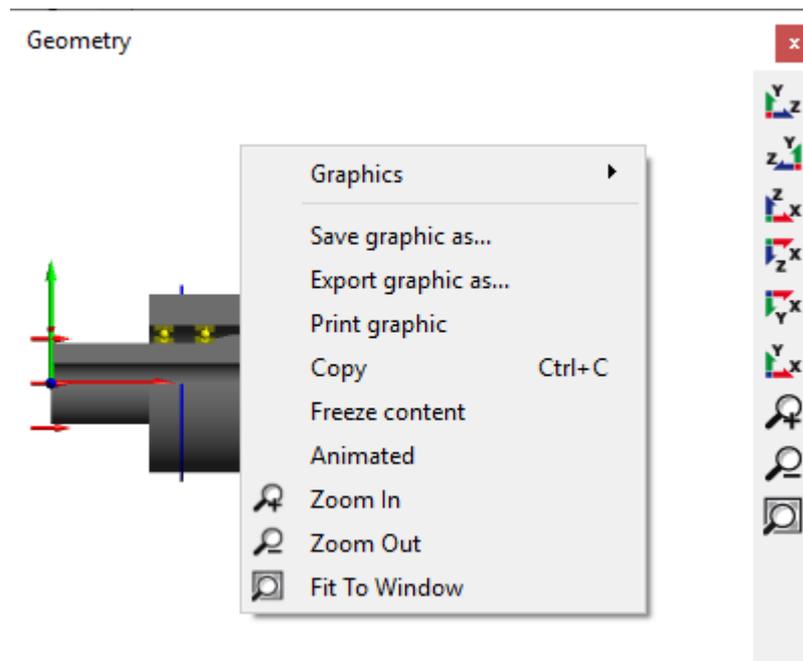
"Save graphics as..." will save the image in a graphics format like PNG. "Export graphic as..." only supported STL for standard graphics, but for graphics generated from submenu CAD also an export as STEP is available.

"Copy" will copy the image to the clipboard, so it can be pasted in other programs.

Activating "Freeze content" will suppress updated of the view on new runs of the calculations. This can be used for comparisons.

The selection "Animated" will rotate the view around the vertical axis.

1.4   Command line parameters

The software supports several command line parameters:

- -highDPIRoundUp rounds a non-integer scale factor up in case of high DPI settings.

- -highDPIRoundDown rounds a non-integer scale factor down in case of high DPI settings.

- -highDPIRound rounds a non-integer scale factor in case of high DPI settings.

- -highDPIRoundPreferDown rounds a non-integer scale factor in case of high DPI settings. In case of 50% it will be rounded down.

- -desktopOpenGL tells the software to use hardware OpenGL, which is the default.

- -softwareOpenGL tells the software to use a software driver for 3D-graphics. This can be used if the two settings above fail to work.

- -ini=file.ini tells the software to use 'file.ini' for program settings.

- -license=license.dat tells the software to use 'license.dat' as license file. This can be used it different license files are available.

- -language=xx sets the language for the user interface. Valid options are EN, DE, FR, ES, KO, ZH, JA, TR.

- -lock keeps the *.xml file, which is also defined on the command line, open until the program is closed. This can be used for calls from other programs, which check when the input file is released.

- -generatereport=filename.pdf generates a report if a *.xml file is also passed on the command line. The program is closed afterwards.

- -save or -saveas=filename.xml will save the input file after calculation if also a *.xml file is passed on the command line. The program is closed afterwards. In the shaft calculation also a *.rexs file is supported for input and output in addition to the default xml file format. In addition to the *.xml extension also the new module specific file extensions are possible.

- If a *.xml filename is passed as a parameter, the file is opened by the software. This also allows to drop an input file on a program icon on the desktop. In the shaft calculation also a *.rexs can be used in addition to *.xml.

1.5   Database

As database a SQLite database is used. In addition to the possibilities of the software also standard tools could be used to change the database. Please only add datasets with id > 100000 and don't modify datasets with id < 0 since they are used internally. On future versions datasets with id< 100000 might be changed or added.

In the current version there are database tables for Bearings, Bearing Clearance, Bearing Tolerance, Material, Material Bearings and Lubrication. They can be added using the Menu 'Database' under 'Extras'.

Data is written into the database only when clicking the Apply-button, changes can be removed by selecting 'Reset' before 'Apply' is clicked.

Rows can be added using the ➕-button or a selected row can be deleted using the ➖-button. By using the ➡-button, additional items can be imported from a file. The columns in the file have to be in the same order and unit as shown in the table. Likewise, the table can be exported with the corresponding ➡-button.

1.5.1   Changing names in the database

When a calculation file is loaded, the software searches the database for combinations of id and name of a dataset. If the item is not found the inputs are changed to 'own input'.

Therefore, calculations files that are opened by other installations of the software with a different database, will show 'own input' and give the correct results.

If you change the name of entries in the database the same will happen. You will see 'own input' in old files that used this entry and they will use the original data.

1.5.2   Add bearings to database

For adding bearings to the database please enter the geometry data into the calculation windows of the bearing calculation, run the calculation and then select 'Extras->Database->Add bearing to database'. You will be prompted for bearing name, comment, manufacturer and a database mode.
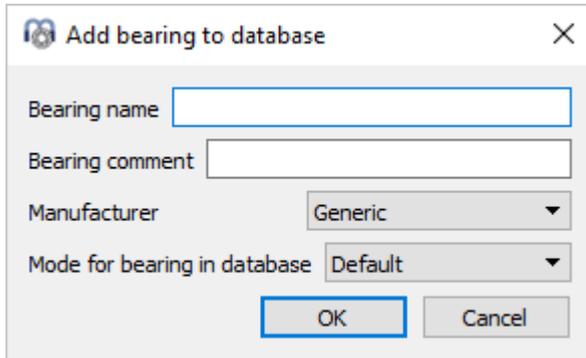
If the combination of name and manufacturer is already found in the database you will be asked if the dataset should be overwritten.

The mode can be

- "Default": Default behavior

- "Approximated": A comment about approximated inner geometry will be shown in the report and damage frequencies will not be shown.

- "Hide inner geometry": Inner geometry will not be available in the report and in input data. This option is for additional encrypted databases.

In case of the shaft calculation the above dialog is shown if 'Extras->Database->Add bearing to database' is selected while the bearing window is active. Else the following dialog is shown:

A window with two tab-pages named "Bearing geometry" and "Material and Lubrication" is opened in the shaft calculation. These tab-pages operate in the same way as the ones in the bearing calculation interface. The feature offers the possibility to create and add a new bearing from scratch by entering all the required data or by selecting and modifying an existing bearing of the database. In both cases, the user does not need to know the load capacity data in advance, since the software finds it out from the required geometrical inputs when pressing the button "Calculate" at the bottom. Once all the data is filled, the "Apply" button must be pressed and the user will be prompted to enter a name for the new bearing.

1.5.3 Database Tables

For some database tables more information about usage is given in the following sections.

1.5.3.1 Bearing Clearance and Bearing Tolerance

For clearance and tolerance definitions two tables have to be used. In table 'BearingClearanceClasses' or 'BearingToleranceClasses' the available names are defined.
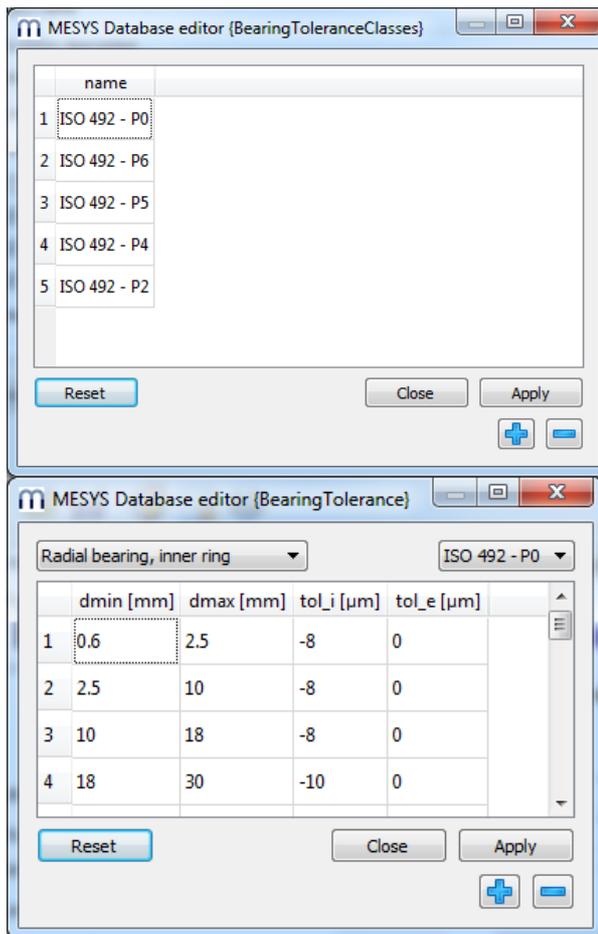
In table 'BearingClearance' and 'BearingTolerance' then the actual values are defined dependent on the diameter.

1.5.3.2 Bearing Materials

The table 'Material Bearings' allows to set additional information about permissible stresses for bearing calculations.

1.5.3.3 Bearing Manufacturers

The tables 'Bearing Manufacturers' allows to set additional names for bearing manufacturers.

1.6 Parameter variations

Using the menu point "Calculation"->"Parameter variation" a dialog for parameter variations is shown. It allows the user to do parameter studies with results provided in tables and graphics. Typical applications are, for example, visualizing

life over clearance or displacement over load. An optional optimization for a parameter is available too.

Several settings can be defined in the parameter variation using the buttons on the lower left.

1.6.1   Generate list

Several input parameters can be added to the table "Generate list" and ranges (Start/End values) can be defined for them. In the bearing calculation dependent on the bearing type the calculation can take up to 0.1 sec for a single calculation, using load spectra and defining many steps can lead to long calculation times.
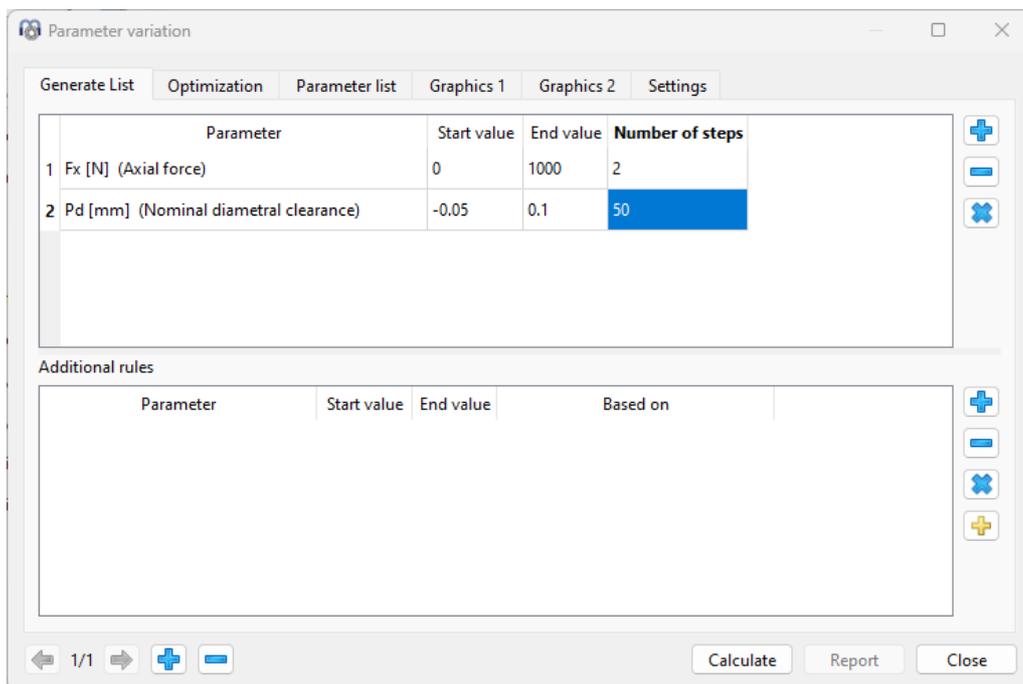
Additional rules can be used to set additional input parameters dependent on parameters on the first list. The additional parameter is interpolated between a start and end value based on another parameter. For example additional parameter y between y1 and y2 based on parameter x in range x1 to x2:

$$y = y_1 + \frac{y_2 - y_1}{x_2 - x_1}(x - x_1)$$

An example for using the rules can be to set the bearing clearances for multiple bearings to the same value within the shaft calculation.

Rules can also be defined using formulas based on the parameters in the first table:
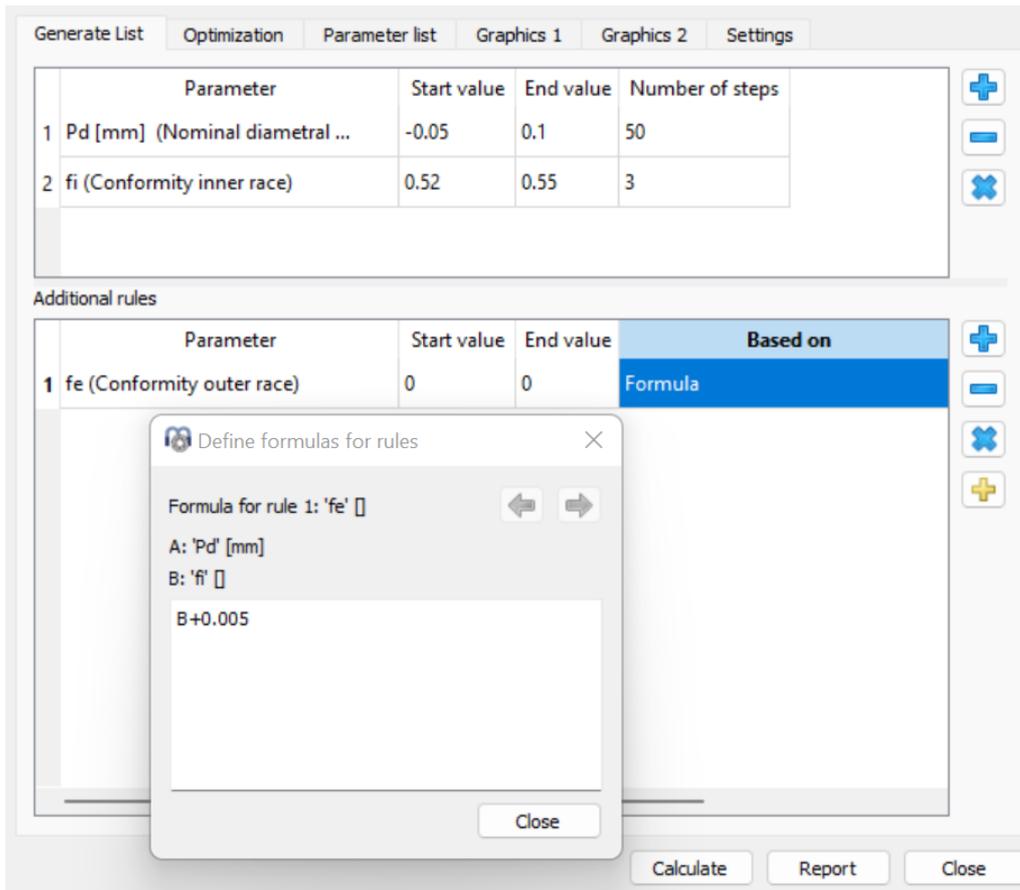
The formulas can be defined using the ⊕-button on the right.

1.6.2 Parameter list

Before we generate a list, it is necessary to define first those outputs or variables we are interested in at the tab window 'Parameter list'.

As shown in the picture, by doing a right mouse-click on the tab window 'Parameter list', a context menu will enable us to add the columns corresponding to the outputs or results we are interested in. The rows will contain the variable values according to the bounds and number of steps.
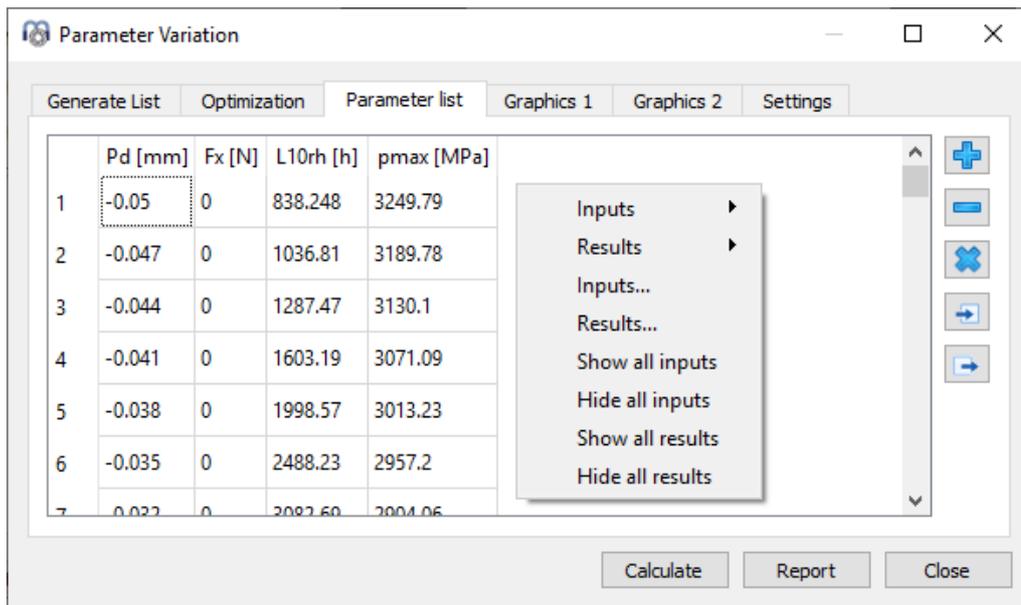
By clicking on 'Calculate' while the "Generate list" tab window is active, a list of all parameter combinations will be generated at the corresponding tab window 'Parameter list' and the analysis will be run. Depending on the chosen number of parameters and steps, the analysis can lead to long calculation times.

A second way to create a table (parameter list) is possible by adding rows and entering values manually with the ✚-button. Moreover, a whole table can be imported from a CSV- or XLS-file using the ➔-button or exported into a file using the ➔-button. Then, in order to run the analysis, we just have to click on "Calculate" while this page is active. Please note that the parameter list is not saved if the dialog is closed.

1.6.3   Graphics

On the pages "Graphics1" and "Graphics2" the results can be shown as graphic.

If the parameter list was generated by "Generate list" and no more than two input parameters are used, lines are used for the diagram. For more parameters or other sources just points are shown.
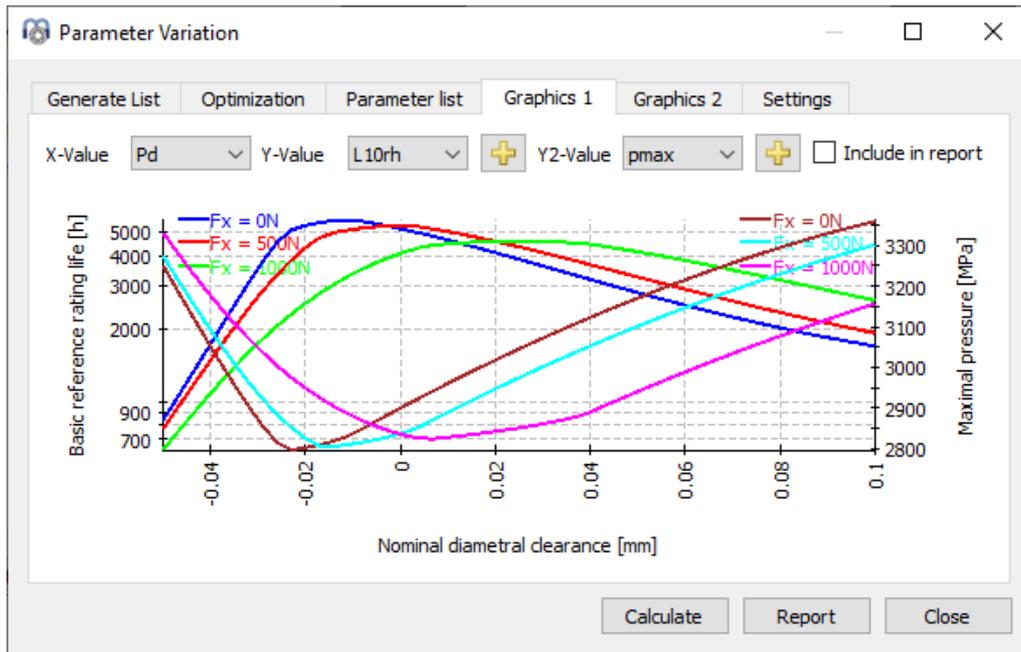
In case two parameters had been varied on "Generate list" multiple curves for the second parameter are shown like in above image showing a variation of nominal clearance and axial force Fx.

The graphic area consists of two different y-axis, thus being possible to compare different types of output results (units) at a time. Furthermore, although the basic life (L10rh) is being primarily analyzed in the graphic, by doing a click on the ➕-button, it could also be possible to add any other outputs with the same units like Lnmrh of one of the two available y-axis.

Under "Settings" an option is available to include input and output values in selections for both axes. This would also allow to swap horizontal and vertical axis.

1.6.4 Optimization

Optionally an optimization step can be used. For the example before the radial force Fy should be maximized to that L10rh=5000h is reached.
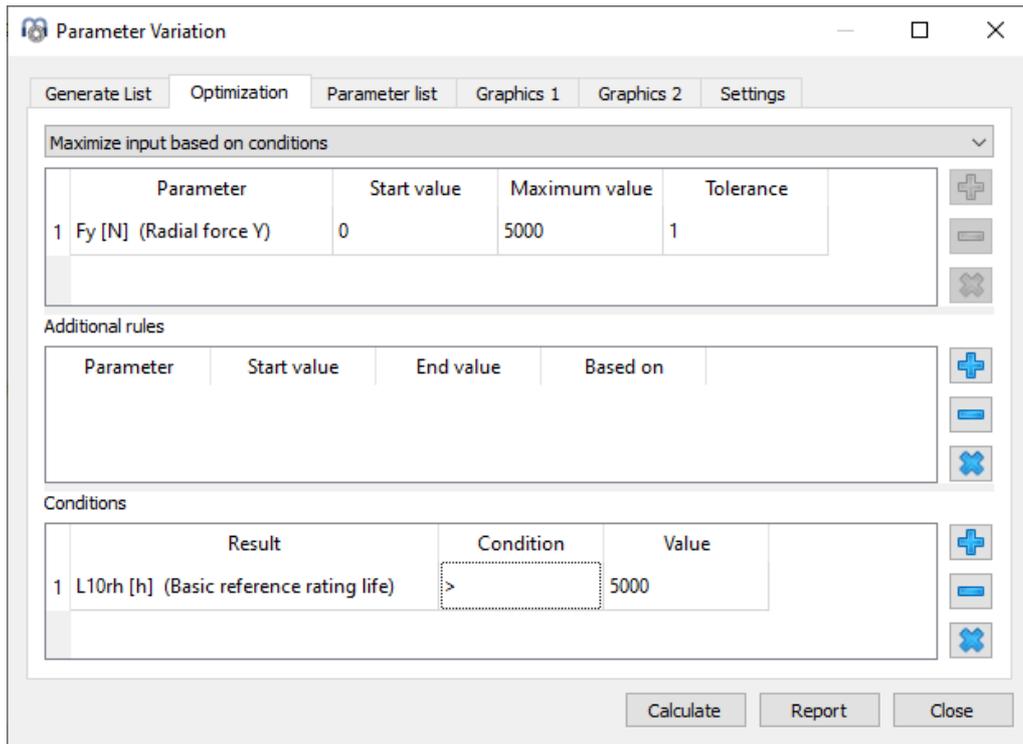
The conditions have to be fulfilled on the start value of the parameter. If this is not the case, the results are set to zero. If the conditions are still fulfilled at the maximum value, the calculation stops and the results are set to zero too.

Additional rules can be defined to set additional inputs based on optimization parameter. The additional inputs are interpolated between start value and end values dependent on the optimization parameter between start value and maximum or minimum.

The results for Fy in below image show an error for small clearance as the 5000h cannot be reached.
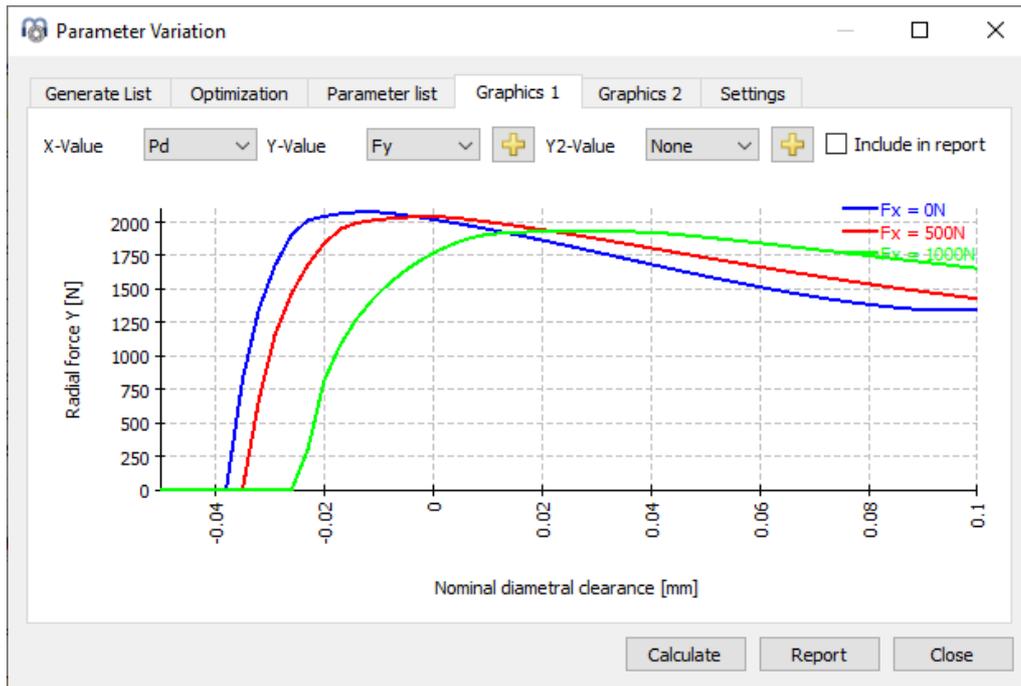
1.6.5   Report

You have to activate the checkbox "Include in report" if the graphic should be added to the report. Moreover, under the tab-page 'Settings' another checkbox is available for including the parameter list in the report.

1.6.6   Settings

Some settings are available in the parameter variation:
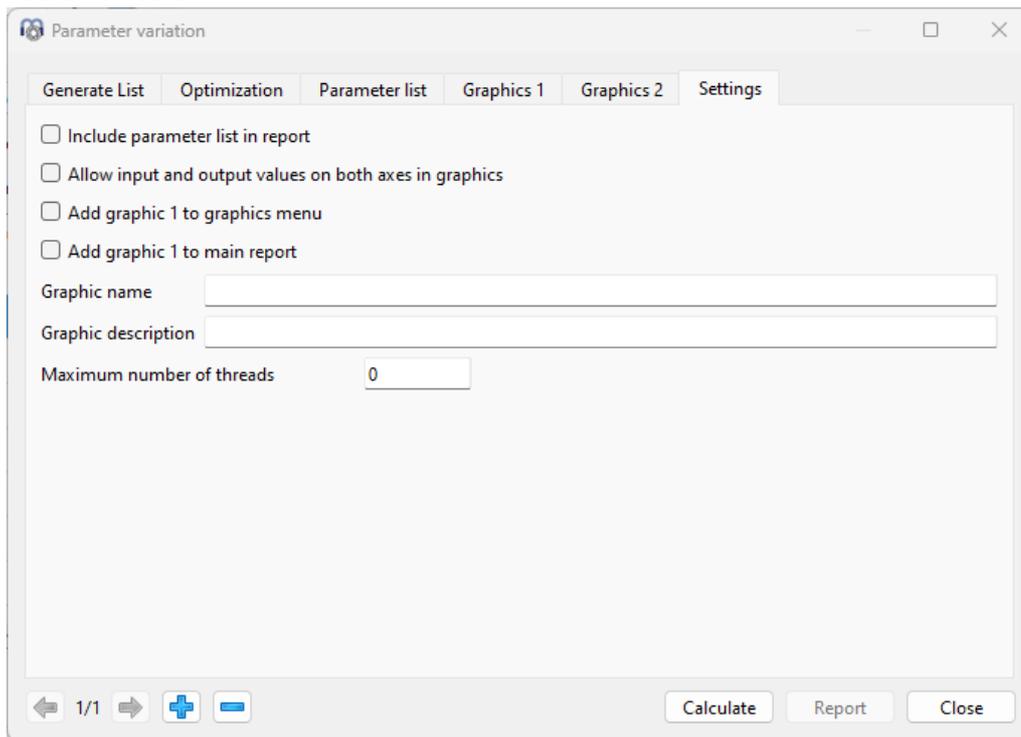
- Include parameter list in report: this adds the result table to the report. Dependent on the inputs it can get quite long.

- Allow input and output values on both axes in graphics: This option allows to switch the axes in the result diagrams.

- Add graphic 1 to graphics menu: The selections for Graphic 1 are used for a parametric diagram the the main graphics menu. This does only work if the data is based on table "Generate list"

- Add graphics 1 to main report: This adds the graphic 1 to the main report. The graphic is not saved, but newly generated on each generated report

- Graphic name and description: These are used for the custom diagrams in the graphics menu and the report.

- Maximum number of threads: With the default setting of zero all available processor cores are used in the calculation. In case of very large models memory usage can be a problem, then a smaller number of threads can be defined here.

1.7   Statistical parameter variations

Using the menu point "Calculation"->"Parameter variation (statistical)" an analogous tool to the default parameter variation is available. The main difference between them lies in the fact that this tool supports statistical evaluations when doing parameter variations.

The default version of parameter variation is helpful to visualize the influence of one parameter on one or multiple results. The statistical version can be used if the influence of multiple uncertainties should be evaluated. Instead of manually checking multiple parameters an automatic calculation of the whole ranges can be done. In order to understand the reasons behind the results it will still be necessary to carry out additional evaluations, but the software shows the ranges of results that have to be expected.

1.7.1 Generate list

As in the normal parameter variation, several input parameters can be added to the table "Generate list" and ranges (Start/End values) can be defined for them. No calculations steps are required for the ranges. However, a limitation on the number of statistical calculations can be set under the tab-page 'Settings'.

Different types of probability distributions can be chosen from a drop-down list which is shown when double-clicking on a cell under the 'Distribution' column for any of the parameters:
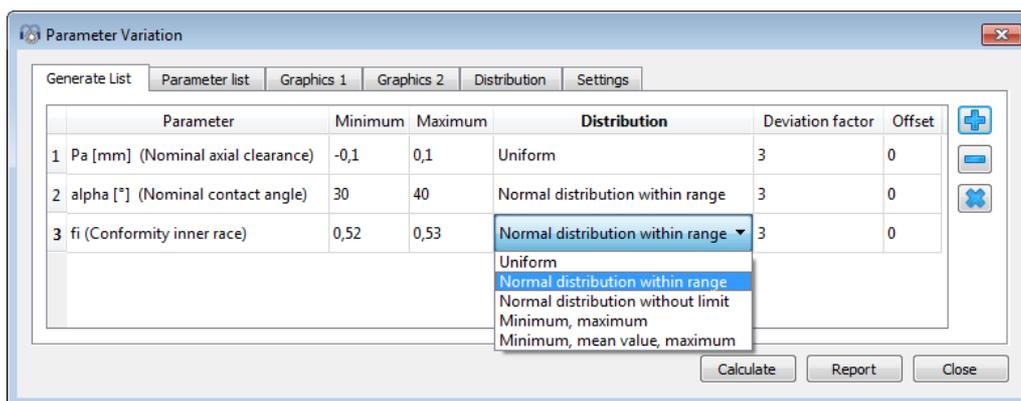
- Uniform: The probability is equally distributed for a set of values between the maximum and minimum value of the parameter.

- Normal distribution within range: The probability is calculated for a set of values ranging from the minimum to maximum by using a normal distribution.

- Normal distribution without limit: The probability is calculated for a set of values theoretically in the range of (-∞, +∞) by using a normal distribution.

- Minimum, maximum: The probability is equally distributed between the maximum and minimum values of the parameter.

- Minimum, mean value, maximum: The probability is equally distributed between the maximum, mean and minimum values of the parameter.

Additionally, a factor can be set for the deviation and an offset value between -1 and +1 can be used to shift horizontally the 'bell curve' in either direction.

1.7.2 Parameter list

The generation of the parameter list is performed as explained for the non-statistical parameter variation. By clicking on 'Calculate' while the "Generate list" tab window is active, a list of all parameter combinations will be generated at the corresponding tab window 'Parameter list' and the statistical analysis will be run. Depending on the chosen number of parameters and the specified number of calculations, the analysis can lead to long calculation times.
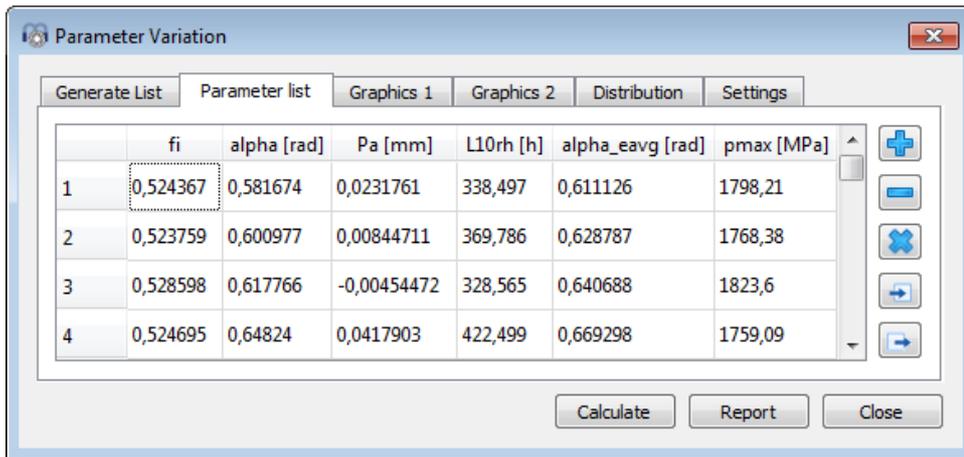
1.7.3 Graphics

On the tab-pages 'Graphics' any variable combination can be visualized in the form of point clouds, where some trends for the relationship between the parameters might be recognized, as shown in the picture below.

1.7.4 Distribution

The probability distribution is graphically represented at the tab-page 'Distribution'. By simply using the ✚-button, it is possible to make comparisons of interest between the statistical results for different mechanical components in a system. As shown in the picture, the probability distribution of basic reference life for the different bearings mounted on the same shaft are compared with each other. The maxima of the curves are close to each other, but the variance for the roller bearing 'B2' is much larger. These analyses are useful to help identify restrictions or weaknesses in the design.

1.7.5 Report

You have to activate the checkbox "Include in report" if the graphic should be added to the report. The report will also show minimum, maximum, mean value and standard deviation for each selected result.

1.7.6 Settings

Some settings are available in the statistical parameter variation:
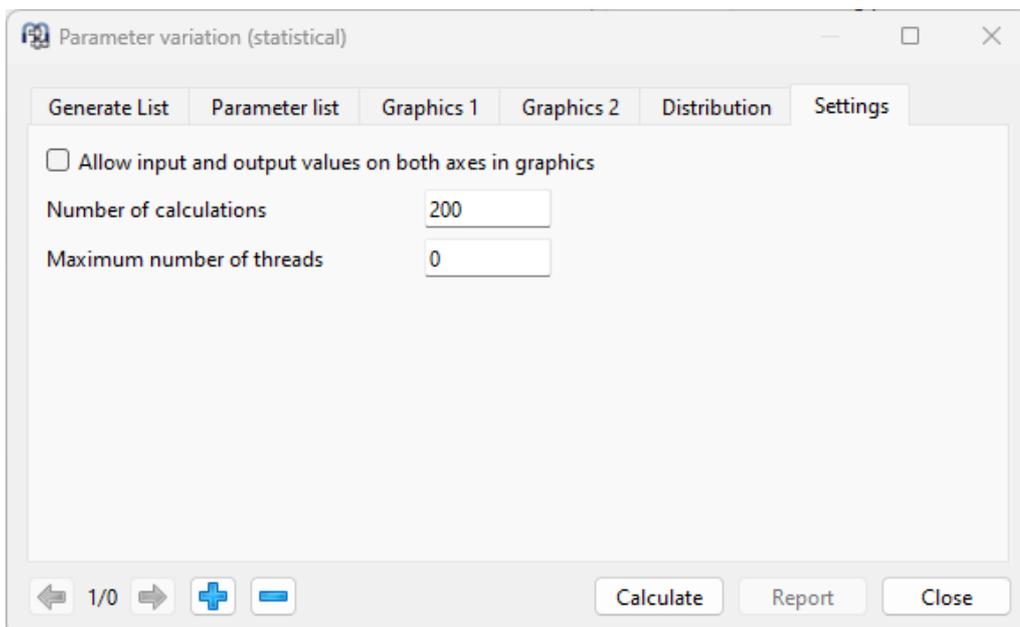
- Allow input and output values on both axes in graphics: This option allows to switch the axes in the result diagrams.

- Number of calculations: This input defined the number of calculations to be done. For nice looking distributions several thousand calculations should be done.

- Maximum number of threads: With the default setting of zero all available processor cores are used in the calculation. In case of very large models memory usage can be a problem, then a smaller number of threads can be defined here.

1.8 Results

Results are provided as a small result overview directly in the software, a main text report in PDF format as standard and optional additional reports dependent on the calculation module. Under Report->'Result tables' a spread sheet with result data can be opened in XLSX format as default.

Graphic windows can be docked onto the main program interface and are automatically updated on each calculation.

1.8.1 Results Overview

The results overview on the bottom of the window shows the most important results. The results overview can be configured under Extras->Results overview.

The list on the left shows all available items for the result overview. Using the arrows selected items can be moved to the list on the right which is used for the result overview. The order of the selected results can be changed by the up/down arrows on the right. Note that only valid results values are actually shown in the result overview. So you will see fewer values than selected in the settings.

The settings for the result overview are saved in user settings not in the calculation file. So they will be the same for different calculations.

1.8.2 Main Report

The main report can be generated selecting Report->'Show report'. It can be saved in different formats using Report->'Save report as'.

In the menu Report->Options the contents of the report can be configured. The graphics to be included can be selected and some sections of the report could be discarded if not of interest. The legend for all the table parameters can be shown in the report.

The logo in the report can be configured in 'mesys.ini'. See Configuration with INI-File 16

1.8.2.1 Template

A template in (.docx) format can be created and defined in 'mesys.ini' so that it is used when generating the software report. For the current version, only the information contained in the header and footer can be edited, in which it is possible to link information to the software such us as 'module title', 'license name', 'date', 'file name', 'project name' or 'description' by means of the text fields option in Microsoft word. Additionally, a company logo can be included, instead of the MESYS logo that it is shown by default at the report:
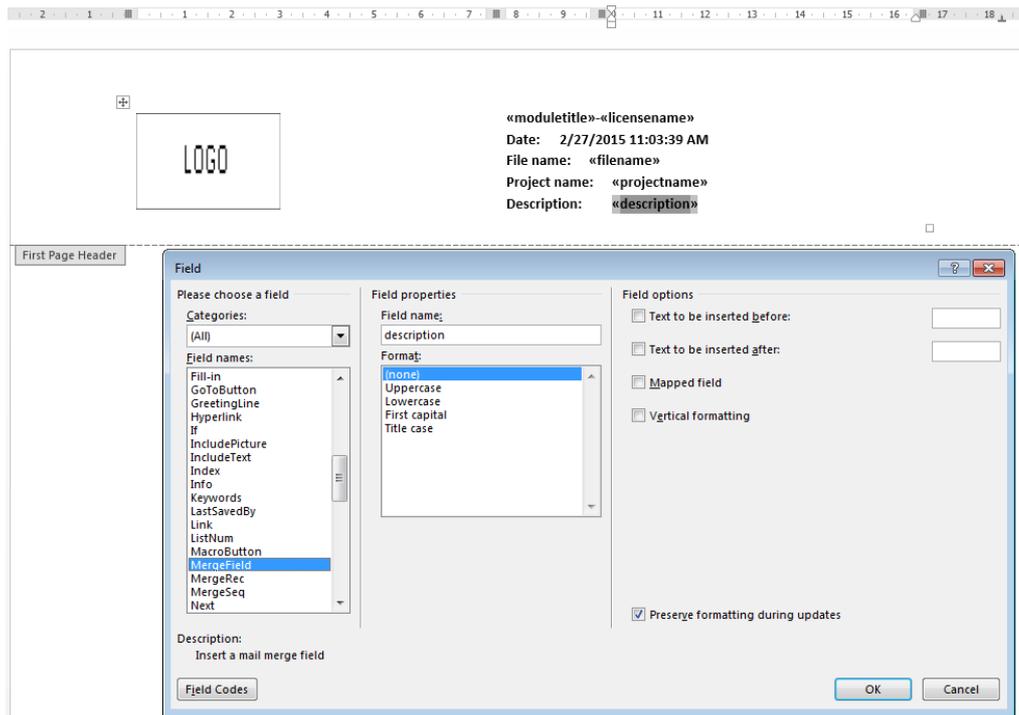
Available field for the 'Mergefield' option are:

- filename

- projectname

- description

- licensename

- username

- moduletitle

1.8.3   Custom Report Templates

Custom report templates can be added in subdirectory "templates/module-id". Reports based on these templates can be generated by selecting the report template under Report->Report templates.

The report templates are word files with DOCVARIABLE fields. These fields are filled on generation of the report.

Possible fields are:

- DOCVARIABLE VAR name [unit]: the variable with the name 'name' is printed. Optionally a unit can be provided in brackets, for example [mm]. Additionally, a format could be added like %6.3g or %6.3f, where 'f' always generates a fixed point output and 'g' generates an exponential output for large numbers.

- DOCVARIABLE VARU name [unit]: the variable with the name 'name' plus the unit are printed. Optionally a unit can be provided in brackets, for example [mm].

- DOCVARIABLE TABLE tablename: A table is inserted.

- DOCVARIABLE GRAPHIC graphicName width=130 height=70: A graphic is inserted. Optionally height and width can be given in millimeters.

Regarding details for variable names, table names and graphic names please ask support@mesys.ch for additional documentation.

1.8.4   Result tables

In the menu Report->Result tables, it is possible to output the result data by means of tables, which can be generated in different file formats as explained before at Configuration with INI-File ⌐16⌐

1.9   COM Interface

The software can be accessed via a COM-Interface under Windows. The installation of the COM-Interface is optional and it can be selected during installation. Before the first use the COM-server has to be registered.

1.9.1   Registering the COM server

The COM server can be registered by running MesysCOM(64).exe with parameter -regserver as administrator. This has to be done once. For unregistering the parameter -unregserver can be used.

In the installation directory two batch files RegisterMesysCOM and UnRegisterMesysCOM are provided so that the registration can be done without a command window. They can be started as Administrator using the right mouse button. This does not work in case the installation is on a network share, in case of a network a command window has to be used.

1.9.2   Methods in COM interface

The server provides the interface MesysCOM which can be used to create calculation modules. Available methods are:

- `void setLanguage([in] BSTR p_lang);`
  `Pass language as two characters ("de", "en", "fr", "es", "tr", "ko", "zh", "ja")`

- `int getVersion();`
  Returns version in format yyyymm.
- `MesysRBC* createRBC();`
  Create bearing calculation module without user interface.
- `MesysSHAFT* createSHAFT();`
  Create shaft calculation module without user interface.
- `MesysHERTZ* createHERTZ();`
  Create Hertz stress calculation module without user interface.
- `MesysBALLSCREW* createBALLSCREW();`
  Create ball screw calculation module without user interface.
- `MesysCYLGEARPAIR* createCYLGEARPAIR();`
  Create cylindrical gear pair calculation module without user interface.
- `MesysGUIRBC* createGUIRBC();`
  Create bearing calculation module with user interface.
- `MesysGUIHERTZ* createGUIHERTZ();`
  Create hertz stress calculation module with user interface.
- `MesysGUISHAFT* createGUISHAFT();`
  Create shaft calculation module with user interface.
- `MesysGUIBALLSCREW* createGUIBALLSCREW();`
  Create ball screw calculation module with user interface.
- `MesysGUICYLGEARPAIR* createGUICYLGEARPAIR();`
  Create cylindrical gear pair calculation module with user interface.

The interfaces MesysRBC, MesysSHAFT and others without GUI create calculation modules without user interface. MesysGUIRBC, MesysGUISHAFT are calculation modules with user interface.

1.9.2.1   General methods

General methods for all calculation modules are:

- `VARIANT_BOOL calculate();`
  Run calculation.
- `VARIANT_BOOL loadFile([in] BSTR p_filename);`
  Load file.
- `VARIANT_BOOL saveFile([in] BSTR p_filename);`
  Save calculation file.
- `VARIANT_BOOL generateReport([in] BSTR p_filename);`
  Generate a report into the filename.
- `VARIANT_BOOL showReport([in] BSTR p_filename);`
  Generate the report into the filename and call a program to view it.
- `VARIANT_BOOL generateSpecialReport([in] BSTR p_type, [in] BSTR`
  `p_filename);`
  Generate a special report into the filename. Use type as "resultTables" for result tables.
- `VARIANT_BOOL generateImage([in] BSTR p_imageID, [in] BSTR p_filename,`
  `[in] int p_dpi, [in] int p_widthinMM, [in] int p_heightinMM);`
  Generate an image and save it as file.
- `VARIANT_BOOL generateImage3D([in] BSTR p_imageID, [in] BSTR p_filename,`
  `[in] int p_dpi, [in] int p_widthinMM, [in] int p_heightinMM, [in]`
  `double p_dirX, [in] double p_dirY, [in] double p_dirZ, [in] double`
  `p_upX, [in] double p_upY, [in] double p_upZ);`
  Generate an image of a 3D-scene and save it as file. dir is a vector in view direction, up is a vector for the up-direction of the view.
- `SAFEARRAY(VARIANT) getDiagramData([in] BSTR p_imageID);`
  Get the diagram data for diagrams. The format of the return value is
  `[["name", ["x", "abscissa label", "abscissa unit", [1, 2, 3, ...],`

```
["y", "ordinate label", "ordinate unit", [1, 2, 3, ...]],
["name", ...]]
```

- `VARIANT_BOOL saveVariableList([in] BSTR p_path);`
  Save the list of available variables into an XLSX file. The available
  variables depend on the input data.
- `SAFEARRAY(VARIANT) getVariableList();`
  Get the available variables as list. The available variables depend on
  the input data. The format is [["name", "type", "unit", "description"],
  ...]
- `VARIANT_BOOL runParameterVariation([in] int p_index, [in] BSTR
  p_resultFilePath);`
  Call the parameter variation with defined input settings for given
  index and saves the result table
- `SAFEARRAY(VARIANT) runParameterVariation_1([in] int p_index);`
- Call the parameter variation with defined input settings for given
  index and returns the result table
- `VARIANT_BOOL setVarDouble([in] BSTR p_name, [in] double p_value);`
  Set variable to double value.
- `VARIANT_BOOL setVarBool([in] BSTR p_name, [in] VARIANT_BOOL p_value);`
  Set variable to boolean value.
- `VARIANT_BOOL setVarInt([in] BSTR p_name, [in] int p_value);`
  Set variable to integer value.
- `VARIANT_BOOL setVarString([in] BSTR p_name, [in] BSTR p_value);`
  Set variable to character value.
- `VARIANT_BOOL getVarDouble([in] BSTR p_name, [in,out] double *p_value);`
  Read back double variable.
- `VARIANT_BOOL getVarBool([in] BSTR p_name, [in,out] VARIANT_BOOL
  *p_value);`
  Read back boolean variable.
- `VARIANT_BOOL getVarInt([in] BSTR p_name, [in,out] int *p_value);`
  Read back integer variable.
- `VARIANT_BOOL getVarString([in] BSTR p_name, [in,out] BSTR *p_value);`
  Read back character variable.
- `VARIANT_BOOL setVar([in] BSTR p_name, [in] VARIANT p_value);`
  Set variable using variant
- `VARIANT getVar([in] BSTR p_name);`
  Get variable using variant
- `VARIANT_BOOL resizeArray([in] BSTR p_name, [in] int p_count);`
  Resize array variable

Each calculation module with GUI provides following methods:

- `void showWindow();`
  Show user interface.
- `void hideWindow();`
  Hide user interface.
- `void exec();`
  Show user interface and process events until the window is closed.
- `void setLanguage([in] BSTR p_lang);`
  Pass language as two characters ("de", "en", "fr", "es", "tr", "ko",
  "zh", "ja")
- `MesysModule* getCalcModule();`
  Get the calculation module from the user interface.
- `void setCalcModule(MesysModule*);`
  Set the calculation module for the user interface.

The names for variables of the calculation modules are the same as provided for the [custom report templates](#) 44 . Currently the use for the shaft calculation is limited as additional functions for changing loads and supports are missing.

1.9.2.2   Additional methods for shaft calculation

Additional methods for the shaft calculation are:

- `VARIANT_BOOL importREXS([in] BSTR p_path);`
  `Import system in REXS format`
- `VARIANT_BOOL exportREXS([in] BSTR p_path);`
  `Export system in REXS format`
- `void resizeLoadSpectrum([in] int p_count);`
  `Resize load spectrum, which needs to be active.`
- `VARIANT_BOOL setLoadSpectrumElement([in] int p_id, [in] BSTR`
  `p_component, [in] int p_index, [in] double p_value);`
  `Set data for load spectrum element`
- `VARIANT_BOOL setPosition([in] int p_id, [in] double p_position);`
  `Set position for force or support`
- `SAFEARRAY(VARIANT) getShaftIds();`
  `Get list of shaft IDs`
- `SAFEARRAY(VARIANT) getForceIds();`
  `Get list of force IDs`
- `SAFEARRAY(VARIANT) getSupportIds();`
  `Get list of support IDs`
- `SAFEARRAY(VARIANT) getSectionIds();`
  `Get list of section IDs`
- `SAFEARRAY(VARIANT) getBearingIds();`
  `Get list of bearing IDs`
- `SAFEARRAY(VARIANT) getBallscrewIds();`
  `Get list of ballscrew IDs`
- `SAFEARRAY(VARIANT) getGroupIds();`
  `Get list of group IDs`
- `SAFEARRAY(VARIANT) getElasticPartIds();`
  `Get list of elastic part IDs`
- `int getShaftIdByName([in] BSTR p_name);`
  `Get ID for first shaft with given name. Returns zero if not found.`
- `int getForceIdByName([in] BSTR p_name);`
  `Get ID for first force with given name. Returns zero if not found.`
- `int getSupportIdByName([in] BSTR p_name);`
  `Get ID for first support with given name. Returns zero if not found.`
- `int getSectionIdByName([in] BSTR p_name);`
  `Get ID for first section with given name. Returns zero if not found.`
- `int getBearingIdByName([in] BSTR p_name);`
  `Get ID for first bearing with given name. Returns zero if not found.`
- `int getBallscrewIdByName([in] BSTR p_name);`
  `Get ID for first ballscrew with given name. Returns zero if not found.`
- `int getElasticPartIdByName([in] BSTR p_name);`
  `Get ID for first elastic part with given name. Returns zero if not`
  `found.`
- `BSTR getName([in] int p_id);`
  `Get element name for ID`
- `VARIANT getIDVar([in] int p_id, [in] BSTR p_name);`
  `Get variable for element with given ID`
- `VARIANT_BOOL setIDVar([in] int p_id, [in] BSTR p_name, [in] VARIANT`
  `p_value);`
  `Set variable for element with given ID`
- `VARIANT_BOOL resizeIDArray([in] int p_id, [in] BSTR p_name, [in] int`
  `p_count);`
  `Resize array variable for element with given ID`

- `MesysRBC* getBearingModule([`in`] `int` p_id);`
  Get bearing calculation for given ID as copy
- `VARIANT_BOOL setBearingModule([`in`] `int` p_id, [`in`] MesysRBC* p1);`
  Set bearing calculation for given ID
- `MesysBALLSCREW* getBallScrewModule([`in`] `int` p_id);`
  Get ballscrew calculation for given ID as copy
- `VARIANT_BOOL setBallScrewModule([`in`] `int` p_id, [`in`] MesysBALLSCREW* p1);`
  Set ballscrew calculation for given ID
- `double getForceValue([`in`] `int` p_id, [`in`] BSTR p_name);`
  Get parameters for force elements like 'x', 'b', 'Fx', 'Fy', 'Fz', 'Mx', 'My', 'Mz'
- `VARIANT_BOOL setForceValue([`in`] `int` p_id, [`in`] BSTR p_name, [`in`] double p_value);`
  Set parameters for force elements like 'x', 'b', 'Fx', 'Fy', 'Fz', 'Mx', 'My', 'Mz'

1.9.2.3   Additional methods for bearing calculation

Additional methods for bearing calculation are:

- `VARIANT_BOOL calculateBearing([`in`] VARIANT_BOOL p_enterFx, [`in`] double p_Fx_or_Ux, [`in`] VARIANT_BOOL p_enterFy, [`in`] double p_Fy_or_Uy, [`in`] VARIANT_BOOL p_enterFz, [`in`] double p_Fz_or_Uz, [`in`] VARIANT_BOOL p_enterMy, [`in`] double p_My_or_Ry, [`in`] VARIANT_BOOL p_enterMz, [`in`] double p_Mz_or_Rz, [`in`] double p_ni, [`in`] double p_ne, [`in`] double p_Ti, [`in`] double p_Te);`
  Run bearing calculation with given loading.
- `VARIANT_BOOL setupLoadSpectrum([`in`] `int` p_count, [`in`] VARIANT_BOOL p_enterFx, [`in`] VARIANT_BOOL p_enterFy, [`in`] VARIANT_BOOL p_enterFz, [`in`] VARIANT_BOOL p_enterMy, [`in`] VARIANT_BOOL p_enterMz);`
  Setup load spectrum definition.
- `VARIANT_BOOL setLoadSpectrumLoad([`in`] `int` p_loadCase, [`in`] double p_frequency, [`in`] double p_Fx_or_Ux, [`in`] double p_Fy_or_Uy, [`in`] double p_Fz_or_Uz, [`in`] double p_My_or_Ry, [`in`] double p_Mz_or_Rz, [`in`] double p_ni, [`in`] double p_ne, [`in`] double p_Ti, [`in`] double p_Te, [`in`] double p_TOil);`
  Set load spectrum load for each load case
- `VARIANT_BOOL selectBearingFromDatabase([`in`] BSTR p_manufacturer, [`in`] BSTR p_bearingName);`
  Selects a bearing based on name and manufacturer from the database
- `VARIANT_BOOL getStiffnessMatrix([`in,out`] double *p_c11, [`in,out`] double *p_c12, [`in,out`] double *p_c13, [`in,out`] double *p_c14, [`in,out`] double *p_c15, [`in,out`] double *p_c21, [`in,out`] double *p_c22, [`in,out`] double *p_c23, [`in,out`] double *p_c24, [`in,out`] double *p_c25, [`in,out`] double *p_c31, [`in,out`] double *p_c32, [`in,out`] double *p_c33, [`in,out`] double *p_c34, [`in,out`] double *p_c35, [`in,out`] double *p_c41, [`in,out`] double *p_c42, [`in,out`] double *p_c43, [`in,out`] double *p_c44, [`in,out`] double *p_c45, [`in,out`] double *p_c51, [`in,out`] double *p_c52, [`in,out`] double *p_c53, [`in,out`] double *p_c54, [`in,out`] double *p_c55);`
  Reads back the bearing stiffness matrix with units N, Nm, mm, rad. The order of rows and columns is like in the report ux, uy, uz, ry, rz.
- `SAFEARRAY(VARIANT) getStiffnessMatrixAsVector();`
  Reads back the bearing stiffness matrix as vector with units N, Nm, mm, rad. The order of the elements is per row.
- `SAFEARRAY(VARIANT) getRollingElementResults([`in`] `int` p_loadCase, [`in`] `int` p_bearing, [`in`] `int` p_row, [`in`] `int` p_rollingElement, [`in`] `int` p_section);`
  Returns results for single rolling elements. The parameter 'section' is only used for roller bearings, for section==-1 the forces acting on the

roller are returned. The parameter 'loadCase' is only used if a load
spectrum is defined.

1.9.2.4   Additional methods for ball screw calculation

Additional methods for the ball screw calculation are:

- `VARIANT_BOOL calculateBallScrew([in] VARIANT_BOOL p_enterFx, [in]
  double p_Fx_or_Ux, [in] VARIANT_BOOL p_enterFy, [in] double p_Fy_or_Uy,
  [in] VARIANT_BOOL p_enterFz, [in] double p_Fz_or_Uz, [in] double p_Rx,
  [in] VARIANT_BOOL p_enterMy, [in] double p_My_or_Ry, [in] VARIANT_BOOL
  p_enterMz, [in] double p_Mz_or_Rz, [in] double p_ni, [in] double p_ne,
  [in] double p_Ti, [in] double p_Te);`
  Run ball screw calculation with given loading
- `VARIANT_BOOL setupLoadSpectrum([in] int p_count, [in] VARIANT_BOOL
  p_enterFx, [in] VARIANT_BOOL p_enterFy, [in] VARIANT_BOOL p_enterFz,
  [in] VARIANT_BOOL p_enterMy, [in] VARIANT_BOOL p_enterMz);`
  Setup load spectrum definition
- `VARIANT_BOOL setLoadSpectrumLoad([in] int p_loadCase, [in] double
  p_frequency, [in] double p_Fx_or_Ux, [in] double p_Fy_or_Uy, [in]
  double p_Fz_or_Uz, [in] double p_Rx, [in] double p_My_or_Ry, [in]
  double p_Mz_or_Rz, [in] double p_ni, [in] double p_ne, [in] double
  p_Ti, [in] double p_Te);`
  Sets load spectrum load for each load case
- `VARIANT_BOOL getStiffnessMatrix([in,out] double *p_c11, [in,out] double
  *p_c12, [in,out] double *p_c13, [in,out] double *p_c14, [in,out] double
  *p_c15, [in,out] double *p_c21, [in,out] double *p_c22, [in,out] double
  *p_c23, [in,out] double *p_c24, [in,out] double *p_c25, [in,out] double
  *p_c31, [in,out] double *p_c32, [in,out] double *p_c33, [in,out] double
  *p_c34, [in,out] double *p_c35, [in,out] double *p_c41, [in,out] double
  *p_c42, [in,out] double *p_c43, [in,out] double *p_c44, [in,out] double
  *p_c45, [in,out] double *p_c51, [in,out] double *p_c52, [in,out] double
  *p_c53, [in,out] double *p_c54, [in,out] double *p_c55);`
  Reads back the stiffness matrix with units N, Nm, mm, rad. The order of
  rows and columns is like in the report ux, uy, uz, ry, rz.
- `SAFEARRAY(VARIANT) getStiffnessMatrixAsVector();`
  Reads back the bearing stiffness matrix as vector with units N, Nm,
  mm, rad. The order of the elements is per row.
- `SAFEARRAY(VARIANT) getRollingElementResults([in] int p_loadCase, [in]
  int p_bearing, [in] int p_thread, [in] int p_rollingElement);`
  Returns results for single rolling elements. The parameter 'loadCase'
  is only used if a load spectrum is defined and should be set to zero
  otherwise.

1.9.3   Using the COM server

The COM server can be used to call functions within the MESYS software from
other programs.

A simple example in a VisualBasic function:

```
Public Sub test()
Dim mesys As MesysCOM
Set mesys = New MesysCOM
Dim rbc As MesysRBC

Set rbc = mesys.createRBC
Dim d As Long
Dim pmax As Double
```

```
Call rbc.setVarInt("inputType", 2)
Call rbc.setVarInt("Z", 12)
Call rbc.setVarDouble("Dw", 5)
Call rbc.setVarDouble("Dpw", 50)
Call rbc.setVarDouble("Fy", 1000)
Call rbc.setVarDouble("ni", 500)

result = rbc.Calculate
Call rbc.showReport("c:/temp/report.pdf")
Call rbc.getVarDouble("pmax", pmax)
Call rbc.getVarInt("Z", d)
Call rbc.calculateBearing(True, 0, True, 1000, True, 0, False, 0, False, 0, 500, 0, 20, 20)
Call rbc.getVarDouble("pmax", pmax)
Set rbc = Nothing
Set mesys = Nothing

End Sub
```

And here the same example in Python using the members with variants as [in, out] parameters do not work from python:

```
import comtypes.client

mesys = comtypes.client.CreateObject("MesysCOM64.MesysCOM")
rbc = mesys.createRBC()
rbc.setVar("inputType", 2)
rbc.setVar("Z", 12)
rbc.setVar("Dw", 5)
rbc.setVar("Dpw", 50)
rbc.setVar("Fy", 1000)
rbc.setVar("ni", 500)
rbc.calculate()
rbc.showReport("c:/temp/report.pdf")
pmax = rbc.getVar("pmax")
z = rbc.getVar("Z")
rbc.calculateBearing(True, 0, True, 1000, True, 0, False, 0, False, 0, 500, 0, 20, 20)
pmax2 = rbc.getVar("pmax")
stiffness = rbc.getStiffnessMatrixAsVector();
rbc = None
mesys = None
```

1.10   Tools

In menu "Extras"->"Tools" some addition calculation tools are available.

1.10.1   Interference fit

This tool allows the calculation of multi-layered interference fits considering thermal expansion and centrifugal forces.

Calculate interference fit

Inner diameter	d	0	mm	Number of layers		2	
Internal pressure	pi	0	MPa	External pressure	pe	0	MPa
Rotation speed	n	0	rpm	Width	b	0	mm

	1	2
Diameter D [mm]	50	70
Interference lw [µm]	5	0
Temperature T [°C]	20	20
Youngs modulus E [MPa]	207000	207000
Poisson number ν [-]	0.3	0.3
Density rho [kg/m³]	7850	7850
Thermal elongation coeff.α [10^{-6}/K]	11.5	11.5
Friction coefficient µ	0.1	0.1
Inner diameter change Δd [µm]	0	4.14279
Outer diameter change ΔD [µm]	-0.857214	3.57137
Radial stress at inner diameter σri [MPa]		-5.0693
Radial stress at outer diameter σre [MPa]	-5.0693	0
Tangential stress at inner diameter σti [MPa]		15.6303
Tangential stress at outer diameter σte [MPa]	-5.0693	10.561
Maximal axial force Fmax [N]	0	
Maximal torque Tmax [Nm]	0	

Calculate   Report   Close

The inner diameter for the inner section can be defined on top of the window, each section then only has the outside diameter as input. Also an internal pressure can be defined on the inner layer and an external pressure on the outer layer. The interference is an additional input for each layer and is not determined by diameters.

If a friction coefficient and a width are defined, then a maximal permissible axial force and torque are shown on the last two rows.

MESYS Shaft Calculation

2   MESYS Shaft Calculation

The shaft calculation (Version 06/2024) calculates the deflections, internal forces and the natural frequencies of several shafts connected by boundary conditions. The following features are supported:

- Definition of multiple coaxial shafts is possible

- Definition of parallel shafts (with extension for shaft systems)

- Definition of shafts in arbitrary direction (with extension for advanced shaft systems)

- Shaft geometry is defined by cylindrical and conical elements

- Inner and outer geometry can be defined independently

- Shear deformation can be considered optionally

- A nonlinear shaft model can be used optionally

- The weight of the shaft can be considered optionally

- An arbitrary number of loads can be defined on each shaft either as point or line loads. Loads (without masses) may be defined outside of the shaft geometry also. Available loading elements are:

  o Force element with three components each for force and moment

  o Eccentrical force element with three force components in polar coordinates

  o Helical Gear element using gear data and torque as input

  o Bevel/Hypoid Gear element using gear data and torque as input

  o Worm/Worm Gear element using gear data and torque as input

  o Coupling element for entering a torque only

  o Pulley element for torque and pretension

  o Mass elements for introducing weight and inertia for natural frequencies

  o Imbalance and dynamic force as periodic loads for harmonic response

- An acceleration can be defined for the system and is considered as inertia force

- An arbitrary number of boundary constraints can either be defined between a shaft and a rigid housing or between two coaxial shafts.

- Several housing stiffness matrices can be considered either with three or six degrees of freedom per node

- Import of shaft geometry as 2D DXF or 3D STEP file

- Strength calculation according DIN 743 (with extension for shaft strength)

- Connections by cylindrical gear pairs (with extension for shaft systems)

- Connections by bevel gear pairs (with extension for advanced shaft systems)

- Connections by worm-worm gears (with extension for advanced shaft systems)

- Planetary gear sets (with extension for advanced shaft systems)

- Centrifugal forces are considered on planet shafts

- Calculation of shaft speeds considering given constraints (with extension for shaft systems)

- Data exchange with several programs for gear calculation (with extension for shaft systems)

- Nonlinear rolling bearing stiffness can be considered

- Bearings can be selected from a database

- MESYS Rolling Bearing Calculation [146] is fully integrated in the shaft calculation

- MESYS Ballscrew Calculation [242] is fully integrated in the shaft calculation

- MESYS Cylindrical Gear Pair [300] is fully integrated in the shaft calculation

- Calculation using load spectra

- Natural frequencies are calculated considering torsion, bending and axial modes (with extension for modal analysis)

- Campbell diagram (with extension for modal analysis)

- Harmonic response to periodic forces (with extension for modal analysis)

- Frequency response over a given range (with extension for modal analysis)

- Thermal expansions, optionally with varying temperature on a shaft

- Parameter variations

- 3D axisymmetric elastic parts considered by 3D FEA (with extension for 3D elastic parts)

- 3D elastic planet carriers defined parametrically or imported as STEP file (with extension for 3D elastic parts)

- Import of housings as STEP file and consideration as stiffness matrix and optionally by modal reduction (with extension for 3D elastic parts)

- Consideration of elastic deformation of bearing rings (with extension for 3D elastic parts and full bearing calculation)

- Consideration of gear body deformation (with extension for 3D elastic parts and shaft systems)

2.1 Input Parameters

The input parameters are shown on several pages which can be selected in a system tree.

2.1.1 System Inputs

If the element 'System' is selected in the system tree on the left, some general data can be defined.
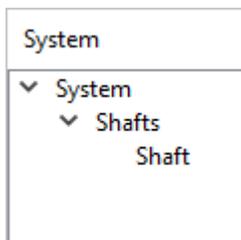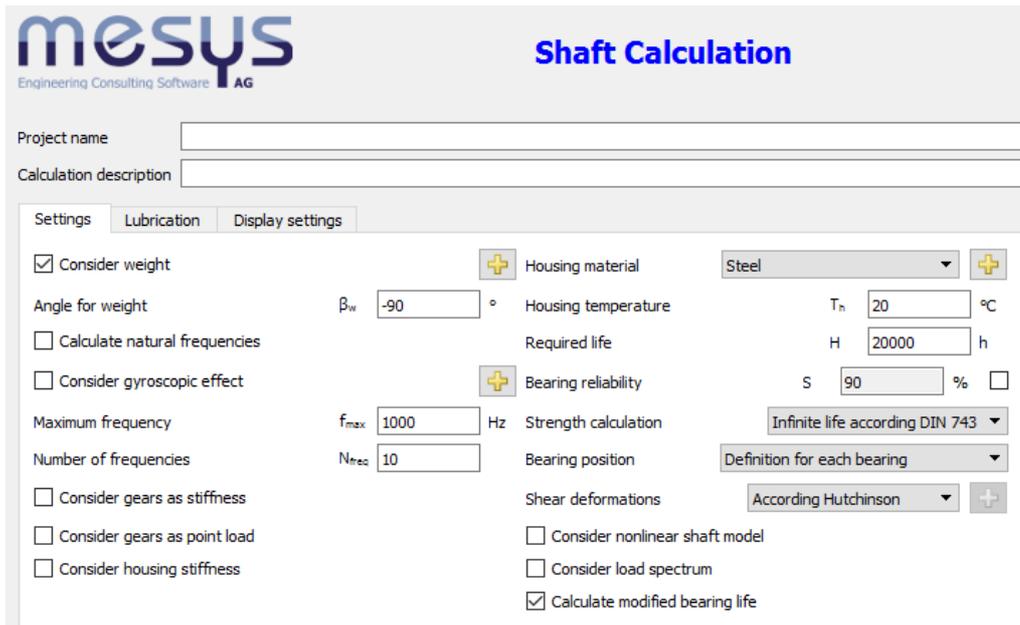
2.1.1.1 Project name and calculation description

The project name and the calculation description are just inputs which are shown in the report header. They can be used to enter information about the purpose of the calculation for documentation.

2.1.1.2 Settings

2.1.1.2.1 Consider weight and accelerations

The weight of shafts and additional masses is considered in the calculation if this option is set.
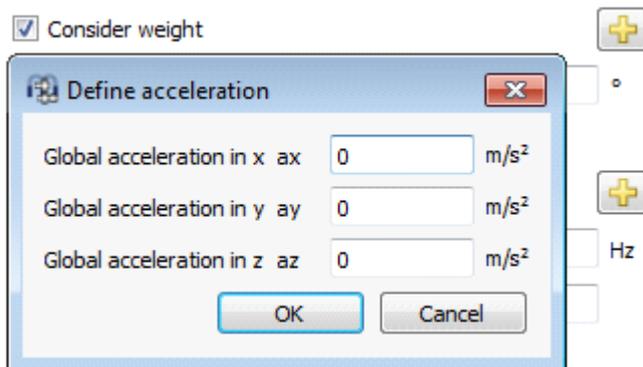
The ✚-button next to the weight checkbox allows the definition of global accelerations for the three axes. The accelerations are also available within the definition of a load spectrum.

The weight is already considered by setting the flag "Consider weight". It does not have to be entered as acceleration. The input for accelerations is thought for additional accelerations like in gearboxes in vehicles.

2.1.1.2.2 Angle for weight

The direction of the weight can be defined by this setting. The angle is in the x-y-plane, a rotation around z-axis. A value of zero results in a weight in the direction of shaft axis. The weight direction is also shown as an arrow in the shaft graphics and it can be varied within the load spectrum.

Angle 0°   Angle -90°   Angle 180°
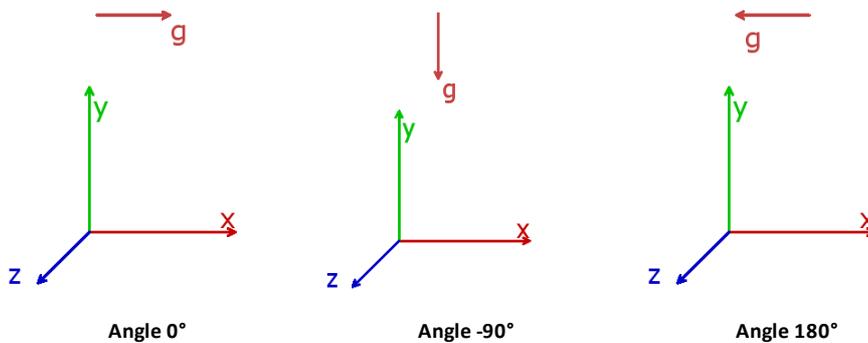
2.1.1.2.3 Calculate natural frequencies

The calculation of natural frequencies can be activated here. Running a calculation is faster if natural frequencies are not calculated. If a load spectrum is active, the natural frequencies are calculated only for the selected result element.

Calculation of natural frequencies also has to be activated if an harmonic response to dynamic loads should be calculated.

2.1.1.2.4 Consider gyroscopic effect

If this setting is activated, a gyroscopic matrix is considered in the calculation of natural frequencies. The mass inertia around x-axis and the speed of the shaft are required for this gyroscopic matrix.

The ✚-button allows some additional inputs for the Campbell diagram:

- The diagram is generated for speeds between start and end factor times the current input value for speed.

- The factor for frequency range defines the length of the ordinate of the Campbell diagram. More frequencies are calculated for a given speed range in case of a larger factor.

- The number of speeds is the number of calculation points. Reduce it for faster calculation. For harmonic response over speed a larger value leads to smoother curves.

- The number of harmonics shows additional lines for multiples of shaft speed.

- The Campbell diagram is calculated in parallel as default. The maximum number of threads can be used to limit the number of threads that are used. This can be applicable if the system memory is not sufficient for multiple parallel calculations of large systems.

- The accuracy for critical frequency is a tolerance for calculation of critical speeds as intersections of shaft speed and natural frequency.

- If 'Increase accuracy for mode changes' is selected, additional points are calculated when an axial mode changed into a radial mode for example.

- If 'Calculate critical frequencies' is selected, the critical frequencies are calculated when the Campbell diagram is shown. Else they are only calculated for the report of critical speeds. The diagram harmonic response over speed gets more accurate in this option is set because additional calculation points at the critical speeds are considered.

- If 'Consider gear frequencies' is selected, additional lines for gear tooth frequencies are shown and the corresponding critical frequencies are calculated.

- If the simplified method for harmonic response is activated, the harmonic response is calculated as superposition of mode shapes. The accuracy of the solutions depends of the calculated number of mode shapes in this case. For large systems the calculation can be faster if the option is activated. The result is more accurate if it is not activated.

- Coefficients for global Rayleigh damping are used to define a global damping matrix as D=α*M+β*K using the global mass matrix M and the global stiffness matrix K. Using the sizing button, the values can be calculated so that a user defined damping ratio will result for two given frequencies.

2.1.1.2.5 Maximum frequency

The maximum value for natural frequencies that are of interest can be entered here. If this maximum frequency is set too small, in some cases no results for natural frequencies might be shown.

2.1.1.2.6 Number of frequencies

The number of natural frequencies that should be calculated can be specified here. Normally only the first few modes are interesting. Both limits, the number of frequencies and the maximum are considered. If the Campbell diagram shows not enough lines, these restrictions are often the reason.

2.1.1.2.7 Consider dynamic loads in static equlibrium

If this option is set, dynamic forces and imbalance are considered in the static equilibrium. This approximation is good enough in case the shaft speed is well below the critical speed. The value of the force is taken for time zero, so the direction of an imbalance is in direction of the input phase.

2.1.1.2.8 Housing material and housing temperature

The material data of the housing is used together with the temperature of the housing to calculate the axial displacement for boundary conditions connected to the housing. For the shaft calculation only the thermal elongation coefficient is important, the other data is only used to transfer it to the bearing calculation.

2.1.1.2.9 Required life

The required life is passed to gear calculations which are connected to the system, and it is used for the shaft strength calculation.

2.1.1.2.10 Bearing reliability

The bearing reliability can be optionally defined on system level and is then passed to all bearing calculations.

2.1.1.2.11 Strength calculation

The method for shaft strength calculation can be selected. Currently, only DIN 743 (2012) is available. It can be selected if the calculation should be done considering infinite or finite life. For finite life the number of cycles is calculated using the input for "Required life".

2.1.1.2.12 Bearing position

The position of the first rolling element for rolling element bearings can be defined globally using this settings. See Position of first rolling element 152.

In case of bearings with clearance and very small loads it might happen, that only one rolling element is loaded. This can lead to convergence problem as there is zero stiffness normal to the loading. Then setting the option to "Load direction between rolling elements" might be helpful. Else the setting "First rolling element on load direction" leads to the maximum contact stress and the minimum bearing life in most cases. Usually there is only a noticeable effect on bearing with a low number of rolling elements.

2.1.1.2.13 Shear deformations

Shear deformation should normally be taken into account, since there are these deformations. For comparison with other calculations the shear deformations can be ignored. For nonlinear shaft model the shear deformations should be considered to improve convergence.

There are different options available. Either a fixed value can be entered. Usually 1/1.1 is used for cylinders. The options 'according Cowper' and 'according Hutchinson' are also considering an inner diameter for hollow shafts and the Poisson number of the material. Hutchinson considers higher order terms in his formula than Cowper according to (Hutchinson, January 2001) (Cowper., June 1966):

- Hutchinson (Circular Cross Section):

$$k = \frac{6(1 + v)^2}{7 + 12v + 4v^2}$$

- Hutchinson (Hollow Circular Cross Section):

$$k = \frac{6(a^2 + b^2)^2(1 + v)^2}{7a^4 + 34a^2b^2 + 7b^4 + v(12a^4 + 48a^2b^2 + 12b^4) + v^2(4a^4 + 16a^2b^2 + 4b^4)}$$

where 'b' is the outer radius, 'a' the inner radius and 'v' the Poisson's ratio.

- Cowper (Circular Cross Section):

$$k = \frac{6(1 + v)}{7 + 6v}$$

- Cowper (Hollow Circular Cross Section):

$$k = \frac{6(1 + v)(1 + m^2)^2}{(7 + 6v)(1 + m^2)^2 + (20 + 12v)m^2}$$

where $m = b/a$ is the ratio of inner to outer radius and 'v' the Poisson's ratio.

2.1.1.2.14 Consider nonlinear shaft model

A nonlinear shaft model can be taken into account. The nonlinear model calculates equilibrium of loads in the deformed state.

A beam fixed on one side and radially loaded on the other side will show only radial deflections in the linear model. In the nonlinear model the end point will also move axially and the length of the beam will not increase. A beam fixed on both sides will show an increased stiffness against loading in the center, the stiffness will increase with the loading because of generated axial loads. The nonlinear model is only needed on large deflections; normal shafts in mechanical engineering do not require a nonlinear model.

2.1.1.2.15 Consider load spectrum

If a load spectrum should be taken into account, select this option. An additional item for the load spectrum will be shown in the tree on the left. See Calculation with load spectrum for the definition of the load spectrum.

2.1.1.2.16　Consider gears as stiffness

As default a gear is just considered as a load element and it adds neither mass nor stiffness to the shafts.

Optionally the gear body can be considered as mass and stiffness. Available options are:

- "Increase shaft diameter": If this option is selected, the shaft diameter is increased to root diameter plus 0.4*module automatically. For the root diameter a dedendum of the reference profile of 1.25 is assumed. The shaft will be too stiff using this option as it cannot deform inside the gear body.

- "3D-model using central nodes": A 3D-FEA model is used for the gear body with the restriction that the inner and the outer cylinder will stay cylindrical. The torsional and bending deformations at the inner and the outer diameter can be different because of the 3D-model. The outer diameter of the 3D-model is set to the root diameter plus 0.4*module. The same material properties as for the shaft are used.

- "3D-model": A 3D-FEA model is used for the gear body as if a 3D-elastic parts as elastic gear is used. The license for 3D-elastic parts is required. The torsional and bending deformations at the inner and the outer diameter can be different because of the 3D-model. The outer diameter of the 3D-model is set to the root diameter plus 0.4*module. The same material properties as for the shaft are used.

- "3D-model with teeth": A 3D-FEA model is used for the gear body as if a 3D-elastic parts as elastic gear is used and the teeth are added to the 3D-model. The license for 3D-elastic parts is required. The torsional and bending deformations at the inner and the outer diameter can be different because of the 3D-model. This option will need much more initial calculation time and needs to be updated on every change of gear data. Therefore it should only be activated in the end phase of the design. The same material properties as for the shaft are used.

2.1.1.2.17　Consider gears as a point load

If this option is selected, the load line from the meshing of gears is converted to a point load. This is useful for calculating bearing loads when the user wants to consider perfectly centered gears loads, that is, no misaligments in tooth mesh.

2.1.1.2.18　Calculate modified bearing life

This setting is passed on to the bearing calculation and activates or suppresses output of modified life in results. It can be disabled if no information about lubrication is available.

2.1.1.2.19 Consider configurations

If multiple groups are used, an additional option is shown to activate configurations. Using the new input page for configurations the status of connections can be changed to allow different options for power flow. See Configurations [105] for usage.

2.1.1.2.20 Consider housing stiffness

When this option is set, an additional page for definition of a housing stiffness matrix is shown in the system tree. See Housing stiffness matrix [106] for details of usage.

A housing stiffness can also be considered by using 3D elastic part as housing [114], this would be easier to use compared to a raw stiffness matrix.

2.1.1.3 Lubrication

The data for lubrication is only to transfer it to bearing or gear calculations. For details regarding the inputs, please consult the documentation of the rolling bearing calculation. The 'FZG load stage' is an additional input which is passed to a gear calculation.

2.1.1.4 Display settings

There are different view options that can be set under the tab-page 'Settings'.

Cutaway in 3D: It is possible to generate a cutaway view for all the existing shafts. If needed, the cutaway can be only applied to the hollow shafts. This setting can be overridden for individual shafts.
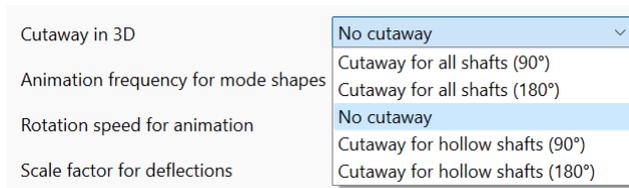
With regard to the 3D animation settings, both the frequency for modes shapes and the rotation speed are adjustable. The default frequency of 0.2Hz for the mode shape above, will lead to a period of 5 seconds in the animation. The 3D pressure distribution of the bearings can be easily exaggerated with a scale factor 'Sp'. And with the 'Su' the deformed shape in 3D can be also scaled. Setting 'Su' to zero will show the deformed shape in 3D without scaling, so scale factor of one.

The look of the shaft geometry in 2D can be emphasized by applying a lighting factor 'fl'. The factor 'fld' is used for the shaft background graphic in 2D diagrams.

The input value for 'sigV' allows the user to specify the maximum permissible stress that any shaft shall reach under loading. In this way, the software shows graphically an approximation of required diameter of the shafts subjected to a maximum value of equivalent stress. Please refer to the 'Graphics area' section of this document on how to visualize the required diameter diagram.

Colors for bearings can be specified on the right of the window. The left button is for bearings, which are connected to one shaft and the housing, while the right button is for bearings connected to two shafts. The upper color is for bearings from the database with approximated internal geometry, then for bearings from the database including internal geometry and then bearings with custom geometry input.

In addition, a default color for shafts and gears can be defined.

2.1.1.5 System options in the system tree

Several options are available in the system tree under "Shafts" and "Bearings" using the context menu.

The context menu under "Shafts" provides following options:

- "Add shaft" adds a shaft to the system.If groups are used shafts can only be added to groups.

- "Add group" adds a group to the systems.

- "Add coaxial group" adds a coaxial groups. The difference between coaxial groups and groups is, that coaxial groups only have an axial position and support elements can be used between shafts in different coaxial groups.

Add Shaft
Add Group
Add Coaxial Group
Add Planetary Group
Add elastic part as housing
Add elastic part as shaft
Import shaft system
Export shaft system
Export geometry
Sort groups and shafts

- "Add planetary group" adds a planetary group. A planetary group rotates about a carrier shafts and a number of planets can be defined on the group. All elements are generated "number of planets" times.

- "Add elastic part as housing" adds a 3D-elastic part as housing. This part does not rotate. See 3D elastic part as housing 114 for details.

- "Add elastic part as shaft" adds a 3D-elastic part as shaft. This part can rotate and it can also be a planet carrier. See 3D elastic part as shaft 110 for details.

- "Import shaft system" allows to import an existing shaft system. Either a file of a MESYS shaft calculation can be selected of a file in REXS format.

- "Export shaft system" allows the export of the system in REXS format. This format allows the data exchange with other CAE software.

- "Export geometry" exports the system as STEP file. Only a simplified geometry is used for gears and bearings. The export could be used to check it it would fit into a housing for example.

- "Sort groups and shafts" sorts shafts and groups alphabetically. Initially the elements are sorted in creation order.

- "Add system" generated a cylindrical gearbox or a planetary gearbox.
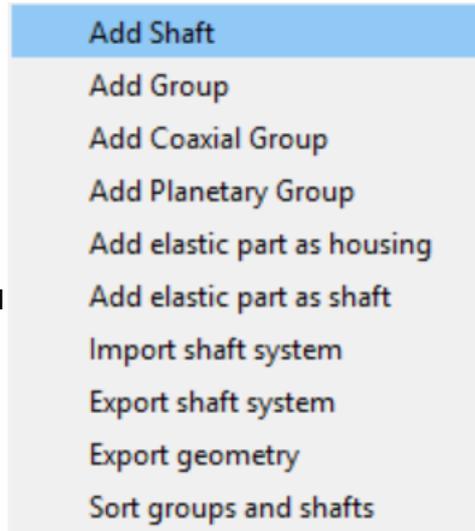
The context menu under "Bearings" allows to activate or deactivate settings for all bearing calculations at the same time:

Activate 'Extended pressure distribution' for all roller bearings

Deactivate 'Extended pressure distribution' for all roller bearings

Activate 'Consider centrifugal force' for all bearings

Deactivate 'Consider centrifugal force' for all bearings

Select minimum clearance for all bearings

Select medium clearance for all bearings

Select maximum clearance for all bearings

Select 'Elastic ring expansion is not considered' for all bearings

Select 'Elastic ring expansion based on mimimum radial force' for all bearings

Select 'Elastic ring expansion based on medium radial force' for all bearings

Select 'Gyroscopic moment is not considered' for all ball bearings

Select 'Gyroscopic moment based on outer raceway control' for all ball bearings

Select 'Gyroscopic moment based on outer raceway control, distributed force' for all ball bearings

Select 'Gyroscopic moment based on friction' for all ball bearings

2.1.2   Definition of shafts
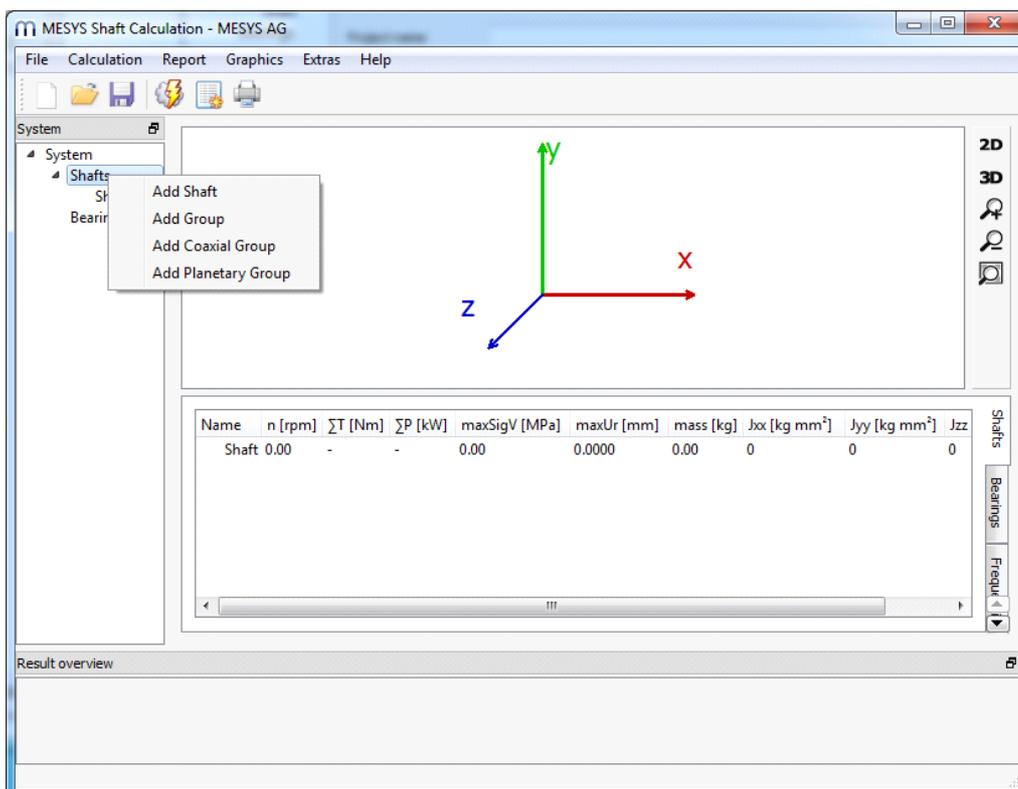
In the system tree using a right click on "Shafts", additional shafts can be added to the system. The shaft can be defined by selecting it in the system tree.

Either shafts can be directly added, if only single or coaxial shafts are used. For general shaft systems groups or coaxial groups can be used to build shaft systems. Planetary groups are used for planet shafts to model planetary gear sets.

2.1.2.1   General shaft data

On the first page "General" some general data and data for the strength calculation can be defined for each shaft.

2.1.2.1.1 Name

A name for each shaft can defined which is used in the system tree, messages and the report to identify the shaft.

2.1.2.1.2 Material

The material can be selected from the data base. The ✚-button allows to see the material data or to change the data if user-input is selected.

2.1.2.1.3 Position

If several shafts are defined, they can have different axial positions. The value for position defines the position of the start point or the left end of a shaft relative to the group. All loads and boundary conditions are then defined relative to this start point.

The shaft position cal also be changed by dragging the shaft in the 2D view using left mouse button together with Shift and Control keys.

2.1.2.1.4 Speed

The rotation speed of the shaft has an influence on bearing life and natural frequencies with gyroscopic effect. If the checkbox behind the input is not selected, the software is trying to calculate it using constraints. It is set to zero if the shaft is not connected to others.

A positive speed is a clockwise rotation looking in the x-direction.

2.1.2.1.5 Temperature

The temperature of the shaft can be defined here. The temperature is transferred to the bearing calculation and will induce axial stresses into the shaft.

In case that the shaft temperature is not uniform, the user must set the flag corresponding to 'Consider temperature per element' at the tab-page 'Settings'. That allows the user to add a temperature difference 'ΔT'(±) for each element created at the outer geometry on the temperature previously set at the tab-page 'General'. The reference temperature with no stress field is 20°C.

2.1.2.2 Data for strength calculation

There are several inputs for the strength calculation according DIN 743 which are defined for each shaft.

2.1.2.2.1 Load factor (static)

For the calculation of static safety factor, the maximal load is multiplied with the static load factor.

2.1.2.2.2 Load factor (fatigue)

For the calculation of safety factor against fatigue the maximal load is multiplied with the load factor (fatigue).

2.1.2.2.3 Overload case

The overload case defines how the permissible stress is calculated. Either "Constant stress ratio" or "Constant mean stress" can be selected. "Constant stress ratio" leads to smaller safety factors and is therefore on the safe side.

2.1.2.2.4 Diameter at heat treatment

The diameter at heat treatment is used to calculate the tensile strength and the yield point for the given material. The software sets it to the maximum shafts diameter automatically, but for hollow shafts the wall thickness can be chosen manually, if heat treatment was done for the hollow shaft.

2.1.2.2.5 Number of load cycles

If a calculation for finite life is done, the number of load cycles is derived from rotation speed and required life as default. But for non-rotating shafts this is not possible. Then the number of load cycles can be entered manually.

2.1.2.2.6 Factor for surface hardening

In DIN 743, a factor for surface hardening KV increases permissible stress. It can be set for hardening or mechanical treatment like shot-peening. As the standard provides a wide range for this value, it has to be defined by the user.

2.1.2.2.7 Stress ratio

The stress ratio R is usually defined as minimum stress value divided by maximum stress value, which leads to values between -∞ and +1.

The software is using a value between -1 and +1 instead and it defines the stress ratio using the following definition:

$$R = \begin{cases} \dfrac{\sigma_1}{\sigma_2} & |\sigma_2| > |\sigma_1| \\[2mm] \dfrac{\sigma_2}{\sigma_1} & |\sigma_1| > |\sigma_2| \end{cases}$$

Using this definition R = 0 can be used for pulsating loads between zero and a positive or a negative value. For alternating load R = -1 and for constant load R = +1.

The stress ratio can be defined for tension, bending and torsion independently. For a rotating shaft the stress ratio for bending should usually be -1. If the stress ratio for torsion is not given, make a calculation with two cases: constant and pulsating for example. It depends on the current loading which case is conservative.

2.1.2.3   Geometry data

The geometry data of a shaft is defined on page "Geometry". The geometry can be defined using cylindrical and conical elements for outer and inner geometry. The shaft geometry could also be imported from CAD files, see Import of shaft geometry from CAD files 94 .

There is an input table for both inner and outer geometry. The plus and minus buttons allow adding and removing rows, the arrows can be used to move an element up or down in the table.

Length and diameter 1 have to be defined for each element. If diameter 2 is left empty, a cylinder is used else a cone. No negative values are permitted. The total length is shown above the table.

If a hole from the right shall be defined as inner geometry, enter a first element with diameter zero to get to the start position on the hole.

The dimensions of each element can also be changed in the view. Use Shift and left mouse button for it. For changing the length the click needs to be in the right third of an element. For changing the diameter it has to be in the upper third of an element. To change a cylinder into a cone element the mouse needs to be in the left third of an element. The interactive change is restricted to a 1mm grid. Using Shift and Control buttons will move the shaft instead of changing an element.

The table for outer geometry also allows an input for a temperature difference if the corresponding option under settings is activated.

2.1.2.3.1 Background drawing

Using the contact menu (right mouse button) a background graphic can be added to the 2D-graphic. Here a CAD-drawing can be set as background image if a PDF drawing is available only.

The background drawing is added using "Set from clipboard" after copying a section in the PDF drawing. Then with "Set origin" the location of the drawing is defined and with "Set horizontal scale" or "Set vertical scale" the scale factor is defined. The location on the drawing is the location where the right click is done. So first right click on the location of the origin and select "Set origin", then click on a point with a given vertical or horizontal distance and select the scale.

The background drawing can also be moved to the foreground using the contact menu or it can be rotated.

Then the shaft geometry can be adjusted to the drawing. Either by changing the geometry elements or by using the input as polyline. First define the inner geometry and then the outer geometry.
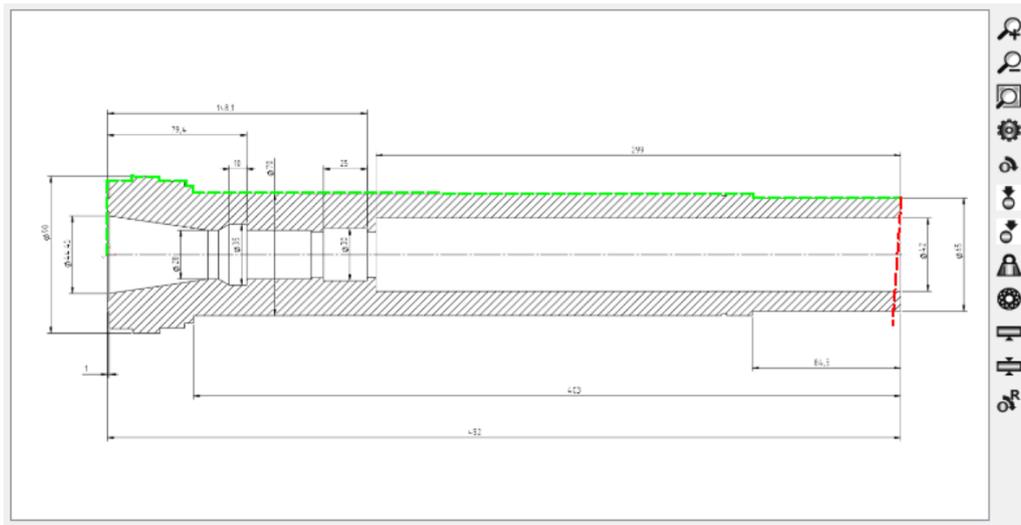
2.1.2.3.2 Define geometry as polyline

Using the context menu an input of inner or outer geometry as polyline can be activated. This is usually done after setting a background drawing, but is independent of it.

Using the left mouse button, points for the polyline can be added. A red line shows an invalid position for the next point and this will not be accepted. Using Shift plus left button the last selected point is removed again. The right mouse button is used to stop the input and the result is set to the table for inner or outer geometry.

The selected points are rounded to a millimeter.

2.1.2.4   Loading

Loads can be defined on the page "Loading" for each shaft. Loads may be defined outside a shaft; masses may not be outside of a shaft. Loads can also be changed by double-clicking them in the graphics. They can be moved in the graphic by using Shift and the left mouse button.
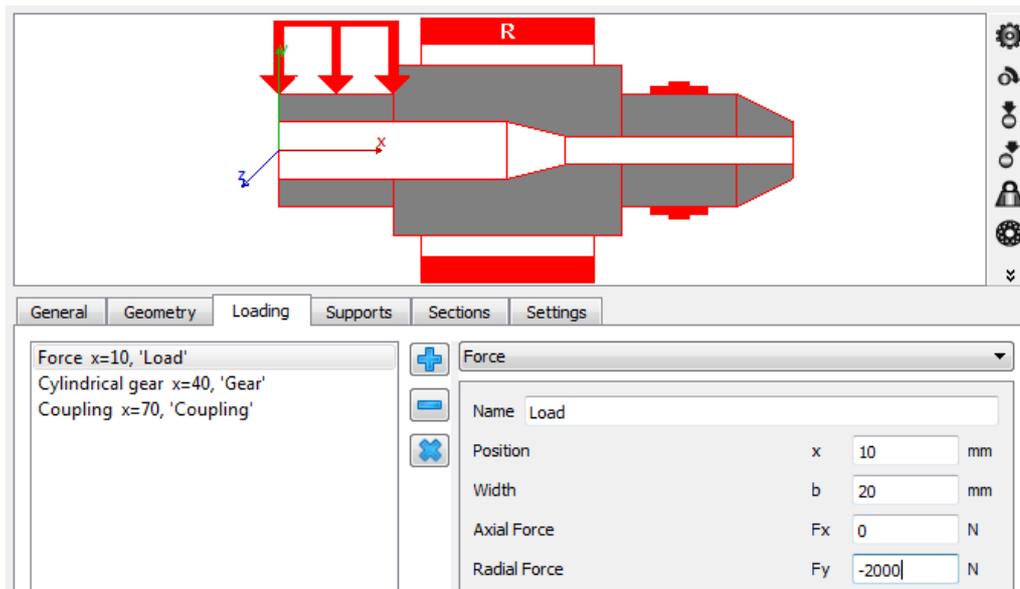
Loads can be added by the plus button next to the list. The type of the loading can be selected using the list on the right. Each load element has a name to identify it, a position relative to the left end of the shaft and a width.

The graphical representation just shows a symbol for the load, not the actual direction, as it can have six components. The coordinate system is shown on the left. The shaft axis is in x-direction, the y-axis goes up and the z-axis to the front. The weight is in negative y-directions a default (angle βw = -90°).

2.1.2.4.1 Force

The load element "Force" allows the definition of a force and a moment with three components each. The moments are moments around an axis. So Fx is the axial force and Mx is the moment around the x-axis so it is a torque.

2.1.2.4.2 Coupling

The coupling only allows a definition of a torque. The direction on the torque can either be defined by its sign or by the selection "Shaft is driven" (torque has the same sign as the rotation speed) or "Shaft is driving" (torque has different sign than the rotation speed).

2.1.2.4.3 Cylindrical gear

The load can be defined by entering geometry data for a cylindrical gear and a torque.

The direction of torque can either be defined by its sign or by the selection "Shaft is driven"/"Shaft is driving". The contact point is given by an angle of contact, which is zero on the y-axis and 90° on the z-axis.

The gear is defined by the usual data on a gear drawing: Number of teeth, normal module, normal pressure angle, helix angle with its direction. Optional inputs are the number of teeth for the mating gear and the center distance. If they are given, the loads are calculated for the operating center distance, which is more accurate than on the reference diameter.

Name	Gear	
Position	x	40 mm
Width	b	30 mm
Torque	T	0 Nm
Direction of torque		Own Input
Angle to contact	ζ	0 °
Number of teeth	z	25
Normal module	mn	2 mm
Normal pressure angle	α_n	20 °
Helix angle	β_n	0 °
Helix direction		Spur gear
Number of teeth of mating gear	z2	25
Center distance	a	50 mm

For internal gears the number of teeth has to be entered as negative value, the center distance is always positive as defined in ISO 21771 for gear geometry.

Optionally a gear body can be defined for external gears. The gear body information can be defined using an axial position for the left and right width of the gear body over the diameter. The first points are mapped down to the shaft diameter. The last points are connected to the root diameter of the gear. Internally the gear body is saved as factors to shaft and gear diameters and the gear facewidth. So if the gear data is changed, also the gear body will change. The gear body is only considered if the option Consider gears as stiffness [63] is set to consider elastic gear bodies.

Definition of gear body

	d [mm]	ΔxL [mm]	ΔxR [mm]
1	40	5	10
2	40	-10	10

Reference diameter   dref 51.9615

Shaft diameter   dShaft 15

OK   Cancel

2.1.2.4.4 Bevel gear

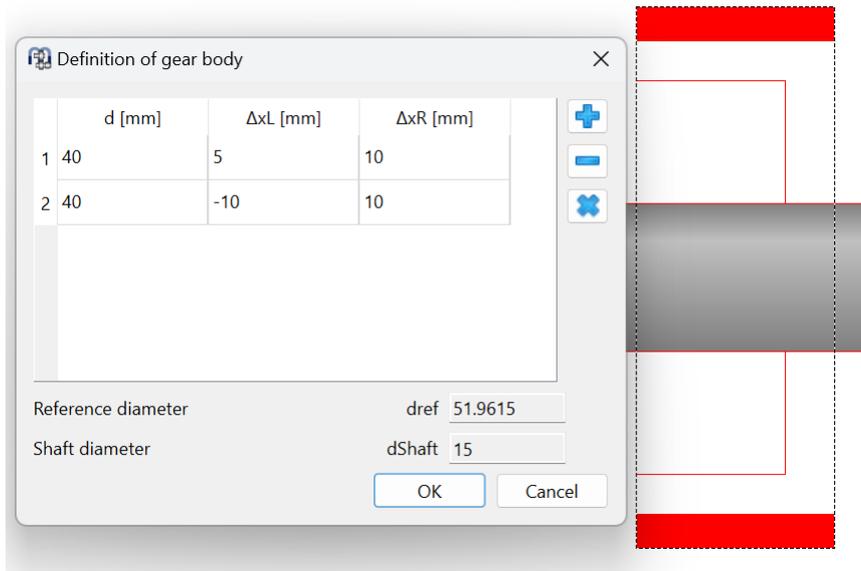The load can be defined using torque and geometry data for a bevel or hypoid gear.

The direction of torque can either be defined by its sign or by the selection "Shaft is driven"/Shaft is driving". The contact point is given by an angle of contact, which is zero on the y-axis and 90° on the z-axis.

Either the pitch angle can be defined directly or it is calculated using axis angle and number of teeth of the mating gear.

Instead of the mean normal module, the outer pitch diameter can be used as an input.

The width is the width of the gear (the length of the flank), the axial length of the loaded shaft is smaller dependent on the pitch angle.

Name	Bevel gear		
Position	x	40	mm
Width	b	20	mm
Torque	T	0	Nm
Direction of torque		Own Input	
Orientation		Apex on the right	
Angle to contact	ζ	0	°
Number of teeth	z	25	
Normal module	mn	2	mm ⦿
Pitch diameter	de	64.1421	mm ○
Profile shift coefficient	x	0	
Generated pressure angle	α_{nD}	20	°
Generated pressure angle	α_{nC}	20	°
Helix angle	β_m	0	°
Helix direction		Spur gear	
Pitch angle	δ	45	° ☐
Number of teeth of mating gear	z2	25	
Axis angle	Σ	90	°
Center distance	a	0	mm

2.1.2.4.5 Worm

The load can be defined using torque and geometry data for a worm.

The direction of torque can either be defined by its sign or by the selection "Shaft is driven"/"Shaft is driving". The contact point is given by an angle of contact, which is zero on the y-axis and 90° on the z-axis. The gear is defined by the usual data on a gear drawing: Number of teeth, axial module, normal pressure angle, lead angle with its direction. The gear friction coefficient, μ_z, can be set for the gear contact.

Name	Worm		
Position	x	1500	mm
Width	b	200	mm
Torque	T	0	Nm
Direction of torque	Own Input		
Angle to contact	ζ	0	°
Number of teeth	z	1	
Axial module	mx	20	mm
Normal pressure angle	α_n	20	°
Lead angle	γ_m	5	°
Helix direction	Helix left hand		
Gear friction coefficient	μ_z	0	

2.1.2.4.6 Worm gear

The load can be defined using torque and geometry data for a worm gear.

The direction of torque can either be defined by its sign or by the selection "Shaft is driven"/"Shaft is driving". The contact point is given by an angle of contact, which is zero on the y-axis and 90° on the z-axis. The gear is defined by the usual data on a gear drawing: Number of teeth, transverse module, normal pressure angle, helix angle with its direction. Optional inputs are the number of teeth for the mating gear and the center distance. The gear friction coefficient, μ_z, can be set for the gear contact.

Name	Worm gear		
Position	x	500	mm
Width	b	200	mm
Torque	T	0	Nm
Direction of torque	Own Input		
Angle to contact	ζ	0	°
Number of teeth	z	40	
Transverse module	mt	20	mm
Profile shift coefficient	x	0	
Normal pressure angle	α_n	20	°
Helix angle	β_m	5	°
Helix direction	Helix left hand		
Gear friction coefficient	μ_z	0	

2.1.2.4.7 Eccentric force

The eccentric force allows entering a force with three components at a point which is not on the shaft axis. The point and the force are defined in polar coordinates. The angle is defined in the same way as the angle of contact in the cylindrical gear (see above).

The radial force is positive if away from the shaft center, the tangential force is positive in the direction of the angle. Moments are calculated by the points and forces given.

2.1.2.4.8 Mass

An additional mass can be defined using the "Mass" element. In addition to its mass, also the mass moment of inertia around three axes can be defined. The weight of the mass is considered and it is considered in the calculation of natural frequencies. The ⭐-button allows to calculate mass and inertia based on a cylinder geometry.

Mass			
Name	Mass		
Position	x	0	mm
Width	b	0	mm
Mass	m	0	kg ⭐
Mass moment of inertia	Jxx	0	kg m²
Mass moment of inertia	Jyy	0	kg m²
Mass moment of inertia	Jzz	0	kg m²
Axial offset	Δx	0	mm
Eccentricity	e	0	mm
Phase	φ	0	°

If a width is entered, the mass is equally distributed on this line. The total mass inertia of this distributed mass are the input values, which results in a minimum value for Jyy and Jzz. A message will be shown if the input values are too small. The axial offset is the axial distance between center of width of the load element and the center of mass.

If the added mass has an eccentricity, it will cause rotating unbalance and can be considered as a harmonic force according to the formula: $F = me_{cc}\omega^2 \cdot e^{i(\omega t + \varphi)}$ (e_{cc}: eccentricity in mm). A phase shift, ϕ (°), for the imbalance force can be taken into account too. The imbalance will only affect the results for the harmonic response. Bearing life, contact stress or safety factors for shaft strength will not change. Setting the option <u>Consider dynamic loads in static equlibrium</u>[60] will lead to a change of bearing life or safety factors, but this is an approximation only.

2.1.2.4.9 Imbalance

If an imbalance should be only considered as a force without adding mass to the shaft, this element has to be used. The imbalance force will be regarded as a harmonic force and calculated in the same way as with an eccentric mass, that is:

$F = me_{cc}\omega^2 \cdot e^{i(\omega t + \varphi)}$, but the inertia and the natural frequencies of the shaft will remain unaffected. The axial offset allows to define an axial offset between the location of the force element and the center of imbalance. It will lead to an additional moment load.

The imbalance will only affect the results for the harmonic response. Bearing life, contact stress or safety factors for shaft strength will not change.

Setting the option will lead to a change of bearing life or safety factors, but this is an approximation only.

2.1.2.4.10 Dynamic force

This element allows the user to consider harmonic loads (moments or forces) on a shaft according to:

$F = \overline{F} \cdot e^{i(\omega t + \varphi)}$. A factor h is multiplied to the shaft rotation frequency. In case that the frequency is not dependent on the shaft speed, then it can be manually specified by checking the box next to the "Frequency" at the software dialog. A phase shift can be defined for the force as well.

The dynamic force will only affect the results for the harmonic response. Bearing life, contact stress or safety factors for shaft strength will not change. Setting the

option <u>Consider dynamic loads in static equlibrium</u> [60] will lead to a change of bearing life or safety factors, but this is an approximation only.

2.1.2.4.11 Pulley

The load can be defined defining a torque on the pulley and its pretension coming from a belt.

The direction of torque can either be defined by its sign or by the selection "Shaft is driven"/Shaft is driving". The positioning of the contact surface is given by an angle of first contact, which starts clockwise from zero on the y-axis, and followed by the wrap angle as shown in the picture below. The pulley diameter can be set straight away or by means of its teeth number and pitch.

A torque will lead to an increase force on one side of the belt and to a decreased force on the other side. This is the correct model in case no tensioning pulley is used.

2.1.2.5 Supports

Boundary conditions can be defined on the page "Supports". They can also be edited by a double click in the graphics or moved by using Shift and the left mouse button. Support elements including rolling bearings only define boundary conditions, they have no mass or weight. For most supports an excitation can be defined which is considered in calculation of harmonic response, see Excitations 90.
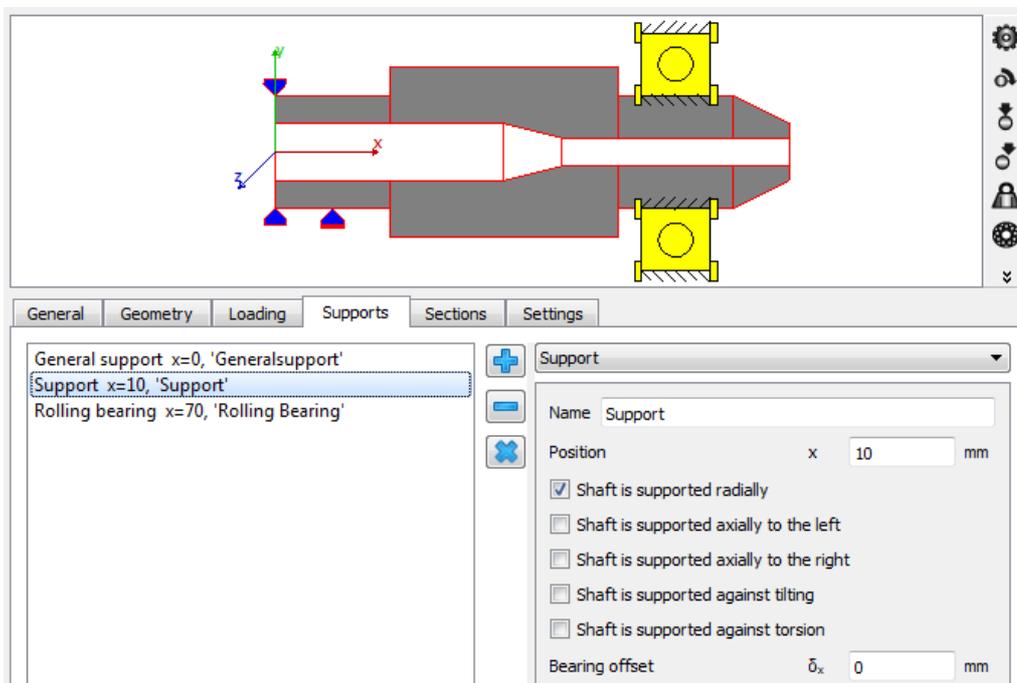
2.1.2.5.1 Support

The "Support" element is an easy-to-use element which allows defining constraints between the shaft and the housing. There are five checkboxes to define in which directions the shaft is supported. For axial direction the movement to the right and to the left can be constrained independently.

A bearing offset can be defined for the three directions. An offset means that the point on the housing is moved in this direction. The resulting force on the shaft is in the direction of the offset.

2.1.2.5.2 General support

The "General support" allows the definition of constraints between a shaft and the housing or between two shafts for all six degrees of freedom independently.

First a selection can be done to which element the shaft is connected to. It can be the housing or a second shaft.

For each degree of freedom, a type of constraint can be defined then:

- No constraint: No constraint in the direction is defined.

- Fixed: The degree of freedom between both elements is coupled in both positive and negative direction. An offset or a clearance can be defined optionally. A positive value for the offset results in a movement of the current shaft in this direction, or a force on the selected shaft in this direction. The clearance is either zero or a positive value, it may not be negative. The clearance is defined as distance between the two end values.

- Fixed to the left (or right): The selection "Fixed to the left" means than the selected shaft is constrained only in the negative direction (left on a horizontal axis). "Fixed to the right" is the constraint in positive direction only. For the unidirectional constraint an offset can be defined.

- Stiffness: The "Stiffness" acts like a spring. An offset and a clearance can be defined. Offset and clearance are defined as explained for 'Fixed' above.

- Stiffness to the left (or right): Like the rigid constraint also stiffness can be defined in just one direction. The stiffness and an offset can be defined.

Using the ▤-button the axial offset in x-direction can be calculated based on a given preload. The offset is varied until the axial force on the support is equal to the preload. The sign of the preload value is important. The calculation only works

if the shaft calculation is fully defined and it is based on an iterative solution of the whole shaft system.

2.1.2.5.3 Rolling bearing

The element "Roller bearing" defines a connection to the rolling bearing calculation.

The current shaft can be connected to the inner or to the outer ring of a roller bearing. The other ring of the bearing can be connected to either the housing or another shaft. The position is the middle of the total bearing width and using the blue arrows the bearing can be moved between shaft shoulders.

By clicking on the button "'Geometry, Material, Temperature, Lubrication' is connected" the user is prompted, by means of a pop-up window, to set those flags of the data to be transferred to bearing calculation. Shaft diameters, material data, lubrication and temperature data can be transferred to the bearing calculation. Displacements, loads and speeds are always connected.

In the default calculation model the rolling element bearing is a nonlinear spring between the nodes on the inner and outer ring shafts. The shaft can be supported radially and axially to the left and the right. This setting changes the connection of the bearing outer ring to the shaft/housing. If the bearing cannot constrain a certain movement, there won't be reaction forces in that direction.

The bearing offsets will move the node of the secondary shaft (or housing) in this direction, it will therefore generate forces to the current shaft in the direction of

the offset. The offset can be used to generate bearing pretension in axial direction or to take misalignment of the housing into account. Using the ▭-button the axial offset can be calculated based on a given preload. The offset is varied until the axial bearing force is equal to the preload. The sign of the preload value is important. The calculation only works if the shaft calculation is fully defined and it is based on an iterative solution of the whole shaft system.

If a damping factor is defined, this will generate a stiffness dependent damping matrix and the factor will override the global Rayleigh damping given in Consider gyroscopic effect 58.

Using the ✚-button next to the bearing type a window of the rolling bearing calculation is shown to select a bearing from the database. This dialog can also be shown on right click on the bearing in the 2D-graphics.

If a rolling bearing element is added, an entry for the bearing in the system tree will be added. If this element is selected, the dialogs of the rolling bearing calculation are active to define the bearing.
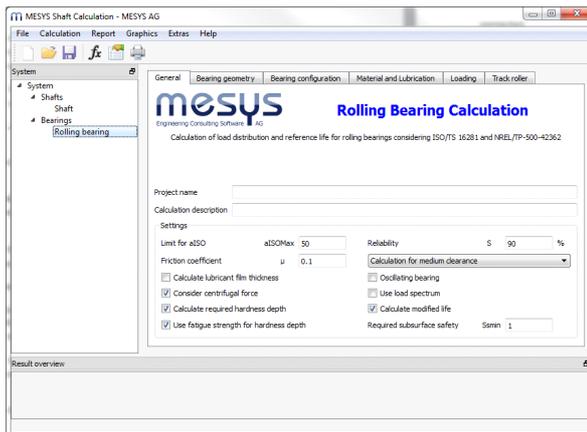
2.1.2.5.3.1 Extended calculation model

The extended calculation model allows to define different elements for axial and radial contacts. Also it allows to consider clearance between bearing ring and shaft or housing. The clearance between bearing ring and shaft or housing is a result of the bearing calculation and it can optionally include elastic expansions of the bearing ring, if activated using Elastic ring expansion [154] in the bearing calculation.

For each ring several options options available:

- "No additional shaft for the inner/outer ring": The ring is connected to the shaft using a single central node like in the default model. Axial and radial forces are transmitted and no clearance (between bearing ring and shaft or housing) is supported.

- "Additional shaft for inner/outer ring, axially free": An additional internal shaft is generated for the ring, which is axially free. In radial direction the clearance defined in the bearing calculation is considered. If there is a radial clearance the ring can also tilt based on clearance and width.

- "Additional shaft for inner/outer ring with axial contact": An additional internal shaft is generated for the ring and axial contact for both sides can be defined to either the same shaft as the radial contact a different shaft or to an adjacent bearing. The connected shaft needs to extend to the bearing ring. An axial clearance can be defined. Optionally an axial stiffness can be defined and a clearance to the block length of the spring. The spring is considered to act on the nominal ring diameter and reaction moment will occur on tilting angles. As before radial clearance as defined in the bearing calculation is considered.
  If the ring is connected to an adjacent bearing ring a distance between the bearing rings is supported. A diameter for a spacer can be defined in this case

which is assumed to have the same material and temperature as the right bearing.

If additional clearance is considered, convergence problems can occur if the system is not statically defined. A bearing without load and without preload can lead to convergence problems if clearance is defined between bearing ring and housing. This case is typically avoided in practice, because it could lead to a bearing damage. Therefore the extended bearing model should only be activated, if additional effects have to be evaluated, but not as standard setting.

If the extended calculation model with an additional shafts is used together with 3D-elastic parts, then only the radial contact may be of type "elastic bearing" in the 3D-part. The axial contacts may only be single nodes defined with type "rigid" or "average".

2.1.2.5.3.2 Selecting bearing from database

The standard interface of the rolling bearing calculation software is shown for selection of a rolling bearing. Either a bearing is selected from the database or custom input is selected. Restrictions to inner and outer diameter can be applied if the bearing is selected from the database; the shaft diameter at the bearing position is set as default value.

For angular contact bearings or taper roller bearings the direction of contact angle can also be defined at the bottom of the dialog.
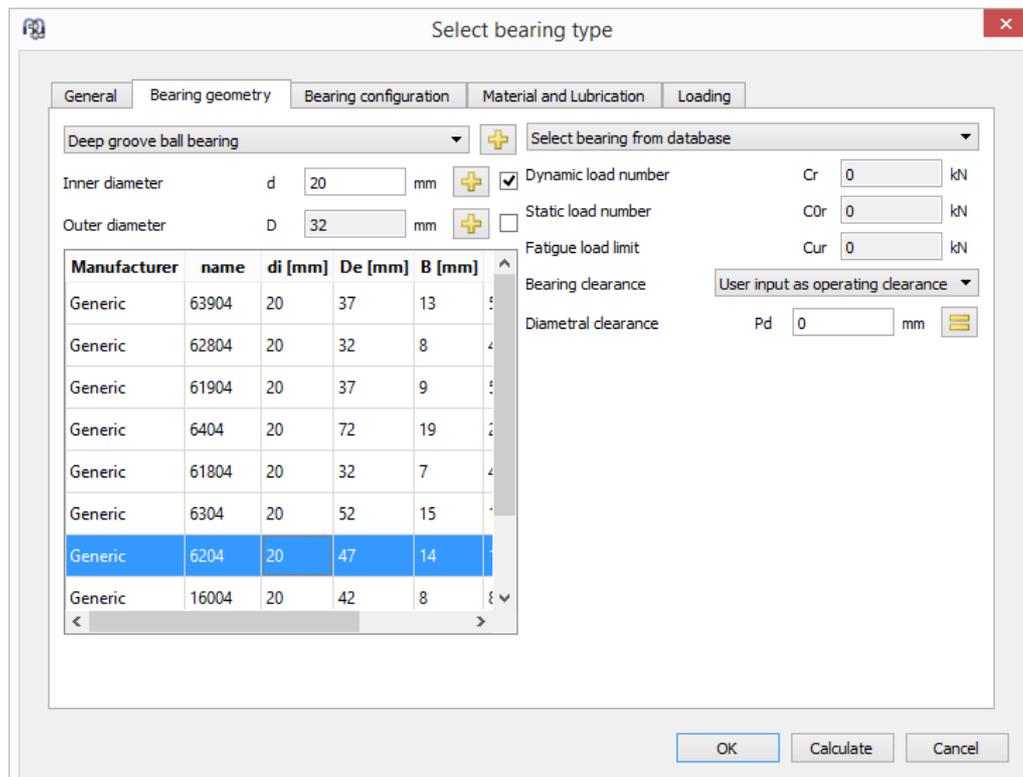
2.1.2.5.4 Radial cylindrical plain journal bearing

The load-displacement behavior of the plain bearing is calculated according to (ISO 7902-2, May 1998). Therefore, the ratio of width to diameter is restricted to $0.25 \leq B/D \leq 1.5$.

The parameters used for the definition of this bearing type are shown in the enclosed figure. Clearance represents the total diametric operating clearance between the journal and the bearing. Any eccentricity is introduced through the corresponding offset inputs in y and x direction.

The optional temperature input does only affect lubricant viscosity. It does not affect clearance.

Surface roughness is the average between the shaft and the bearing and it is used to calculate a minimum required film thickness.

The diameter used for the calculation of the hydrodynamic cylindrical journal bearing is the outer diameter of the shaft where the bearing is placed. The outer ring of the bearing can be connected to either the housing or another shaft.

For modal analysis the unsymmetrical stiffness matrix of the plain bearing is replaced by two orthogonal springs, because of limitations of the FEA library. Therefore, any natural frequencies are an approximation only. No stability analysis can be performed with this software. Damping is not considered either.

2.1.2.5.5 Ball screw

A ball screw can be used as support element and defines a connection to the ball screw calculation. The geometry of the ball screw can be defined at the ball screw element in the system tree or by clicking "Define ball screw".

2.1.2.5.6 Coupling for reaction torque

A coupling for reaction torque fixes the rotation angle of a shaft. The sum of torques will be taken at this element.

An additional checkbox allows to consider the constraint in modal analysis or not to consider it.

2.1.2.5.7 Stiffness matrix

A general symmetric stiffness matrix can be defined between a shaft and the housing or between two shafts.

Stiffness matrix						
Name	Stiffness matrix					
Position				x	100	mm
Connect to housing						
	ux [mm]	uy [mm]	uz [mm]	rx [mrad]	ry [mrad]	rz [mrad]
Fx [N]	0	0	0	0	0	0
Fy [N]		0	0	0	0	0
Fz [N]			0	0	0	0
Mx [Nm]				0	0	0
My [Nm]					0	0
Mz [Nm]						0

2.1.2.5.8 Damping matrix

A general damping matrix can be defined between a shaft and the housing or between two shafts.

Damping matrix						
Name	Damping matrix					
Position				x	0	mm
Connect to housing						
	vx [m/s]	vy [m/s]	vz [m/s]	ωx [rad/s]	ωy [rad/s]	ωz [mrad/s]
Fx [N]	0	0	0	0	0	0
Fy [N]		500	0	0	0	0
Fz [N]			0	0	0	0
Mx [Nm]				0	0	0
My [Nm]					0	0
Mz [Nm]						0

2.1.2.5.9 Weld point

A weld point is fixing all six degrees of freedom. It is easier to use than a [general constraint](#) 81 with all components set to 'fixed' but has the same functionality. The weld point has the additional option to define a width. In case a width > 0 is defined, multiple weld points are considered automatically.

2.1.2.5.10 Planetary support

A planetary support is available for planetary shafts. The planetary shaft is always supported in radial and tangential direction by the carrier. The constraints in axial direction, against tilting or torsion can be activated independently.

2.1.2.5.11 Axial stiffness

The axial stiffness can be used to define a nonlinear spring in axial direction. An axial clearance can be used to define clearance or pretension, the data in the table has to start with (0, 0). If a diameter is defined, the axial stiffness will act as distributed stiffness on this diameter, therefore the support can also support tilting moments if loaded axially.

The axial stiffness will be set to a very large value behind the last point for displacement in the table.

2.1.2.5.12 Cylindrical support

The cylindrical support allows to define a radial clearance over some widths and an axial clearance between two parts. It can be used to model a ring on or in a shaft. The constraint can be rigid or a stiffness can be defined. For the axial constraints a diameter can be defined so it also takes a moment load using a distributed axial stiffness. The cylindrical support should be defined in the inner shaft.
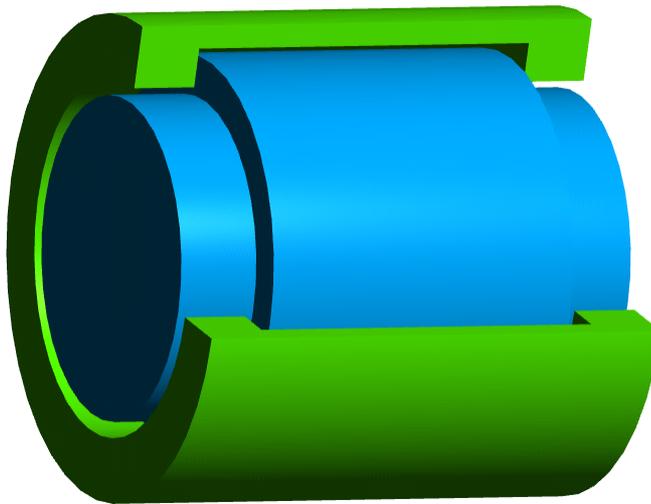
2.1.2.5.13 Contact stiffness

The contact stiffness defines nonlinear springs at several positions around the circumference. It is mainly thought to be used with 3D-elastic parts and elastic faces. The support can be connected to faces of type "elastic bearing". The springs act in a direction normal to the face, where it is defined on. For axial stiffness it is important to defined the correct diameter. The clearance is the clearance for a single spring, so for a radial contact it is based on the radius and not on the diameter.

The stiffness law can be defined in a table with forces for different displacements. The table has to start with values (0, 0). For displacements larger than the last values, the stiffness gets very large.

The spring positions can be defined using an angular position. If the clearance is not defined globally, a clearance can be defined for each spring individually. The ⭐-button allows to fill the table with equidistant springs.
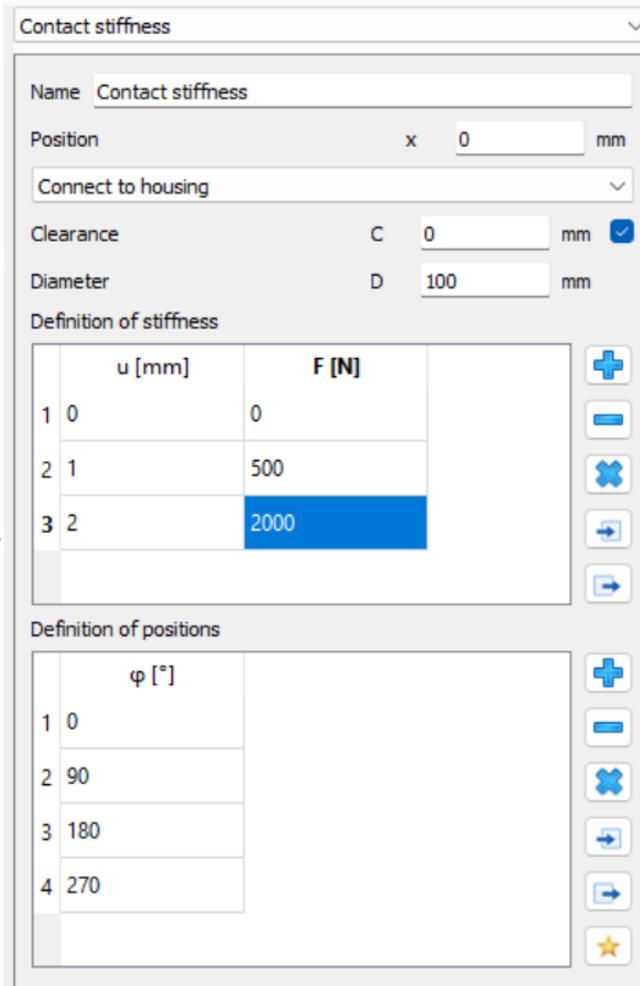
2.1.2.5.14 Excitations

Excitations can be defined for supports which are connected to the rigid environment. The are considered in calculation of harmonic response.

Several types of excitations can be considered:

- A base excitations is a harmonic displacement or rotation of the node in the rigid environment. It has to be defined by giving a frequency. It could be used for vibrations of the grounding.

- An excitation based on shaft rotation will rotate with the shaft. If can be used to model roundness errors. A factor for the harmonic is defined and a factor of one is an excitation with the shaft rotation speed.

- An excitation based on inner ring rotation can be defined for rolling element bearings in case of rigid bearing rings. Here also a harmonic is defined and the excitation rotates with the inner ring. For

- An excitation based on outer ring rotation can be defined for rolling element bearings in case of rigid bearing rings. Here also a harmonic is defined and the excitation rotates with the outer ring.

- An excitation based on cage rotation can be defined for rolling element bearings in case of rigid bearing rings. Here also a harmonic is defined and the excitation rotates with the cage speed. The cage speed is calculated using the nominal contact angle.

2.1.2.6 Sections

Using "Sections" several cross sections can be defined for the strength calculation. One type of section is "Documentation point" which can be used without the module for strength calculation. In this case just displacements and forces are documented in the report, no strength calculation according DIN 743 is performed.

Dependent on the selected type of cross section several inputs are necessary to define details of the notch effect. All types from (DIN 743-2, Dezember 2012) are supported.
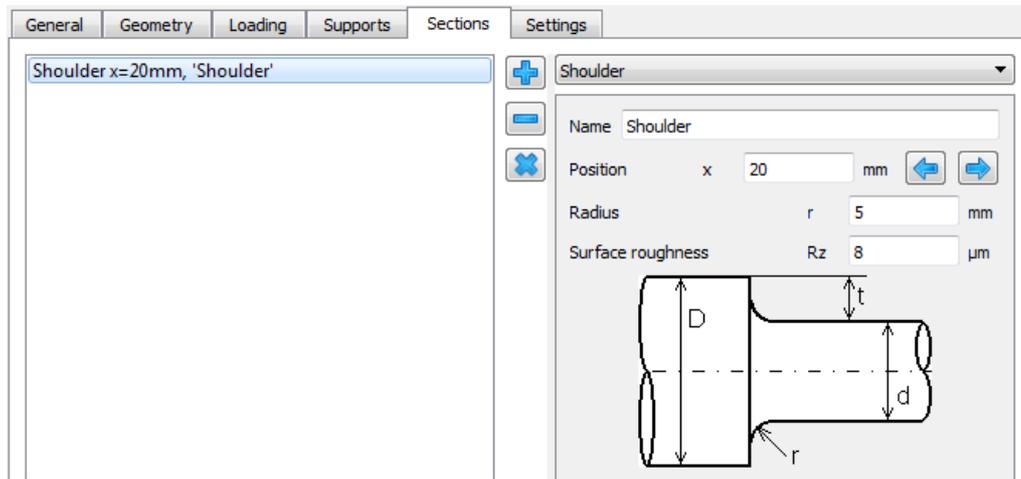
The "Documentation point" has the setting "Calculate stiffness". If the option is activated the stiffness in 6 degrees of freedom for this point of the shaft is shown in the results overview and the report. The stiffness is the reciprocal of the main diagonal of the compliance matrix of the system stiffness reduced to this point. So it is for example cxx=dFx/dux for constant other forces and moment Fy, Fz, Mx, My, Mz.

2.1.2.7   Settings

Some settings can be entered for a single shaft:
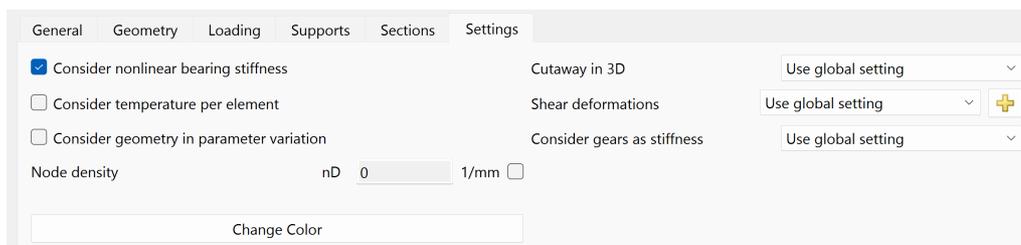
2.1.2.7.1 Consider nonlinear bearing stiffness

As default, the nonlinear bearing stiffness is considered for rolling bearings. For large systems this option can be cleared to decrease calculation time. Without this option, bearings are considered as rigid. The option is available for each shaft, so it can be set where bearing stiffness is important and can be cleared on other shafts.

If bearings with contact angle are used, the nonlinear stiffness matrix should be considered to take into account the coupling of axial and radial loads.

2.1.2.7.2 Consider temperature per element

By setting this flag, the user can add a temperature difference 'ΔT'(±) for each element created at the outer geometry on the temperature previously set at the tab-page 'General'. The temperature difference can also be varied within a load spectrum.

2.1.2.7.3 Consider geometry in parameter variation

If this option is set, the geometry of this shaft can be varied in the parameter variation. It is not active as default because in can lead to many input parameters in the parameter variation in cases of gearboxes with many shafts.

2.1.2.7.4 Node density

The user can overwrite a node density for the FEA model. For a static linear calculation, the software adds additional nodes for improved resolution of diagrams. Results are only affected for conical geometry not for cylindrical elements. For modal analysis the number of nodes can make a difference.

Reducing the node density decreases calculation time.

2.1.2.7.5 Cutaway in 3D

For each shaft the cutaway setting for the 3D view can be set independently of the general option selected under the tab-page 'Display settings'.

2.1.2.7.6 Shear deformations

Here it is possible to define which shear coefficient value (kappa) is being taken into account for the selected shaft. By default, the one defined at the global settings is used. The other approaches for the calculation of kappa can be selected from the drop-down list. If 'user input for kappa' is selected, then the user has to press the ✚-button in order to enter the desired value.

2.1.2.7.7 Consider gears as stiffness

For each shaft the setting for consideration of gears as stiffness can be set individually independent from the global setting at Consider gears as stiffness 63.

2.1.2.7.8 Change color

The color of the shaft can be changed. The transparency is only considered on 3D view, not in 2D view.

2.1.2.8 Options in system tree

Delete
Duplicate
Mirror
Split shaft at
Append shaft to ▸
Export Deflection
Change Color
Move to group ▸
Import geometry
Export geometry
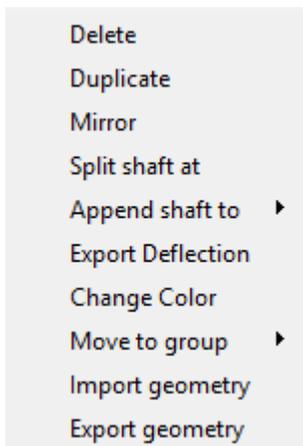
The context menu with the right mouse button on the shaft in the system tree provides some additional options:

- "Delete" deletes the shaft.

- "Duplicate" duplicates the shaft. This is helpful if several similar shafts are needed in a gearbox.

- "Mirror" mirrors the shaft geometry and positions of forces and supports. The orientation of forces and supports is not changed.

- "Split shaft at" asks for a position on the shaft and splits the shaft into two shafts.

- "Append shaft to" appends the shaft to another shaft in the same group.

- "Export deflection" exports a *.csv file for the deflection of a shaft. The "Result tables" in menu report also contain this information, but for all shafts instead of one only.

- "Change color" allows to change the color of a shaft.

- "Move to group" allows to move a shaft into another group. This is often used after "Duplicate".

- "Import geometry" allows an input of shaft geometry as 2D DXF or 3D STEP, see below for details.

- "Export geometry" exports the shaft as 3D STEP or IGES. Gears and bearings are included on the export.

2.1.2.8.1 Import of shaft geometry from CAD files

The shaft geometry can be imported from 2D DXF or 3D STEP files. In case of a 3D-STEP file an intersection of the part in the x/y-plane, the x/z-plane and the y/z-

plane is done and the resulting edges are considered as in the DXF input. Relief grooves in the CAD model can lead to many imported elements.

For the DXF file a normal drawing can be selected. By using the left mouse button, some geometry can be selected. The selected geometry can either be deleted or it can be cut. This allows to select the correct geometry for the import. The ➡-button allows to switch through different layers within the DXF file or through different parts or sections in STEP import.

If an inner geometry should be imported too, the flag at the bottom of the window has to be activated.

The DXF file only contains lines. Sometimes the software cannot detect the correct connections between these lines. In this case the wrongly used lines can be selected and deleted. The lower part of the window shows the currently detected geometry input. Selecting some segments with undercut and pressing the key "c" allows to remove the undercut. Selecting two lines with a gap and pressing the key "c" allows to close the gap. Selecting two lines with the same axial start end end point and pressing the key "c" will add vertical lines which are used to define the position of the axis.
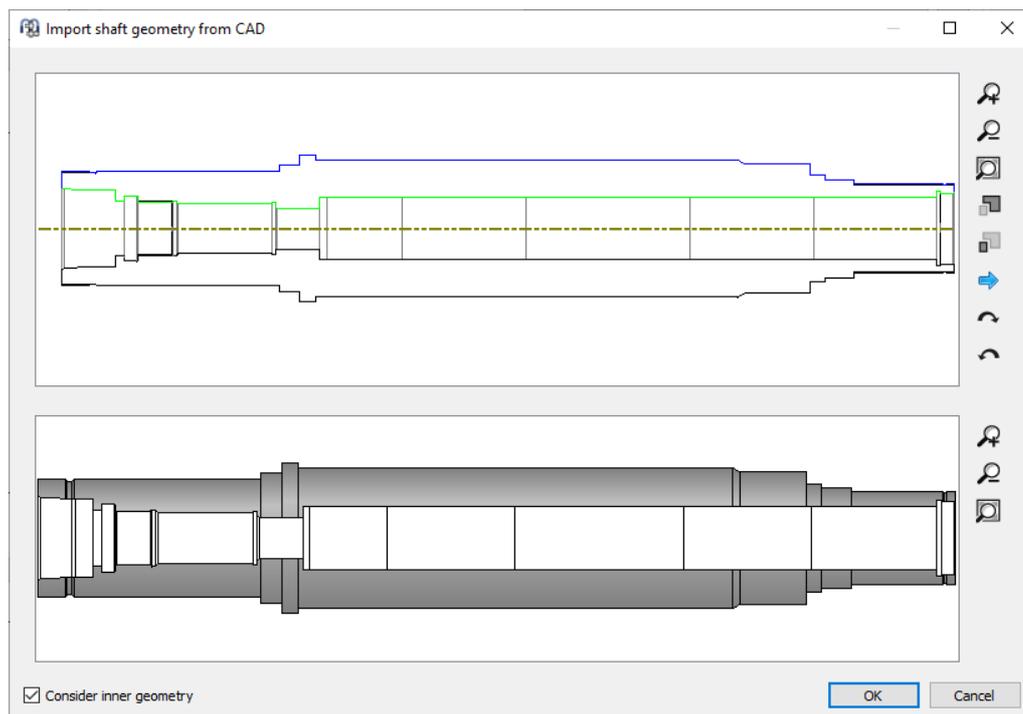
2.1.3 Shaft Groups

For systems with shafts that are not coaxial, shaft groups have to be added. For each group a position can be defined.
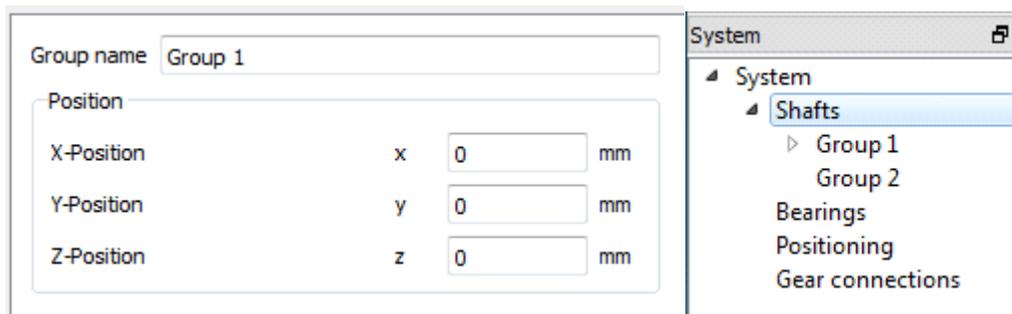
The positions can also be calculated by the software if constraints are used for positioning. See below.

There are three types of groups:

1. Standard group: The standard group can have an arbitrary position and it can hold subgroups. Shafts in different groups can not be connected by supports.

2. Coaxial group: The coaxial group can only be positioned axially relative to its parent group. Shafts in different coaxial groups can be connected by rolling bearings or other supports.

3. Planetary group: A planetary group is needed to define planetary gear sets. The number of planets can be defined in addition to position. Shafts in a planetary group rotate around a planet carrier. The centrifugal loads caused by this rotation are considered. For bearings and other supports in planetary groups the result for each planet is shown in the results.

2.1.4   Positioning

If multiple groups are used, an additional entry for positioning is shown in the system tree.

The page "Positioning" can be used to define constraints for group positions. New constraints are added using the ➕-button, they can be sorted using the arrows. The constraints are evaluated once per calculation, so the order can be important.
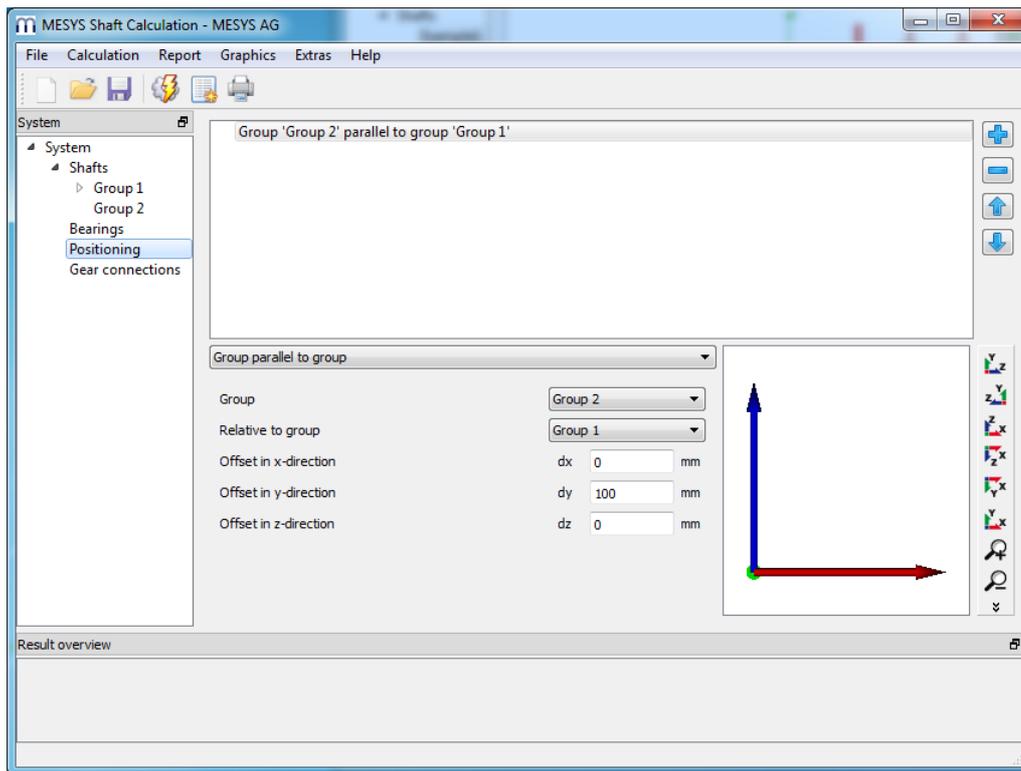
Different constraints are available:

- Group parallel to group: Specify an offset between two groups.

- Group according to gear pair: The selected group is positioned according to the selected gear pair. An additional angle has to be specified, zero is for a center distance on the y-axis. An axial offset can be entered if the centers of the gears should not align axially.

- Group according to two gear pairs: One group is positioned according to the center distances of two gear pairs. The axial position is defined by the first gear pair.

- Group according bevel gear pair: One group is positioned according to geometry data of a bevel gear. The angle of contact for the reference group can be defined.

- Shaft according to gear pair: This constraint is moving a shaft axially, so that the centers of the gears are aligned. An axial offset can be entered optionally.

- Gear according to gear pair: This constraint is moving a gear axially so that it is aligned with its mating gear. Here an axial offset can be entered optionally, too.

- Group according worm gear pair: The selected group is positioned according to the selected worm gear pair. An additional angle has to be

specified, zero is for a center distance on the y-axis. An offset can be entered axially to the worm.

2.1.5 Gear connections

If multiple groups are defined, a page for gear connections is shown. Gear connections can be added using the ➕-button. The shafts and gears in contact can be defined and the basic data for the gear pair is shown. The data for the gears can be modified in this window in addition to the inputs at the single shaft but using the connection allows to change data for both gears at the same time.
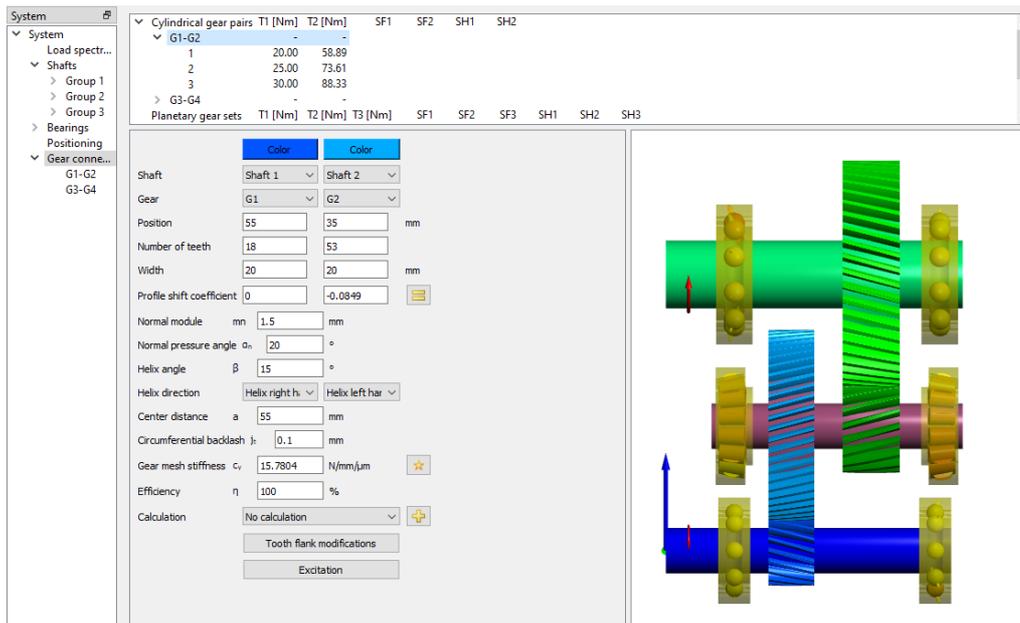
The torque on each gear pair is shown in the list at the top; the safety factors are shown if a connection to a gear calculation program is set up.

With a click on the items for the gear pairs in the system tree on the left the gear calculation program can be started to define details for the gear pair. The shaft calculation is passing basic gear geometry (z, b, x, m_n, α_n, β, x, a) and loading to the gear calculation; it is reading back the same geometry data and the safety factors. The colors of the gears can be changed by clicking on the "Color" button shown in the picture.

If a gear is considered in a connection, its torque input is hidden at the corresponding input window under the page 'Loading'.

For cylindrical gear pairs a sizing button allows to select the mesh stiffness according to ISO 6336-1. Either $c\gamma\alpha$ or $c\gamma\beta$ can be selected. CB and CR are assumed to be equal one for the calculation of mesh stiffness.

2.1.5.1 Tooth flank modification

With the goal of ensuring optimum tooth flank contact, the teeth of gear pairs can undergo flank corrections based on three different approaches according to the standard (ISO 21771:2007, September 2007), i.e. flank line crowning, flank line slope modification and flank line end relief. By pressing on 'Tooth flank modifications' at the page 'Gear connections', a pop-up window allows the user to enter the corresponding geometrical data in relation with these approaches. For each gear it is possible to have either symmetric or unsymmetrical modifications by setting or clearing the available flags respectively at the bottom. By activation of an unsymmetrical flank modification, the corresponding data column is added in the window.
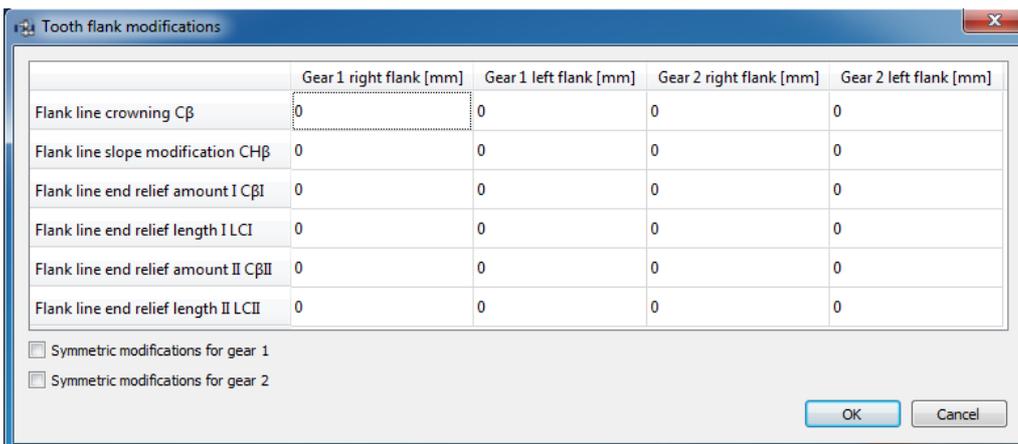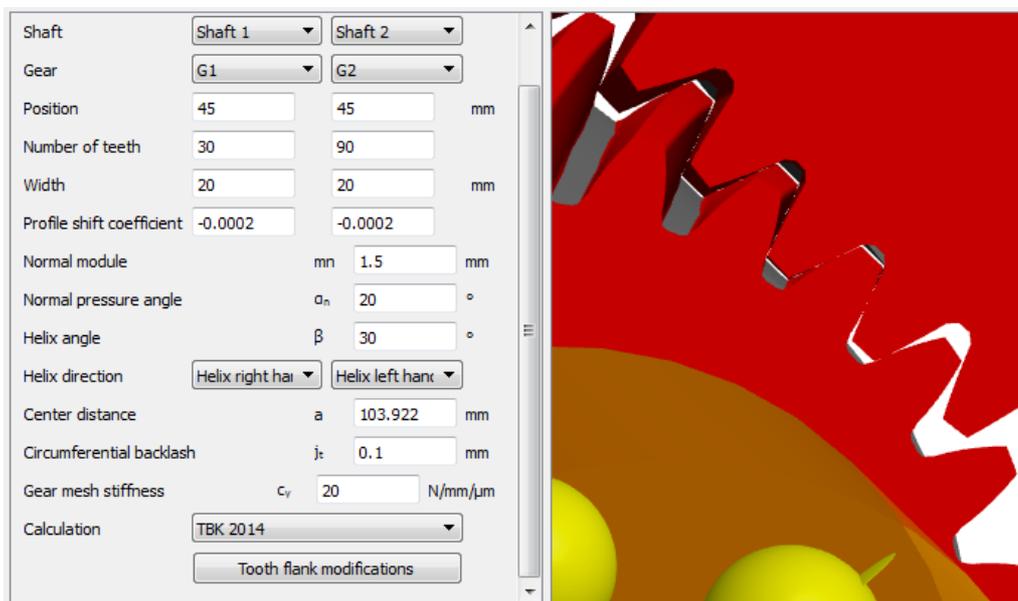
2.1.5.1.1 Flank line crowning

It consists of a relief based on a circular arc which is tangent to the mid-point of the tooth width. The relief amount, which is measured from the end of the tooth, must be specified.
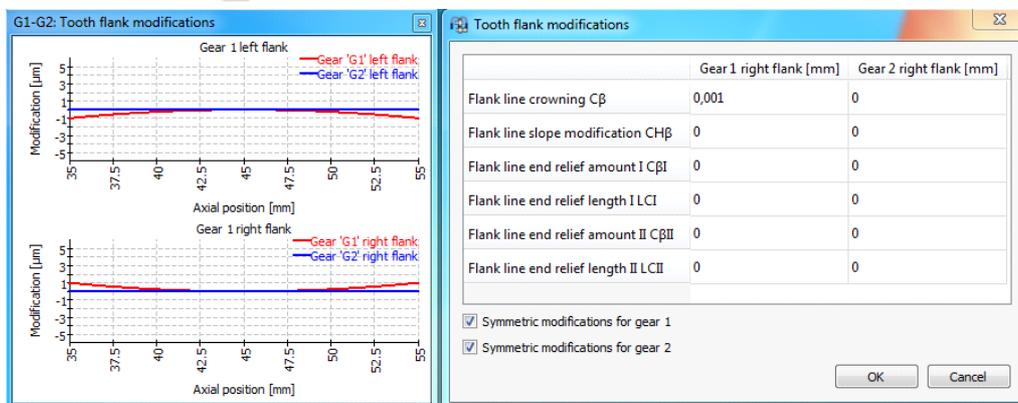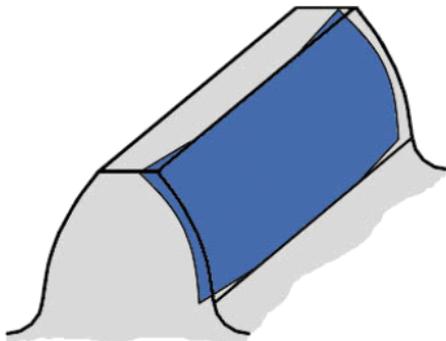
2.1.5.1.2 Flank line slope modification

It consists of an increasing linear relief over the whole tooth width. So the slope of the relief line starting from one end to the other must be specified.
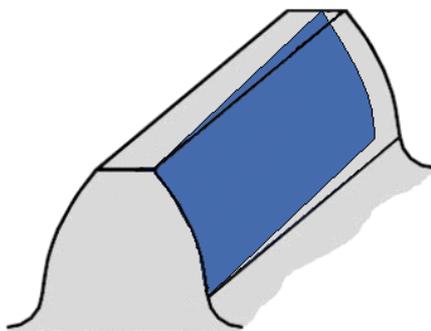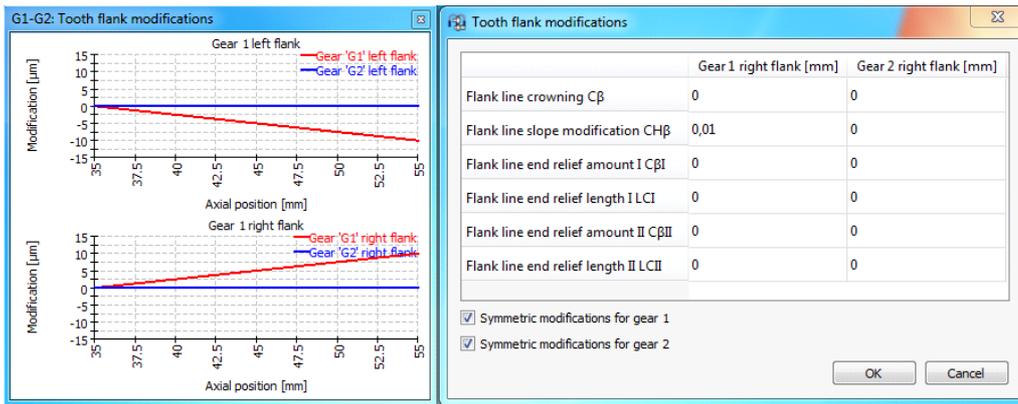
2.1.5.1.3 Flank line end relief

It consists of an increasing linear relief starting from a certain point until the end of the flank line. The relief amount and length, which are measured from both ends of the tooth, must be specified.
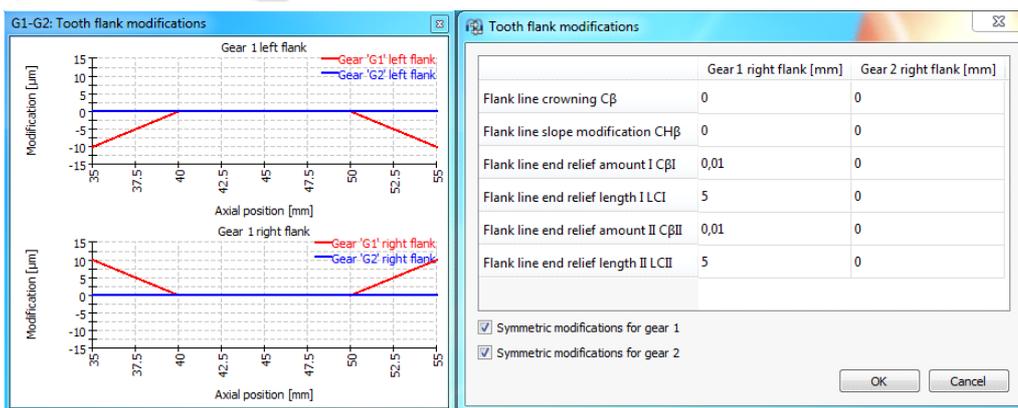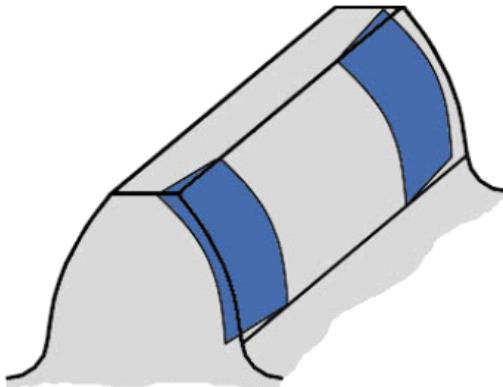
All in all, the user can make use of the line load and gap width graphics as guidance to stablish a flank correction proposal. If, for instance, we start from a gear tooth mesh condition like this:
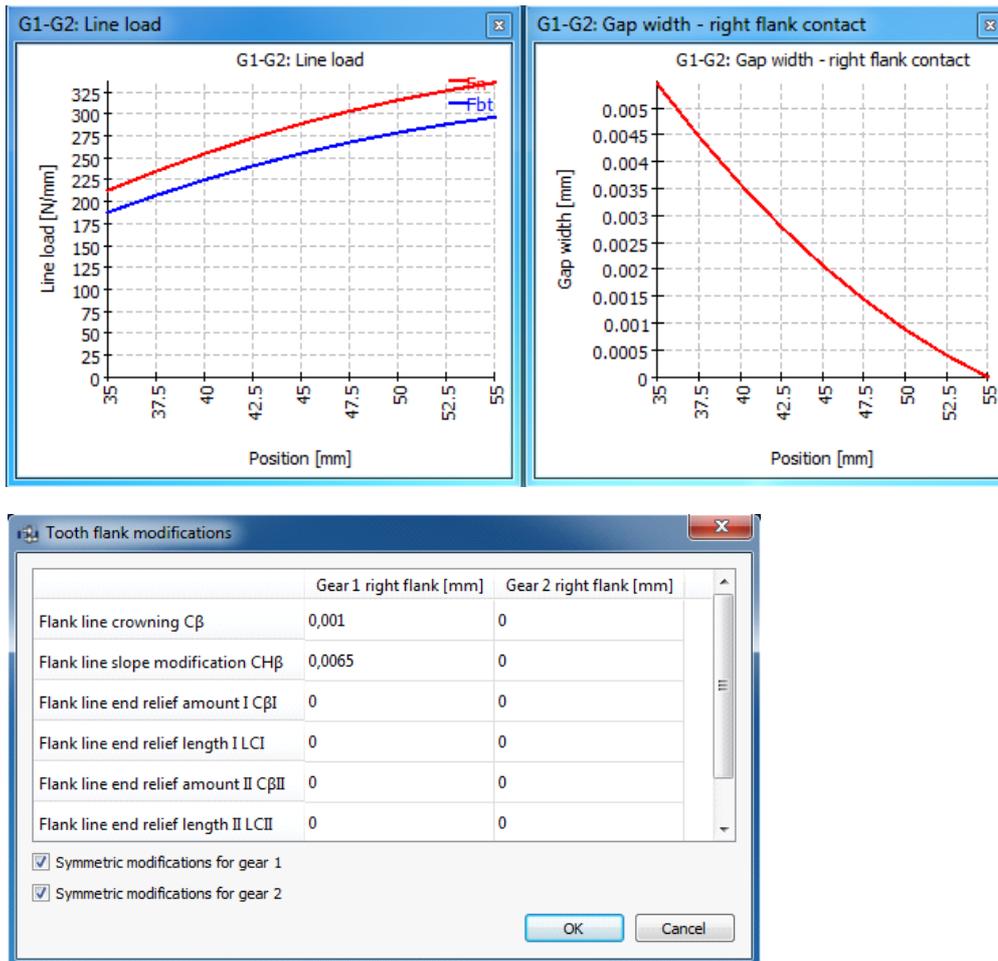
And we apply a combination of flank line crowning and flank line slope modification, it significantly helps to have an homegenous line load as shown in the following pictures:
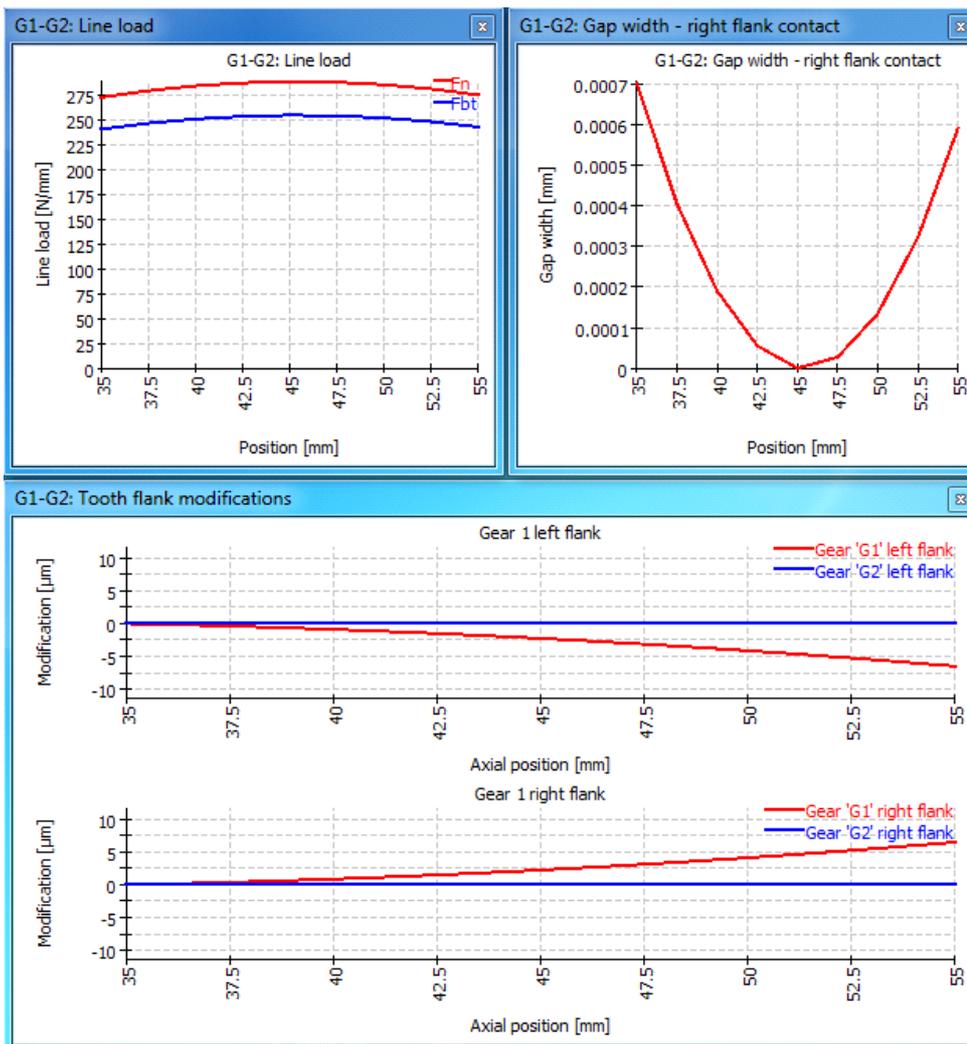
2.1.5.2 Excitations

For cylindrical gear pairs three types of excitations can be defined:

- A mesh excitation is defined on the path of contact. Pitch errors will lead to excitations based on gear mesh frequencies. Higher orders can be defined using the input of harmonic.

- An excitation based on roundness errors of the gear 1 can be defined. Here a variation of the radius is used. The base frequency is the rotation speed of the gear.

- An excitation based on roundness errors of the gear 2 can be defined like for gear 1.

The excitations are considered in the calculation of harmonic response. They will not affect the results of quasi-static calculation and will not affect safety factors for gears.
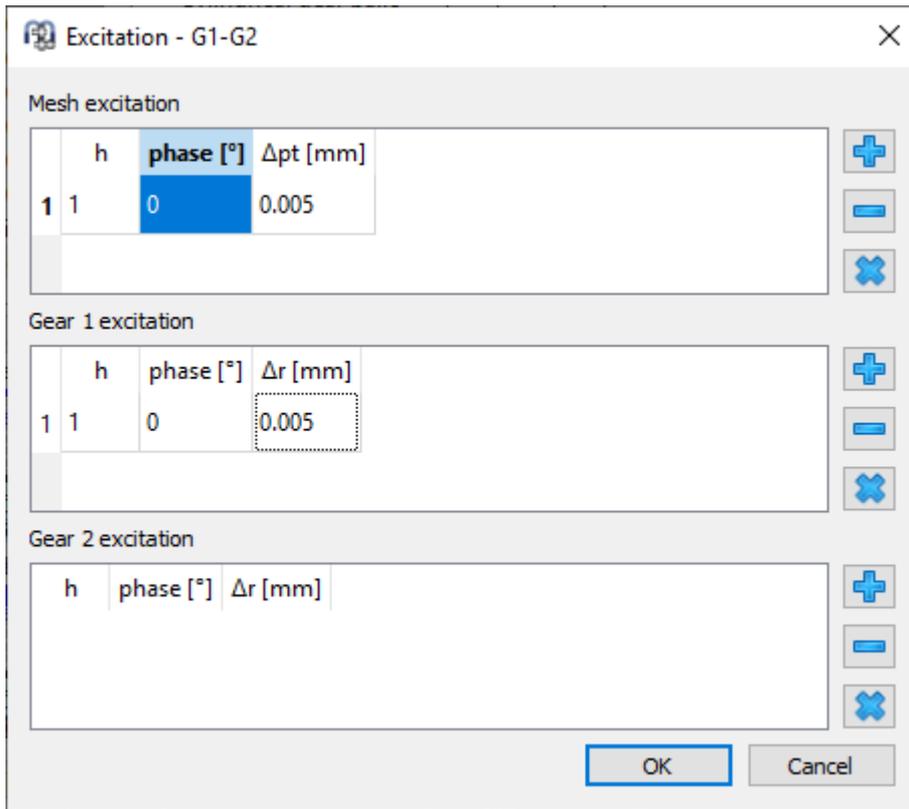
2.1.6 Calculation with load spectrum

If the flag for the calculation with load spectrum is set on the 'System' page, an additional item titled load spectrum is shown in the system tree.
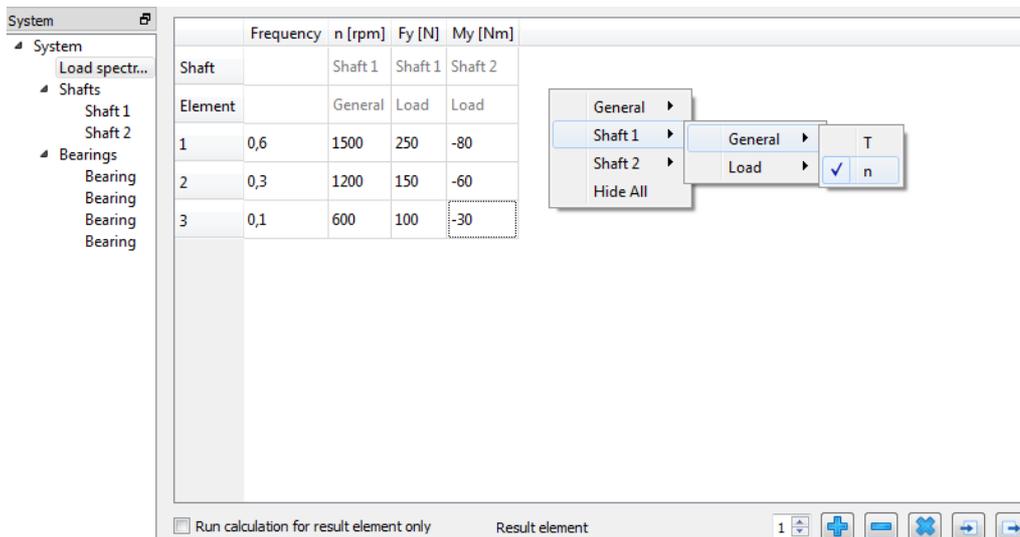
By default, the table is empty. Only the column header for the load frequency is shown. By right-clicking somewhere on the window, different calculation parameters for the load cases can be added optionally as column headers. These parameters are classified as "General" and "Load". Concerning the rows in the table, they can be added using the ➕-button and also deleted by selecting it and using the ➖-button. These rows are called "Element" and are taken into account as

a load case where the values (inputs) corresponding to the calculation parameters can be entered. The ✖-button clears all the inputs of the table, also the elements (rows).

You can hide any column by right-clicking on the header and selecting "Hide". For all load components that are not shown in the table the default value is used.

By using the ➡-button, the load spectrum is read from a file. The columns in the file have to be in the same order and unit as shown in the table. Likewise, a created load spectrum table can be exported with the corresponding ➡-button.

The default graphics for deflection are shown for the selected "Result element". Natural frequencies are also calculated for this "Result element".

2.1.6.1 Calculation for a single load spectrum element only

It is possible to run the calculation for one element (load case) only if the user sets the flag 'Run calculation for result element only'. The desired load case is chosen by either using the arrows of the small box next to the ➕-button or typing the element number into it directly. The graphics will be only shown for the selected element.

This allows using the load spectrum calculation as input interface for independent load cases, which are calculated independently.

2.1.7 Configurations

Configurations can be activated at settings on the first input page if multiple groups are defined.
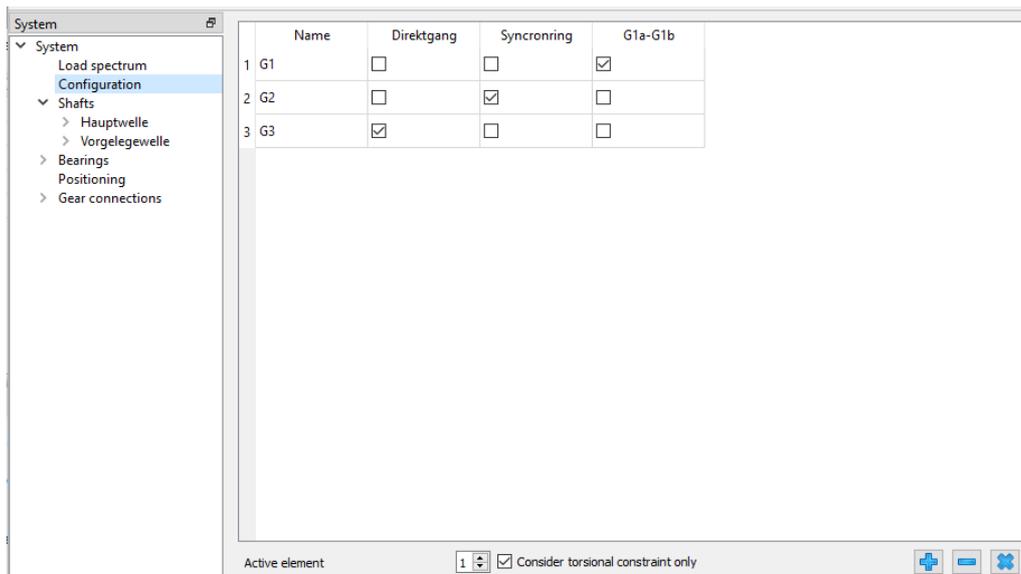
Using the right mouse button additional columns can be added to the view. Available are gear connections, reaction couplings, general constraints, supports

and cylindrical constraints. The constraints are only available if the torsion is fixed in the constraint.

Rows can be added using the ➕-button and with the check box the active elements for each configuration can be selected.

If the option "Consider torsional constraint only" is active, clearing the flags for the constraints only disables the torsional constraint of general constraints, constraints in all other directions are not affected. If the option is not active the constraints is deactivated for all directions.

The active configuration can be selected at the bottom of the window or within a load spectrum.

2.1.8 Housing stiffness matrix

A symmetric stiffness matrix can be defined between the supports (only those which are connected to the housing) and the housing.
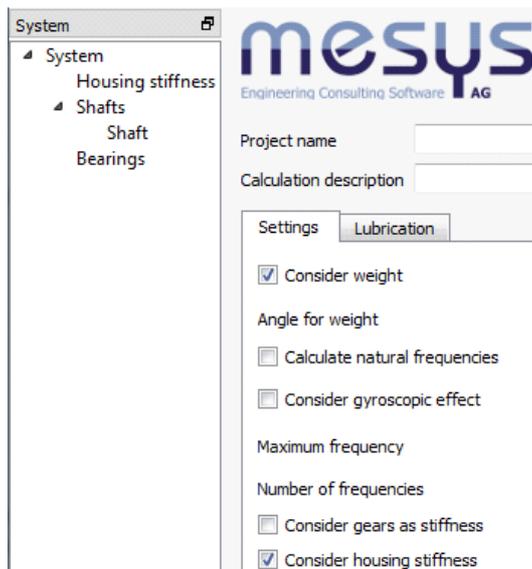
In order to make use of this option, the corresponding flag of 'Consider housing stiffness' must be activated at the tab-page 'Settings' which it is shown when selecting 'System' from the software tree.

The ➕-button, located under the matrix, can be used to create an additional stiffness matrix, which is referenced with a number shown in a box titled 'Selected stiffness matrix'. Any one or all of these created matrices can be excluded for the calculation at any time by using the flag titled 'Consider stiffness'. In this way, any matrix data can be kept, even though it is not being used for the analysis. In case we do want to erase any of the matrices, the ➖-button can be employed for it. The entry field can be used to name them and their names will be shown in the report.
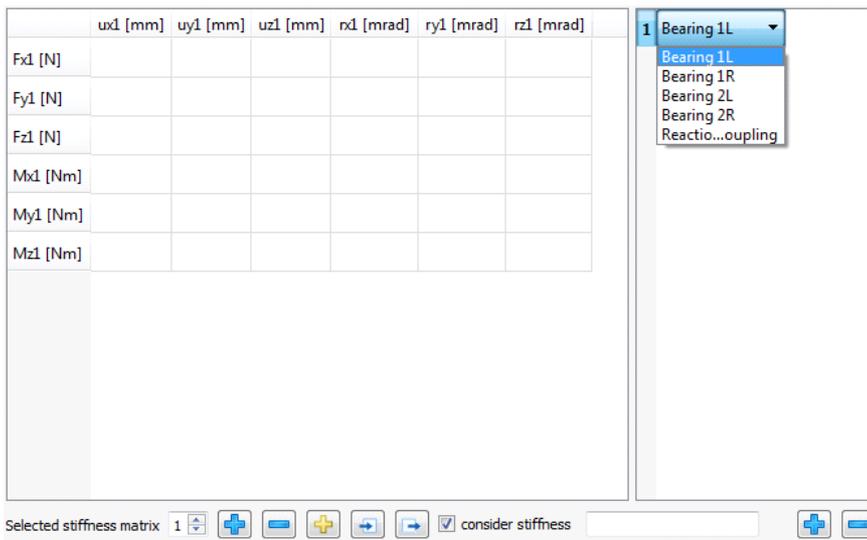
By clicking once the ✚-button on the right lower corner, a new numbered element is listed on the window above it and a matrix is shown on a bigger window. With a double click on this element a drop-down list is shown letting us choose the support to consider. By repeating this procedure, we could add other existing bearings or supports on the list, and therefore their corresponding lines and columns of the stiffness matrix would be shown and taken into account.

All the values for the stiffness can be entered manually with double click on the cells or also be loaded as a matrix, which is sometimes obtainable from a finite element analysis, by using the import button ➡. Equally, a created matrix can be exported with the corresponding export button ➡.

If we use the ✚-button, a pop-up window allows activating four additional options. As shown in the following picture, we can for instance omit the consideration of rotation stiffness, and activate the possibility to define for the connecting point, in any available direction required, a force or a moment with 'Enter force/moment', and also a certain displacement or rotation with 'Enter displacement'.

	ux1 [mm]	uy1 [mm]	uz1 [mm]	F/Mext
Fx1 [N]	2e+11	0	0	0
Fy1 [N]		2e+11	0	0
Fz1 [N]			2e+11	0
du/dr	0	0	0	
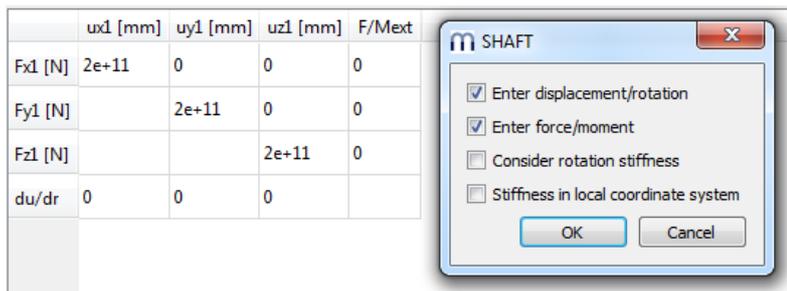
The usage of the last option lets us to consider the stiffness in the local coordinate system of the connecting point instead of the global coordinate system.

2.1.9   3D elastic parts

The context menu under "Shafts" or under groups allows to insert "3D elastic parts as shaft" or "3D elastic parts as housing". A new part will then be shown in the shaft/group tree.

The 3D elastic parts combine the FEA beam model used for the usual shafts with 3D solid meshes. The stiffness of the 3D solids is reduced to a stiffness matrix, which is then considered in the solver for the beam model. On the first use of the 3D elastic part the reduced stiffness matrix is generated, which can take a while dependent on the size of the mesh. Afterwards the reduced stiffness is saved with the file and only needs to be regenerated if material, mesh or selections for faces are changed.

The 3D elastic parts are considered as stiffness and optionally by modal reduction. No mass is taken into account without modal reduction. Thermal expansion and weight are considered. A "3D elastic parts as shaft" must have the x-axis as rotation axis and a rotation speed can be defined. A "3D elastic parts as housing" can not rotate and should be fixed in space.

If the option "calculate displacements" for the 3D elastic parts is selected, the displacement field of the surface is calculated for each connected degree of freedom on initial calculation of reduced stiffness. This increases the initial calculation time and the file size, but allows to visualize the deformed shape afterwards. The same is valid for the option "calculate mode shapes"; this is used for visualization only, too.

In case modal analysis is supported, the displacement field and the mode shapes need to be calculated. Therefore options "calculate displacements" and "calculate mode shapes" won't affect the required time for reduction. They only affect the required file size.

If the option "use second order mesh" is activated quadratic shape functions are used instead of linear shape functions. An eight node hexahedral will get a 27-node hexahedral instead. As the number of nodes increase, reduction time and

memory usage will increase too. Generally the second order mesh will be more accurate. It is better to use a larger mesh size and a second order mesh instead of a fine linear mesh.

On geometry import or generation an option "Generate second order mesh" in available at several locations. If this option is set, then a second order surface mesh will be generated. It will require approximately four times the memory compared to a linear mesh, but will lead to a better geometry approximation if "use second order mesh" is activated. The option at the import only affects the surface mesh while the option "use second order mesh" under "General" defined what type of 3D-mesh is used.

Selected faces are reduced to one node (or elastic bearing, gear or contact) and can be connected to a support element. Following options are available:

- An existing support can be selected. This support is then connected to the 3D-elastic part instead of the rigid environment

- A new support can be defined as possible for shafts. Only a subset of the available types in the shaft calculation is supported so far. It can be used to support an imported housing by a stiffness.

- A force can be added on the reduced node

- A shaft can be selected. This will lead to a connection to the shaft in all six degrees of freedom. It is the same as adding a weld-point to the shaft and connection this to the 3D-elastic part. The shaft can be selected for the reduction types 'rigid', 'average' or 'elastic contact'. In case of an 'elastic contact' the face will be kept cylindrical in each section but can deform over the axial position.

- "Set as fixed" is available for "3D-elastic parts as housing". This node will be connected to the rigid environment.

For the reduction of a face five options are available as connection type:

- average: The reaction force will be distributed on all nodes of the face and an average value of the displacement of the face will be used. This option will not stiffen the part.

- rigid: A rigid connection of the reduced node to all nodes of the face will be added. This will stiffen the part.

- elastic bearing: This option may only be connected to a roller bearing and will consider the elastic deformations of the part in the load distribution of the bearing. A license for the bearing calculation software including ring deformations is required. Currently an elastic outer ring is only permitted for axially fixed bearings.

- elastic gear: This option may only be connected to a cylindrical gear so far and will consider elastic deformations of a gear body. The shaft system option is required for this.

- elastic contact: This option may only be used with either a weld point or a cylindrical constraint for connection of faces in two parts both reduces as elastic contact. The dimensions of both contact faces have to be identical. In addition the connection to a shaft is possible.

2.1.9.1   3D elastic part as shaft

A 3D elastic part as shaft is a rotating part with one axis which can be connected to other shafts. Currently, axisymmetric parts defined by a polygon or a STEP import can be considered.
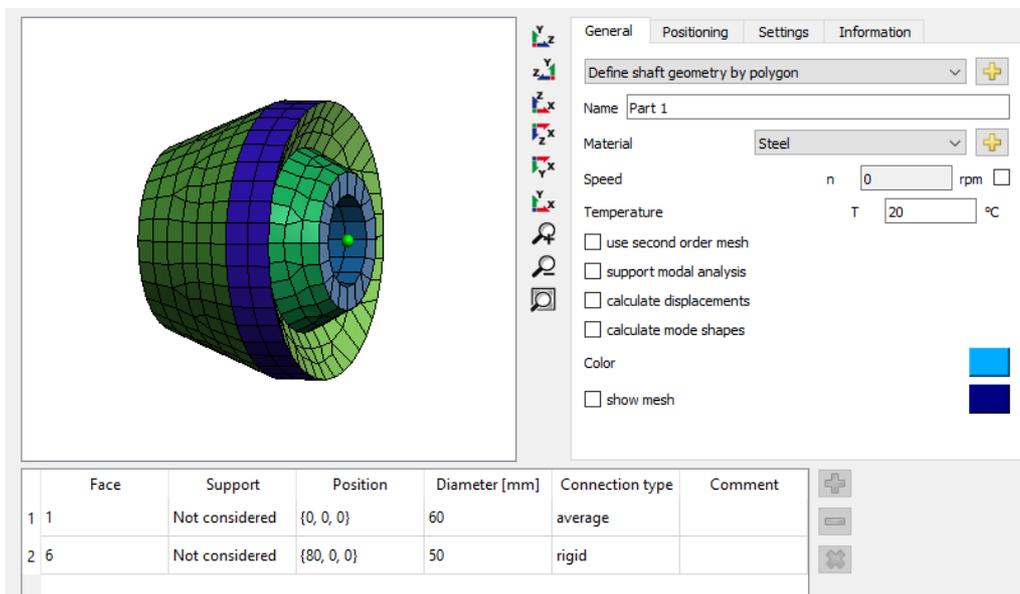
Using the ✚-button behind "Define shaft geometry by polygon", a dialog opens to define the geometry using a polygon. It should be defined clockwise. "x" is the axial coordinate, "y" the radius. Optional connections to normal shafts can be defined as "Considered as face" or "Considered as edge". Either all points on a face or only points on an edge will be connected to a shaft support element.
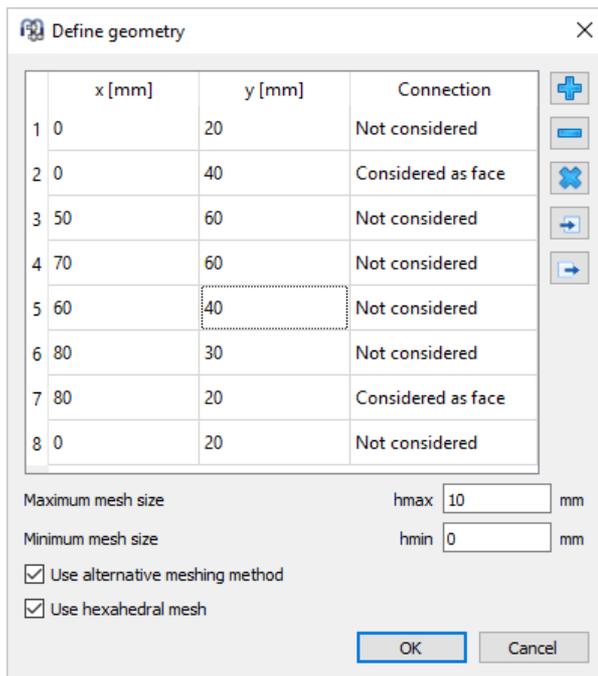
In addition, the mesh size can be defined in this dialog.

Additional settings are the material, the rotation speed, the option to select a second order (quadratic) mesh and the option if the displacements should be calculated. Using a quadratic mesh is recommended for better accuracy. The calculation of displacements increases the calculation time and would only showthe deformed shape in the 3D view.

Under "Positioning" the axial position can be defined.

In the table of the main view the support elements of the shafts have to be selected under "Support". The connection type "rigid" enforces a rigid connection of the selected face/edge to the beam node of the shaft. The connection type "average" enforces an averaged connection to the node, which leads to less stiffening of the 3D elastic part.

In case of a STEP import faces can be selected by double click in the 3D-view. Faces can be merged by selecting two faces, marking them in the table and selecting "Merge faces" in the context menu (right mouse button). Only neighbouring faces can be merged. For an imported shaft the x-axis must be the rotation axis. Under Positioning the orientation can be changed selecting "Permanent coordinate transformation". If faces are inside the part a clipping of front faces can be activated using the context menu in the 3D-view. Using Alt-Button and left mouse button the clip-plane can be moved.

2.1.9.2   3D elastic planet carriers

In case of a 3D elastic shaft, also planet carriers can be defined. Either imported as STEP file or as parametric planet carrier. For a parametric planet carriers, the following basic shapes are available:
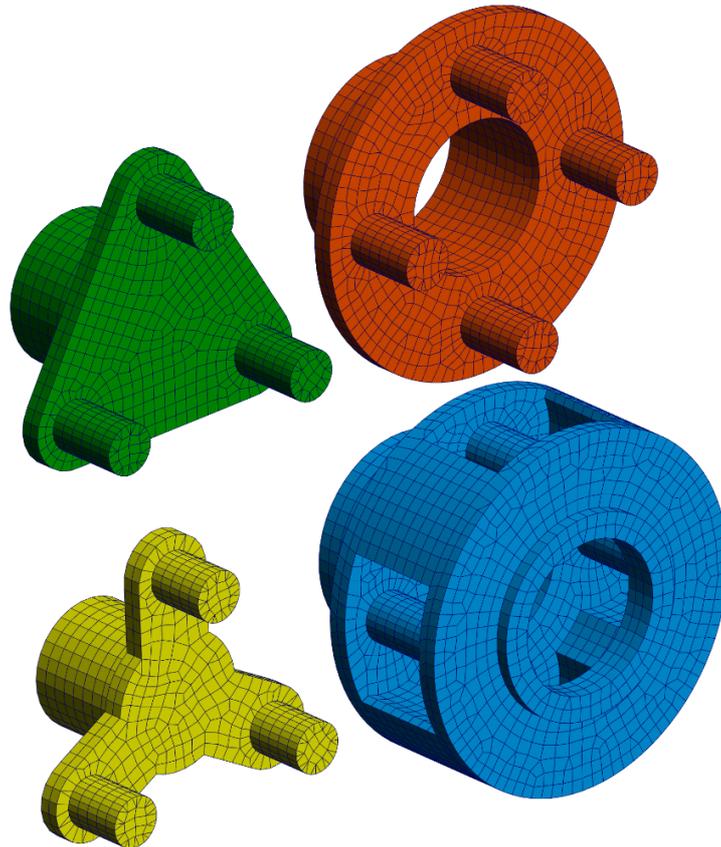
For an imported planet carrier, the rotation axis has to be the x-axis. On page "Positioning" a permanent coordinate transform can be used to change the imported orientation to the x-axis as rotation axis.

2.1.9.2.1 Parametric circular planet carrier

A parametric circular planet carrier can be defined either single or double sided. The geometry is shown in following image:

2.1.9.2.2 Parametric polygonal planet carrier

The geometry of the polygonal parametric carrier is shown in following image:

2.1.9.2.3 Parametric star-type planet carrier

The geometry of the star-type parametric carrier is shown in following image:
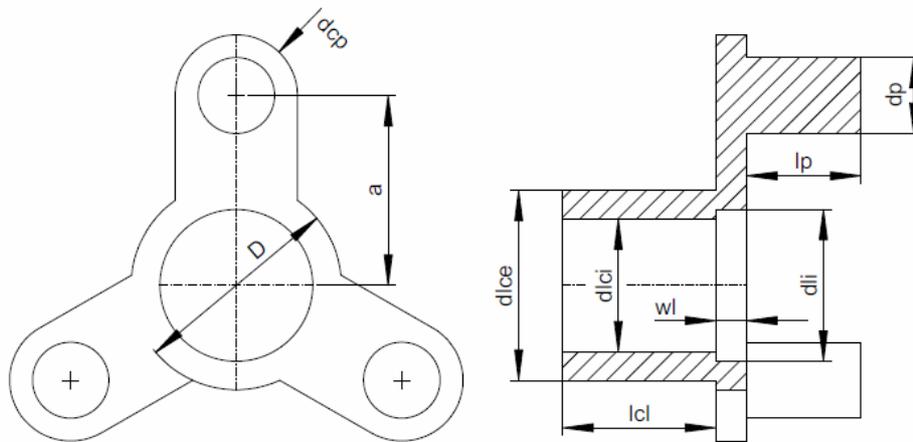
2.1.9.3   3D elastic part as housing

A 3D elastic part as housing can be connected to shafts in different groups, it does not rotate and at least one face has to be set as fixed. Currently, it can only be imported from CAD geometry as STEP file or as Nastran mesh.

Using the ✚-button behind "Import CAD geometry" a STEP file can be selected. An assembly with connected parts is glued together automatically. Afterwards the required mesh size can be entered. Alternatively, a Nastran mesh can be imported; only the surface mesh is kept in this case.

In the 3D view for the parts now multiple faces can be selected by mouse double click on a face. The faces can then be connected to a shaft support element or "Set as fixed". As above a rigid connection or an averaged connection can be selected. To merge two connected faces, select both faces in the table, press the right mouse button and select "Merge faces". If faces are inside the part a clipping of front faces can be activated using the context menu in the 3D-view. Using Alt-Button and left mouse button the clip-plane can be moved.

Under settings as before the material and options for a second order (quadratic) mesh and for calculation of the displacements can be selected. In addition, a 3D elastic housing can be supported in modal analysis; if "calculate mode shapes" is selected, the mode shapes can be shown, too. Under "Positioning" the position and rotation of the 3D part can be defined.

The selected support element will be connected to the nodes of the selected faces. A rigid or averaged connection will be considered, no contact and no clearance will be considered.

2.1.9.4   Import CAD geometry

For geometry import from CAD a preview dialog is shown first:
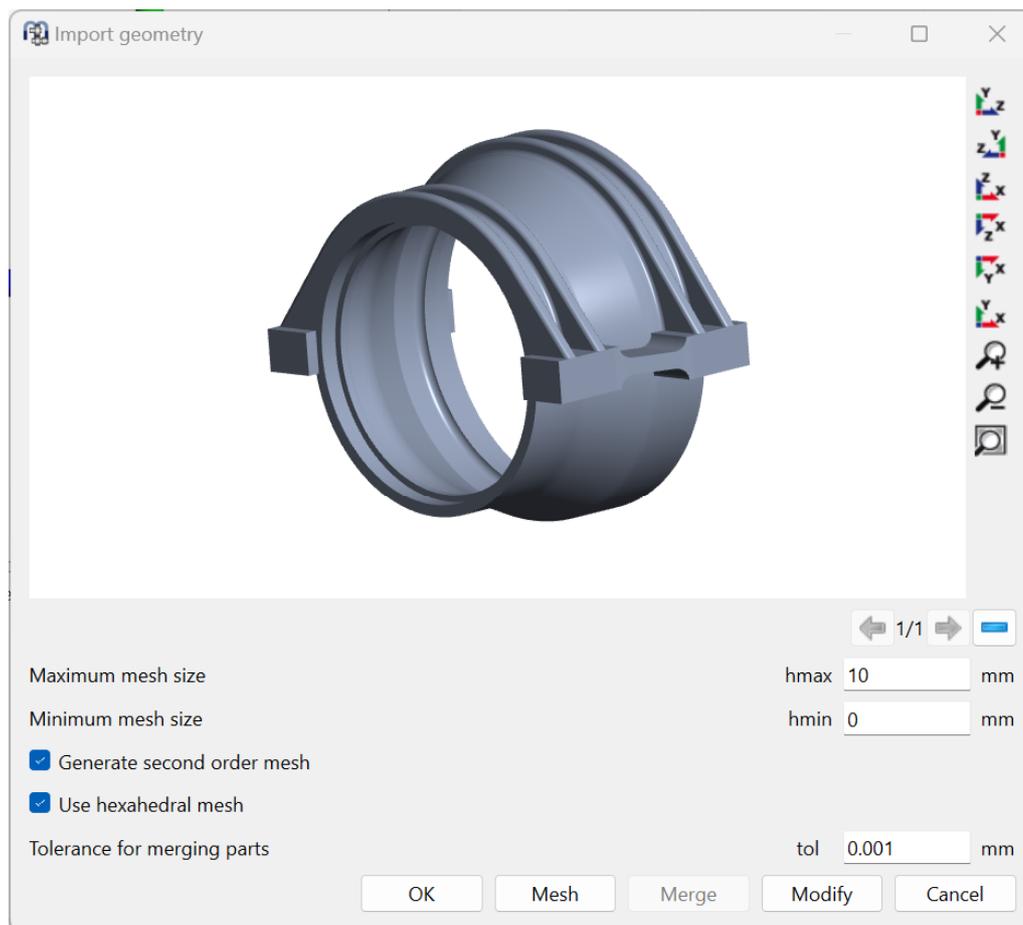
This dialog shows the number of parts available in the file and allows to delete parts and merge the remaining parts. The part can be remeshed with different settings to evaluate the mesh quality.

Using "Modify" chamfers, fillets and small holes can be removed from the geometry to simplify the model. This can reduce the required number of nodes and therefore the calculation time.

Only the currently selected part is meshed and imported. To combine parts the "Merge" function has to be used and a reasonable merge tolerance has to be set. After meshing the number of surface nodes is shown in the window to get an idea about mesh complexity.

If the option "Generate second order mesh" is set then a second order surface mesh is generated and saved. The setting "Use hexahedral mesh" will lead to mainly quads instead of triangles in the surface mesh.

2.1.9.5   Selecting faces on imported geometry

After importing a STEP file or a mesh multiple faces can be selected by mouse double click on a face. To merge two faces, select both faces in the table, press the right mouse button and select "Merge faces". If faces are inside the part a clipping of front faces can be activated using the context menu in the 3D-view. Using Alt-Button and left mouse button the clip-plane can be moved.

New faces can be defined using the context menu for the table and selecting "Add face". A new face can be selected based on a cylinder a bearing or a box:
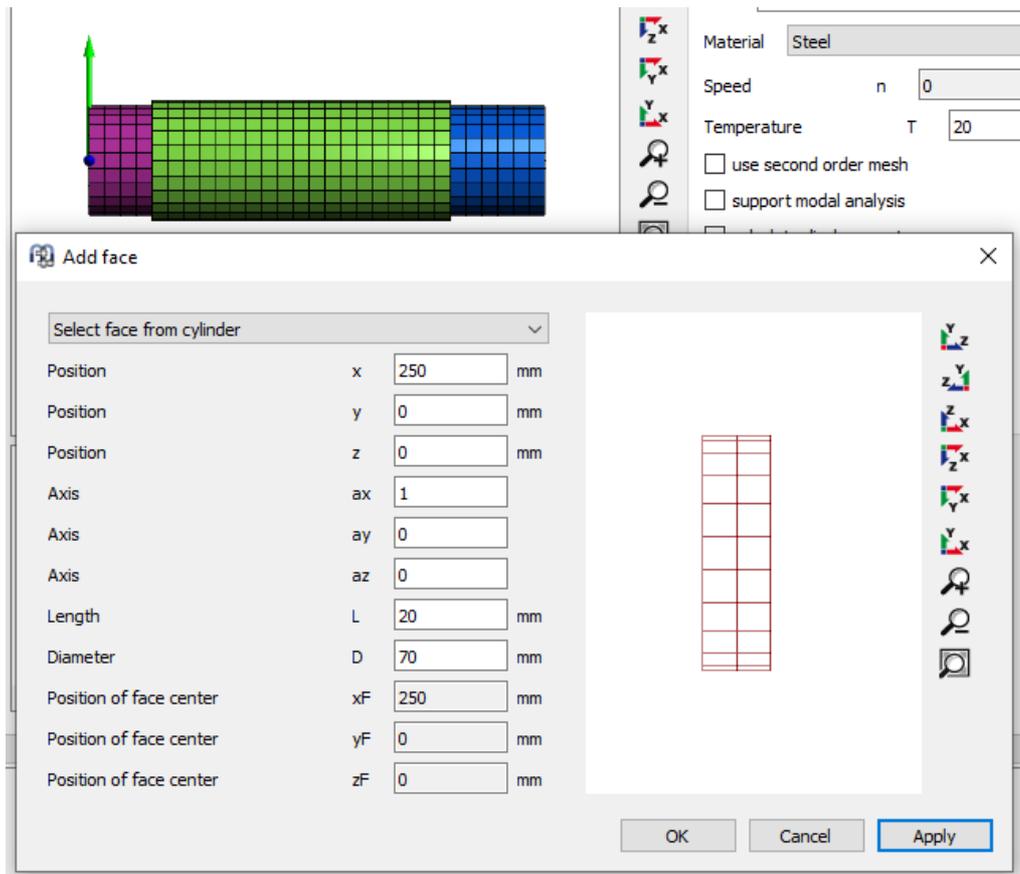
Define the position of the center of the cylinder and its diameter and length. Clicking "Apply" will select all elements in the range and will display the new elements in red. Existing selected faces are shown as mesh in black. For a box distances dx, dy, dz in 3 directions need to be defined instead of diameter and length. For a bearing, position, diameter and lengths are set automatically. This allows to add faces to an imported part without doing a new import.

2.1.9.6 Settings

There are some settings available for the 3D-elastic parts.

On page "General" the material, speed, temperature can be defined. A change of material requires a new reduction for the part.
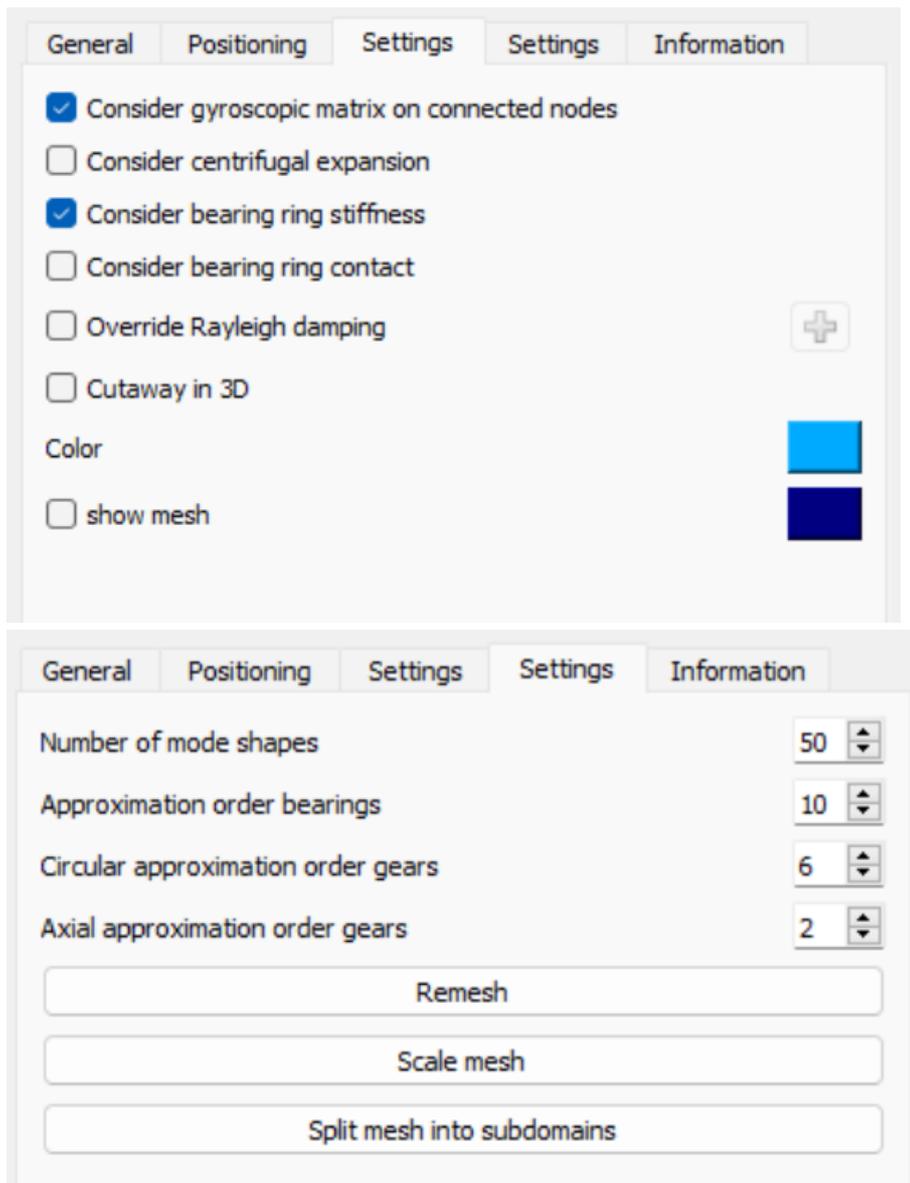
There are two pages for settings. All settings on the first page do not require a new condensation and all settings on the second page require a new condensation.

The color for the part and the mesh can defined which is used in the standard 3D-geometry views.

The reduction of the 3D-elastic parts lead only to mass and stiffness matrices. No gyroscopic matrix is considered. As approximation the polar inertia can be considered a gyroscopic matrix on the connected reduced nodes on the x-axis. The polar inertia will be distributed equally on all available nodes if the option"Consider gyroscopic matrix on connected nodes" is set.

The option "Consider centrifugal expansion" will add the effect of centrifugal expansion to the part. This is only active for 3D-elastic shafts.

The option "Consider bearing ring stiffness" will add the stiffness of the bearing rings to elastic bearings. If the ring is already included in the 3D-elastic part, this option should be disabled. Using "Consider bearing ring contact" will add a contact model between bearing ring and part. Then the fitting calculation will not directly set an operating clearance of the bearing, but the fitting will be influenced by expansion or compression of the elastic parts. If the stiffness of the part is not uniform, there will be already a deformation of the bearing ring after mounting.

The option "Override Rayleigh damping" allows to define a different Rayleigh damping for the 3D-elastic part than for the global system.

Activating the option "Cutaway in 3D" will lead to a cut view in the 3D-view for the system like the same setting available for shafts.

On the second page "Settings" the number of mode shapes used for modal reduction can be defined. A larger number increases accuracy but required more memory and reduction time. Usually a value larger 20 should be used. The maximum frequency is shown on page "Information". This frequency should be larger than the maximum frequency that is of interest for the system.

An approximation order for elastic bearing rings and elastic gears can be defined here. The transverse approximation order for gears of 2 means a parabola is used for transverse deformations. A factor of 1 would only use a straight line. For gears with a large face width larger orders than 2 might be appropriate. The approximation order can be overridden using the contact menu in the table for faces.

The button "Remesh" allows to remesh the part. Either to change the mesh size or the type hexahedral/tetrahedral. Note that this is a remesh of the previous mesh, generally it would be better to do a new import of the CAD file in case the mesh size is changed. "Scale mesh" allows to apply a scale factor on the mesh. This could help, in case the unit system of the import was not correct. "Split mesh into subdomains" allows to split the mesh into subdomains based on selection of a cylinder or a box, for each subdomain then different material properties can be selected.The use of either button requires a new reduction of the part.

2.1.9.6.1 Details for bearing clearance in case of elastic bearing rings

The deformations of the selected face for a bearing contact is used as deformation for the bearing ring. To avoid double consideration of thermal expansions, the thermal expansion based on the nominal diameter is subtracted on the bearing deformation. For the thermal expansion the shaft/housing material and temperature defined in the bearing calculation is used.

The centrifugal expansion might be considered twice. Once in the deformation of the part, if activated there, and once in the operating clearance of the bearing, if fitting is considered. Use the contact model for bearing rings to avoid this or disable centrifugal expansion for the part.

If the bearing ring is considered as additional stiffness and no contact model is used, the thermal expansion coefficient of the part is used instead of the coefficient of the bearing ring. This is done to avoid additional deformations. The thermal expansion is subtracted from the bearing deformation. The contact model should be used to include fitting in the FEA model.

If the contact model is used the ring deformations considered in calculation of operating clearance in the bearing calculation are subtracted from the elastic deformation of the FEA-model. Therefore fitting is not considered twice. Centrifugal expansion should be activated on the 3D-elastic part if that would be relevant. In the case that the bearing has neither inner ring nor outer ring also the clearance changes calculated in the bearing calculation are subtracted from the FEA deformations to avoid double consideration.

The diagrams for deformations of the face always show the deformations of the part without these mentioned corrections. The deformations of the bearing rings will show a reduced value corrected by influence of temperature and fitting.

If temperature effects are considered it is recommended to consider the bearing ring contact for the 3D-elastic parts.

2.1.9.6.2 Split mesh into subdomains

"Split mesh into subdomains" under Settings allows to split the mesh into subdomains based on selection of a cylinder or a box. For each subdomain then different material properties can be defined. All selected faces will be lost on this option.

Using a box or a cylinder, elements of the current mesh can be selected and assigned to a new subdomain. Currently it is not possible to join subdomains again directly, but it could be done by selection of a box containing all elements.
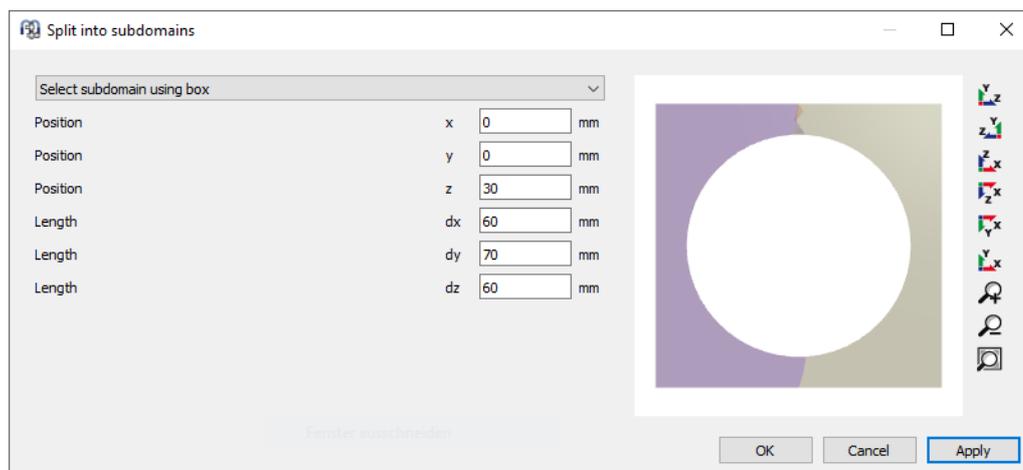
2.2 Calculation of natural frequencies and harmonic response

Optionally natural frequencies and harmonic response can be calculated using the software. The setting Calculate natural frequencies |58| has to be activated for this.

First the nonlinear static equilibrium is calculated. Then the natural frequencies and the harmonic response are calculated using the linear equations of motion with the tangent stiffness matrices from the static equilibrium. Therefore the results are only valid for small amplitudes around the static solution.

The natural frequencies are calculated using the given rotation speed. The critical frequencies are the intersections between shaft speed and natural frequency at this shaft speed. The critical frequencies are shown in a special report or they can be found in the Campbell diagram.

In the menu Calculations an additional calculation of frequency response is available. To use this, one or more dynamic forces have to be placed on a shaft and the calculation of natural frequencies needs to be active. It allows to define one dynamic force and to vary its frequency over a given range. Results for displacement, velocities and forces are available for documentation points or supports.

2.3 Interface to rolling bearing calculation

The dialogs of the MESYS Rolling Bearing Calculation |146| are fully integrated into the shaft calculation.
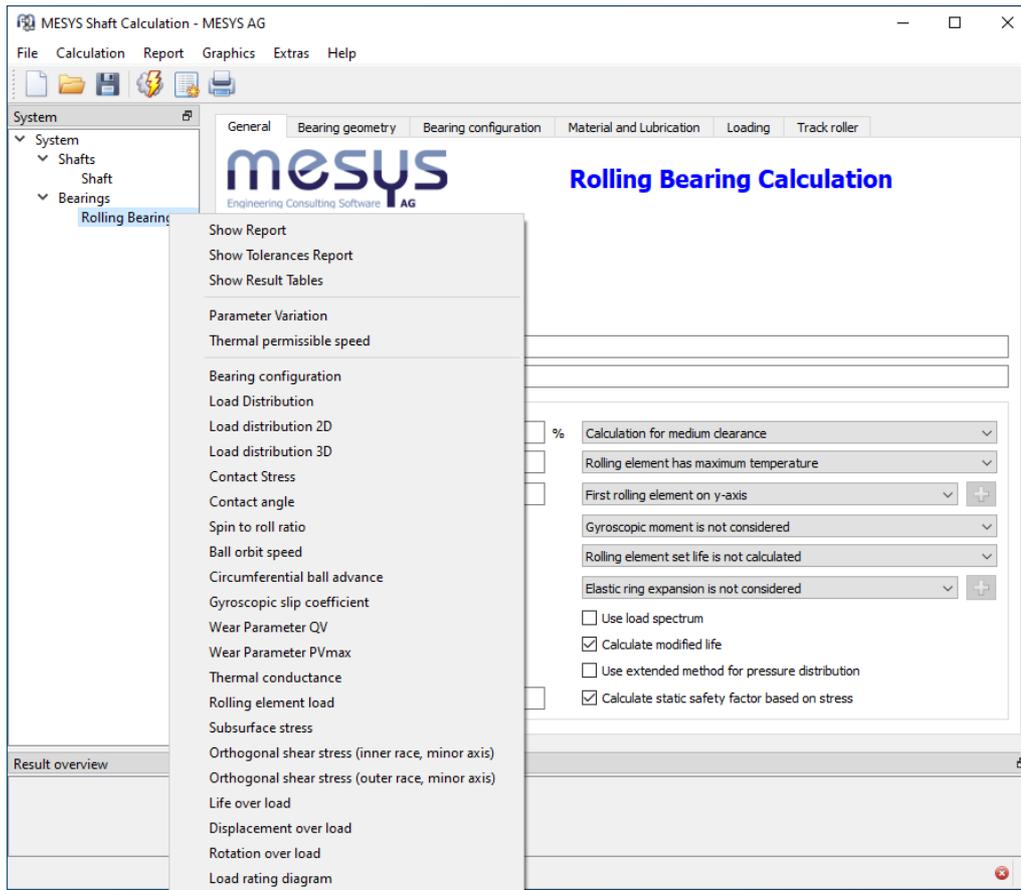
While the bearing calculation is open, the functions for file operations, calculation and report generation are used for the bearing calculation only. Therefore, only the bearing will be calculated and you will get results for the bearing in the results overview. However, it exists the possibility to run the whole shaft calculation, while the bearing calculation is in use, by pressing the key combination CTRL+F5.

Graphics and the tolerance report can be opened using the right mouse button in the system tree. Graphics are also available in the menu for graphics. Graphics from different bearings can be shown at the same time.

2.4   Interface to ball screw calculation

All dialogs of the ball screw calculation are fully integrated into the shaft calculation like for the rolling bearing calculation.
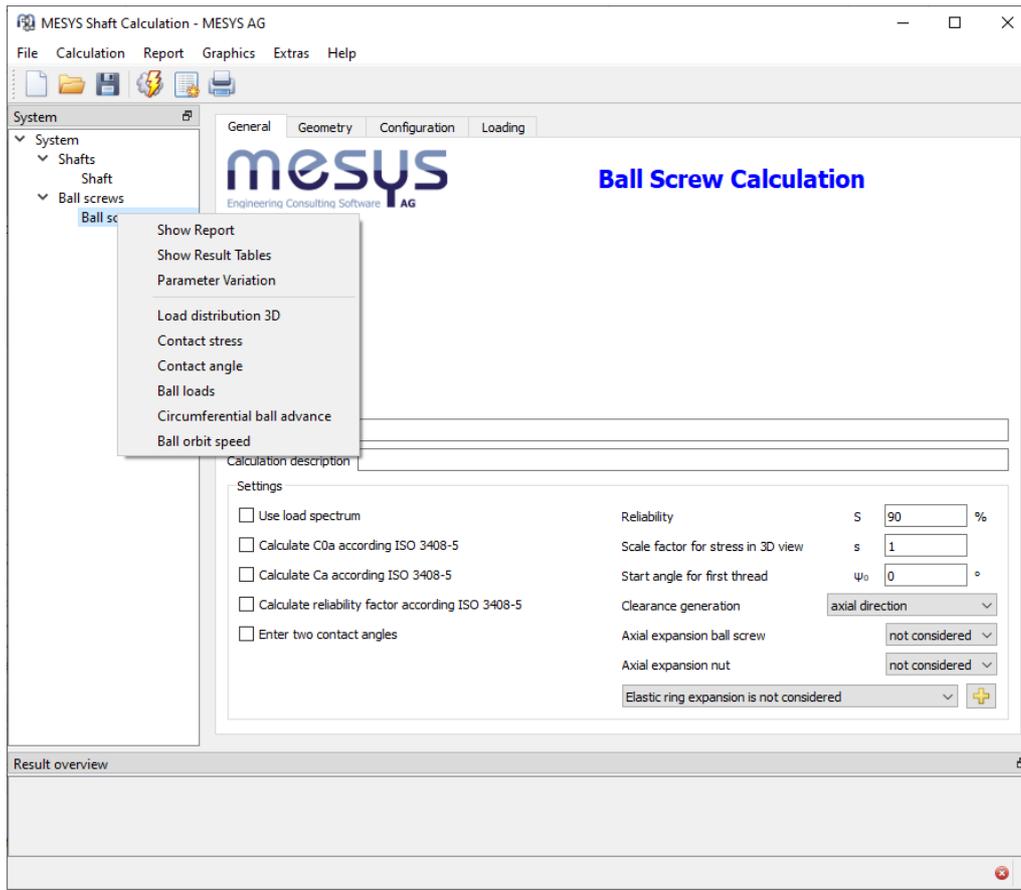
While the ball screw calculation is open, the functions for file operations, calculation and report generation are used for the ball screw calculation only. Therefore, only the ball screw will be calculated and you will get results for the ball screw in the results overview. However, it exists the possibility to run the whole shaft calculation, while the ball screw calculation is in use, by pressing the key combination CTRL+F5.

Graphics and can be opened using the right mouse button in the system tree. Graphics are also available in the menu for graphics. Graphics from different ball screws can be shown at the same time.

2.5 Interface to SKF bearing module

In menu "Extras" the SKF bearing module can be activated:

After accepting the terms of use and clicking the "Check"-button a log-in dialog in the web-browser is shown. If no credentials for the SKF bearing module are available, first register for the service. After successful log-in "Access granted" will be show in the window. Using "Reset" the service is disabled and no further requests are send to the SKF server.

Two services are available:

- The SKF bearing calculation service is available to all registered users. If this is activated additional two columns for SKF bearing life are shown in the result overview and a table with SKF results is added to the report. The SKF modified life rating can be different from the ISO 281 life rating as aSKF is calculated differently than aISO.

- The SKF bearing data service allows to retrieve detailed bearing data from the SKF server. Additional permissions by SKF are required to use this service. Please contact your SKF application support regarding this service. Initially the bearing data service is available for super-precision bearings only and will be extended later. Initially supported bearing types are: Angular contact bearings, single and double row radial cylindrical roller bearings and limited to super-precision bearings.
The SKF data base within the installation only contains approximated internal geometry. With this service the detailed internal geometry from SKF is loaded into memory and will be lost again after closing the software. If the internal geometry is available, the bearing is printed in bold face in the bearing database table and a different color is used in the 2D-shaft editor. These colors can be selected under Display settings 64 in the shaft calculation. If the option "Consider custom bearings for SKF bearing data service" is active, the software also tries to get internal data for bearings in the custom bearing database, if these are set to approximated bearing geometry and the manufacturer name starts with "SKF".

2.6 Results

Results are available in different outputs. There is the default result overview on the bottom of the user interface, an overview of bearing forces and natural frequencies, several graphics and the report.

2.6.1 Results Overview

The results overview on the bottom of the window shows minimal bearing life, minimum static bearing safety and maximum shaft deflection. The results overview can be configured under Extras->Results overview.
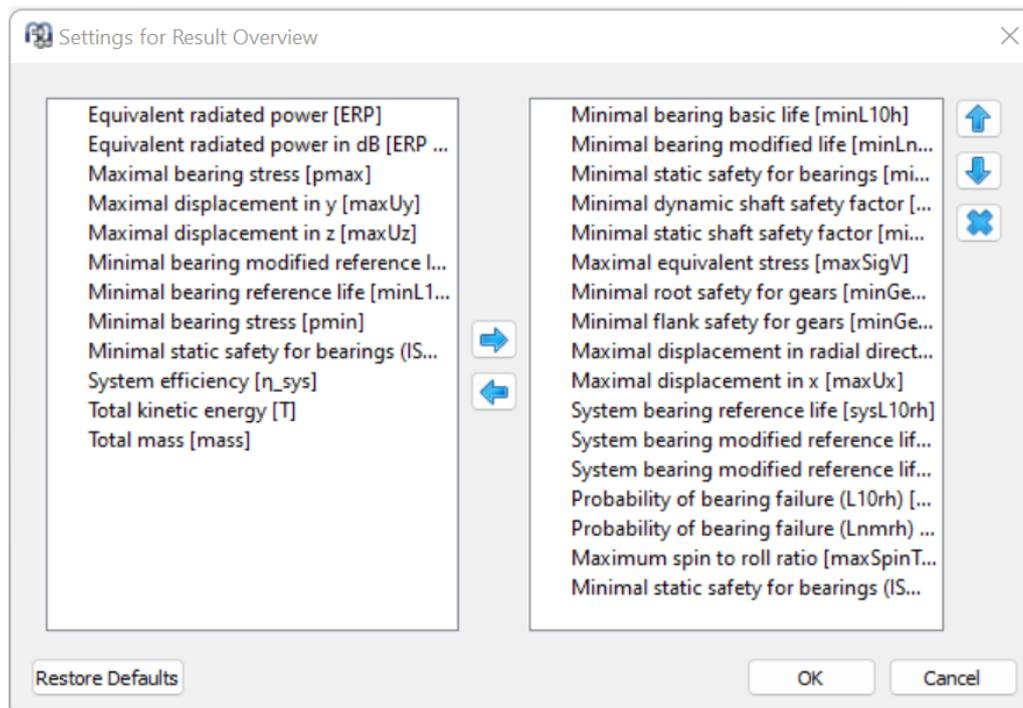
The list on the left shows all available items for the result overview. Using the arrows selected items can be moved to the list on the right which is used for the result overview. The order of the selected results can be changed by the up/down arrows on the right. Note that only valid results values are actually shown in the result overview. So you will see fewer values than selected in the settings.

The settings for the result overview are saved in user settings not in the calculation file. So they will be the same for different calculations.

2.6.2 Result Tables

Several result tables are available if the pages "Shafts", "Bearings" or a group is selected.
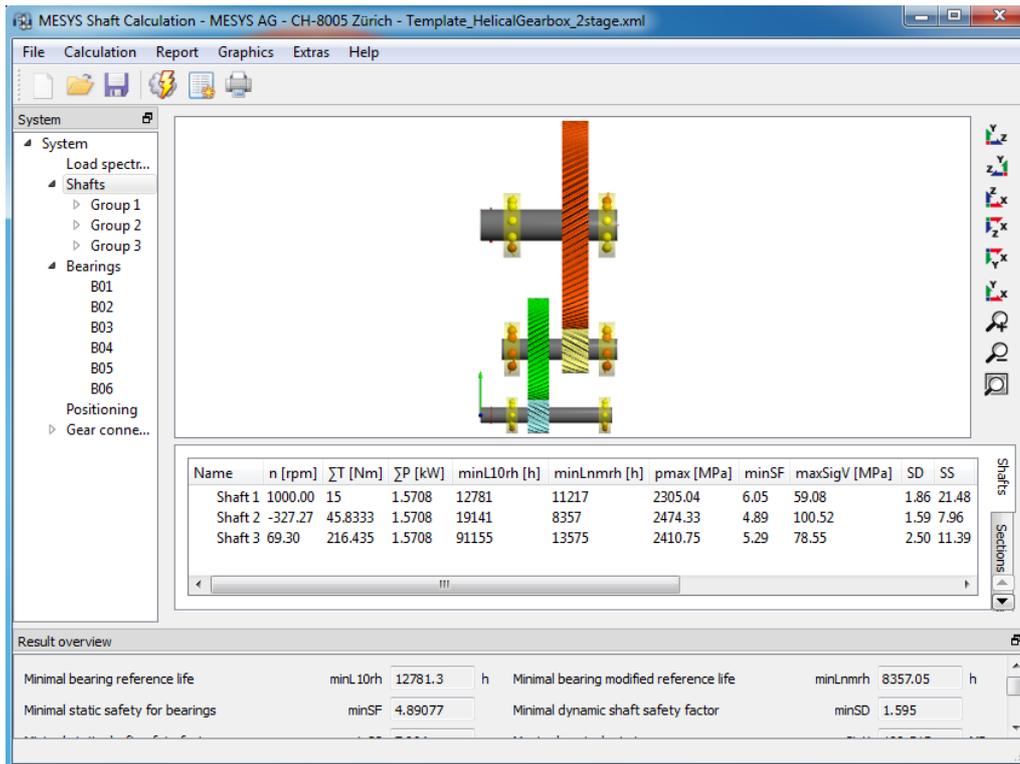
2.6.2.1 Table for shafts

A table for shafts is showing results for each shaft:

Available results are:

- 'n': shaft rotation speed

- '∑T': sum of all torques with the same sign

- '∑P': sum of all power inputs with the same sign

- 'minL10rh': minimum reference life L10rh for all bearings on this shaft. Alternatively L10h according to ISO 281 is shown.

- 'minL10rh': minimum modified reference life Lnmrh for all bearings on this shaft. Alternatively Lnmh according to ISO 281 is shown.

- 'pmax': maximum contact stress for bearings

- 'minS0eff': minimum static safety factor for all bearings on this shaft

- 'maxSigV': maximum equivalent stress with in this shaft

- 'SD': safety for fatigue according to DIN 743 for this shaft

- 'SS': static safety factor according to DIN 743 for this shaft

- 'maxUr': maximum radial displacement of the shaft

- 'mass': mass of this shaft

- 'Jxx, Jyy, Jzz': moments of inertia of the shaft

2.6.2.2 Table for sections

The table for sections shows details for each section for every shaft. The type of notch effect is shown, safety factors for strength, forces, moments and displacements:

Name	Type	x [mm]	SD	SS	Fx [kN]	Fy [kN]	Fz [kN]	Mx [Nm]	My [Nm]	Mz [Nm]	ux [mm]	uy [mm]	uz [mm]	rx [°]	ry [°]	rz [°]
⊿ S1			3.94	8.28												
A	Shoulder	38	5.35	8.28	0.000	-1.034	-2.826	-100.00	-22.57	8.22	0.0048	-0.0107	-0.0282	1.20	0.01	-0.00
B	Shoulder	172	27.83	87.42	0.000	0.403	1.095	0.00	-8.74	3.20	0.0048	-0.0063	-0.0165	1.18	-0.02	0.01
C	Interference fit	55	4.47	10.46	0.000	-1.033	-2.826	-100.00	-70.61	25.78	0.0048	-0.0122	-0.0324	1.19	0.01	-0.00
D	Interference fit	95	3.94	12.44	0.000	0.361	0.995	-2.57	-93.09	34.06	0.0048	-0.0132	-0.0351	1.18	-0.00	0.00

Available results are:

- 'SD': safety for fatigue according to DIN 743 for this section

- 'SS': static safety factor according to DIN 743 for this section

- 'Fx, Fy, Fz': forces in the section of the shaft. For faces of elastic parts the force is acting onto the part.

- 'Mx, My, Mz': moments in the section of the shaft. For faces of elastic parts the force is acting onto the part.

- 'ux, uy, uz': axial and radial displacement for this position

- 'rx, ry, rz': torsion and tilting angles for this position

- 'cxx, cyy, czz': stiffness of the system reduced to this point. The reciprocal of the main diagonal of the compliance matrix is shown.

- 'crx, cry, crz': rotational stiffness of the system reduced to this point. The reciprocal of the main diagonal of the compliance matrix is shown.

2.6.2.3 Table for bearings

A table for bearings shows life and loading of bearings and supports. The sign convention for the loads is that the force from the shaft to the bearing is shown.

For rolling bearings the life and pressure is shown in addition to forces and moments. Using the right mouse button additional results like operating clearance or spin-to-roll ratio can be shown.
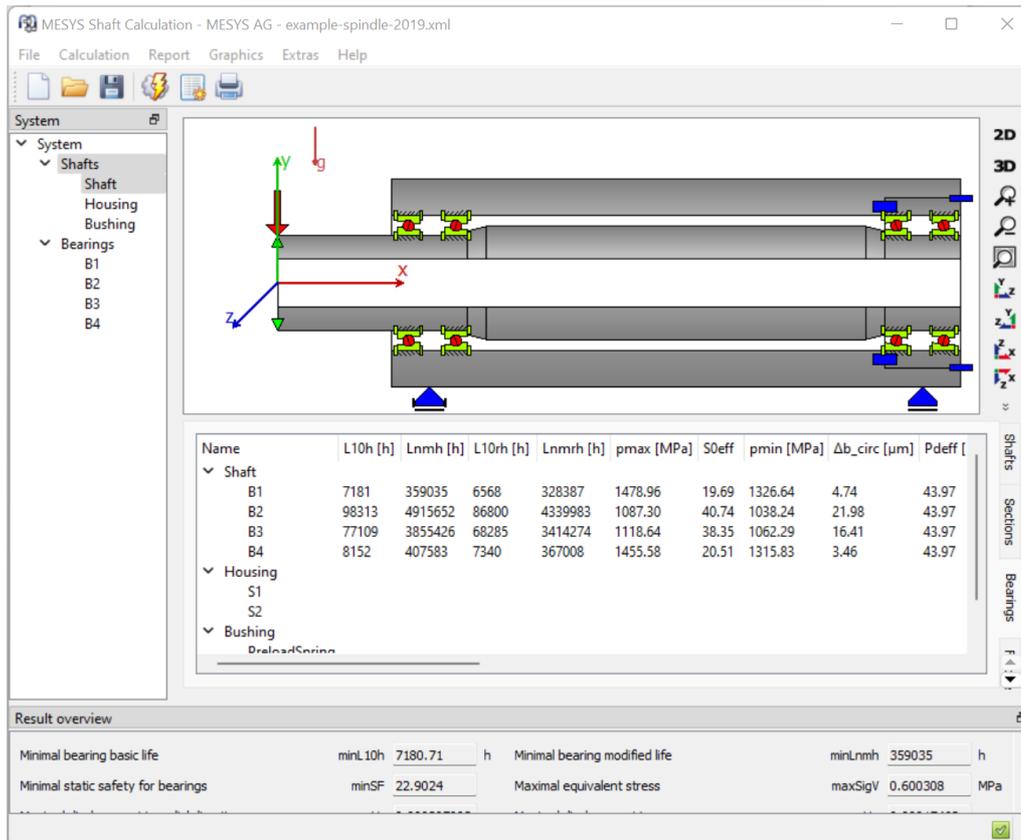
Available results are:

- 'L10h': nominal life according to ISO 281

- 'Lnmh': modified life according to ISO 281

- 'L10rh': nominal reference life according to ISO/TS 16281

- 'Lnmrh': modified reference life according to ISO/TS 16281

- 'pmax': maximum contact stress of the bearing

- 'S0eff': static safety factor based on the load distribution within the bearing

- 'pmin': minimum contact stress of the bearing. If the value is larger zero all rolling elements are loaded.

- 'ω_spin/ω_roll': spin to roll ratio for ball bearings

- 'Δb_circ': circumferential ball advance for ball bearings

- 'ΔT': temperature increase according to DIN 732 for this bearing

- 'nt': maximum permissible speed according to DIN 732

- 'κ': viscosity ratio for this bearing. It should usually be between 1 and 4

- 'Pdeff': effective radial clearance of the bearing at operating conditions

- 'Paeff': effective axial clearance of the bearing at operating conditions

- 'α0eff': effective free contact angle for ball bearings

- 'Δα': variation of contact angle between inner and outer ring for ball bearings. this value should be smaller 15° according to some manufacturers.

- 'pFitShaft': pressure in interference fit between shaft and inner ring. This should be larger than zero usually.

- 'eLR_i, eLR_e': extension of contact ellipse for ball bearings. Values below 100% indicate truncation.

- 'Fx, Fy, Fz, Fr': bearing forces, Fr is radial force. The force is acting onto the support.

- 'Mx, My, Mz, Mr': moments, Mr is tilting moment. The moment is acting onto the support.

- 'ux, uy, uz, ur': relative displacements of shaft node to second node (or housing), ur is radial displacement

- 'rx, ry, rz, rr': relative rotation of shaft node relative to second node (or housing), rr is resulting tilting angle

2.6.2.4 Table for frequencies

In the overview also the natural frequencies can be shown. If a frequency is selected, the corresponding mode shape is shown in the graphics. Three displacements and the torsion angle are shown in 2D view. In 3D view the mode shapes are shown animated.
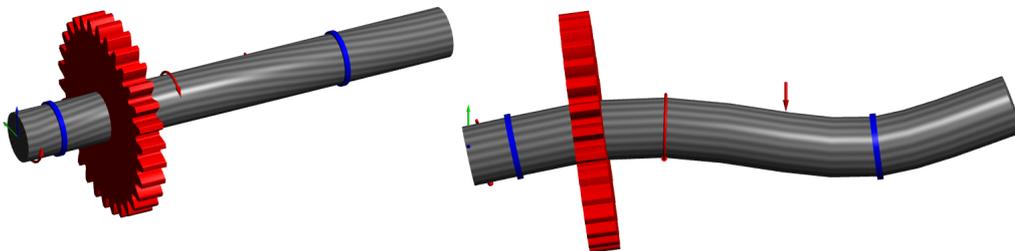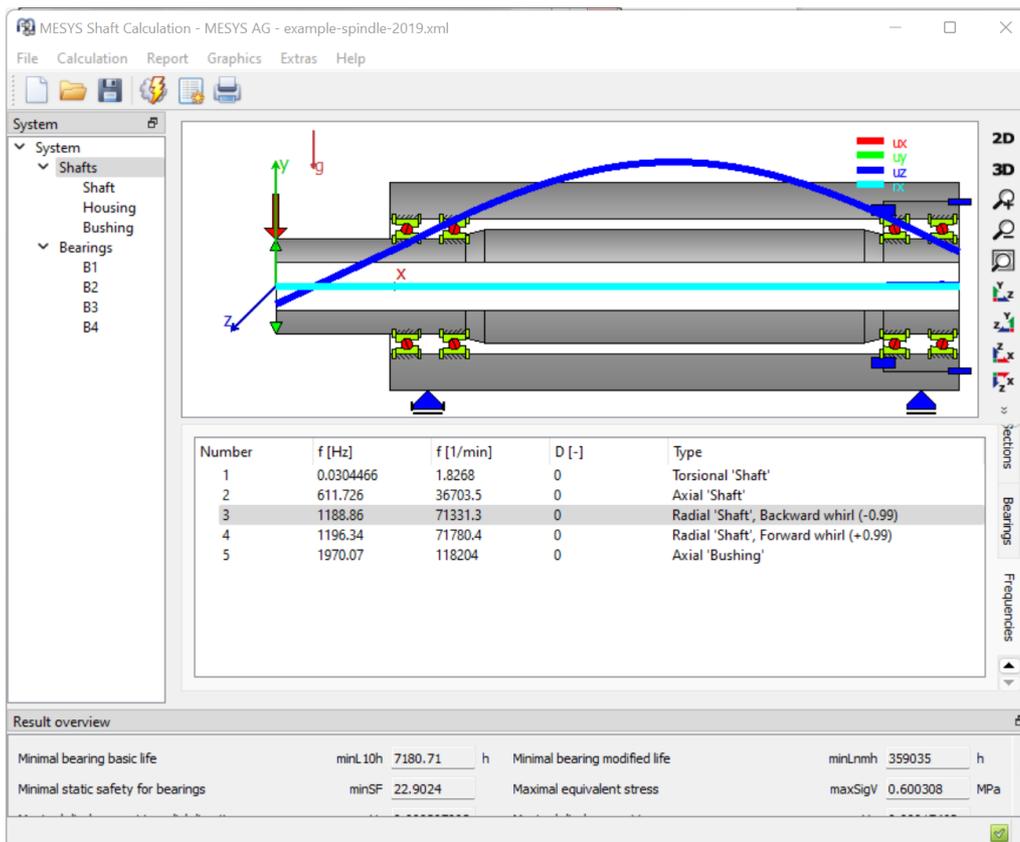
Available results are:

- 'f': frequency in Hz and per minute

- 'D': Damping ratio of the mode in case a damping is defined

- 'Type': Try's to classify the type of mode based on the kinetic energy of the modes. A numeric value for the whirl direction is given. +1 is a circular forward whirl, 0 is a linear orbit and -1 is a circular backward whirl.

2.6.3   Report

Using the toolbar button or "Report"->"Show Report", a report for the shaft calculation is generated which only gives an overview for the bearing results. There is also "Report"->"Full report" which is generating a full report with results of the shaft calculation and the full reports of the bearing calculations.

2.6.3.1   Report options

Using "Report"->"Report options" the contents of the report can be configured.
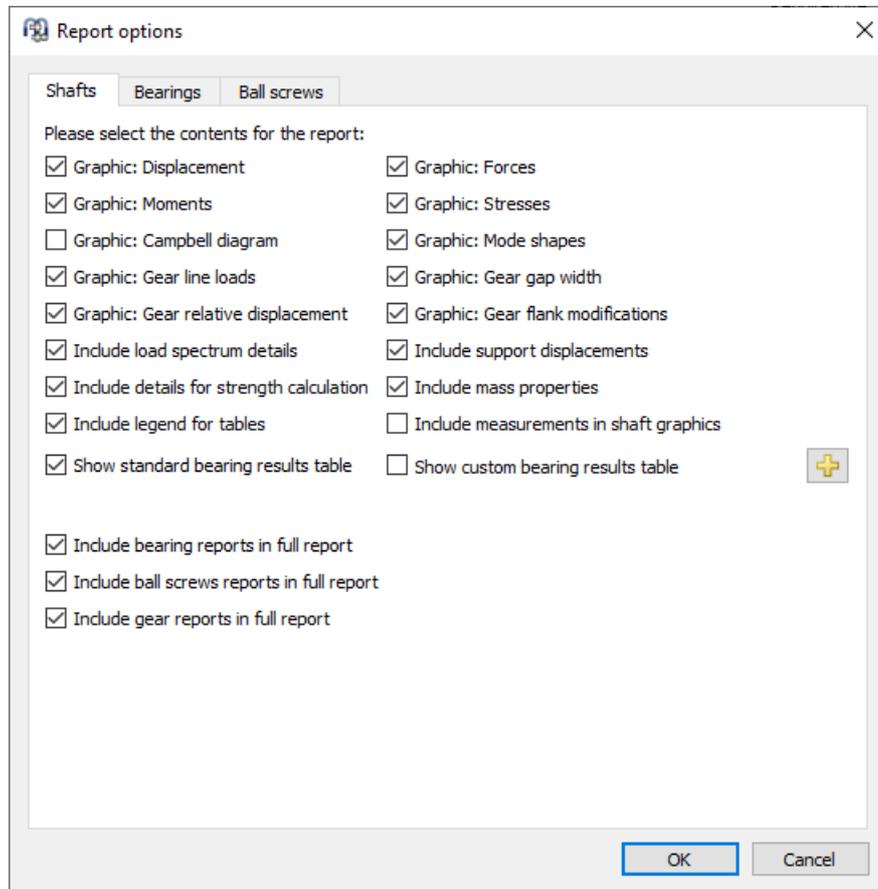
One page is available for outputs of the shaft calculation, one page for the bearing calculation and one page for the ball screw calculation. The settings for bearings and ball screws are passed to the corresponding calculations. The legend for all the table parameters can be shown in the report. Custom tables for bearing results can be configured if the license for the bearing calculation is available.

2.6.4   Result tables

In the menu "Report"->"Result tables", it is possible to output the result data by means of tables, which can be generated in different file formats as explained before at "Configuration with INI-File".

Results tables with detail results for bearings and ball screws can be opened with a right click on the bearing or ball screw in the system tree.

2.6.5 Custom Report Templates

Custom report templates can be added in subdirectory "templates/shaft". Reports based on these templates can be generated by selecting the report template under "Report"->"Report templates".

The report templates are word files with DOCVARIABLE fields. These fields are filled on generation of the report.

Possible fields are:

- DOCVARIABLE VAR name [unit]: the variable with the name 'name' is printed. Optionally a unit can be provided in brackets, for example [mm]. Additionally a formal could be added like %6.3g or %6.3f, where 'f' always generates a fixed point output and 'g' generates an exponential output for large numbers.

- DOCVARIABLE VARU name [unit]: the variable with the name 'name' plus the unit are printed. Optionally a unit can be provided in brackets, for example [mm].

- DOCVARIABLE TABLE tablename: A table is inserted.

- DOCVARIABLE GRAPHIC graphicName width=130 height=70: A graphic is inserted. Optionally height and width can be given in millimeters.

Regarding details for variable names, table names and graphic names, please ask support@mesys.ch for additional documentation.

2.6.6 Graphics

Several results can be shown as graphics. Most graphics are also appended to the report, so that a full documentation is provided in one document.
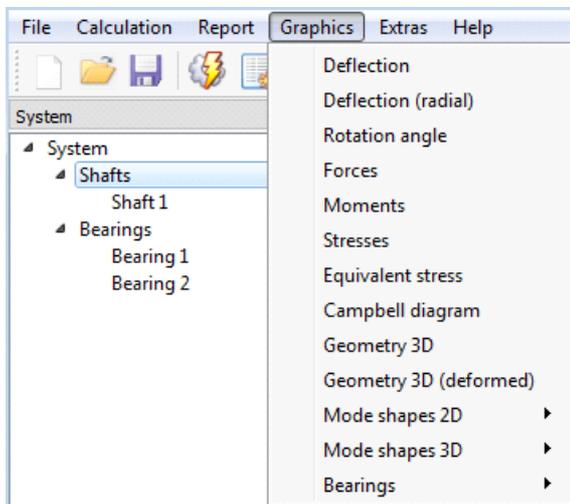
For all diagrams diagram options are available using the right mouse button. It is possible to change coordinate system and units. Also export and printing is available in the context menu.

2.6.6.1 Shaft deflection

The deflection of the shafts is shown for the three displacement coordinates for each shaft.
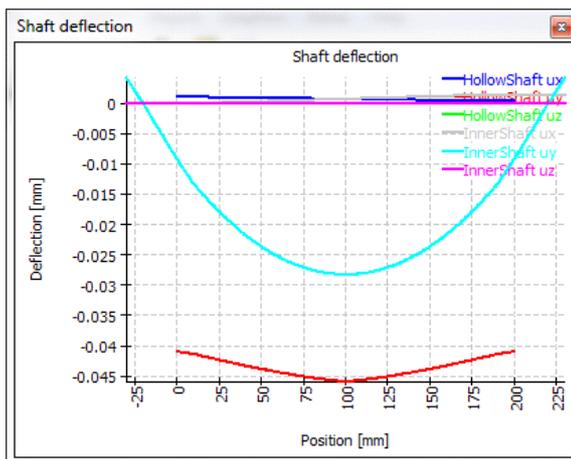
2.6.6.2 Shaft deflection (radial)

The radial deflection chart shows the absolute value of total radial deflection for the shafts.
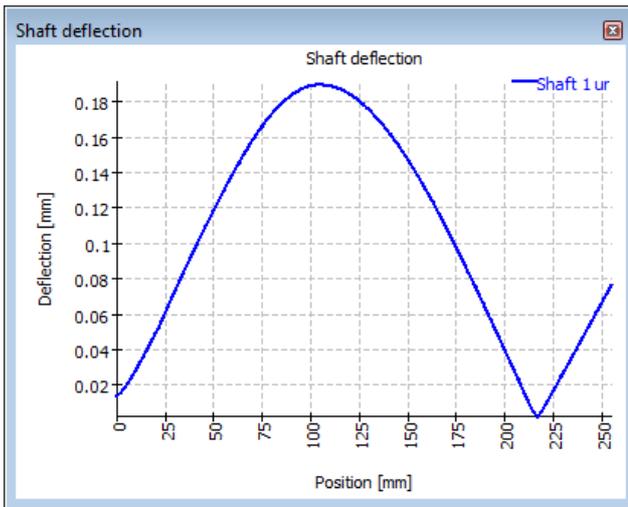
2.6.6.3   Shaft forces

The three force components are shown for each shaft.
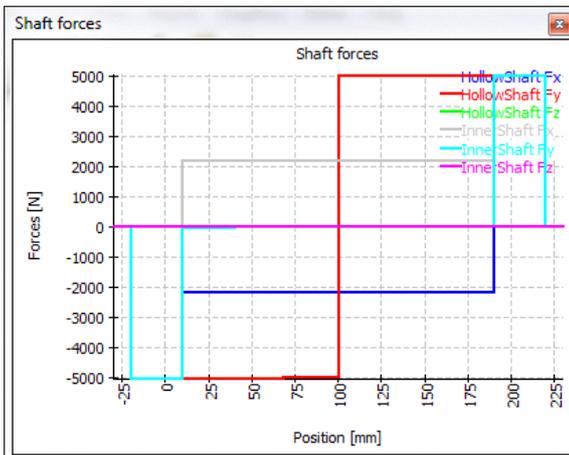
2.6.6.4 Shaft moments

The two bending moments and the torque are shown for each shaft.
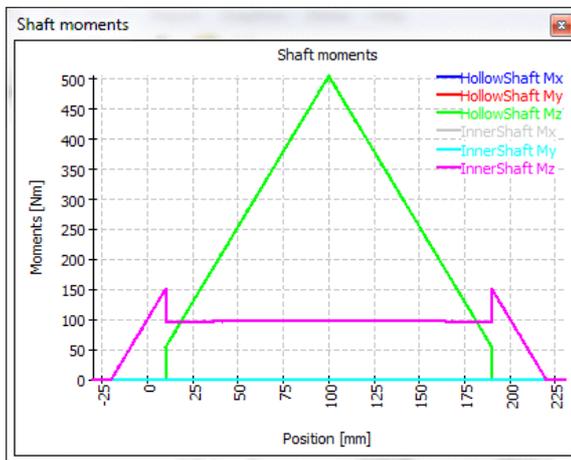
2.6.6.5 Shaft stresses

The stress components are shown for all shafts. Since there are too many curves in that diagram if several shafts are used, there is also a diagram which only shows the equivalent stress.

Shaft stresses

Shaft equivalent stress
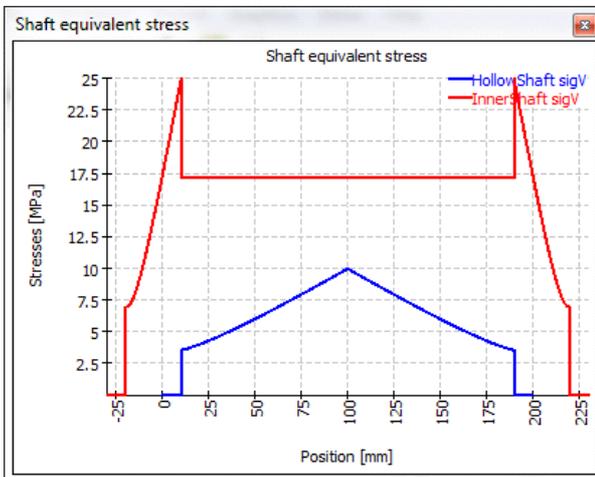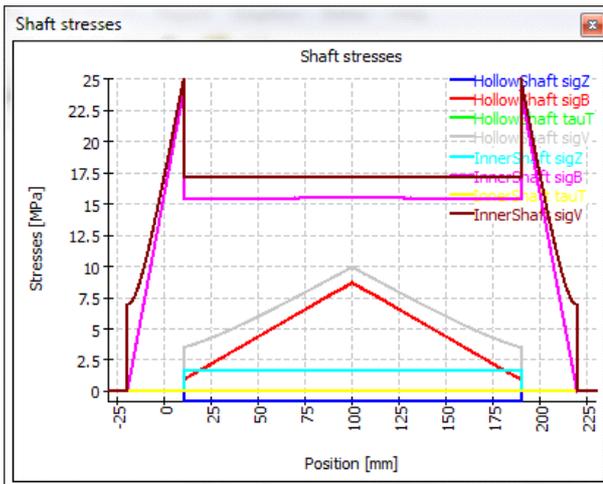
2.6.6.6 Campbell diagram

The Campbell diagram shows the change of natural frequencies over the shaft speed. The speed of all shafts is multiplied by the same factor in this calculation. The speed of the shafts is shown using dashed lines.

This calculation is always done considering the gyroscopic effect.
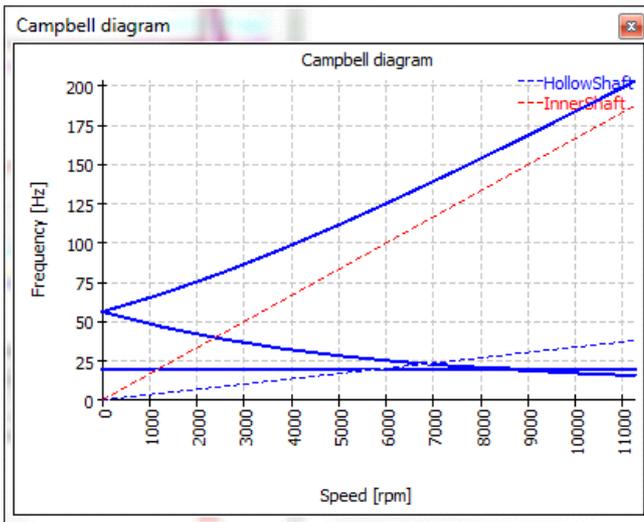
2.6.6.7 Harmonic response over speed

Harmonic response can be shown over speed. The calculation is done together with the Campbell diagram.

There are diagrams for forces available at all supports and diagrams for displacements available at positions of loads and sections of type "Documentation point"
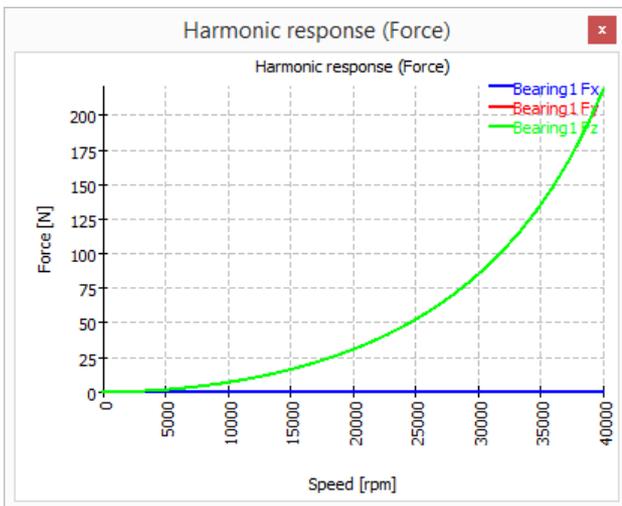
2.6.6.8 Harmonic response over time

Harmonic response over time is showing the variation of reaction forces and displacements over time. This calculation is done for the current rotation speed.
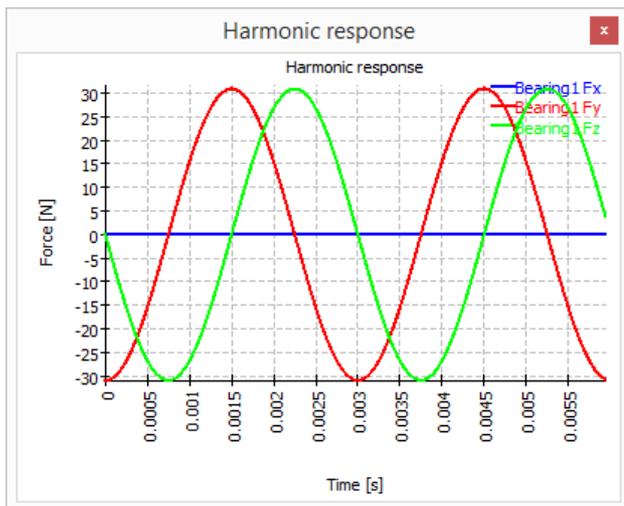
2.6.6.9   Harmonic response 3D

Harmonic response 3D is showing the dynamic displacements over time as 3D view.

2.6.6.10   Diagrams for bearing analysis

In addition, there are several diagrams for bearing analysis. Please check the documentation of the bearing analysis for details. See result graphics for bearings 223 .

2.6.6.11   Bearing overviews

Several graphics for bearings are available as overview graphics, showing the same diagram for each bearing. This allows to get a quick overview over the behavior of all bearings.

2.6.6.12 Bearing system reliability

The system reliability considering all bearings is shown in a diagram. The reliability is shown for the four different results for bearing life.
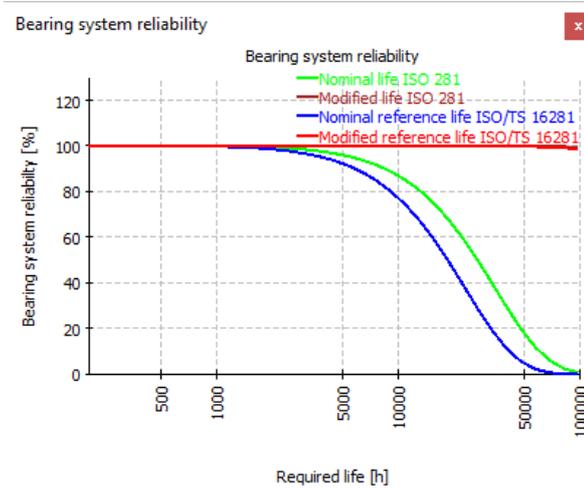
2.6.6.13 Geometry 3D

The 3D geometry can be shown without deformations or with exaggerated deformations. The diagram is available for the whole systems and for each group.
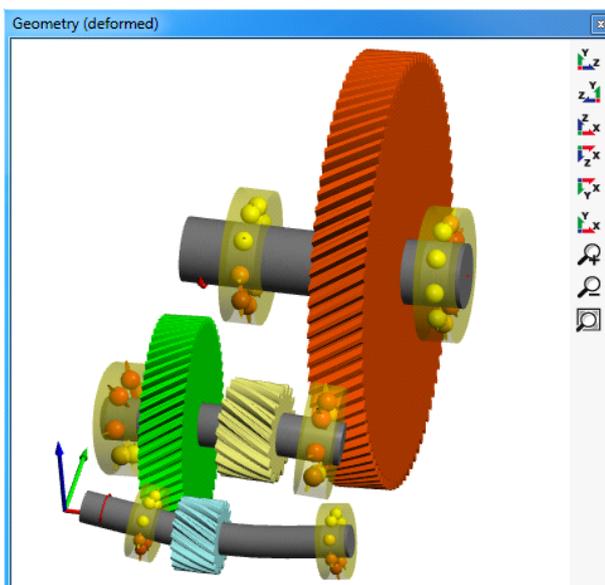
2.6.6.14 Geometry 3D (power flow)

The power flow of transmission systems can be shown in the 3D graphics.
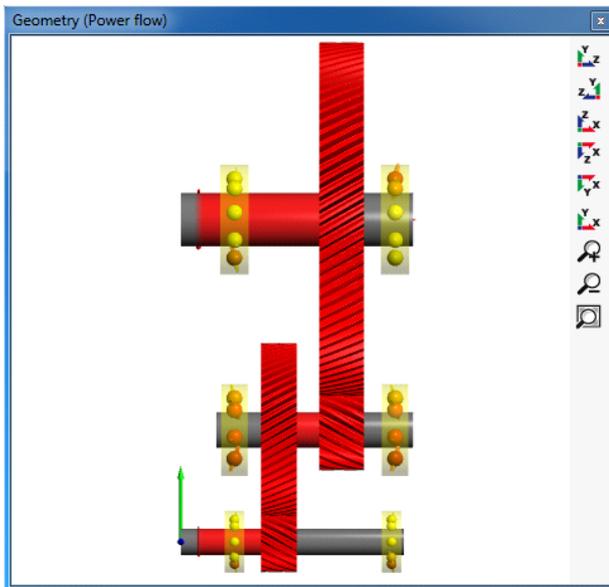
2.6.6.15 Gear line load and gap width

The line load on a gear pair can be shown and also the gap width. For these diagrams either the gears should be considered as stiffness under settings or the shaft diameter has to be large enough to take into account the stiffness of the gear body. The gap width can be used for a proposal of flank line corrections. Both diagrams are also available for load spectra showing results for all elements.
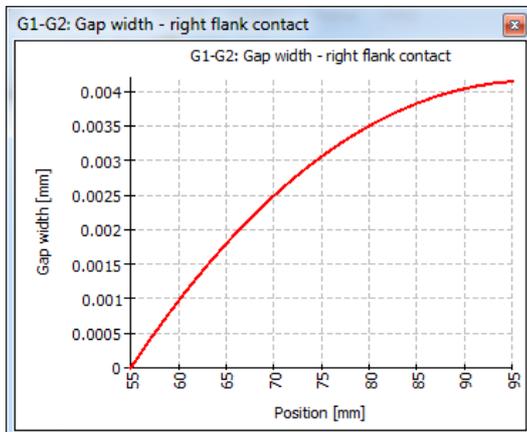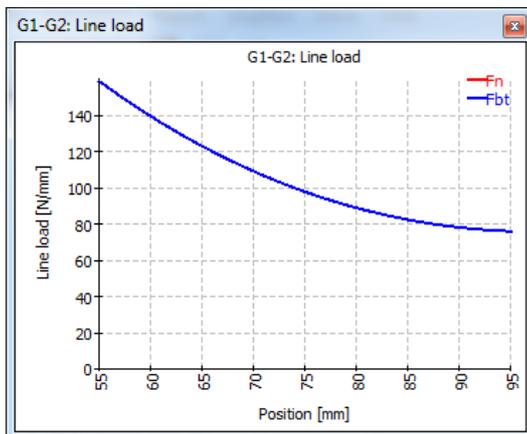
2.6.6.16 Tooth flank modifications

It shows the resulting profile of the tooth flank from the corrections applied in order to obtain an optimum tooth contact. In this way, the x-axis represents the tooth width in mm and the y-axis the geometrical variation in μm which has been undergone by the flank.
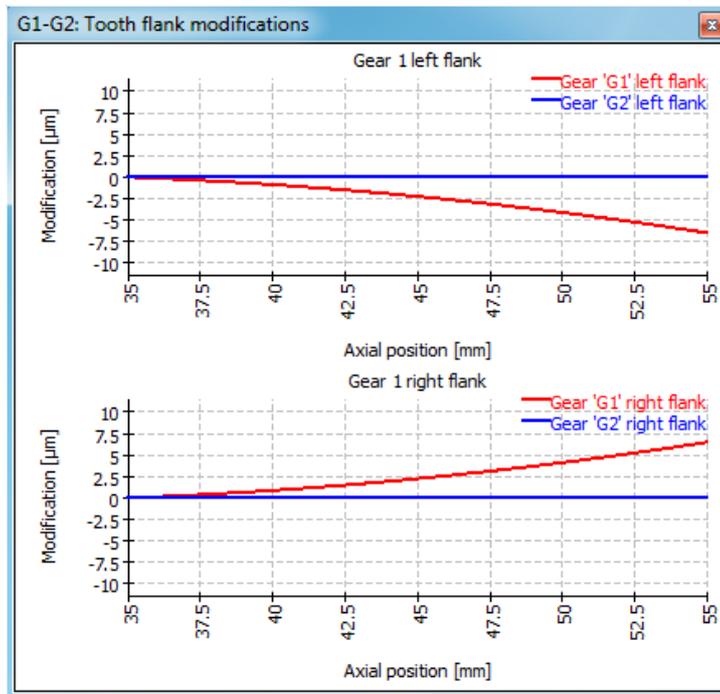
2.6.6.17 Relative displacements

A diagram for relative displacement of shafts is shown for each cylindrical gear pair. It shows the relative displacement of both shafts over the width of the gears. The axial displacement is a change of the center distance, the normal displacement a translation normal to the plane of the two shafts.

The diagram helps to understand required modifications or changes or clearance.

2.6.6.18 Connection overviews

Several graphics for connections are available as overview graphics, showing the same diagram for each connection. This allows to get a quick overview over the behavior of all connections.
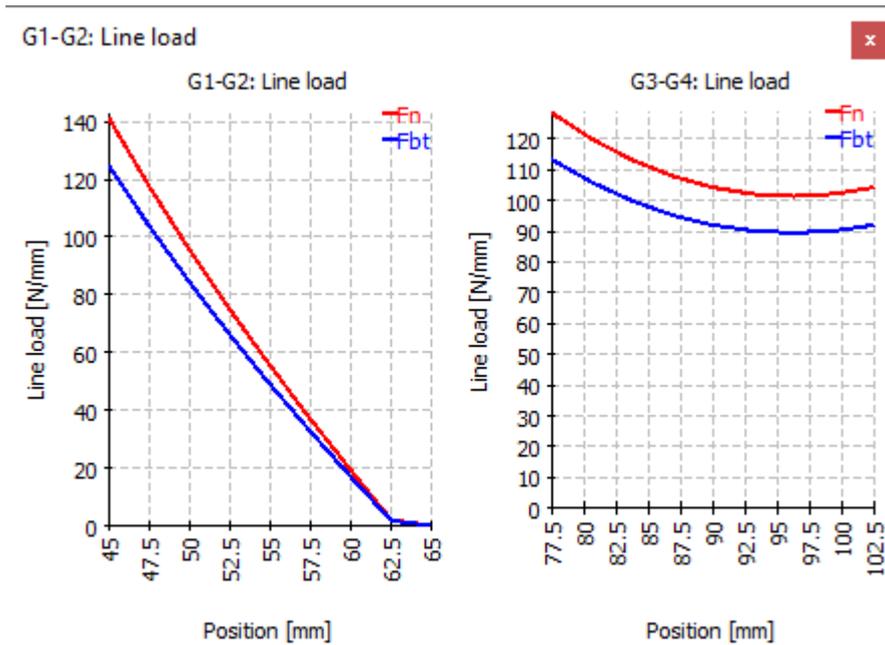
2.6.7 Graphics area

By right-clicking the main graphics area in 2D mode, a context menu called 'Diagram' offers the possibility of overlaying some shaft-related diagrams on the actual geometry of the shaft.
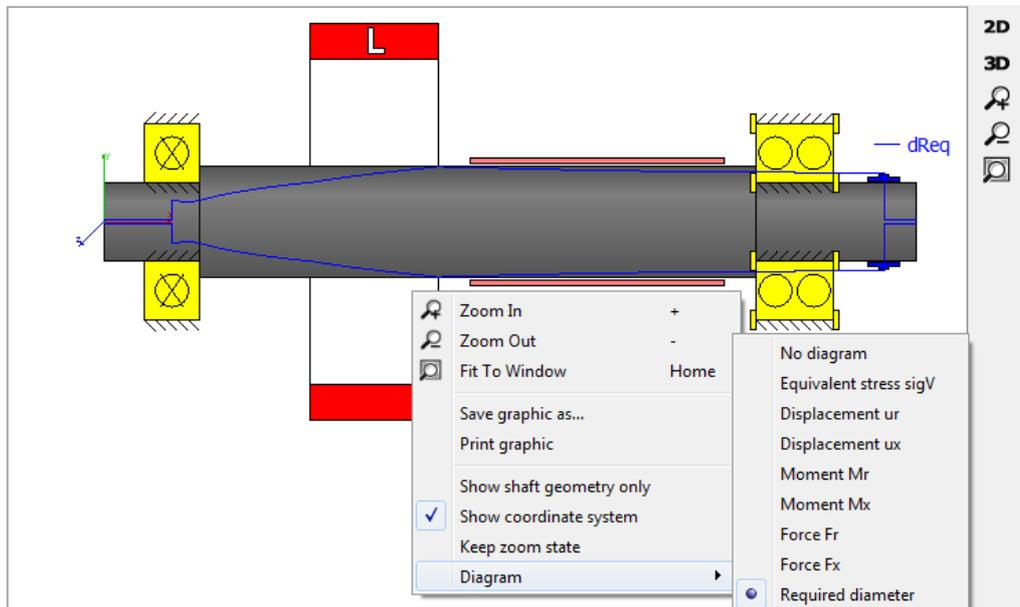
An additional diagram called 'Required diameter' can be overlaid to the 2D geometry of the shaft. It provides a clue as to how the size of the diameter should be in order not to exceed a maximum equivalent stress value (see the image above), which is defined under the tab-page 'Display settings'.

2.6.8 CAD export

Several graphics can be selected in menu Graphics->CAD. In the shaft calculation these are shaft geometry or shaft geometry with simplified or detailed supports and forces. These graphics can be exported as STEP. These STEP files are not to be used for manufacturing as not all geometry details are available in the software and simplifications are made.

The export can be used as a starting point for geometry generation or for comparisons.

2.7 Bibliography

Cowper., G. R. June 1966. The Shear Coefficient in Timoshenko's Beam Theory. *Journal of Applied Mechanics – Transactions of the ASME.* June 1966.

DIN 743-1. Dezember 2012. *Tragfähigkeitsberechnung von Wellen und Achsen - Grundlagen.* s.l. : DIN, Dezember 2012.

DIN 743-2. Dezember 2012. *Tragfähigkeitsberechnung von Wellen und Achsen - Formzahlen und Kerbwirkungszahlen.* s.l. : DIN, Dezember 2012.

DIN 743-3. Dezember 2012. *Tragfähigkeitsberechnung von Wellen und Achsen - Werkstoffe-Festigkeitswerte.* s.l. : DIN, Dezember 2012.

DIN 743-4. Dezember 2012. *Tragfähigkeitsberechnung von Wellen und Achsen - Schädigungsäquivalente Spannungsamplitude.* s.l. : DIN, Dezember 2012.

Hutchinson, J. R. January 2001. Shear Coefficients for Timoshenko Beam Theory. *Journal of Applied Mechanics – Transactions of the ASME.* January 2001.

ISO 21771:2007. September 2007. *Gears – Cylindrical involute gears and gears pairs – Concepts and Geometry.* s.l. : ISO, September 2007.

ISO 7902-1. November 2013. *Hydrodynamic plain journal bearings under steady-state conditions - Calculation procedure - Second edition.* s.l. : ISO, November 2013.

ISO 7902-2. May 1998. *Hydrodynamic plain journal bearings under steady-state conditions - Functions used in the calculation procedure - Second edition.* s.l. : ISO, May 1998.

ISO 7902-3. July 1998. *Hydrodynamic plain journal bearings under steady-state conditions - Permissible operational parameters - Second edition.* s.l. : ISO, July 1998.

MESYS Rolling Bearing Calculation

3   MESYS Rolling Bearing Calculation

This rolling bearing calculation (Version 06/2024) calculates the load distribution, the reference life and the modified reference life according ISO/TS 16281 (DIN 26281) for the following types of bearings:

- Deep groove radial ball bearings

- Double row deep groove ball bearings

- Axial deep groove ball bearings

- Radial angular contact bearings

- Axial angular contact bearings

- Double row radial angular contact bearings

- Double row axial angular contact bearings

- Single row spherical ball bearings

- Double row spherical ball bearings

- Four-point ball bearings considered as radial bearings

- Four-point ball bearings considered as axial bearings

- Three-point bearings with split inner ring

- Three-point bearings with split outer ring

- Duplex ball bearings

- Radial cylindrical roller bearings

- Double row radial cylindrical roller bearings

- Axial cylindrical roller bearings

- Axial cylindrical roller bearings (double row)

- Barrel roller bearings

- Toroidal roller bearings

- Needle bearings

- Tapered roller bearings

- Double row tapered roller bearings

- Axial tapered roller bearings

- Radial spherical roller bearings

- Radial half spherical roller bearings

- Axial spherical roller bearings

- Cross roller bearings considered as radial bearings

- Cross roller bearings considered as axial bearings

- Angular roller bearings considered as radial bearings

- Angular roller bearings considered as axial bearings

Additional bearing types will be added in the future.

The inner geometry of the bearings can be approximated by the software or provided by the user. The operating clearance can be specified. The influence of interference fits, temperature and centrifugal load on the clearance can be taken into account. Centrifugal loads and gyroscopic moments on the rolling elements can be considered for ball bearings only.

In extension of ISO/TS 16281 the influence of reduced material hardness can be considered according NREL guideline DG03. This includes a check of the case core interface.

The lubricant film thickness can be calculated by the software. This is done according to Harris: Rolling bearing analysis.

The loading can be specified as force/moment or displacement/rotation independently for each of the five degrees of freedom. Bearing sets can be defined by using a configuration with multiple rows. This can be used for spindles or multiple row cylindrical roller bearings.

The following results can be found in the report:

- Reference life according ISO/TS 16281

- Modified reference life according ISO/TS 16281

- Basis life and modified life according ISO 281 for comparison

- Load distribution between rolling elements

- Reaction forces/moments and displacements/rotations

- Contact pressure for each contact

- Required shoulder height for contact ellipsis in ball bearings

- Static safety factor

- Maximum subsurface shear stress and stress at case core interface

- Load dependent friction torque for ball bearings based on coulomb friction

- Change to clearance because of interference fit and temperature

- Wear parameters like PV and QV for ball bearings

In addition to the report, the results are shown in several graphics. Report and user interface are both available in metric or US customary units. Supported languages for user interface and report are English, German, French, Spanish, Chinese, Japanese and Korean.

Parameter variations can be made using ranges for several parameters. The results of parameter variations are given as tables or graphics. Proposals are provided for several inputs and conversions like axial clearance in radial clearance are possible.

Elastic deformations of the outer ring can be considered with an extension of the base software. The loading can be specified on several points in radial, axial or tangential direction and the life and load distribution are calculated with a deformed outer ring. The main use of this feature is for track rollers but also deformations in a planetary gear as outer ring can be considered. The elastic outer ring can be considered with following bearing types: Deep groove ball bearing, radial angular contact bearing, four-point ball bearing, three-point ball bearing, radial cylindrical roller bearings and double row tapered roller bearings. Multi row bearings can be considered using bearing configurations.

3.1 Input Parameters

The input parameters are shown on five tab pages.

3.1.1 General

On the first input page in addition to a project name, several settings can be done.

3.1.1.1   Project name and calculation description

The project name and the calculation description are just inputs which are shown in the report header. They can be used to enter information about the purpose of the calculation for documentation.

3.1.1.2   Reliability

As default, the bearing life is calculated for a reliability of 90%. The required reliability can be changed here with limits 90% and <100%. The life modification factor for reliability a_1 is calculated according to this input using the three

parameter Weibull relationship as given as formula in (ISO/TR 1281-2, 2008). The table in ISO 281 (2007) is also calculated according to this formula and has changed to previous versions of the standard.

3.1.1.3   Limit for aISO

The life modification factor for systems approach a_{ISO} takes into account

lubrication properties and fatigue limit of the bearing and it is multiplied to the L10r life to get the modified reference life. According to ISO 281, this factor is limited to a maximum value of 50. In some cases, for example in wind turbines, a smaller maximum limit of 3.8 is required.

The maximum limit for the a_{ISO} factor can be specified here. The input is only an

upper limit for the a_{ISO} factor which is calculated according to (ISO 281, 2007).

3.1.1.4   Friction coefficient

The friction coefficient is used to calculate the friction moment of the bearing assuming coulomb friction in the contacts. In the actual version, this is only available for ball bearings. The friction moment is only considering the load dependent part of the friction; the no load part is not considered.

A proposal by (Harris, et al., 2007) is a value of 0.1 for the friction coefficient.

3.1.1.5   Calculate lubricant film thickness

The calculation of modified reference life requires a viscosity ratio κ. This ratio is calculated by the lubricant viscosity and a reference viscosity ν_1 or it can be calculated from the lubricant film parameter Λ as $\kappa = \Lambda^{1.3}$ according (ISO 281, 2007) and (ISO/TR 1281-2, 2008).

Since the viscosity ratio is calculated using standard settings for surface roughness, loading, pressure viscosity coefficient of the oil and geometric properties, the usage of lubricant film thickness takes into account more parameters of the actual bearing. For the definition of the reference viscosity ν_1 see the derivation in (Baalmann, 1994) or (Heemskerk, 1980).

If this setting is activated the lubricant film thickness is calculated according (Harris, et al., 2007). For ball bearings the minimal film thickness is calculated using equation 4.60 (Harris, et al., 2007) by Hamrock and Dowson. For roller bearings equation 4.57 (Harris, et al., 2007) according Dowson and Higginson is used. The same formulas are used for the calculation of Λ in (ISO/TR 1281-2, 2008). The software uses the minimal film thickness for the calculation of Λ, in contrast to (Heemskerk, 1980) who is using the central film thickness.

In addition, a thermal correction is applied from version 08-2016 on. The thermal correction factor is calculated according to (Koch, 1996) using the formula 4.62 from Wilson and Murch. Different literature provides different correction factors, see (Baly, 2005) or (Gohar, 2001). As temperature the input for the lubricant temperature is used. For high speed the thermal correction factor reduces the film thickness.

This calculation requires the input of surface roughness, oil density and oil pressure viscosity coefficient. The minimal film thickness for all contacts is then used for ball bearings. For roller bearings the minimal film thickness for each section is calculated and used for the calculation of life modification factor for systems approach a_{ISO} for this section.

3.1.1.6 Consider centrifugal force

The centrifugal force can be considered for ball bearings and radial cylindrical roller bearings in this version of the software. It will also be added for other roller bearings in the future.

Centrifugal forces will increase the pressure at the outer race, but decrease the pressure at the inner race. It will lead to different contact angles on inner and outer race and an increased spin to roll ratio.

3.1.1.7 Consider temperature gradient in fits

If this option is activated, the shaft and housing temperatures are available as inputs in addition to inner and outer ring temperatures. This option should only be activated if a temperature gradient between shaft and inner ring or between outer ring and housing should be considered.

If this option is activated be sure to define all four temperatures especially if used from the shaft calculation. The shaft temperature in the shaft calculation will set the shaft temperature for the bearing, but the inner ring temperature still needs to be defined by the user.

3.1.1.8 Oscillating bearing

Some bearings are not fully rotating but they are just oscillating. In this case, the effective number of load cycles is smaller than for a rotating bearing which can be considered by the software. The calculation is done according (Harris, et al., 2009) which is based on (Harris, et al., 2007).

An oscillating angle and an oscillating speed (oscillations per minute) have to be provided in this case. The oscillation angle is defined as the angle between the two end points of oscillation, so it is twice the amplitude.

The rotation speeds n_i and n_e are used for the calculation of centrifugal forces and lubrication film thickness, oscillation speed f_{osc} is used for the calculation of life in h. For the speed used to calculate the lubrication factor (Houpert, 1999) proposes $n = f_{osc} \cdot \theta_{osc}/180°$. The speed has to be entered by the user.

In case of $\Theta_{osc} < \Theta_{crit}/2$ fretting corrosion can occur; the bearing should be rotated by a larger angle from time to time to redistribute the lubricant. A warning is shown in this case.

3.1.1.9 Calculate required hardness depth

The hardness depth is an input for material parameters. If it should be calculated by the software or if it is of no interest for through hardened bearings for example, then set this flag.

If the flag is not set, a check is made if the hardness depth is large enough.

3.1.1.10   Use fatigue strength for hardness depth

If this flag is set, the hardness depth is calculated using the fatigue strength of the core. If it is not set, the yield point is used.

If the calculation is done using a maximal load, the yield point can be used. If it is done using the equivalent load the fatigue strength is recommended.

3.1.1.11   Required subsurface safety

A required safety factor for the subsurface stresses against the yield point can be defined. It will be used for the calculation of the required hardness depth.

3.1.1.12   Selection for clearance

You can select which clearance is taken into account for the calculation. For interference fit and clearance a range of tolerances is given. The calculation can either use the minimum, minimum probable, medium, maximum probable or maximum value of the clearance. For other needs, a user input for the position in the tolerance field between 0 (minimum clearance) and 1 (maximum clearance) can be defined or the operating clearance can be specified directly as a number.

The probable values are calculated assuming a normal distribution within the tolerance field and an independence of tolerances for inner ring fit, outer ring fit and bearing clearance. In addition linearity is assumed which is not true in case a fit changes from interference to clearance within the tolerance range. For comparison probable values could be calculated using the statistical parameter variation 37 .

3.1.1.13   Rolling element temperature

As default, the temperature of the rolling element for calculation of operating clearance is set to the maximum temperature of inner and outer ring. This default setting can be changed to either the inner or outer ring temperature or a user input can be selected, which will be shown on page "Loading" together with the inner and outer ring temperatures.

If the heat is generated by the bearing, the maximum temperature is a reasonable selection. If the heat is generated by other heat sources, the mean temperature could be a better choice.

3.1.1.14   Position of first rolling element

As the circumferential position of the rolling elements within the bearing may affect the results, an option to define the position of the first rolling element by means of the angle is available. If a custom angle is desired, it can be entered by using the ✚-button.
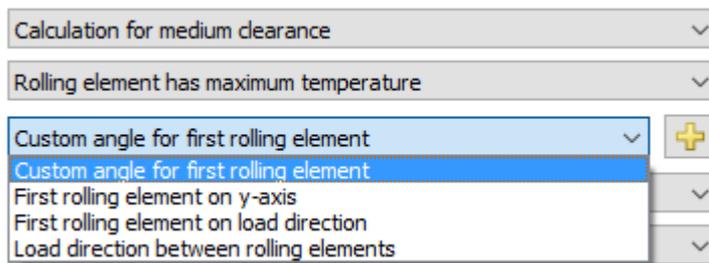
The selection "First rolling element on load direction" uses the radial displacement for definition of the angle. For pure moment load the angle is set to zero. In most cases this settings leads to the maximum contact stress and the lowest life.

The selection "Load direction between rolling elements" uses the radial displacement for definition of the angle and adds half the pitch angle. For pure moment load the angle is set to zero.

The option is not yet available for track roller calculation with elastic outer ring.

3.1.1.15 Options for gyroscopic moment

In case of different directions of forces on inner and outer ring contact, the gyroscopic moment on the ball will affect the load distribution. For high speed ball bearings often "outer race control" is assumed which means that spinning speed at outer race contact is zero. The available options are:

- "Gyroscopic moment is not considered": This is the same behavior than older versions of the software. No gyroscopic moment is considered and spinning speeds are calculated by Coulomb friction.

- "Gyroscopic moment based on outer race control": The ball rotation vector is calculated assuming no spinning at outer race contact. The gyroscopic moment is only causing a friction force at outer race contact.

- "Gyroscopic moment based on outer race control, distributed force": The ball rotation vector is calculated assuming no spinning at outer race contact. The gyroscopic moment is causing a friction force at inner and outer race contact. The friction forces at each contact are proportional to normal force.

- "Gyroscopic moment based on inner race control": The ball rotation vector is calculated assuming no spinning at inner race contact. The gyroscopic moment is only causing a friction force at inner race contact.

- "Gyroscopic moment based on inner race control, distributed force": The ball rotation vector is calculated assuming no spinning at inner race contact. The gyroscopic moment is causing a friction force at inner and

outer race contact. The friction forces at each contact are proportional to normal force.

- "Gyroscopic moment based on mixed control": Dependent on the loading for each ball the rotation vector of the ball is calculated based on inner or outer race control assumption. The gyroscopic moment is causing a friction force at inner and outer race contact. The friction forces at each contact are proportional to normal force.

- "Gyroscopic moment based on friction": On each iteration for ball equilibrium the frictional forces based on Coulomb friction are calculated in the contact ellipses. The ball rotation vector is based on this calculation. The resulting gyroscopic moments is distributed on both contacts proportionally to the normal loads. This options leads to much longer calculation time and it can lead to convergence problems if not all balls are loaded.

For high speed ball bearing usually "outer race control" is assumed, this limits the selection to two options only. For low speed the gyroscopic has usually only very little influence and can be ignored.

For three-point bearings and four-point bearings the options based on outer ring, inner ring or mixed control can only be used on two point contact. In case of more contact points the gyroscopic moment is set to zero and a warning is shown.

3.1.1.16 Life for rolling elements

The bearing life calculation usually only takes into account fatigue life of the bearing races.

Life for rolling elements can optionally be calculated. The rolling element life is calculated analogous to (ISO/TS 16281, 2008) like described by (Correns, 2015). For the combination of life for single rolling elements to the life of the rolling element set, two options for the Weibull exponent can be considered, either e=10/9 for ball bearings and e=9/8 for roller bearings as in (ISO/TS 16281, 2008) and (ISO/TR 1281-1, 2008), or e=1.5 as in (ISO/TR 1281-2, 2008) section 4.2.1 for the reliability factor a_1. This is different to (Correns, 2015) where an exponent of e=10/3 for ball bearings and e=9/2 for roller bearings is used without giving a reason for these exponents.

If this option is activated, the usual bearing life is not changed just the life of the rolling element set L10r_RESet is calculated in addition.

3.1.1.17 Elastic ring expansion

As default, only the elasticity of the contact between rolling element and race is considered. If, for example, an angular contact bearing with clearance between

outer ring and housing is under axial pretension, the outer ring can expand radially. This will reduce the pretension.

In the current version, two options for consideration of elastic expansion of bearing rings are available. Both options are based on the assumption of a thick-walled ring as in the calculation of interference fits.

Either the minimal or the mean radial force in the load distribution is converted into a constant radial pressure which is then taken into account as in a calculation for interference fits. Both rings inner/outer ring and shaft/housing are considered in the calculation of the fit. This calculation approach is only valid if the variation of radial forces is small. So the axial load should normally be larger than the radial load.

The ring diameter is taken as Dpw±Dw as default, so a stiffening effect of shoulders is not taken into account. Using the ➕-button behind the [bearing type]157 instead the use of an equivalent cross section can be activated. Then the ring diameter is defined so that the cross section is equivalent to the cross section of the bearing ring. Both options for the ring diameter can be compared with FEA results using the graphic ["Radial expansion of races"]236.

An additional factor for the ring stiffness of the inner and outer ring can be defined using the ➕-button behind the selection box. This factor is applied to the surface used for conversion of the rolling element force to a pressure: p = F/(fcr*A).

Enter parameters		
Factor for stiffness for elastic expansion (inner ring) fcri	1	
Factor for stiffness for elastic expansion (outer ring) fcre	1	
	OK	Cancel
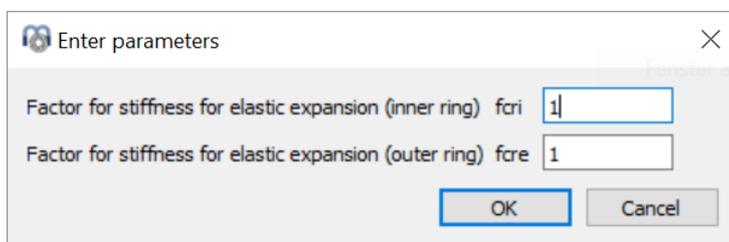
The elastic ring expansion is set to zero in case deformations from the shaft calculation are considered as the ring expansion should not be considered twice.

3.1.1.18 Use load spectrum

A load spectrum can be used instead of a single load case. This option can be activated here. For each load case a full calculation using all factors is made. The resulting life is calculated using the life of each element as $L_{10rh} = 1 \Big/ \sum \dfrac{h_i}{L_{10rh,i}}$.

Result graphics are only shown in the report for the selected result element of the load spectrum. However, the result graphics corresponding to any element number (load case) of the load spectrum can be shown under the menu 'Graphics'->'Load spectrum', as we can see in this picture:
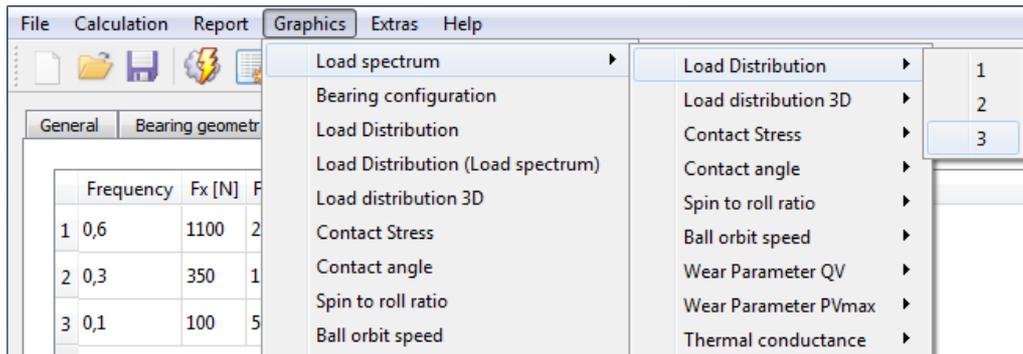
All other graphics available under the menu 'Graphics are only valid for the selected element.

Additionally, under 'Graphics'->'Load distribution (Load spectrum)', all the load distribution cases are superimposed on the chart at the same time. This is also possible for the roller graphics.

Please note that more intermediate results are printed in the report if a calculation is done for a single load case.

3.1.1.19   Calculate modified life

If this flag is set, the modified reference life L_{nmrh} will be calculated. You can clear the flag if no information about lubrication is known or lubrication should not be considered in the calculation.

3.1.1.20   Use extended method for pressure distribution

In (ISO/TS 16281, 2008) a simplified approach to calculate edge stresses in roller bearings is used and extended method according other literature are recommended. If this option is set the pressure distribution is calculated according to (de Mul, et al., 1986), which is the newest literature in the proposals of the standard.

This extended method is available for all supported types of roller bearings.

3.1.1.21   Calculate static safety factor based on stress

If this options is set the static safety SF is calculated based on maximal contact stress. For ball bearings $SF = \left(\dfrac{p_{perm}}{p_{max}}\right)^3$ is used, for roller bearings $SF = \left(\dfrac{p_{perm}}{p_{max}}\right)^2$ is used, so that the safety factor is proportional to the load. The permissible stress is based on (ISO 76, 2006).

If the option is not set, the static safety factor is calculated based on maximum rolling element load and Q_{max} in (ISO/TR 10657, 1991).

For ball bearings both options usually show the same results, but for roller bearings edge stresses are only considered if the static safety factor is based on stresses.

3.1.2   Bearing geometry

3.1.2.1   Bearing types

Different types of ball and roller bearings can be calculated using the software. In addition to the calculation of a single bearing, a configuration of several rows can be defined using "Bearing configuration".

By using the ⊕-button behind the bearing type, some options for the selected bearing type can be defined.

- For all bearings, it can be selected that inner ring and shaft or outer ring and housing are identical. This affects the material input and the input of tolerances.

- The option "Use ring diameter for equivalent cross section for calculation of fits" will lead to a larger ring thickness of inner and outer ring for the calculation of fits. Both variants can be compared using the graphic "Radial expansion of races" 236. By using the 🟡-button behind the option, the race diameters can be overridden.

- If the option "Calculate load capacity for hybrid bearings automatically" is active, then the next option is automatically set for hybrid bearing (Er >= 300 GPa, Ei, Ee < 260GPa)

- The option "Calculate load capacity for hybrid bearings" will use (ISO 20056-1, 2017) and (ISO 20056-2, 2017) for the calculation of load capacities which leads to an increased static load capacity because of higher permissible stresses. The Youngs modulus of the rolling elements is required to be larger than 300GPa for the increased static permissible stress.

- If the option "Calculate X/Y-factors based on free contact angle", then the effective free contact angle will be used to calculate the X/Y-factors in ISO 281 instead of the nominal contact angle. This will mainly affect deep groove ball bearings, where the free contact angle is increased with clearance.

- For ball bearings the "Permissible ellipse length ratio" affects a warning about truncation and an output for a permissible axial force in the report.

- For ball bearings the "Lower stress limit for truncation" ignores truncation for all contacts with a contact stress less than this limit.

- For ball bearings the option "Use ISO conformity in case of small conformity" will use fi = fe = 0.52 for calculation of load ratings in case fi < 0.52 and fe < 0.53 for radial ball bearings and fi = fe = 0.535 if fi < 0.54 and fe < 0.54. These are the limits were tabulated values for fc in ISO 281 may be used.

- For ball bearings, a limit for the conformity used in calculation of load capacity can be defined. The load capacity is calculated by the maximum value of the given limit and the actual geometry input. This input is only available in the above option regarding ISO conformity is not set.

- For deep groove ball bearings a tolerance for the conformity can be defined. This will lead to additional outputs in the Tolerances Report 223.

- If a friction coefficient for fitting is defined, an axial mounting force is shown in the tolerance report.

- The reduction of the load capacities because of hardness can be calculated according to Harris or Schaeffler. The option according to Schaeffler leads to larger load capacities for very low hardness.

In the current version, the following bearing types are supported:

3.1.2.1.1 Deep groove radial ball bearing
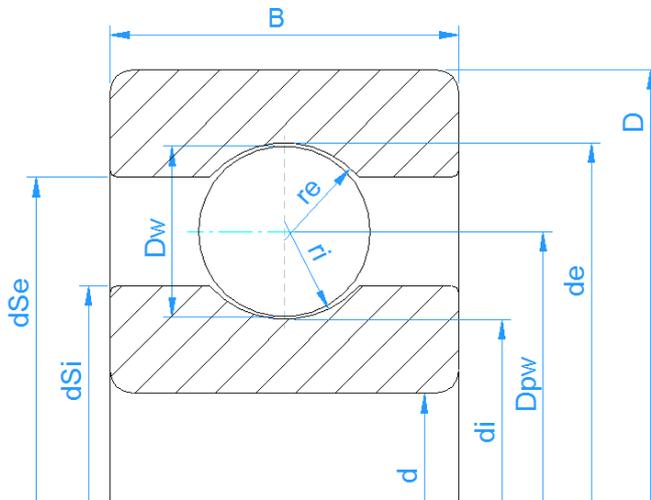
Deep groove radial ball bearings are cheap standard bearings. Both radial and axial loads can be transmitted. The nominal contact angle is 0° which will increase under axial load depending on the radial clearance in the bearing. The free contact angle $\alpha 0$ is shown in report and result overview.

The geometry is described with number and diameter of balls, pitch diameter, conformity of inner and outer race and the diametral clearance. The number of balls in the bearing is restricted, to be able to assemble the bearing.

The conformity is the input parameter defining the radius of the races. (ri = fi*Dw, re = fe*De)

Usually the conformity is fi = fe = 0.52 for deep groove radial ball bearings.

The diametral clearance is defined as Pd = de- di -2*Dw.

The ✚-button behind the bearing selection also offers some other settings depending on the bearing type. For deep groove radial ball bearings, it can be selected if a "filling slot" bearings is used which influences the calculation of the load capacity with factor bm in (ISO 281, 2007).

The permissible ellipsis length ratio controls the warning about cut off of contact ellipsis. A value below 100% would allow some amount of cut off, a value larger 100% adds an additional distance. This permissible value is used for the calculation of permissible axial load limit given in the report too.

3.1.2.1.2 Double row deep groove ball bearing

For double row deep groove ball bearings, a distance between rows 'δR' has to be defined in addition to the parameters of the single row deep groove ball bearing.
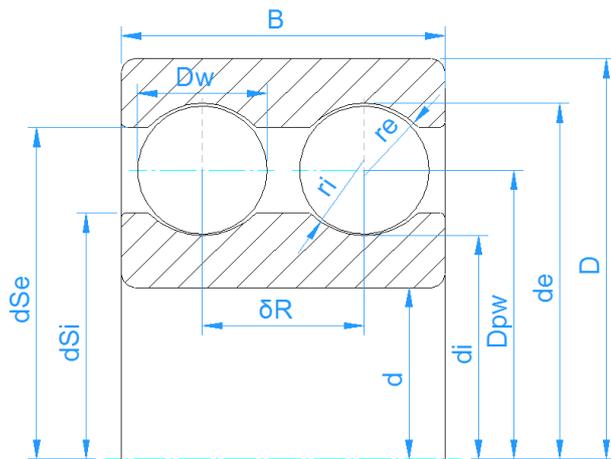
3.1.2.1.3 Axial deep groove ball bearings

For axial deep groove ball bearings, the nominal contact angle is 90°. The default conformity is fi = fe = 0.535.
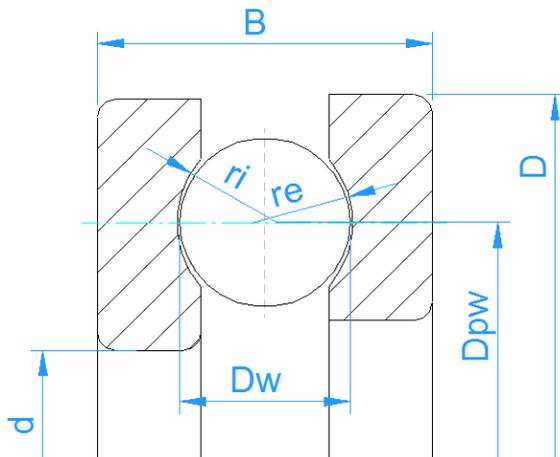
In the bearing type options dialog, it can be defined if the left or the right ring shall be considered as inner ring. This also changes the sign of the axial force which is given for the inner ring.
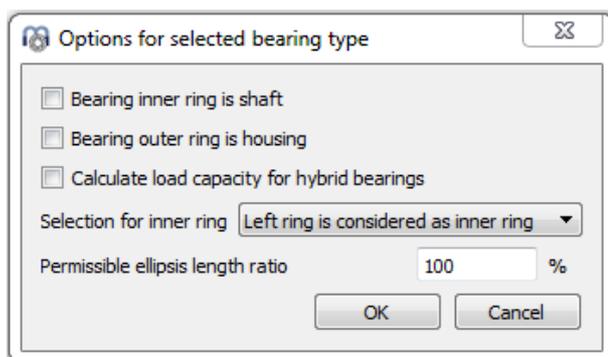
No tolerances are considered for axial deep groove ball bearings.

3.1.2.1.4 Radial angular contact bearing

The angular contact bearing is similar to the deep groove ball bearing, but the contact angle is larger. Standard bearings have contact angle of 15°, 25° or 40°.
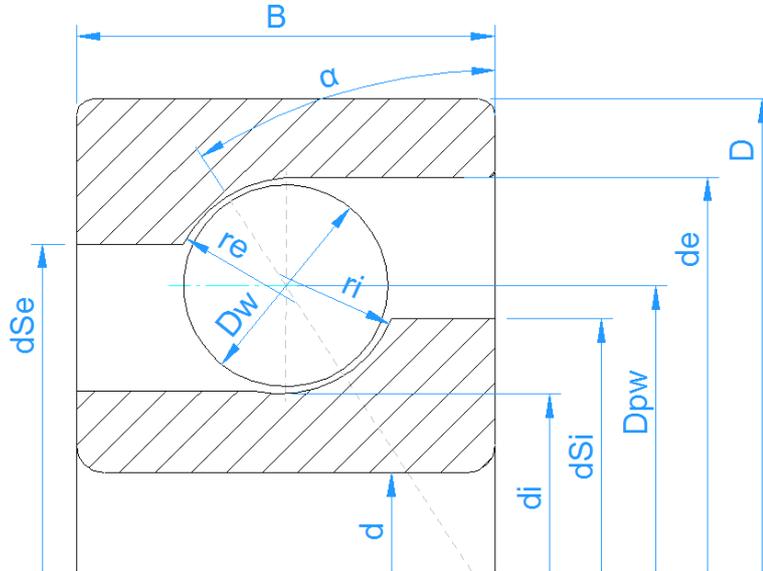
Double row angular contact bearings can be either defined using a single row bearing and a configuration of two bearings or a double row angular contact bearings can be defined directly. This allows the input of radial clearance of the two row bearing. In both cases life modification factor a_{ISO} is calculated for both rows separately.

The direction of contact angle can be defined using the ➕-button behind the contact angle. Additionally, the input of direction of contact is available in the bearing type options.

The ☰-button for axial clearance allows to calculate the axial clearance for given pretension or for given radial clearance in case of double row angular contact bearing.

3.1.2.1.5 Double row radial angular contact bearing

For double row angular contact bearings, the configuration of contact angles can be defined in the options dialog.
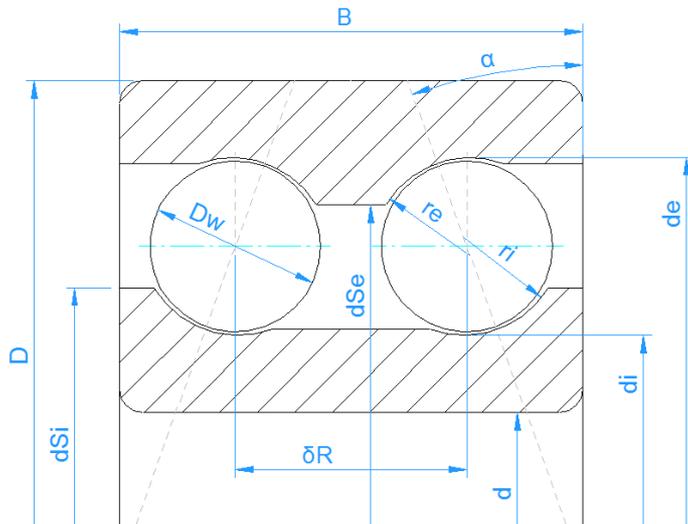
The ▤-button for axial clearance allows to calculate the axial clearance for given pretension or for given radial clearance. The clearance can be generated in three different ways. See at four-point bearing below.
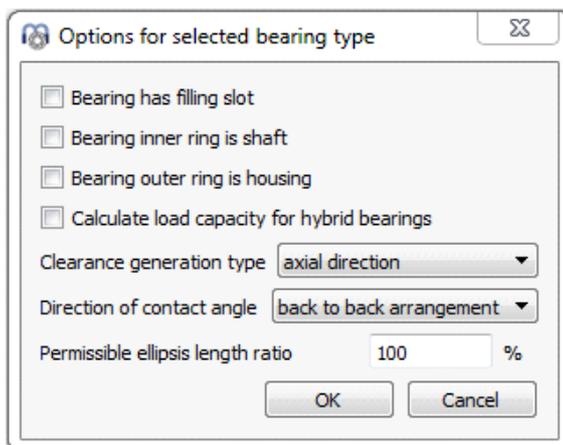
In addition to the inputs for the single row angular contact bearing, the distance between rows has to be entered.

3.1.2.1.6 Axial angular contact bearing
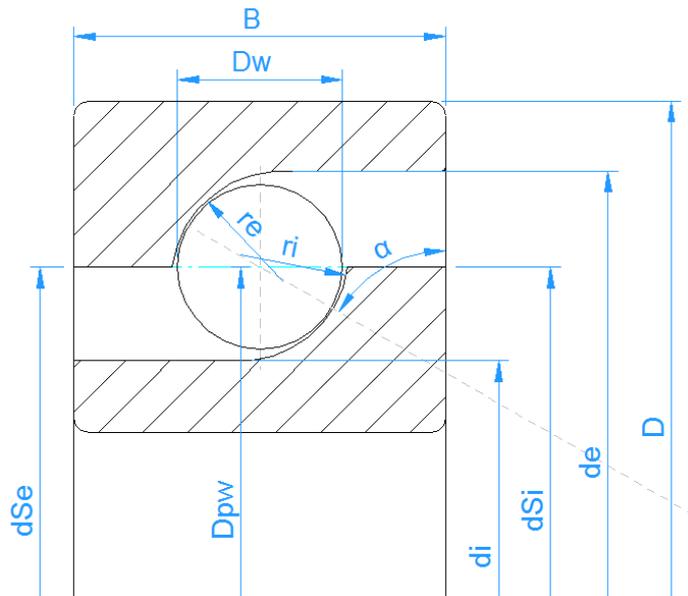
The description of geometry for axial angular contact bearings is the same as for radial angular contact bearings. The only difference in geometry is a value of 0.535 for the conformity, instead of the 0.52 used for radial bearings.

The results for load distribution are the same for the selection of axial or radial angular contact bearings, but for axial bearings additional reduction factors are considered in the calculation of load capacity. Therefore, the resulting life is smaller if the bearing is calculated as axial bearing.

Standard axial angular contact bearings have a contact angle of 60°.

3.1.2.1.7 Double row axial angular contact bearing
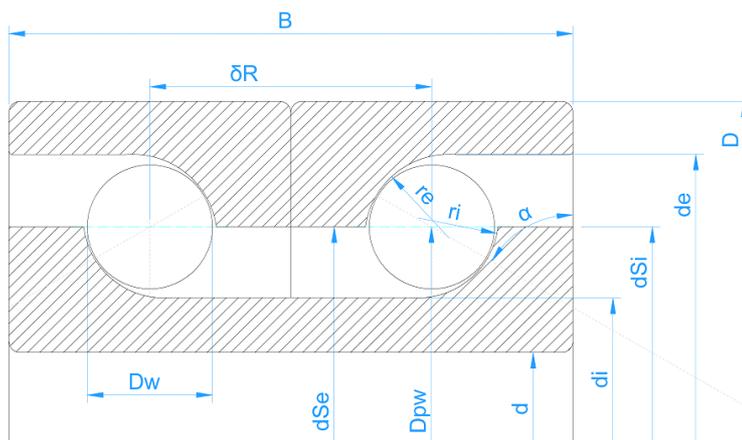
Just as for the radial case, double row axial angular contact bearings can be either defined using a single row bearing and a configuration of two bearings or a double row angular contact bearings can be defined directly.

3.1.2.1.8 Four-point bearing considered as radial bearing
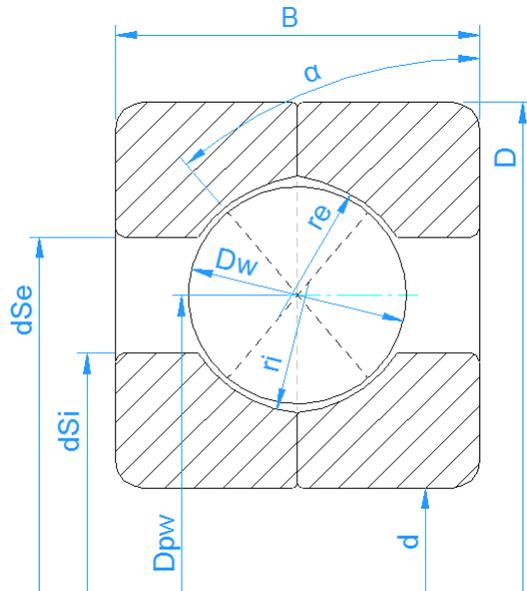
This can be used for standard QJ bearings with 35° contact angle. Load capacity for a four-point bearing is calculated as a double row bearing, as usually done in bearing catalogues. The results are reported for both rows. In case of two contact points, there is one contact on the inner ring of one row and on the outer ring of the other row. All possible four contacts are considered in life calculation.

Using the ⊕-button behind the bearing type a "filling slot" can be defined and the method clearance is generated for the bearing. There are three ways to generate clearance in the software:

1. Centers of curvature are moved in axial direction. This leads to a decreased contact angle for radial load. It corresponds to a four-point considered as angular contact bearing.

2. Centers of curvature are moved in radial direction. This leads to an increased contact angle for axial load.

3. Centers of curvature are moved along the nominal contact angle.
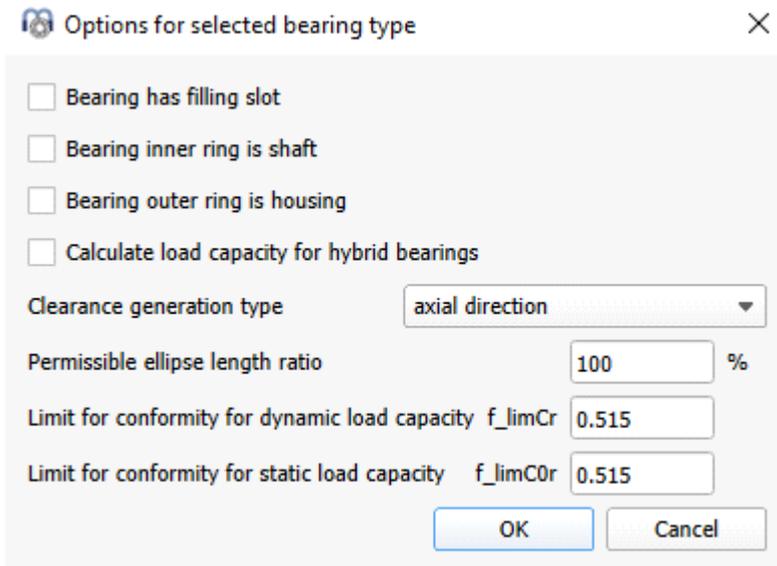
3.1.2.1.9 Four point bearings considered as axial bearing

This selection can be used for slewing rings as done in (Harris, et al., 2009). Because of additional reduction factors for axial ball bearings, the calculated life will be smaller than considered as a radial bearing. The load distribution is the same as for the four-point bearing considered as radial bearing.
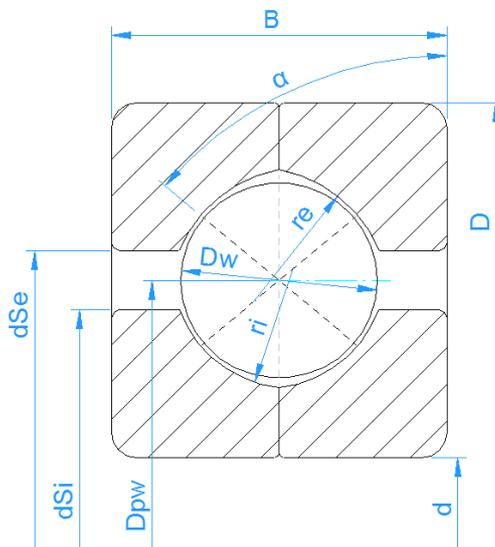
3.1.2.1.10 Three point bearings

Three point bearings with split inner ring or split outer ring can be considered. The split ring has two radii like in a four-point bearing and the second ring has a full radius. Therefore there can be two or three contact points.

In contrast to the other bearing types, the input for the contact angle is the contact angle on the split ring under radial contact. The input field is therefore shown as shim angle. The operating contact angle will depend on radial clearance. The radial clearance can be calculated from the required free contact angle.

The free contact angle is calculated based on two point contact. If the resulting free contact angle is less than the shim angle, there will be three point contact under low axial load.

The load capacity is calculated based on a single row radial angular contact ball bearing using the free contact angle. Because the free contact angle is used, the load capacity is dependent on the nominal clearance.

3.1.2.1.11 Self-aligning ball bearing

Spherical ball bearings can be selected as single or double row bearings. A single row bearing has a contact angle of zero.
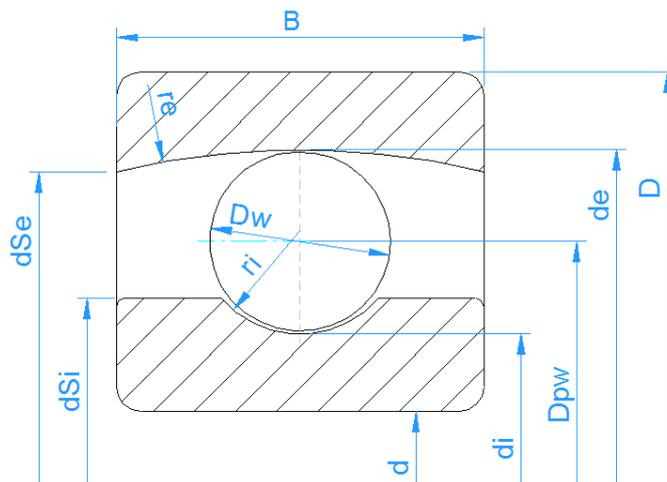
3.1.2.1.12 Self-aligning ball bearing (double row)

For double row bearings the distance of rows is defined by the contact angle. The distance between rows is then

$$\delta_R = D_{pw} \cdot \tan \alpha$$

For spherical ball bearings, the conformity of the outer race is defined as

$$f_e = \frac{r_e}{D_{pw}/\cos\alpha + D_w}$$

So it is the ratio of radius and outer race diameter which is 0.5 as standard.

The shoulder diameter of the outer ring is automatically limited by the width and the radius of the outer race. Still a larger value as the limit could be entered by the user. The life modification factor a_{ISO} is calculated for both rows separately.
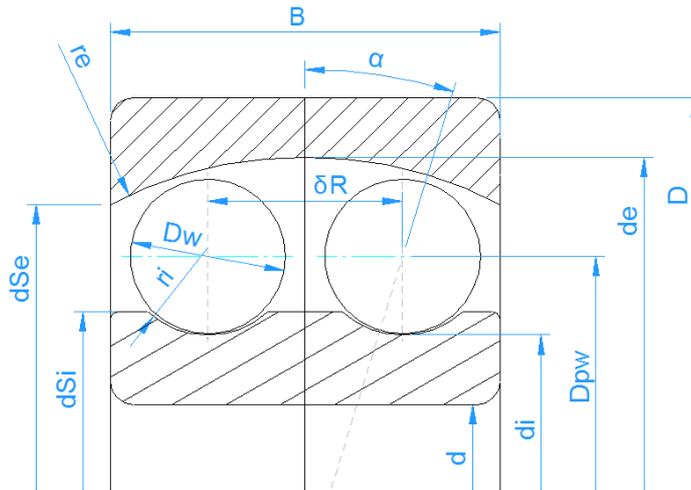
3.1.2.1.13 Duplex bearings

Two deep groove ball bearings can be calculated as a set by selecting "Duplex bearings". The same could be done by using a single deep groove ball bearing and a configuration of two bearings but the input is more flexible using "Duplex bearings".
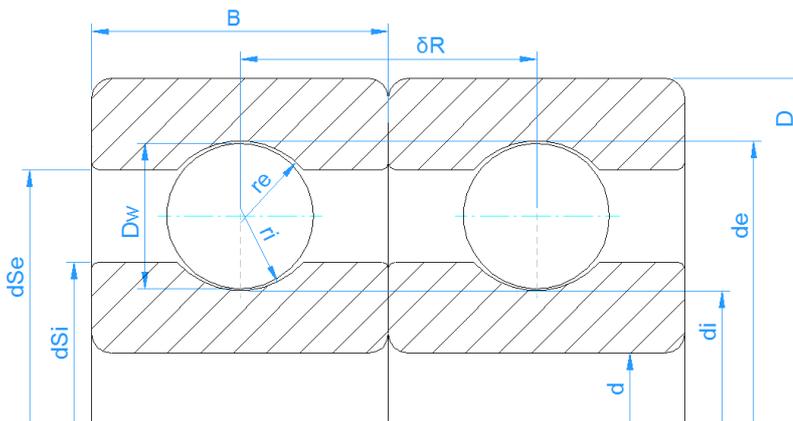
The geometry data is defined for a single deep groove ball bearing, additionally the distance between the two rows is an input value. The input of the diametral clearance is for the single bearing only.

The clearance of the configuration can then be changed by an axial offset δ_{CC} between inner and outer ring of each bearing. Using the ☰-button it can be calculated from a given axial clearance, radial clearance or pretension force for the bearing configuration.

The free contact angle of the bearing α_0 and α_{0eff} are shown in the report and in the results overview.

Options for the bearing allow selecting of face-to-face or back-to-back configuration. This has an influence of the load distribution on moment loads or a tilted bearing.
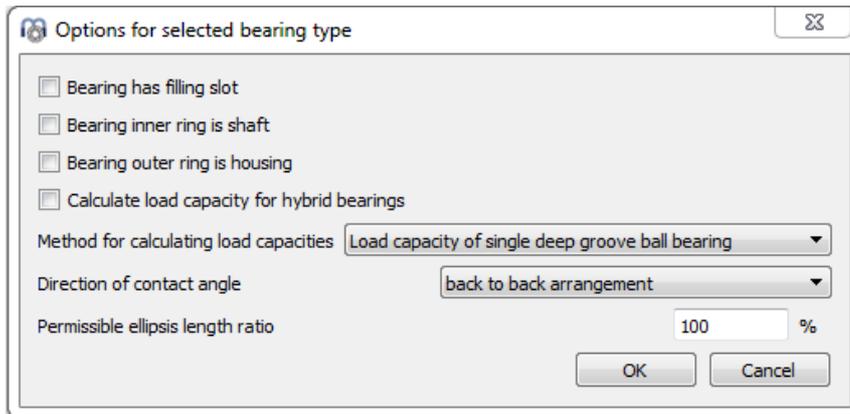
There are four options for the calculation of load capacities:

1. *Load capacity of single deep groove ball bearing*: The input values are for a single bearing only. These are the values which are given in the documentation of the single bearing. In this case, the load capacities of the pair are shown as C_{sys} in the report.

2. *Load capacity of paired deep groove ball bearings*: Here the calculation is done using load capacities for two paired deep groove ball bearings using factors of $2^{0.7}$ for dynamic and 2 for static load capacity.

3. *Load capacity of double row deep groove ball bearing*: Here the load capacities are calculated using the factors for a double row bearing. The dynamic load capacity is smaller than for the second case.

4. *Load capacity of double row angular contact ball bearing*: Here the load capacity is calculated using the free contact angle of the bearings.

If thermal effects are considered, the axial offset δ_{CC} will be modified during the calculation to take the different axial elongation of inner and outer ring into account. The results are the same as if using a single radial deep groove ball bearing with a configuration of two bearings.

3.1.2.1.14 Radial cylindrical roller bearings
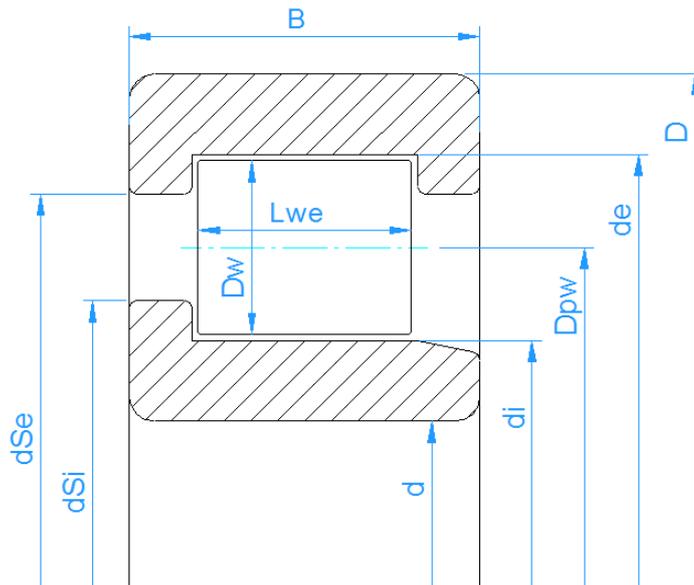
The radial cylindrical roller bearing is a bearing providing a high load capacity for radial loads but it does not support high axial loads or misalignment of inner and outer ring.

In addition to the parameters used for ball bearings, the effective length of the roller L_{we} is a required input parameter. The effective length is a little smaller than the length of the roller because of radii at the end of the roller. The contact angle is always zero for cylindrical roller bearings.

The roller profile is considered as defined by ISO/TS 16281 as default. If the extended calculation for pressure distribution is selected, an input of profile for races and roller is possible using the ✚-button behind the roller effective length. Several options are available including reading the profile from a file.

The axial load is considered as shown by (Harris, et al., 2007). The axial load brings a tilting moment to the roller and an unsymmetrical load distribution on the races occurs. The axial forces are considered at the half height of the shoulders. If 20% is entered for the height of the shoulder, the axial force is acting at 10% of roller diameter.
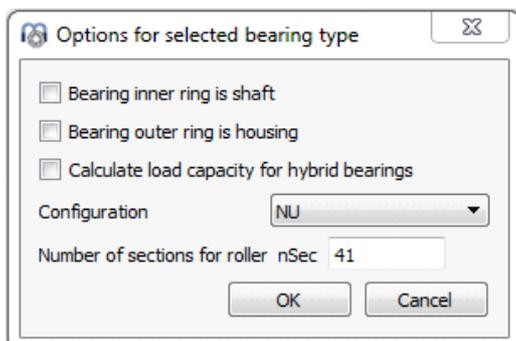
The type of cylindrical roller bearing NU, NJ, NUP … can be selected in the options dialog of the bearing.

For cylindrical roller bearings which support axial loads, a radial and axial clearance can be specified. It is important to enter a value for axial clearance if tilting occurs. For NUP type, the axial clearance has influence on the reaction moment. The Y-factor for ISO 281 life calculation is not specified in the standard, a value of Y=0.6 with e=0.2 is used like in some bearing catalogs.

For NUP type, the axial clearance is measured between left and right positions of the rings like for deep groove ball bearings. For directional bearings like NJ, the axial clearance is between reference position and one side like for angular contact ball bearings or taper roller bearings.

For types which do not support axial loads like NU, the axial displacement 'ux' has to be selected as input instead of 'Fx'.

The number of sections for the calculation of load distribution can be changed in bearing options, too. The minimum is 31 sections. A larger value reduces the edge stresses by the approximation function in ISO/TS 16281 if the extended method for pressure distribution is not active.

3.1.2.1.15 Radial cylindrical roller bearings (double row)
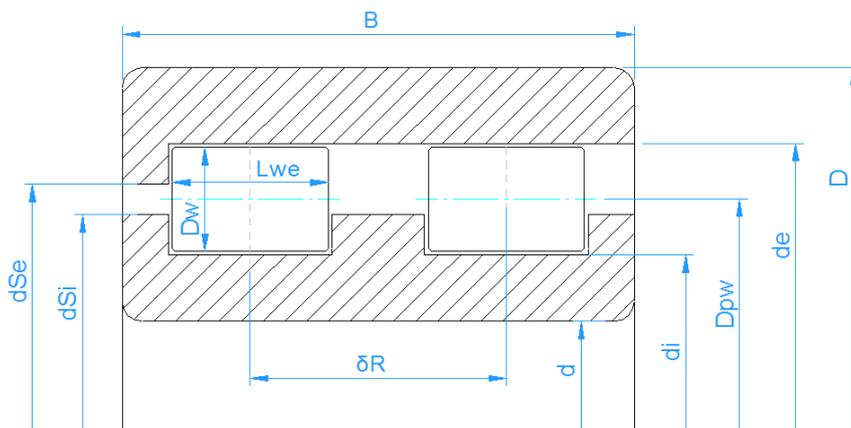
In addition to the inputs for a single row bearing, the distance between the row centers has to be defined. Generally, the same results are obtained as if using a single row bearing with a bearing configuration of two rows; however, the results will differ if the configuration shows different positions of shoulders under axial load. An additional difference is the load capacity which is shown for the double row bearing.

The different bearing types can be selected in the options dialog as well.

The Y-factor for ISO 281 life calculation is not specified in the standard, a value of Y=0.6 with e=0.2 is used like in some bearing catalogs.

3.1.2.1.16 Needle bearings
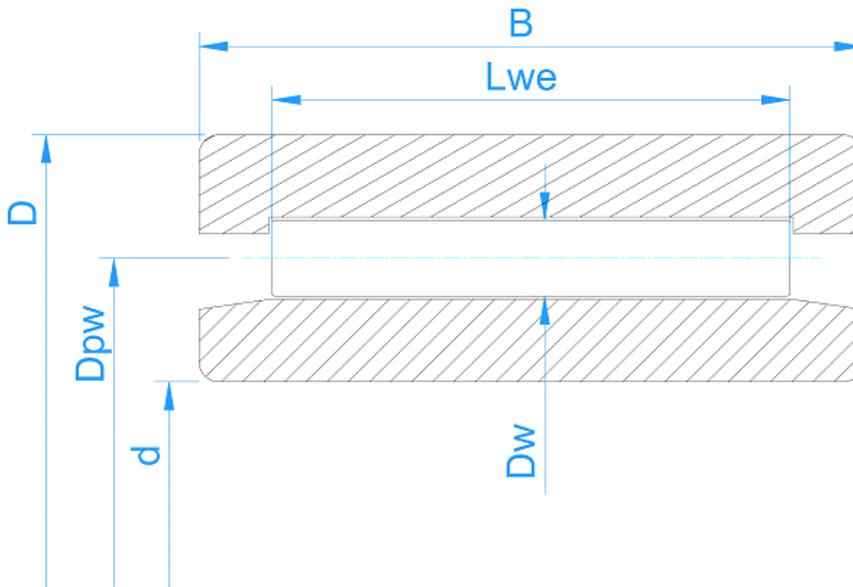
Needle bearings can be calculated using the selection 'Cylindrical roller bearing', too. The type 'needle bearing' was only added so the types can be separated in the database.

Needle bearings do not support axial forces. Therefore 'ux' has to be selected as input instead of 'Fx'. The axial clearance is not available as an input.

For drawn cup needle roller bearings the b_m-factor for the dynamic load capacity should be set to 1.0 instead of 1.1 in case of machined bearing rings. The type can be selected under options for the bearing type using the ✚-button behind the bearing type.

3.1.2.1.17 Axial cylindrical roller bearings

Axial cylindrical roller bearings have a contact angle of 90°. They only allow axial forces and bending moments. No radial forces can be used. Therefore, uy, uz have to be entered instead of Fy, Fz.
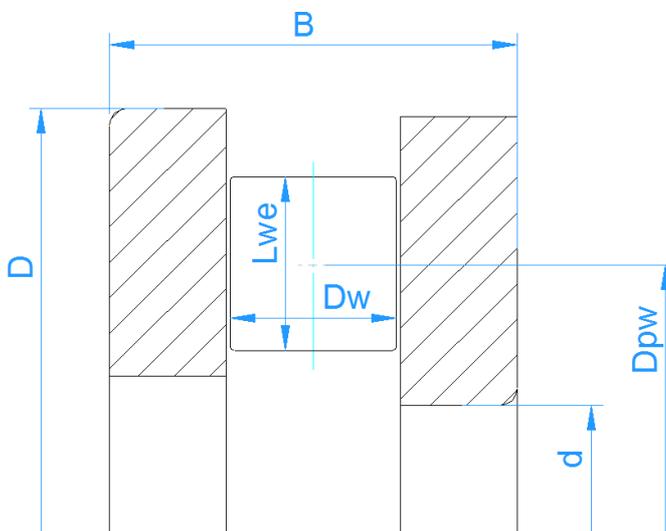
The options dialog (➕ -button behind bearing type) allows to specify if the left or the right ring shall be considered the "inner ring" on which loading applies. The default setting "Left ring is considered as inner ring" results in a positive axial load to be entered.
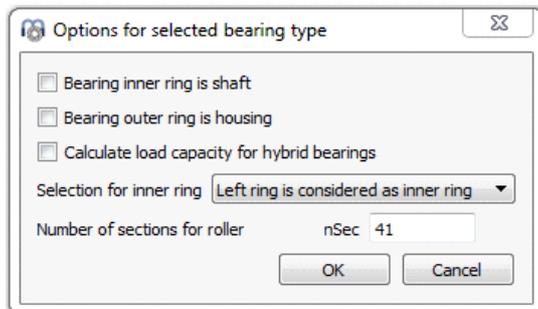
An axial clearance can be entered; its only influence is an offset to the axial displacement.

No tolerances are considered for axial cylindrical roller bearings.

3.1.2.1.18 Axial cylindrical roller bearing (double row)

Like single row axial cylindrical roller bearings, the double row bearings have a contact angle of 90°. They only allow axial forces and bending moments. No radial forces can be used. Therefore, uy, uz have to be entered instead of Fy, Fz.

The mean diameter has to be set for Dpw. Local load capacities are calculated based on Dpw + Lw and Dpw - Lw. Both rows have the same roller geometries.

3.1.2.1.19 Radial tapered roller bearings

Tapered roller bearings use a conical roller instead of a cylindrical roller. The input roller diameter is given for the middle of the roller and also the pitch diameter D_{pw} is defined for the middle of the rollers. The clearance is defined as axial clearance.
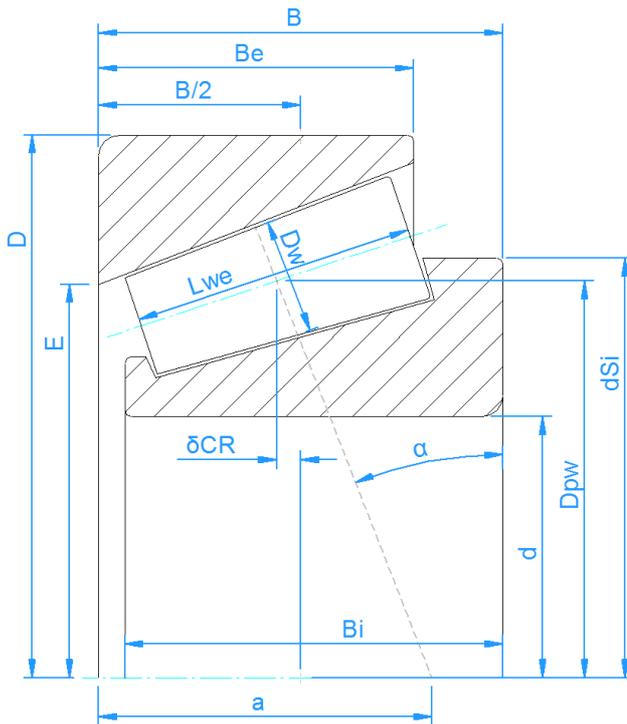
The contact angle should be the direction of the load. Therefore, the angle of the outer ring, the cup, has to be specified for the contact angle. The angles for the roller and the cone are then calculated so that all cones intersect on no load condition.

If the axial force is too small, a calculation error can occur since the bearing will fall apart. You have to enter an axial force large enough or to specify an axial displacement instead.

The direction of contact angle can be defined using the ✚-button behind contact angle or bearing type.

The diameter of the shoulder of the inner ring can be defined. The force is assumed to be at the medium height of the shoulder.
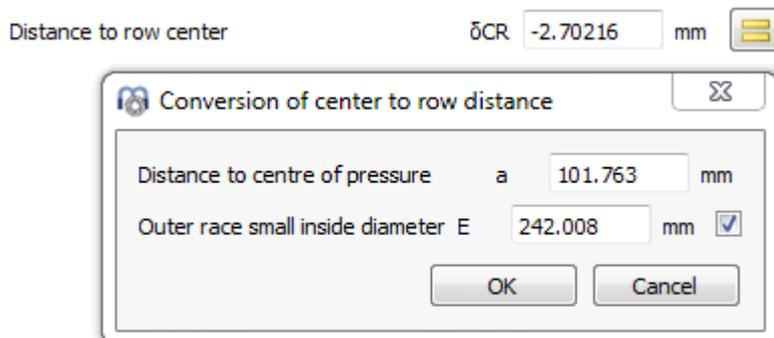
The distance between center of bearing and center of roller 'δCR' can be defined in two different ways by using the ▭-button. It can be converted either from the

distance to the center of pressure 'a', or from the outer race small inside diameter 'E' by setting the corresponding flag. Note that defining 'E' will in turn modify 'Dpw' according to their geometric relations.

3.1.2.1.20 Radial tapered roller bearing (double row)

As for other double row bearings types, the arrangement of contact angles can be defined in the options dialog. Unlike the single row radial tapered roller bearing, the distance between rows measured from the center of the corresponding rollers has to be entered. The same results are calculated as if using a single row bearing with a bearing configuration of two rows. The only difference is the load capacity which is shown for the double row bearing. As in the other bearing types, the clearance can be also defined as radial clearance.
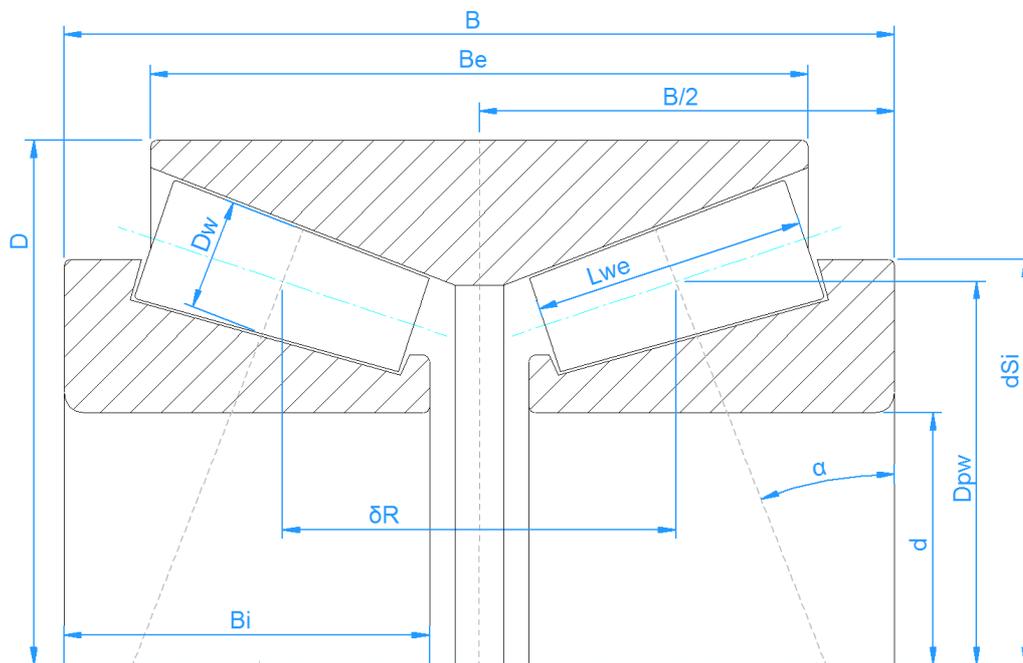
3.1.2.1.21 Axial tapered roller bearing
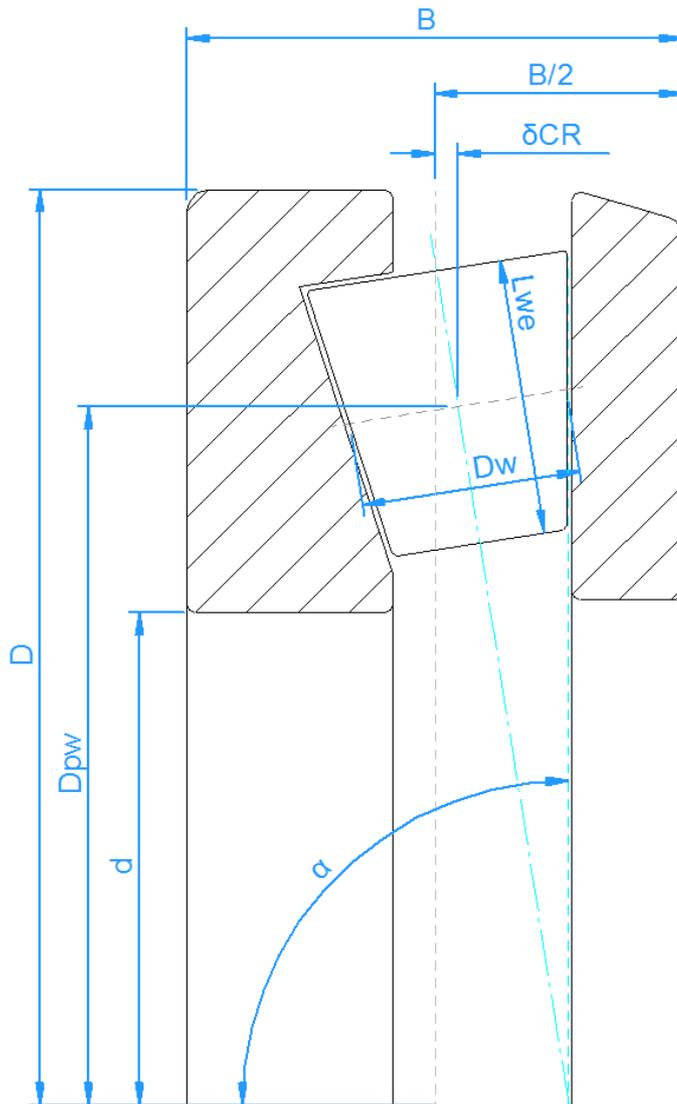
The inputs which define a tapered roller thrust bearing are a bit simpler than for the radial case of tapered roller bearings, since the distance between center of bearing and center of roller 'δCR' is directly entered by the user. The clearance is also defined as axial clearance.

As shown in the picture, a contact angle ranging from 90° to 0° can be set, but it should always be between 90° and 45° in order to suit its axial load-carrying design.

No radial forces can be accommodated, so the radial displacements 'uy' and 'uz' under loading have to be set to zero.

The direction of contact angle can be defined using the ⊕-button behind contact angle or bearing type.

The height of the shoulder of the inner diameter can be defined through the factor 'fSi' (%) in terms of percentage of the roller diameter. The force is assumed to be located at the medium height of the shoulder.

No tolerances are considered for axial tapered roller bearings.

3.1.2.1.22 Barrel roller bearings
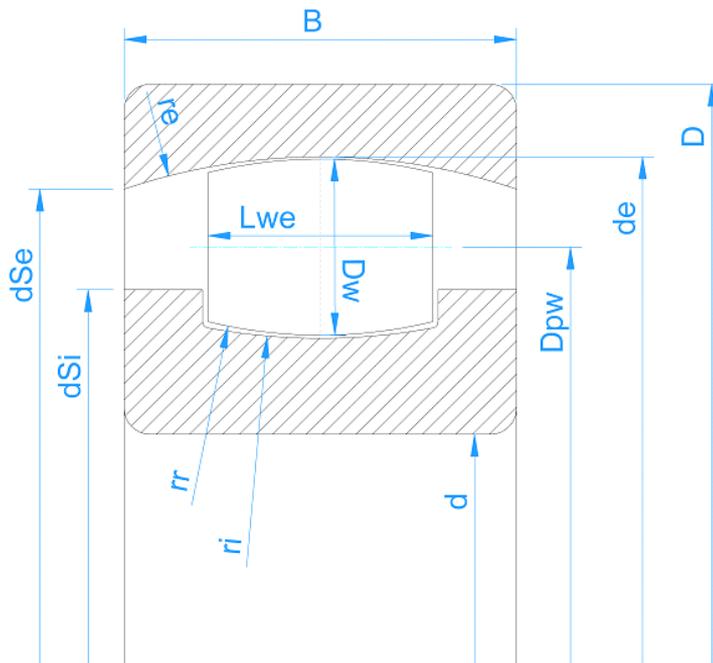
Barrel roller bearing are single row spherical roller bearings. As for double row spherical roller bearings, the outer race is a sphere and the bearing does not support moment loads. Therefore, the tilting angle has to be defined instead of moment load.

The radius of inner race, outer race and roller can be specified as ratio to the nominal diameter of the outer race: $d_e = D_{pw} + D_w$. Default parameters are: fe = 0.5; fi = 0.5; fr = 0.485. If the radius for the outer race is chosen differently, the bearing cannot rotate freely any more.

The dynamic load capacity is calculated using bm=1 as this seems to be the case in catalog data of major manufacturers.

3.1.2.1.23 Toroidal roller bearing

Toroidal roller bearings have an outer race radius which is larger than for a sphere. The don't have shoulders, therefore the roller can float axially.

The radius of inner race, outer race and roller can be specified as ratio to the nominal diameter of the outer race: $d_e = D_{pw} + D_w$.

The dynamic load capacity is calculated using bm=1.15 like for spherical roller bearings as this seems to be the case in catalog data of major manufacturers.

3.1.2.1.24 Spherical roller bearing
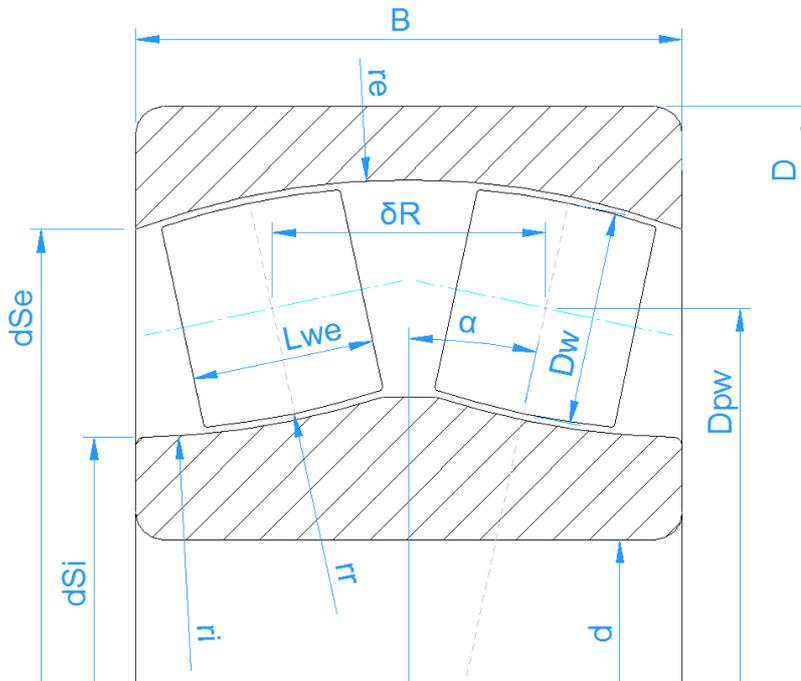
As the outer race of a spherical roller bearing is a sphere the bearing does not support moment loads. Therefore, the tilting angle has to be defined instead of a moment load. The distance between both rows is determined the contact angle and the pitch diameter. In contrast to the other bearing types, the nominal diametral clearance is only applied to the inner race.

The radius of inner race, outer race and roller can be specified as ratio to the nominal diameter of the outer race: $d_e = D_{pw}/cos(\alpha) + D_w$. Default parameters are: fe = 0.5; fi = 0.5; fr = 0.485. If the radius for the outer race is chosen differently, the bearing cannot tilt freely anymore.

3.1.2.1.25 Radial half spherical roller bearing

This is half a spherical roller bearing with only a single row. It can be used to assemble asymmetric spherical roller bearings using the shaft calculation.

Like for spherical roller bearings the radius of inner race, outer race and roller can be specified as ratio to the nominal diameter of the outer race: $d_e = D_{pw}/cos(\alpha) + D_w$. Default parameters are: fe = 0.5; fi = 0.5; fr = 0.485. If the radius for the outer race is chosen differently, the bearing cannot tilt freely anymore.

The defined the axial position of the roller. It should be defined to that the center of curvature is on one side of the bearing. The distance to the center of pressure 'a' should be equal to the bearing width.

3.1.2.1.26 Axial spherical roller bearing
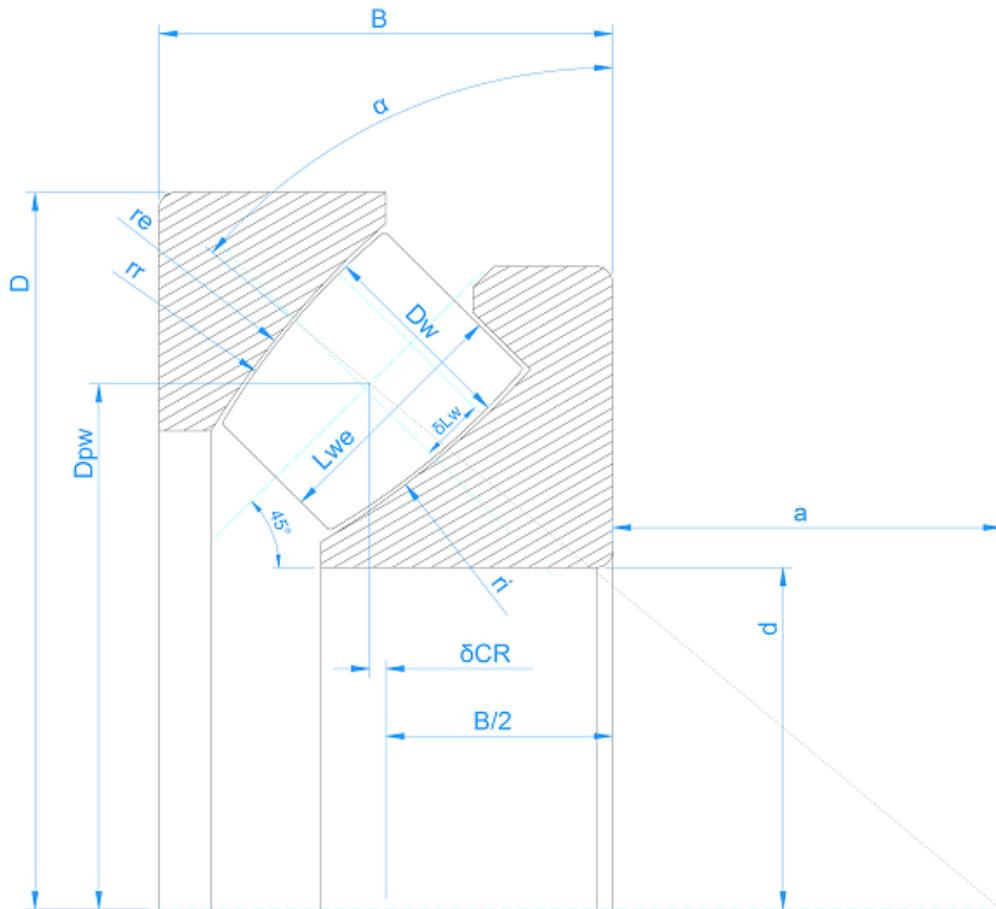
The pitch diameter D_{pw} is defined as intersection of contact angle α and axis of roller. Usual values for contact angle are 45° to 50°.

Radii of inner race and roller are defined by factors f_i and f_r like for the radial spherical roller bearings. They are defined as $r_i = f_i*(2*r_e)$ and $r_r = f_r*(2*r_e)$. Default values are $f_i = 0.5$ and $f_r = 0.485$. If the three radii r_i, r_e, r_r are entered using the ⬒-button behind f_i, f_r, the factors f_i, f_r and the pitch diameter D_{pw} are calculated.

The unsymmetry of the roller is defined by δL as offset between center of roller and position of maximum roller diameter D_w. A sizing button is available to calculate δL so that the contact point is at the center of the roller.

The load capacities are calculated using the roller diameter D_{we} at the contact point. The contact stress on the roller is always shows from inner roller end to outer roller end, instead of left to right.

3.1.2.1.27 Cross roller bearings
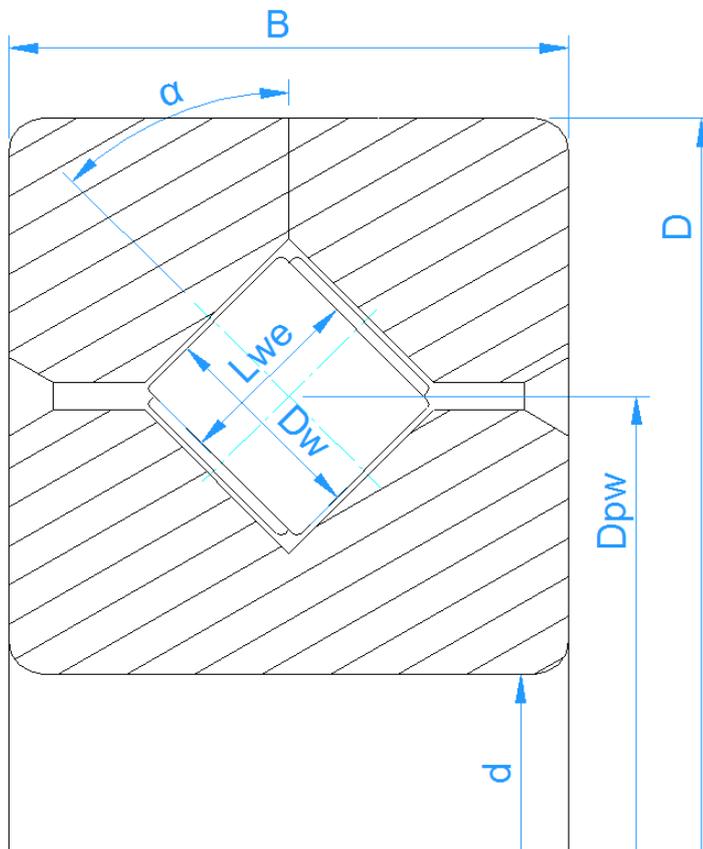
Cross roller bearings have a contact angle of 45°. They can be selected as radial or axial bearings. Differences are the calculation of load capacity and the input of clearance as radial or axial clearance.

The number of rollers has to be entered for one row; so it is half of the total number of rollers. The length of the roller has to be smaller than its diameter.

3.1.2.1.28 Angular roller bearings

Angular roller bearings are similar to cross roller bearings, but all rollers are mounted in one direction. Therefore, the restriction on roller length does not apply anymore and the contact angle is available as input.

3.1.2.2   Approximation of bearing geometry

If the inner bearing geometry is not available, it can be approximated by the software. Four possibilities are available:

Enter outside geometry only

In this case only the outside geometry of the bearing is defined with inner diameter d, outer diameter D and width B. Additionally, the contact angle and the clearance have to be defined.

The number and size of rolling elements are approximated by the software. The load capacities are then calculated using this inner geometry. This does not lead to accurate results, because the real bearing geometry is not used. But still influences of moment loads and other parameters could be seen.

Enter outside geometry and load capacities

In this case, the inner geometry is approximated as before, but the load capacities are provided by the user. The load capacities are usually available in bearing catalogues.

Enter inner geometry

Using this selection, you have to enter all the dimensions for inner geometry. The load capacities are calculated according to the standards.

Enter inner geometry and load capacities

Since bearing manufacturers often use load capacities larger than calculated according to standards, it is possible to enter both: the inner geometry and the load capacities. The load capacities are then used for the calculation of life.

Select bearing from database

Instead of entering bearing geometry by the user, it can be selected from a database.

Inner and outer diameter can be defined optionally. This restricts the number of bearings shown in the list. By clicking on the titles of the columns, the data can be selected according to this column.
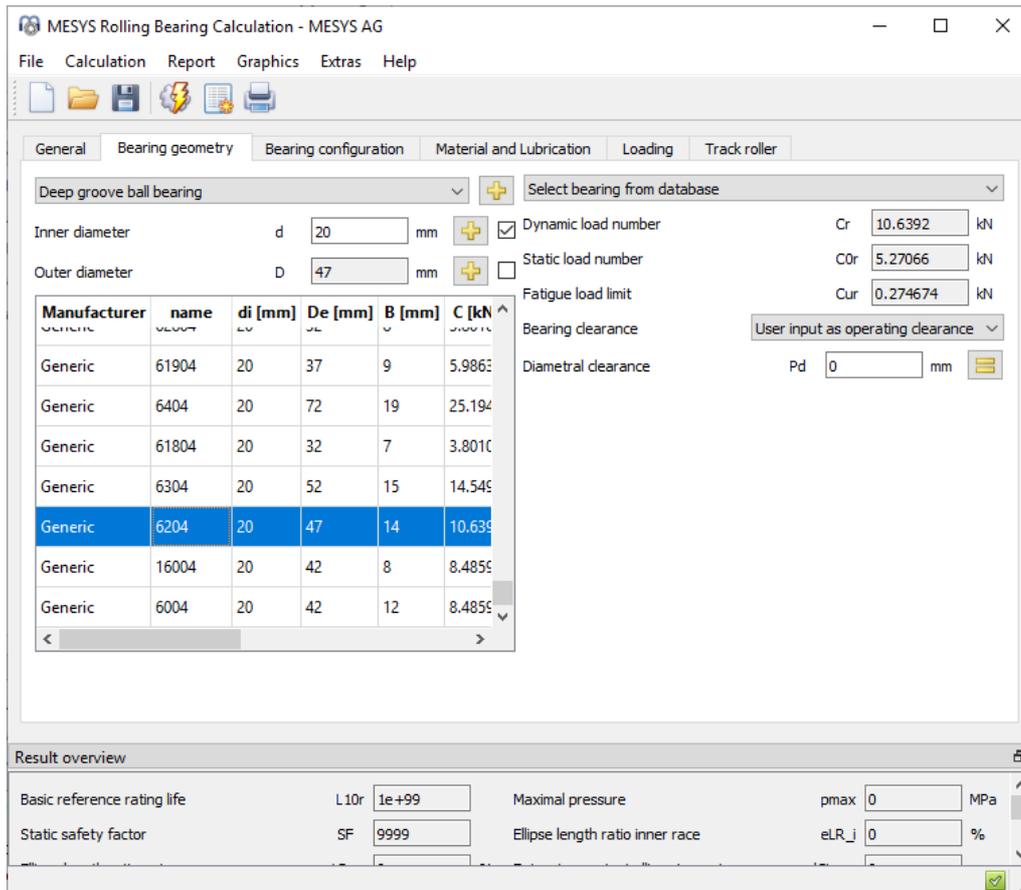
Double-clicking of a row reads in the bearing data and updates the values which are read from the database.

The data of inner geometry of the bearings provided with the database are approximated from outer dimensions. Additional databases from bearing manufacturers are available. Catalog data with approximated inner geometry is available from Schaeffler and SKF and is included in the installation. Encrypted bearing databases including inner geometry are available from HQW/Barden, CSC, GMN and IBC and other manufacturers. The databases from HQW/Barden and CSC are included in the installation, please contact the bearing manufacturer for further databases.

Bearings with approximated bearing inner geometry and bearings with manufacturer setting "Generic" are shown with default font, custom bearings and databases with inner geometry information are shown in bold font.

3.1.2.3   Load capacities

Dependent on the setting for the approximation of bearing geometry, the load capacities can be an input or an output. If they are given by the user, they will not be changed because of surface hardness of the material. The surface hardness is only considered if the values are calculated by the software.

3.1.2.3.1 Dynamic load number

The dynamic load number is used for the calculation of bearing life. It is calculated according (ISO 281, 2007) with factor fc according (ISO/TR 1281-1, 2008). The factor b_m can be influenced for filling slot bearings using the options dialog for the bearing type (-button behind bearing type). If the option for hybrid bearing is selected in the options dialog, the dynamic load capacity is calculated according to (ISO 20056-1, 2017), leading to larger bm and smaller fc, but unchanged load capacity compared to steel bearings.

The dynamic load number can be influenced by a modification factor available at material properties.

3.1.2.3.2 Static load number

The static load number is calculated according (ISO 76, 2006) and (ISO/TR 10657, 1991). It is only for documentation and only used in the calculation of static safety factor if the static safety is not calculated based on stresses (see settings "Calculate static safety factor based on stresses"). It is also used in one case for track roller calculation, see that section for details.

If the option for hybrid bearing is selected in the bearing options dialog, the static load capacity is calculated according to (ISO 20056-2, 2017), leading to larger static load capacities in case of silicon nitride rolling elements with Youngs Modulus larger than 300GPa . Note that (ISO 20056-2, 2017) also uses higher values for recommendations of static safety factors.

The static load number is based on a permissible stress which can be changed at inputs for material data.

For bearings with low speed higher loads are permitted sometimes. For example, (ISO 1002, 1983) allows a radial force of over five times the static load capacity for nonrotating ball bearings.

3.1.2.3.3 Fatigue load limit

The fatigue load limit is calculated according (ISO 281, 2007) section B.3.2.1.2 for ball bearings and according section B.3.2.1.3 for roller bearings. For roller bearings, the standard calculation according ISO/TS 16281 is used, not the extended method for pressure distribution.

The fatigue load limit is based on fatigue strength of 1500MPa and it is used for the calculation of modified life. The fatigue strength can be modified using inputs for material data.

3.1.2.4 Inner, outer diameter and width

The inner diameter, outer diameter and width are only needed for documentation and for the approximation of inner geometry. They are only used for the calculation of operating clearance after fitting.

If the inner geometry is provided and no calculation of operating clearance is required, these values could be set to zero.

3.1.2.5 Deformations of rings

If deformations of outer and inner ring are, for instance, known by means of FE analysis, they can be entered into the software to evaluate their influence on bearing life and contact stress by clicking on the ✛-button behind both diameter input fields. The deformations change the local bearing clearance for each rolling element within the calculation of load distribution.

Inner diameter	d	100	mm	✛
Outer diameter	D	140	mm	✛

Define deformations of inner ring

Type of input			Fourier coefficients	
	u_r [mm]	φ_r [°]	u_x [mm]	φ_x [°]
u*cos(0*ψ + φ)	0,09	0	0,001	0
u*cos(1*ψ + φ)	-0,03	-30	0,0015	-45
u*cos(2*ψ + φ)	0,015	60	0,009	270

OK   Cancel

Define deformations of outer ring

Type of input: Point data

Not considered
Fourier coefficients
Point data

	ψ [°]	u_r [mm]	u_x
1	0	0,02	-0,01
2	90	-0,01	-0,012
3	135	0,015	-0,016
4	225	0,06	-0,0115
5	270	0,045	-0,011
6	315	0,01	-0,014

OK   Cancel

Deformation of rings

Deformation of rings

ur_i
ua_i
ur_e
ua_e

They are two possibilities. On one hand, with the option 'Point data', as many points of deformation as needed can be added to the data table. Any point is positioned circumferentially by its angle and both the axial and radial deformation

can be defined. Note that the deformations along the rings between any defined points will be linearly interpolated. On the other hand, there is an option to define an approximate deformation curve by 'Fourier coefficients' as shown in the picture above. Additionally, a whole table can be imported from a CSV-file using the 🡒-button and any created table can be exported into a file using the 🡒-button.

In case of shaft calculations with 3D-elastic parts a shaft deformation can be transferred to the bearing calculation. In this case a check-box "Consider shaft deformations" is shown. If this check-box is cleared the shaft deformations are removed and not considered any more. If the bearings are using within the shaft calculation this dialog also allows to specify a gap width between bearing ring and shaft or housing. The definition for deformation is used for the race and the gap is used for the contact between bearing ring and shaft/housing.

These deformations and deformations transferred from the shaft calculation are not considered in outputs for bearing clearance or the tolerance report. They only affect the calculation of load distribution and life.

In the current version, the definition of deformations for the outer ring cannot be used together with track roller calculation with elastic outer ring.

3.1.2.6 Number of rolling elements

The number of rolling elements has to be specified. A minimum number is three; the maximum number depends on the bearing pitch diameter. A warning is shown if the rolling elements overlap, the distance between rolling elements is available in the results overview and report.

For cross-roller bearings, this is the number of rollers for one row.

Using the ➕-button the number of rolling elements can be calculated automatically based on a maximum fill angle and a minimum distance between the rolling elements. For deep groove ball bearings, the fill angle would be 180-200°, the minimum distance can be based on requirements for the cage.
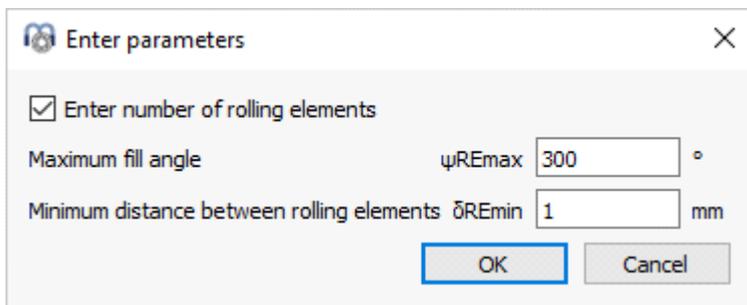
This option is mainly thought to allow the variation of the rolling element diameter in the parameter variation with an automatic setting of the number of rolling elements.

3.1.2.7 Diameter of rolling elements

The diameter of the rolling elements is specified here. For taper roller bearings, the diameter in the middle of the roller is used.

Using the ▭-button, the number of rolling elements, roller diameter and contact angle can be calculated from given damage frequencies. This can be used if damage frequencies for a bearing are given and geometry data is missing.

3.1.2.8 Pitch diameter

The pitch diameter is the diameter of the centers of rolling elements when they have the same amount of clearance to both races. The diameters of inner and outer race are calculated by this value, the diameter of rolling element, the contact angle and the clearance.

Usually, the mean value of inner and outer diameter of the bearing is taken, but it can be different. For example, it is not valid for a needle bearing without inner ring. The mean value can be selected using the ☆-button behind the input.

3.1.2.9 Contact angle

The contact angle has to be provided for angular contact bearings, four-point ball bearings, spherical ball bearings, taper roller bearings ad spherical roller bearings. For taper roller bearings, the cup angle is used since this is the direction of the force. Using the ✚-button, the direction of contact angle can be selected.

For double row angular contact bearings, the back-to-back arrangement has the larger width between the centers of pressure and it is also called O-configuration, while the face-to-face arrangement is the same as the X-configuration.

For three point bearings this input is used for the shim angle of the split ring. The ⊟-button allows to calculate the shim angle by given shim width. The resulting contact angle for axial contact is available as free contact angle in the results.

3.1.2.10 Conformity of inner and outer ring

The conformity is the ratio between radius of curvature of a race and the ball diameter. For geometric reasons, the value has to be larger than 0.5. The values used for the calculation of tables in ISO 281 are $f_i=f_e=0.52$ for radial ball bearings and $f_i=f_e=0.535$ for thrust ball bearings as stated in (ISO/TR 1281-1, 2008).

According to (ISO/TS 16281, 2008) usual values are $f_i=0.52$ and $f_e=0.53$ for radial ball bearings and $f_i=f_e=0.54$ for thrust ball bearings. (ISO 76, 2006) also uses $f_i=f_e=0.54$ for thrust ball bearings.

For spherical or barrel roller bearings values of $f_i=f_e=0.5$ should be used. Here the diameter of the outer race d_e is used as reference.

According to (ISO 281, 2007) and (ISO 76, 2006) a larger conformity reduces the load capacity and therefore a reduction of fc is required, but a smaller conformity does "not necessarily increase the load-carrying ability". Therefore, the software allows to enter a limit for the conformity used for the calculation of load capacity. See the ✚-button behind the bearing type for ball bearings.

For spherical ball bearings, the diameter of outer race d_e is used as reference for f_e. So $f_e=0.5$ should be used as default. The ⊟-button allows to enter the radii and to optionally adjust the pitch diameter D_{pw} so that the outer race is a sphere with $f_e=0.5$

Using the ⊟-button behind the input field, the conformity can be calculated from a given radius. Ensure that you input the correct roller and pitch diameter before.

3.1.2.11 Conformity of roller

For spherical or barrel roller bearings, the conformity of the roller has a usual value of $f_r=0.485$. The diameter of the outer race d_e is used as reference, so $r = f_r * d_e$.

3.1.2.12 Distance between rows

For double row bearings, the distance between rows is shown in the user interface. It is the axial distance between the centers of balls. For double row angular contact bearings, it is an input, for the other bearing types it is an output since the distance is calculated by pitch diameter and contact angle. The distance between rows is

$$\delta_R = D_{pw} \cdot \tan \alpha$$

3.1.2.13 Effective length of roller

The effective length of the roller is the length that can be loaded. A radius at the end of the roller has to be subtracted to get the effective length.

If the extended method pressure calculation for pressure distribution is selected on page 'General', the roller profile can be defined using the ➕-button.

3.1.2.13.1 Defining roller profiles

Profiles can be defined for inner race, outer race and roller.

As general inputs, the difference between effective and total roller length can be defined. As it increases the stiffness of the roller, it has an influence on the calculation.

A different profile definition length Lwd can be selected for each profile for roller, inner race and outer race. Either the effective roller length Lwe can be used or the total roller length Lw or a custom input between these two limits ($L_{we} \leq L_{wd} \leq L_w$). Different lengths for the profile definition can be used for

example in cases with a larger length for the roller and the outer raceway and a smaller length for the inner raceway because of relief grooves.

The number of sections for the calculation has to be larger 30 and has to be an odd number. The calculation accuracy and the calculation time increase with a larger number of sections.

Several options are available for the definition of profiles:

1. Standard profile according ISO/TS 16281:
   This is using the defaults of the standard, which is logarithmic profile of the roller and no profile on the races.

2. Logarithmic profile:
   The amount of relief can be defined and a cylindrical part that should not be modified. The following formula is used:

$$P(x) = pr \cdot \ln\left(\cfrac{1}{1 - \left\{\cfrac{2|x| - L_{wu}}{L_{wd} - L_{wu}}\right\}^2}\right)$$

   For the roller, a sizing button provides the default relief according ISO/TS 16281. The default relief of 0.00035*Dw for short cylindrical roller bearings corresponds to a contact stress of 3000MPa.

3. Modified logarithmic profile:
   The amount of relief is calculated according following formula derived from (Fujiwara, et al., 2010):

$$P(x) = A \cdot \ln\left(\cfrac{1}{1 - \left\{1 - e^{-\frac{pr}{A}}\right\}\left(\cfrac{2|x| - L_{wu}}{L_{wd} - L_{wu}}\right)^2}\right)$$

   using

$$A = 2D_w \cdot \left(\frac{p \cdot (1 - v^2)}{E}\right)^2$$

   The maximum relief is not infinity at the edge of the roller like in the logarithmic profile according ISO/TS 16281 but limited to the maximum value of pr.
   Using p = 3000MPa and pr=Infinity, the results are the same as for the logarithmic profile according ISO/TS 16281 above.
   The material data for this profile is always taken from the contact roller/inner race.

4. Tangential crowning:
   A circular arc tangential to the cylinder is used to generate the profile.

The amount of relief and a percentage of unmodified cylinder length can be defined.

5. Crowning:
   A circular arc with center positioned axially on the center of the roller is used to generate the profile. The amount of relief and a percentage of unmodified cylinder length and a transition radius can be defined.

6. Two tangential radii:
   A first cylindrical part with width "Lwu" is extended by two tangential radii. The width of the first radius is defined as percentage "Lwr1". A requirement is "Lwr1" > "Lwu". "Lwu" can be set to zero to have two radii only.

7. Exponent
   An exponent "e" and an unmodified length "Lwu" for the profile can be specified. The exponent has to be $e \geq 2$. The following formula is used for the profile:

$$P(x) = pr \cdot \left(\frac{2|x| - L_{wu}}{L_{wd} - L_{wu}} \right)^e$$

8. Read file:
   The profile can be read from a file. The data has to be specified with two values on each line, an axial position and the profile relief. The axial position is scaled with the roller length and the relief is scaled with the input for the profile relief. So a range from -1 to +1 can be used for the axial position and 0 to 1 for the relief.
   A very simple file is shown here. It was selected for the inner race profile on the right:

The unsymmetrical profile is used from left to right for most bearings. For taper roller bearings and axial spherical roller bearings, the profiles is used from left to right if direction of contact angle is left. For contact angle to the right, the profile is mirrored. So mounting conditions do not influence the profile.

9. Read file without scaling:
   Like option "Read file" the data is read from a file. Using this option no scaling will be done. The axial extension has to be defined from -Lwd/2 to +Lwd/2 or from 0 to Lwd. The profile is defined in "mm" and won't be scaled too.

10. User Input:
    Instead of defining a file name, the points for the profile can be defined in a table. Again as above, use a range of -1 to +1 for the axial position and a value between 0 and 1 for the profile relief.
    The table in the diagram on the right shows the same data as the file above.

11. User defined equation:

    A user defined equation can be defined in ECMAScript language. The variable 'x' has the range $-1 < x < +1$ and variables 'Dw', 'Lwe', 'Lw', 'Lwd' are defined and in 'mm'. The result of the equation is considered in 'mm' and a positive value increases the gap.

    Examples are "return 0.01*x**4;" or "return -0.00035*Dw*Math.log(1-x*x);" or using conditions like

    ```
    if(x < -0.5) {
            return 0.01*(-0.5-x);
    }
    if(x > 0.5) {
            return 0.01*(x-0.5);
    }
    return 0;
    ```

The profiles can be shown in the graphics for roller profile.

3.1.2.14 Offset for roller center

For axial spherical roller bearings, an offset for the roller center δL can be defined. It is the distance between the center of the roller and the position of the maximum diameter D_w.

3.1.2.15 Shoulder diameter of inner and outer ring

For ball bearings and radial cylindrical roller bearings, the shoulder diameter of inner and outer ring can be defined. For tapered roller bearings, the shoulder for the inner race can be defined. If an input of zero is given, it will not be considered.

The shoulder diameter is compared to the maximum extension of the contact ellipsis. A warning is shown if the ellipsis would extend above the shoulder. The Hertzian stress is not valid in that case.

The required shoulder diameter is shown in the report together with a length ratio eLR_i, eLR_e which is showing a safety of shoulder length. The length ratio is defined as the length from lower end of contact ellipsis to shoulder (the green line the graphics) divided by the length of the contact ellipsis 2a (red line in the graphics). The value should therefore be larger than 1 or 100%.

For spherical ball bearings, the shoulder diameter of the outer race is limited by the radius of outer race and bearing width automatically if the bearing width is entered.
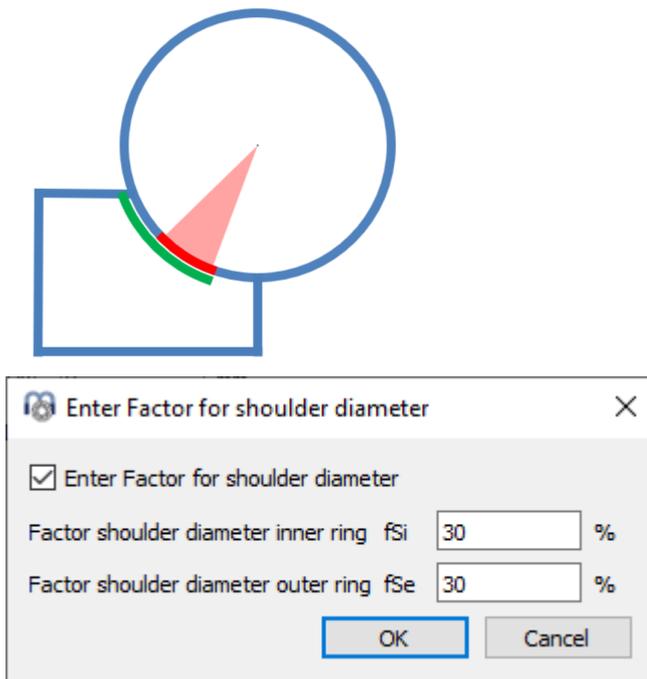
Instead of using an absolute value, the shoulder height can be defined using a percentage of ball diameters. A factor of 50% would be a shoulder up to the pitch diameter, so the factor should be between 10% and 40% for most bearing types. Using this factor allows to have a default geometry on changing ball diameter or pitch diameter.

For radial cylindrical roller bearings and tapered roller bearings, it is assumed that the axial force is acting at half the height of the shoulder. A shoulder opening angle can be defined. A value of 0° means the force is acting parallel to the axis of the roller.

3.1.2.16 Distance to row center

This input is only available for taper roller bearings, radial half spherical roller bearings and axial spherical roller bearings. It is the distance between center of roller and center of bearing. It can be calculated from the distance to center of pressure 'a' provided in bearing catalogs.

For radial half spherical roller bearings the distance 'a' should usually be equal to the bearing width, so that the center of curvature of the outer ring is on one side of a bearing and two combined bearing have a sphere as outer race.

3.1.2.17 Bearing clearance

The bearing clearance can be set automatically according (ISO 5753, 2009) (C2…C5) for deep groove ball bearings, four-point bearings, self-aligning ball bearings, spherical and cylindrical roller bearings.

In addition, there are the settings 'from database', 'own input as operating clearance' and 'own input':

- If 'From database' is selected, the minimal and maximal values for the bearing clearance are taken from the bearing database. An error message is shown if the bearing data is not selected from database. It is not necessary to use the clearance from database, also the other options for clearance can be selected.

- Selection 'user input as operating clearance' means that no modification to the clearance will be done by the software. No interference fit or temperature is taken into account. This is the recommended setting if you know the clearance you want to consider.

- Selecting 'user input' allows entering the clearance of the bearing before mounting. Influence of temperature or interference fits will be considered additionally.

- Selecting 'user input as range' allows entering the clearance of the bearing before mounting as a range using the ➕-button. Influence of temperature or interference fits will be considered additionally. The load distribution will be calculated using the clearance according to the setting as minimum, medium or maximum value of the range. The ▤-button in the dialog will use the minimal clearance position for sizing of 'Pdmin' or 'Pamin' and the maximal clearance position for sizing of 'Pdmax' or 'Pamax'.

In case of radial angular contact bearings a preload can be defined in the database. Then additional selections can be available: "Extra light preload", "Light preload", "Medium preload", "Heavy preload". These entries are only shown if a preload is defined in the database.

3.1.2.18 Diametral clearance

The diametral clearance P_d is shown for deep groove radial ball bearings, spherical ball bearings, cylindrical roller bearings and spherical roller bearings. It is the distance between the upper and the lower position the inner ring can have without load in a fixed outer ring. For single row bearings it can be calculated as

$$P_d = d_e - d_i - 2 \cdot D_w$$

For pretension, a negative clearance can be entered.

The same value is sometimes described as radial clearance (in contrast to axial clearance), but the measurement is still a difference of diameters.

A ▤-button allows the input of an axial clearance instead of a diametral clearance. For deep groove ball bearings, the clearance can also be calculated for given free contact angle.

3.1.2.19 Axial clearance

The axial clearance P_a is shown for angular contact bearings, four-point bearings, cylindrical roller bearings and all axial bearings. The axial clearance is the difference between the possible axial displacement in positive and negative direction for-four point bearings and cylindrical roller bearings.

For single row angular contact bearings or tapered roller bearings, it is the axial distance between inner and outer ring when the bearing rings are moved together with a very small axial load. So in this case, the clearance is between center and one direction and for double row angular contact bearings, four-point bearings and cylindrical roller bearings, it is the maximum distance between left and right end positions.

For cylindrical roller bearings, a value larger than zero should be entered for the axial clearance, since the bearing usually has axial clearance. If the calculation is done with clearance zero, you will get some axial loads on the rollers if there is a tilting angle of the bearing. This should be the case for a locating cylindrical roller bearing, but not for a non-locating bearing. So the correct clearance should be entered in both cases.

A pretension can be entered by using a negative value for the axial clearance Pa. A ▤-button allows the input of diametral clearance instead of axial clearance for certain bearing types. For angular contact bearings or tapered roller bearings, the axial clearance can also be calculated for a given pretension force.

For roller bearings, the calculation of axial clearance from pretension force does not yet take into account the extended pressure calculation or changes of profiles.

Dependent on the bearing type, several options for the pretension force are available. The calculation with "Fp" is using nominal dimensions for the bearings, the calculation using "Fpu" is using nominal dimensions together with a radial elastic expansion without limit; this option can be used for the case of loading an unmounted bearing axially during manufacturing. The mounted pretension force "Fpm" is calculated using the dimensions after fitting but no temperature or speed. The effective pretension force "FpEff" is calculated using the dimensions after fitting and considering temperature and speed.

3.1.2.20 Bearing tolerance

The bearing tolerance can be considered according ISO 492 (P0…P2). After defining the tolerance class of the bearing, the following inputs allow the definition of fitting to the shaft and the housing.

In addition to tolerance classes some special inputs are available:

- 'Not considered': If selecting 'Not considered', the interference fit is not considered for operating clearance, in this case only temperature will affect the operating clearance, centrifugal forces are not considered.

- 'Nominal dimensions': If selecting 'Nominal dimensions', the tolerances of the bearing are considered zero. Then the interference fit can be defined with the inputs for the shaft and housing tolerances only.

- 'User input': This option allows to define a custom tolerance field for bearing inner and outer ring using the ➕-button.

- 'Define interference': Selecting 'Define interference' allows to define the interference between shaft/inner ring and between housing/outer ring directly without using tolerances. Using the ➖-button behind the inputs for interference a required interference for a given operating interference can be calculated. The rotation speed can be overwritten for this calculation.

- 'Define multi-layer interference': This option allows to define a multi-layer interference fit for both inner and outer ring. Using the ➕-button, the layers can be defined. The diameters and materials for the interference fit are not coupled with the shaft calculation in this case.

3.1.2.20.1 Multi-layer interference fit

A multi-layer interference fit can be defined for both, inner and outer rings. For the inner ring fit the layers needs to be ordered from large to small diameter and for the outer ring contact from small to large diameter. The first layer is always in contact with the bearing ring. Optionally a tolerance for the interference can be defined.
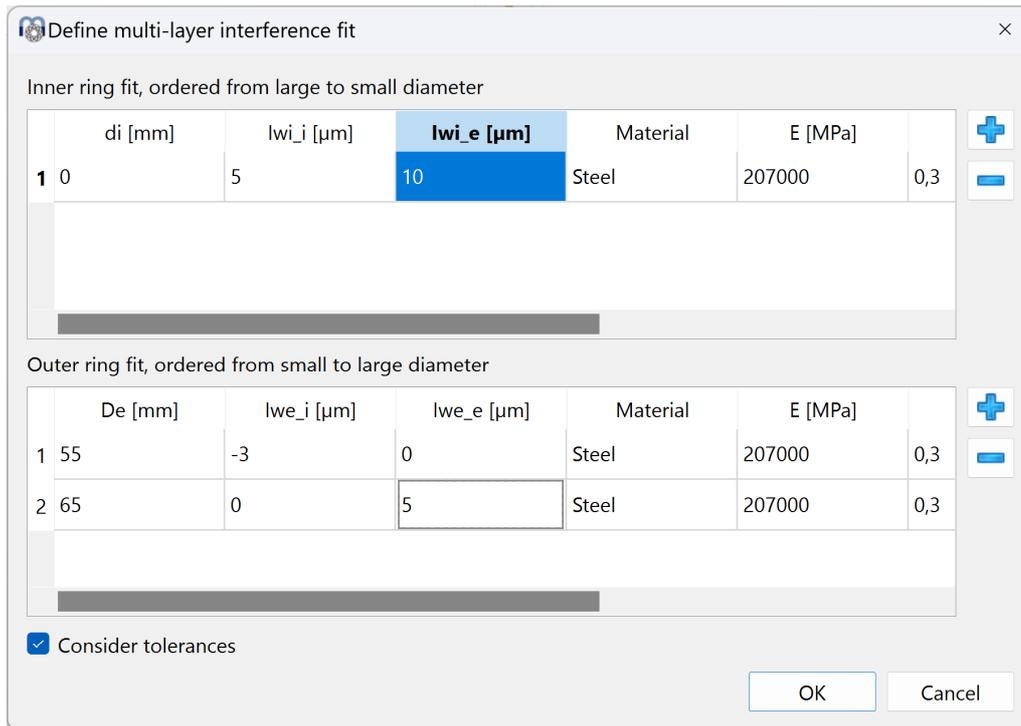
If the bearing calculation is used from the shaft calculation, note that the diameters and materials are not coupled to the information within the shaft calculation. All layers for each ring have the same temperature.

3.1.2.21 Fit to shaft/housing

The fit between inner ring and shaft can be defined as a tolerance according (ISO 286-1, 2010). For example, it can be specified as 'k6'. The same is possible for the fit between outer ring and housing.

A numerical value can be specified directly using the ✚-button next to the input field. Selecting "own input" allows the input of arbitrary tolerances:
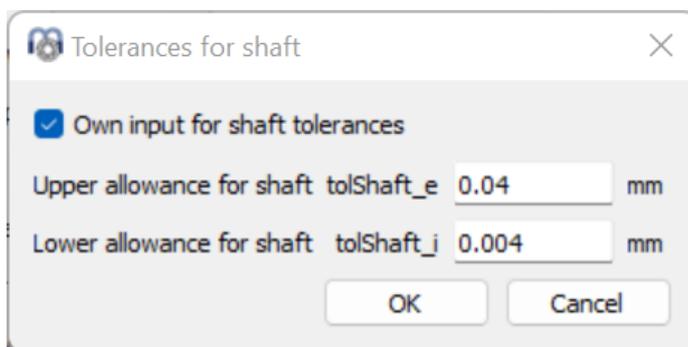
3.1.2.22 Shaft inner diameter/Housing outer diameter

The inner diameter of the shaft can be specified for a hollow shaft. This has an effect on the interference fit.

For the housing, the outer diameter can be specified. If zero is entered, a value of 2·D will be used in the calculation.

3.1.2.23 Surface roughness shaft and housing

The surface roughness of the shaft and the housing is only used for the calculation of the interference fit. The effective interference of the parts is reduced by 0.4·Rz according (DIN 7190-1, 2017). In previous versions, a reduction of 0.8·Rz was used as in the previous version of the standard.

More precisely for the surface roughness of the shaft, the sum of roughness of shaft and inner ring should be entered. Similar for the outer ring. As the roughness of the bearing is usually smaller that of shaft and housing, this might be ignored in most cases.

3.1.3 Bearing configuration

A group of bearings can be considered in the calculation. The bearings are assumed to be connected to a rigid shaft and housing. The loads or displacements are given for the origin of the bearing group instead for each single bearing.

A relative position can be entered for each bearing. An axial offset allows the definition of pretension in the system. The offset is an additional movement of the outer ring of a bearing. The displacements, forces and moments under loading are applied at the reference position 0. So in most cases, the positions should be chosen symmetrically around zero.

For bearings with a contact angle, the position of the cone center can be specified. In the picture, an O-configuration (or back-to-back configuration) is defined.
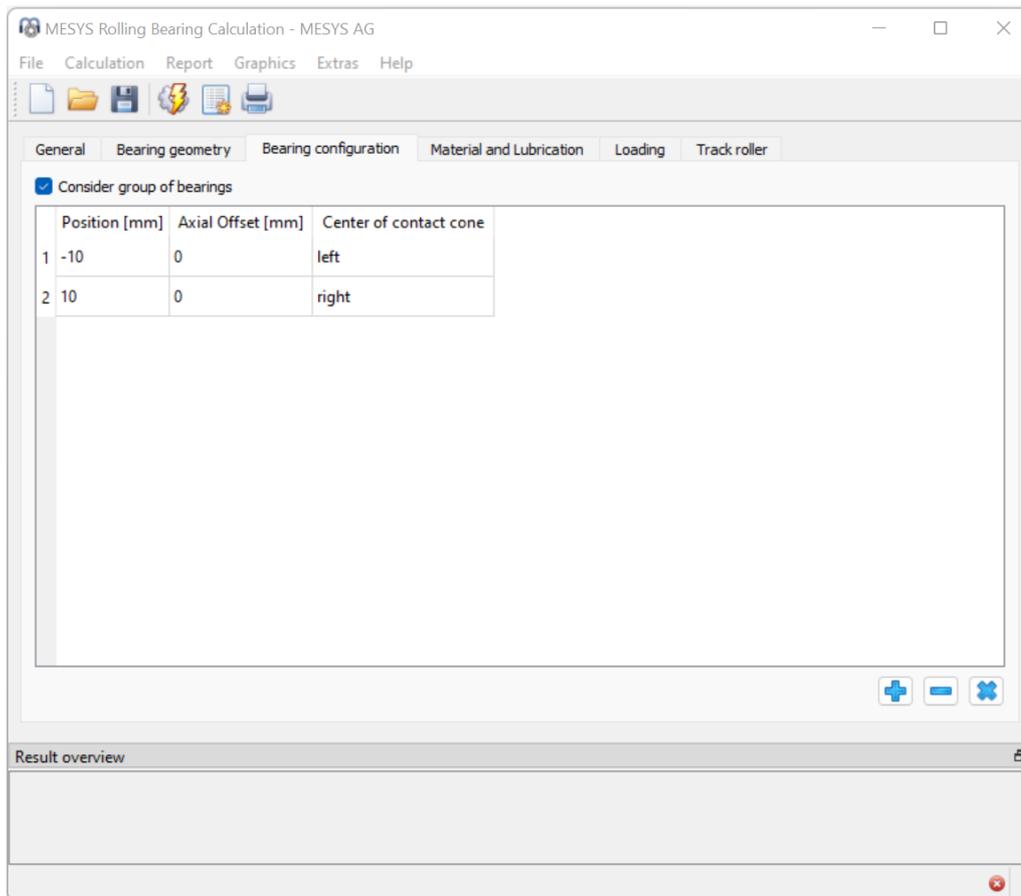
Additional bearings can be added using the ✚-button. A row can be deleted using the ➖- and the ✖-button clears all inputs.

The possibility of bearings groups allows the calculation of 8-point ball bearings as two rows of four-point bearings; it allows sets of angular contact bearings, paired taper rollers or sets of cylindrical roller bearings which could be used in a planetary gear.

With the context menu (right mouse button) in the table, several results values can be shown in the table. This allows seeing quickly how load or life is distributed among the bearing rows.

3.1.4   Lubrication

The lubrication has an effect on the modified life rating. A large oil film thickness prevents direct contact between rolling element and races and therefore can increase the life.

Some lubricants can directly be selected from the list. Mineral oils and PAG-based synthetic oils are provided. For other types of oil, 'own input' can be selected.

Lubrication

ISO VG 220 mineral oil	▾		Oil lubrication with on-line filter ISO4406 -/17/14	▾	➕

Viscosity at 40°C	nu40	220	mm²/s	Temperature	TOil	70	°C	☑
Viscosity at 100°C	nu100	19	mm²/s	Oil density	ρ	890	kg/m³	
☐ contains effective EP additives				Pressure viscosity coefficient	α	0.02	1/MPa	

3.1.4.1   Contamination factor

The contamination factor e_C is used for the calculation of life modification factor a_{ISO}. It is considering the cleanliness of the lubricant and can be calculated according to ISO 281 Annex A with the viscosity ratio, the bearing size and a selection of oil cleanness according to ISO 4406. The selection according to ISO 4406 defines how many particles of certain size may exist. The selections for grease lubrication also use ISO 281 Annex A to calculate the contamination factor e_C.

In addition an own input of the contamination factor can be selected and the factor can be entered using the ➕-button. The contamination factor e_C is in the range from 0.1 to 1 and guide values can be selected according to (ISO 281, 2007):

Contamination level	e_C	
	$D_{pw} < 100mm$	$D_{pw} \geq 100mm$
Extreme cleanliness Particle size of order of lubricant film thickness, laboratory conditions	1	1
High cleanliness Oil filtered through extremely fine filter; conditions typical for bearings greased for life and sealed	0.8.to 0.6	0.9 to 0.8
Normal cleanliness Oil filtered through fine filter; conditions typical for bearings greased for life and shielded	0.6 to 0.5	0.8 to 0.6
Slight contamination	0.5 to 0.3	0.6 to 0.4
Typical contamination Conditions typical of bearings without seals; course filtering; wear particles from surroundings	0.3 to 0.1	0.4 to 0.2
Severe contamination Bearing environment heavily contaminated and bearing arrangement with inadequate sealing	0.1 to 0	0.1 to 0
Very severe contamination	0	0

Note that for small bearings, the values for the contamination factor e_c calculated according to ISO 281 Annex A can be much lower than in the table. A user input might be preferable in this case.

3.1.4.2 Kinematic viscosity

The kinematic viscosity has to be given for two temperatures: ν_{40} at 40°C and ν_{100} at 100°C. The actual viscosity at the given temperature will then be interpolated using these values.

If a single kinematic viscosity at operating temperature is given, it can just be entered in both inputs and therefore will be used in the calculation.

A high viscosity results in a larger lubricant film, but efficiency will be reduced if it is too large.

3.1.4.3 EP-Additives

If the lubricant contains effective EP-additives, the life modification factor a_{ISO} may be calculated using $\kappa=1$ in case of $\kappa<1$ and eC>=0.2 and $\kappa>=\kappa$lim.

A ➕-button allows to increase κ also in case eC<0.2, if the option "Efficiency of EP-additives is proven in actual application" is checked. For very small κ no increase is done. The default limit is set to κlim=0.1, but it can be changed to other values, based on experience.
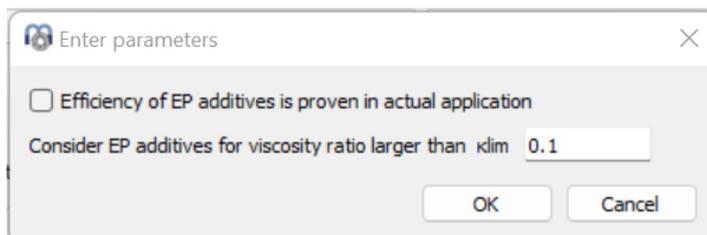
3.1.4.4 Oil temperature

The oil temperature is only used for the calculation of lubricant properties like operating viscosity. It has no influence on clearance. If the oil temperature is not selected as input, it will be set to the temperature of the rolling elements as calculated from the temperatures of the rings dependent on the settings or as given on page "Loading".

3.1.4.5 Oil density

The oil density is used for the calculation of dynamic viscosity. If a lubricant from the list is selected, the density is modified according to the oil temperature according (Niemann, et al., 2005). If 'own input' is selected, the input value is directly used for the calculation.

3.1.4.6 Pressure viscosity coefficient

The pressure viscosity coefficient α is used for the calculation of lubricant film thickness. If the film thickness should not be calculated, you can just enter zero.

According to (ISO/TR 1281-2, 2008) the pressure viscosity coefficient α can be calculated by the kinematic viscosity ν_0 in cm^2/s as $\alpha = 0.1122 \cdot \left(\frac{\nu_0}{10^4}\right)^{0.163}$

To calculate the reference viscosity ν_1, a value of $\alpha = 0.0077 \cdot \nu^{0.204}$ using ν in mm^2/s and α in mm^2/N is used according (Baalmann, 1994).

In the software, the proposal according (AGMA 925, 2003) is used, because different oil types are considered. The pressure viscosity coefficient is calculated as $\alpha = k \cdot \eta_M^s$, where η_M is the dynamic viscosity at operating temperature. The factors k and s are given for different lubricant types as shown in the following table:

Oil type	k	s
Mineral oil	0.010471	0.1348
PAO based synthetic non-VI improved oil	0.010326	0.0507
PAG based synthetic	0.0047	0.1572

This formula is used if a lubricant from the list is selected. If 'own input' is selected, either the formula according to (ISO/TR 1281-2, 2008) 11.8 or according to (AGMA 925, 2003) or a direct input can be selected using the ✚-button next to the input field.

3.1.5 Material

Material properties for rolling element, inner and outer ring, shaft and housing can be defined. This data is used for the calculation of load distribution and interference fit between bearing and shaft/housing. Also hardness and surface roughness can be specified.

3.1.5.1 Surface hardness

The surface hardness is given as Rockwell hardness HRC. If the surface hardness is smaller than 58 HRC, a reduction in dynamic and static load capacity is taken into account according (Harris, et al., 2009). This reduction is only considered as long as the load capacity is calculated by the software. The reduction factor for dynamic load capacity is:

$$C' = C\left(\frac{HRC}{58}\right)^{3.6}$$

An input value for load capacity is not modified because of the hardness value. If a modification factor for dynamic load capacity or the permissible stress for static capacity is defined in material data, a reduction because of hardness is not considered.

The static safety factor is also affected by a reduced hardness. This is also done according (Harris, et al., 2009) using the factor fs:

$$f_s = 1.5\left(\frac{HV}{800}\right)^2 \text{(Ball bearings) or } f_s = 2\left(\frac{HV}{800}\right)^2 \text{(roller bearings), with } f_s \leq 1$$

For this formula, HV is calculated as HV = (223 * HRC + 14500) / (100 - HRC) according (Schlecht, 2010).

3.1.5.2 Core strength of inner and outer race

The ultimate strength of the inner and outer ring core and the corresponding hardness depth are used for a check of case core interface. The actual shear stress at hardness depth is compared with the shear yield stress τ_{yield} and the shear fatigue strength τ_a which are calculated according to (Harris, et al., 2009) from the input value of ultimate strength:

$$\tau_{yield} = 0.425 \cdot R_m; \tau_a = 0.6 \cdot \tau_{yield}$$

If the material is though hardened, the software can be set to calculate the hardness depth automatically. Then no warnings will occur, just a larger hardness depth is calculated on load increase.

3.1.5.3 Surface roughness

The surface roughness R_q is used for the calculation of the specific lubricant film thickness Λ. It is a root mean square roughness R_q or R_{RMS}. According to (Niemann, et al., 2005), an approximation is given as $R_q \approx 1.25 \cdot R_a$.

According to (Baalmann, 1994), an estimate for a usual composite roughness is given by $R_q = 1.2 \cdot 10^{-5} \cdot D_{pw}^{0.55}$. This formula according Baalmann is used when the sizing button next to the input field is clicked.

3.1.5.4 Material for races, rolling element, shaft and housing

The material of the races, the rolling element, shaft and housing can be selected to be steel or Si3N4. The list is taken from the database therefore it can be extended by the user. Also 'own input' is available and the material data can be defined using the ✚-button.
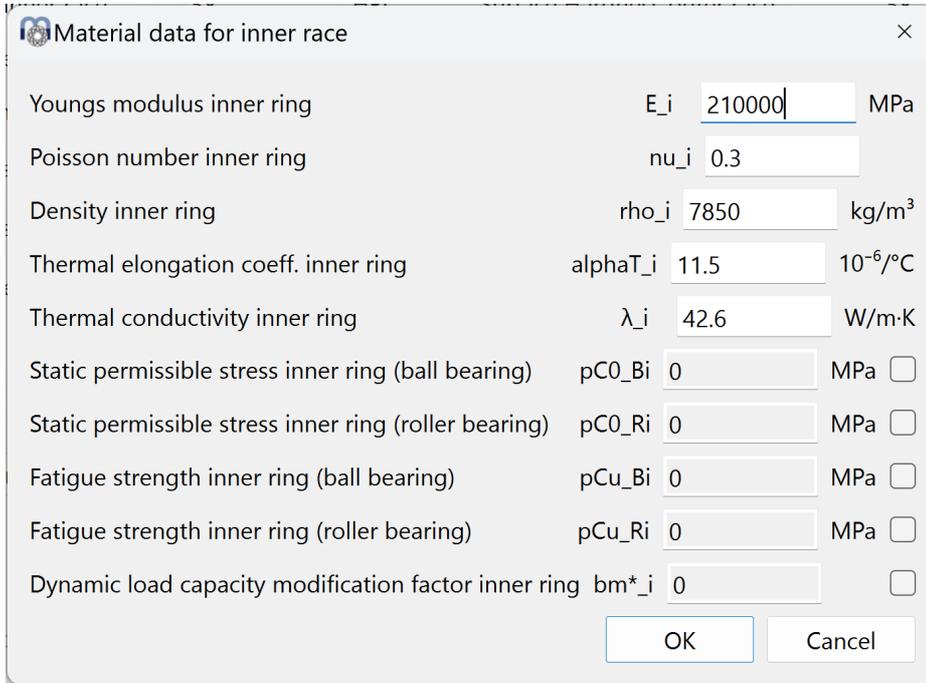
Material data for inner race				×
Youngs modulus inner ring	E_i	210000	MPa	
Poisson number inner ring	nu_i	0.3		
Density inner ring	rho_i	7850	kg/m³	
Thermal elongation coeff. inner ring	alphaT_i	11.5	10^{-6}/°C	
Thermal conductivity inner ring	λ_i	42.6	W/m·K	
Static permissible stress inner ring (ball bearing)	pC0_Bi	0	MPa	☐
Static permissible stress inner ring (roller bearing)	pC0_Ri	0	MPa	☐
Fatigue strength inner ring (ball bearing)	pCu_Bi	0	MPa	☐
Fatigue strength inner ring (roller bearing)	pCu_Ri	0	MPa	☐
Dynamic load capacity modification factor inner ring	bm*_i	0		☐
		OK	Cancel	

For material data, the Young's modulus, the Poisson number, the density, the thermal elongation coefficient and the thermal conductivity can be defined. This data is used to calculate the operating clearance and the load distribution within the bearing. The thermal conductivity is used to calculate the thermal resistance for ball bearings according (Nakajima, 1995).

Optionally, a static permissible stress can be defined which is used for the calculation of static load capacity and static safety factor. The default value is 4000MPa for roller bearings and 4200MPa for ball bearings. For spherical ball bearings, the permissible stress is multiplied by 4600/4200 for the outer ring internally by the software.

The fatigue strength is used for the calculation of the fatigue load limit Cu. The default value is 1500MPa, it can be overwritten by the user.

The dynamic load capacity can be modified dependent on the material selection, too. The bm*-factor is multiplied to the dynamic load capacity according ISO 281. This allows to consider a material influence on dynamic load capacity. A different value can be used for inner and outer race. The value of bm*-factor for the rolling element is only used for the calculation of rolling element set life, it does not affect the dynamic load capacity of the bearing.

3.1.6   Loading

For each coordinate direction, you can select if a force or a displacement should be entered. For example, if a calculation of a tapered roller bearing should be done considering only radial load, the displacement in axial direction can be fixed to zero and the reaction force in axial direction will be calculated.

A moment load or a rotation (misalignment) can be entered for two directions only, because the rotation around the bearing axis is not constrained.

Use the radio buttons to select the elements you want to enter.
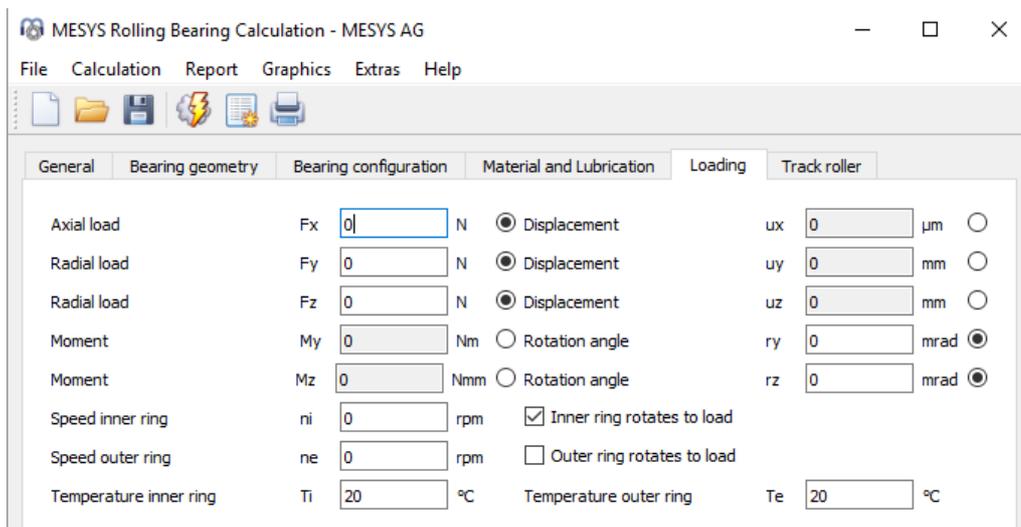
3.1.6.1   Coordinate system

The coordinate system is defined with x as the axial direction. The y-axis points up to the first rolling element and the angle ψ is positive around the x-axis or clockwise in the diagram on the right (looking in the direction of the x-axis). The angle starts with zero at the first rolling element on the y-axis. Moments are positive if acting around the corresponding axis.
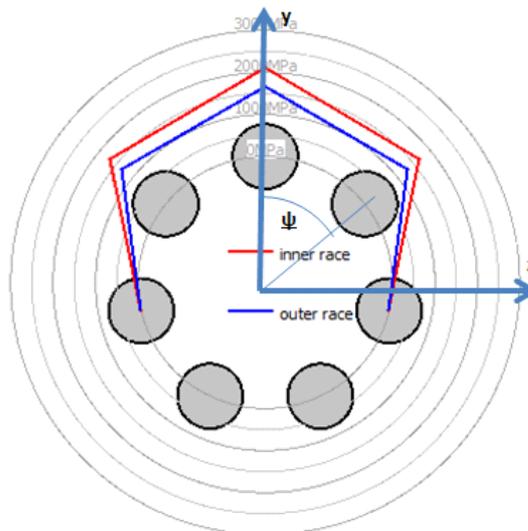
The load is acting on the inner ring (different for track rollers with elastic outer ring), so a positive load in y-direction leads to a loading of the rolling elements on the top as shown in the graphics on the right.

3.1.6.2 Inner ring rotates to load

The resulting life is slightly different if the load distribution on a race is stationary or rotating. For many bearings, the inner ring rotates relative to the load and the outer ring is stationary to the load. This is the case if the inner ring is connected to a rotating shaft and the load is stationary in space.

What is the correct input if only an axial load occurs? In this case, the input doesn't matter since the results are the same for both settings if all the rolling elements have the same loading.

3.1.6.3 Temperature of inner ring

The temperatures of the inner and outer ring are used for the calculation of the interference fit between shaft and inner ring or between housing and outer ring only. They affect the operating clearance. Optionally, the temperatures of the shaft and housing can be entered, too, to consider a temperature gradient, see Consider temperature gradient in fits 151.

The temperature of the rolling element is assumed to be the maximum value of the ring temperatures as default. This can be changed under settings and a manual input of the rolling element temperature is available optionally, too.

3.1.7 Load spectrum

If at the tab-page 'General' the flag for the calculation with load spectrum is set, instead of for a single load case, the calculation will be done for a load spectrum. The load or displacement, the rotation speed and the temperatures can be specified for each element.

	Frequency	Fx [N]	Fy [N]	Fz [N]	ry [°]	rz [°]	ni [rpm]	ne [rpm]	T_i [°C]	T_e [°C]	TOil [°C]
1	0,5	15000	10000	0	0	0	1500	0	60	50	70
2	0,3	10000	8500	0	0,5	0	1000	0	65	60	70
3	0,2	12500	500	7000	0	0	350	0	60	50	70

- ✓ enter Fx
- ✓ enter Fy
- ✓ enter Fz
- enter My
- enter Mz

☑ inner Ring rotates to load ☐ Outer ring rotates to load Results for No 1

Using the context menu that opens clicking the right mouse button, you can select which forces or moments should be used as input values. If a value is not selected, the corresponding displacement/rotation can be entered. If [Consider temperature gradient in fits] 151 is active, additional temperatures for shaft and housing temperature can be defined. The oil temperature is only shown if it is an active input.

Rows can be added using the ➕-button or a selected row can be deleted using the ➖-button. The ✖-button clears all the inputs.

Using the ➡-button, the load spectrum is read from a file. The columns that are currently shown with its units have to be used for the input file. As the data from the file is added to the inputs, you must clear the inputs first to define a full load spectrum. Likewise, a created load spectrum table can be exported with the corresponding ➡-button.

As for the single load case, a selection is available if the ring is rotating relative to the load or not. A ring not rotating to the load would be on the safe side.

The load distribution is calculated for each load case. The resulting life for the whole load spectrum is shown as result and the minimum static safety of all elements. The result graphics are only shown for the element of the load spectrum that is chosen as results on this page.

3.1.8 Track rollers

Elastic deformations of the outer ring can be optionally enabled. This is an extension to the base bearing software.

The outer ring of the bearing is modeled by a Timoshenko beam element. So bending, axial deformations and shear deformations are considered. The shear correction factor is fixed to 1. The calculation should only be used if the approximation as beam seems to be appropriate. The beam does not consider deformations within the cross section for example in axial direction for multi row bearings.

3.1.8.1   Supported bearing types

The calculation with elastic outer ring is only available for several bearing types. The calculation is possible for:

- Deep groove ball bearing

- Double row deep groove ball bearing

- Radial angular contact bearing

- Double row radial angular contact bearing

- Radial four-point ball bearing

- Radial cylindrical roller bearing

- Double row cylindrical roller bearings

- Double row tapered roller bearings

Multi-row bearings can be considered using the "bearing configuration".

3.1.8.2   Track roller geometry

The track roller geometry can be defined using several points. The contour starts at axial position zero on the outside of the bearing and is then defined in clockwise direction.

A radius can be defined in the third column. An arc from the last point to the new point will be generated. For a convex contour, use a positive value, for a concave contour, use a negative value. A clockwise arc has a positive radius, a counterclockwise arc a negative radius.

Either a full geometry is defined back to the start point or only half the geometry is defined for a symmetric outer ring. In the graphic for the symmetric track roller geometry, the mirrored part is shown in grey.

3.1.8.3   Symmetric geometry

A symmetric geometry starts at axial position zero on the outside diameter and ends on axial position zero on the inner diameter.

An unsymmetrical geometry should end with its start point.

3.1.8.4   Consider shear deformations

As default shear deformations in the elastic outer ring should be considered. Since other software and old literature, see (Paland, 1968), is only considering deformations because of bending, the influence of shear deformations can be ignored by clearing this setting. The effective load capacities will be higher because the deformation of the outer ring is reduced.

3.1.8.5   Calculate standard values

Some standard values for track rollers can be calculated automatically using a pure radial load at axial position zero.

Because of elastic deformations of the outer ring, track rollers show a different load distribution than rigid bearings. This leads to a decrease of life which is taken into account be giving reduced effective load capacities in bearing catalogs.

The effective dynamic load capacity C_w is given in bearing catalogs for track rollers. There are two options to calculate this value. See option 'Calculate C_w using $L_{10r} = 1$' below. Also for effective static load capacity, there are two options. See section 'Calculation options for C_{0w} and C_{uw}' below.

The load capacities are calculated using a radial load on the angle of zero, so it is acting directly on a rolling element.

The static and dynamic permissible radial loads are the loads were the permissible bending stress is reached. See below. The permissible static and dynamic loads are calculated using a radial load which is acting between two rolling elements. This is the critical case.

If a bearing configuration is used, effective load capacities and permissible radial loads are calculated for the whole bearing.

3.1.8.6   Calculate Cw using L10r = 1

As the dynamic load capacity is defined in (ISO 281, 2007), a radial bearing loaded with $F_r = C$ should have a life $L_{10} = 1$ (One million load cycles). This definition is used for the definition of the effective dynamic load capacity if the option 'Calculate C_w using $L_{10r} = 1$' is set. The software iterates for a radial load which leads to $L_{10r} = 1$ and this is shown as effective dynamic load capacity C_w.

For bearings with a small number of rolling elements the life $L_{10r} = 1$ is not reached for the rigid bearing loaded with $F_r = C$. The reason is that (ISO 281, 2007) assumes a load zone of 180° which cannot be reached if only six rolling elements are used for example. Therefore, a different definition for C_w is available if the option is not set. In this case, C_w is defined to the radial load which leads to the same life as a rigid bearing with $F_r = C$.

This option will only have an influence for a small number of rolling elements or bearings with clearance. If the option is not set, a load of C_w will lead to the same life for the elastic bearing as a load of C for the rigid bearing.

3.1.8.7   Calculation options for C0w and Cuw

Several options for the calculation of effective static load capacity are available:

Calculate C0w, Cuw using standard stresses

If the option 'Calculate C_{0w}, C_{uw} using standard stresses' is set, the effective static load capacity C_{0w} is defined as the radial load which leads to $p_{max} = 4200$

MPa (or 4000 MPa for roller bearings). The effective fatigue limit is defined similar, but with a stress of 1500 MPa in this case.

Calculate C0w, Cuw using standard stresses, ignoring edge stress

For roller bearings edge stresses can occur on high loading leading to different results than usually found in bearing catalogues. Therefore, a second option exists for the calculation of effective static load capacity C_{0w}.

If this option is selected, the stress in the center of the roller is compared with the permissible stress as above. So edge stresses are not taken into account.

For ball bearings, this option does not affect the results, it only makes a difference for roller bearings.

Calculate C0w, Cuw using roller load

If the option is set, C0w is defined as the radial load which leads to the same maximum roller load than a rigid bearing loaded with $F_r = C_0$. Also for the calculation of C_{uw}, the comparison of roller loading instead of stress is made.

If this option is selected, the resulting C_{0w} has the same meaning than C_0 for the rigid bearing. It can be useful if own input is used for the static load capacity.

3.1.8.8 Permissible static bending stress

The permissible static bending stress is used to calculate the permissible static radial load and a safety factor for the bending stresses in the outer ring. The stress is calculated as maximal value of bending stress plus tension in the outer ring.

$$\sigma_{max} = \sigma_{b,max} + F_t/A$$

If not defined manually, the software automatically selects the point of input geometry which leads to the highest stress. Only points in the geometry list are considered, no intermediate values. This could make a difference for convex radii.

3.1.8.9 Permissible dynamic bending stress

The permissible dynamic bending stress is used to calculate the permissible dynamic radial load and a safety factor for the bending stress amplitude. The stress is considered at the point where the maximal stress occurs.

$$\sigma_{max} = \sigma_{b,max} + F_t/A$$
$$\sigma_{min} = \sigma_{b,min} + F_t/A$$
$$\sigma_m = (\sigma_{max} + \sigma_{min})/2$$

$$\sigma_a = (\sigma_{max} - \sigma_{min})/2$$

The input value is a permissible alternating stress with is reduced by the mean stress influence coefficient ψ. The safety factor is calculated as

$$S_{dyn} = \frac{\sigma_{bDperm} - \Psi \cdot \sigma_m}{\sigma_a}$$

The permissible dynamic radial load is calculated so that a safety factor of 1 is reached.

3.1.8.10   Mean stress influence

The mean stress influence is considering the mean stress for calculating permissible amplitude of bending stress.

The mean stress influence coefficient can be calculated as
$\Psi = \sigma_{bDperm}/(2 \cdot R_m - \sigma_{bDperm})$ using the definition according DIN 743.

3.1.8.11   Axial position for the load

If an asymmetric geometry is defined, an axial position for the load can be defined. It is used for the calculation of standard values like Cw, C0w …

For symmetric geometry, the load is applied at position zero.

3.1.8.12   Manually enter geometry data

It is not mandatory to enter the cross section using point data. Also cross section data can be introduced directly after calculating them by a CAD system for example.

If the cross section data is entered manually, also the critical point for calculating bending stresses has to be defined manually. Unfortunately, this point is load-dependent. If you just have a radial load on a symmetric cross section, this is not a problem, but you have to be careful in other cases.

3.1.8.13   Second momentums

The second momentums are given on polar coordinates for the cross section. They are given relative for the center of gravity, not for the origin of the coordinate system.

The second momentum in axial direction I_{aa} and in radial direction I_{rr} are usually available. For the tangent direction, the torsional moment should be given, but since this is not usually known the polar moment $I_{tt} = I_{aa} + I_{rr}$ can be used as replacement. This is only exact for a circular cross section, but the torsion load is not the main loading to the outer ring, so it could be acceptable.

The deviatory momentum is defined as $I_{ar} = + \iint a \cdot r \, dA$. Depending on the literature, also definitions with a negative sign are usual. The deviatory momentum is zero for symmetric cross sections.

3.1.8.14 Center of gravity

The center of gravity is defined using its axial and radial coordinate. For symmetric cross sections, the axial position is zero.

3.1.8.15 Cross section

The cross section of the outer ring has to be entered here. Since the calculation is considering tension and shear deformations, it has an influence on the load distribution.

3.1.8.16 Critical point

The critical point for the calculation of bending stress has to be entered if the geometry data is entered manually. The bending stress is calculated using this point.

Be sure to enter the correct value for the given load conditions. The critical point is marked with a red circle in the graphics of track roller geometry.

3.1.8.17 Track roller loading

In the case of elastic outer ring, the input of loading is changed. Instead of the load onto the inner ring, the load on the outer ring can be defined on several points. It is not possible to define displacements or rotations directly.

An angle, axial and radial positions define the point where the load is acting. The load can be defined in axial, radial and tangential direction. Since the overall torque has to be zero, an input of tangential load can only be used if several loads are defined.

The coordinate system is a standard cylindrical coordinate system with r-direction pointing to the outside. Therefore, Fr should usually be entered as negative value for pressure.

The axial position can be used if you assume that the radial load is not centric. An eccentrical load leads to torsion of the outer ring.

The first rolling element on the y-axis is at an angle of zero. The results will be different if the load is applied on the position of a rolling element or between them. Both cases should be considered in doubt.
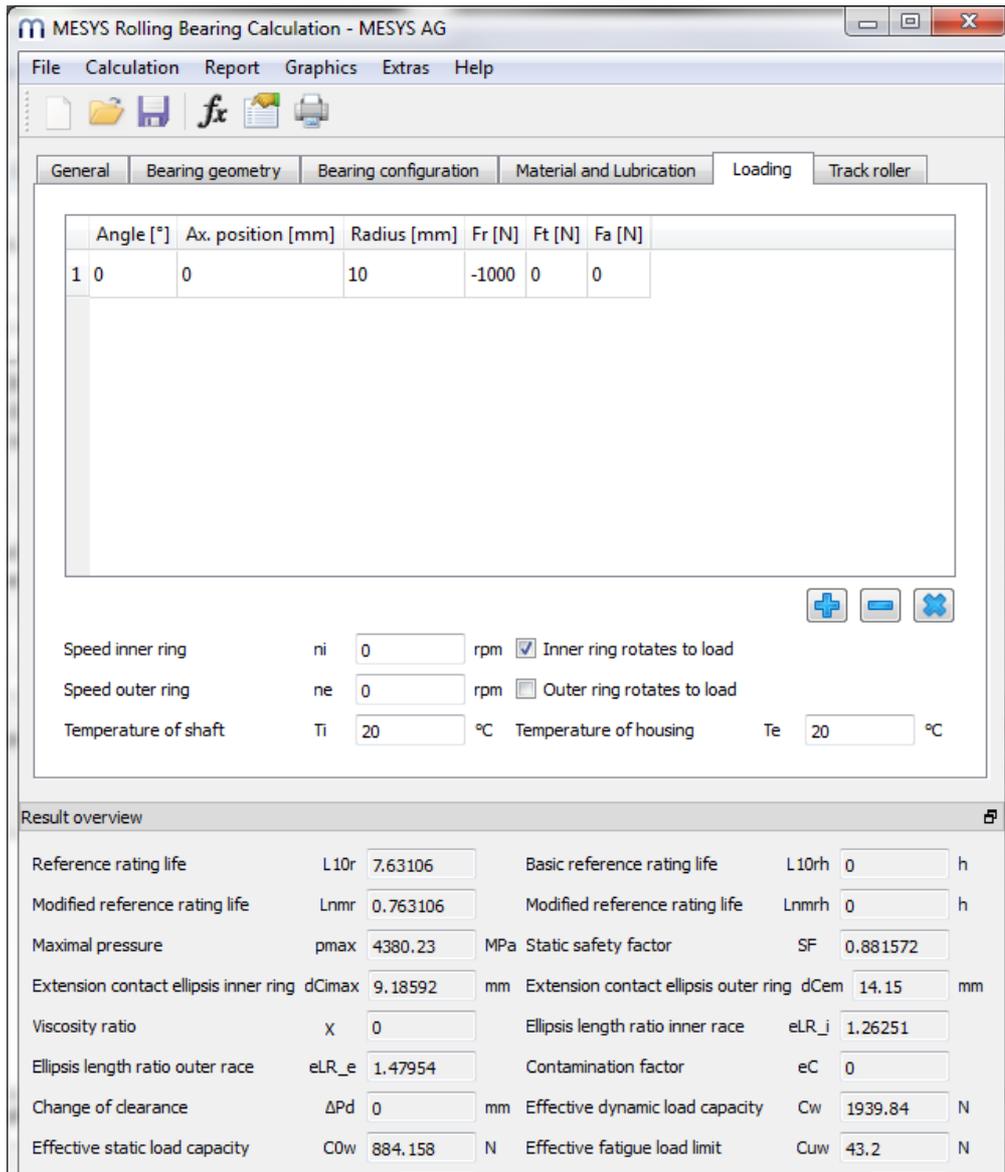
3.1.8.17.1 Load spectra for track rollers

If load spectra are used for track rollers, as a first step, the location of forces has to be defined in the loading page without activating load spectra. If the load spectra are activated afterwards, a column for each force is available as input. Using the context menu (right mouse button) additional columns like the angle can be included.

	Frequency	ni [rpm]	ne [rpm]	T_i [°C]	T_e [°C]	T_r [°C]	TOil [°C]	Fr_1 [N]	Ft_1 [N]	Fa_1 [N]
1	0.6	0	500	20	20	20	70	-1000	0	0
2	0.4	0	200	20	20	20	70	-500	0	0

☑ inner Ring rotates to load   ☐ Outer ring rotates to load   Results for No 1
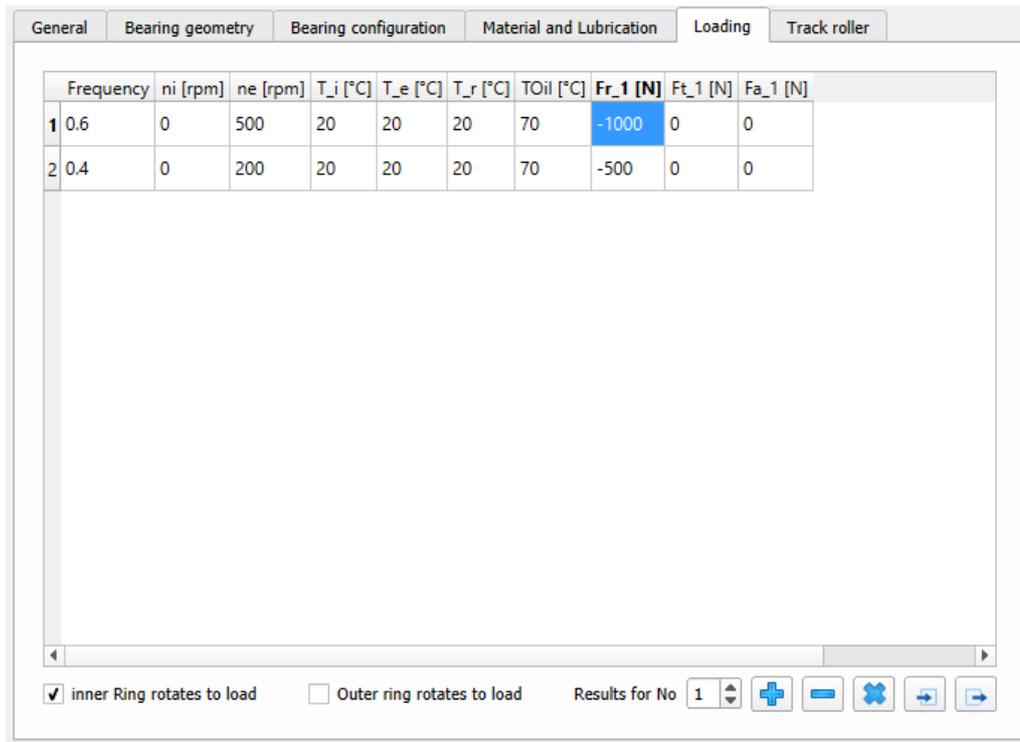
3.2   Thermal permissible speed

The thermal reference speed according (ISO 15312, 2018) and the thermal permissible speed according (DIN 732, 2010) are calculated by the software. The calculation can be run separately under menu 'Calculation'->'Thermal permissible speed'.

Most of the parameters can be overwritten by the user. Note that the calculation is just according to the standard. The equivalent load is calculated by axial and radial load, moment load is not considered. Also clearance is not considered.

For the thermal permissible speed in addition to DIN 732, the friction moments can be derived from ISO/TR 14179-2. The ISO standard provides different factors for grease and oil lubrication. As default the lubricant temperature as defined in the main window is used. Clearing the option "Assume fixed temperature for lubricant" uses the bearing temperature for the oil viscosity. The bearing temperature is the temperature of bearing surroundings plus temperature increase for the bearing.

If a bearing geometry is not considered in the standard, the input values for f0r, f1r and f1 can be entered by the user and the calculation can be run.

In case of bearing configurations, the values of f0, f1 and Ar apply for a single bearing. The oil volume flow applies to the set of bearings. Currently only bearing sets using radial bearings are supported.

The results for thermal reference speed, for thermal permissible speed and for the friction moments are also shown in the main report.

3.3 Grease operating life

The grease operating life is calculated according to Schaeffler using the mean value from given diagrams. The factor kf considering the bearing type has a range for some bearing types. It can by overwritten in the software. The factor KU can be used for additional reduction factors. The grease temperature limit is dependent on the grease. It is the temperature where no reduction in grease operating live is seen. For many standard greases it is 80°C, the default value in the software is 70°C to be conservative. The software takes into account the factor for temperature KT, the factor for loading KP and the factor for oscillation KR.
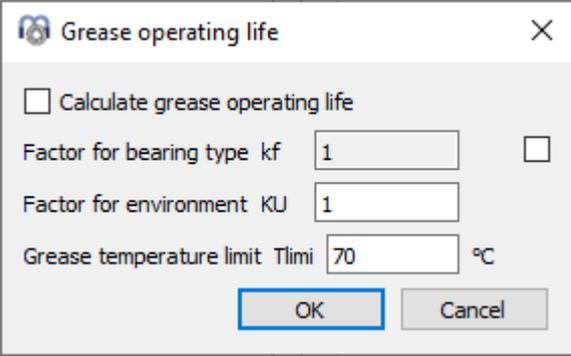
For hybrid bearings the grease operating life is increased by a factor of 3.3. A hybrid bearing is assumed if the Youngs modulus of the rolling element is larger than 270 GPa. Instead of the dynamic equivalent load P the dynamic equivalent reference load Pref is used in the calculation of grease operating life. This allows to consider pretension too. The base grease operating life is limited at 75'000h for small speed as grease life of several years is not reliable.

The result for grease operating life is just a guide value. Usually, a speed limit is also specified for greases, which has to be taken into account and which is not checked by the software.

3.4 Load rating diagram

A load rating diagram can be configured at menu 'Calculation'->'Load rating diagram'. It allow us to show four possible line charts, where the variables of the XY axes are compared against each other (Mz-Fx, Fx-Mz, Fy-Fx, Fx-Fy) while maintaining constant four types of values predefined by the user (see the picture). These diagrams are often used for slewing rings.
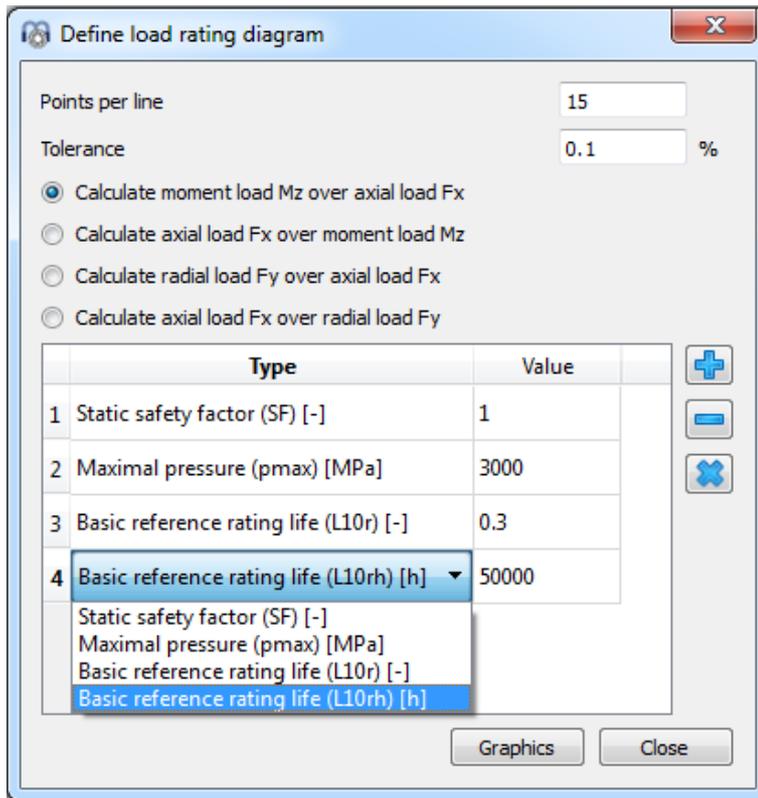
In the following picture it is shown a graphic example of the load rating diagram by which tilting moment (Mz) is compared against axial load (Fx) for a safety factor of 1, maximal pressure of 3000 MPa, 300.000 cycles and 50000 h.
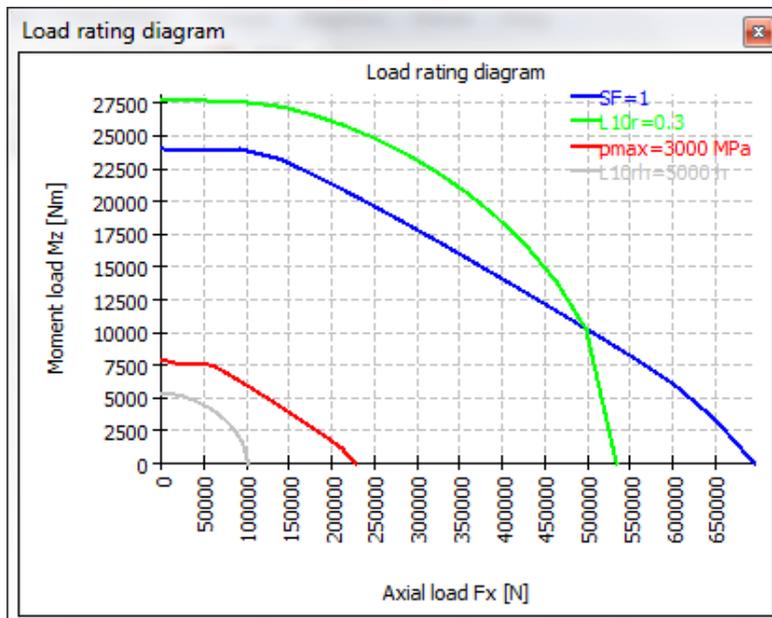
For each axial load, the tilting moment is increased until the selected criteria are reached. The number of points used for one line and a tolerance for the calculation can be defined.

3.5 Results

Results are provided as a small result overview directly in the software, a main PDF text report, a tolerance report and separate graphic windows. Graphic windows can be docked onto the main program interface and are automatically updated on each calculation.
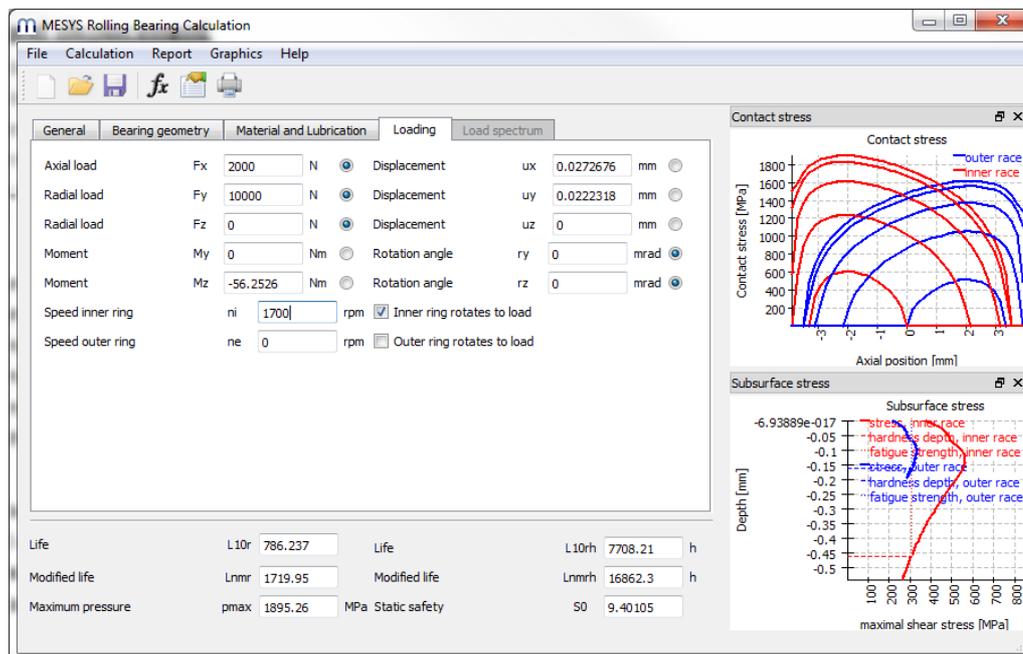
3.5.1 Result Overview

The result overview shows the reference life L_{10r} in 10^6 rotations and the reference life L_{10rh} in hours. The modified reference life L_{nmr} considering reliability and lubrication is also shown in 10^6 rotations and in hours. The maximum pressure and the static safety provide information about the maximum loading.

The results overview can be configured under Extras->Results overview.

The list on the left shows all available items for the result overview. Using the arrows selected items can be moved to the list on the right which is used for the result overview. The order of the selected results can be changed by the up/down arrows on the right. Note that only valid results values are actually shown in the result overview. So you will see fewer values than selected in the settings.

The settings for the result overview are saved in user settings not in the calculation file. So they will be the same for different calculations.

3.5.2   Main Result Report

The text report is provided in PDF format. An example report is available as separate document. The results are provided in sections 'Bearing inner geometry', 'Forces and displacement', 'Life', 'Subsurface stresses' and 'Damage frequencies'.
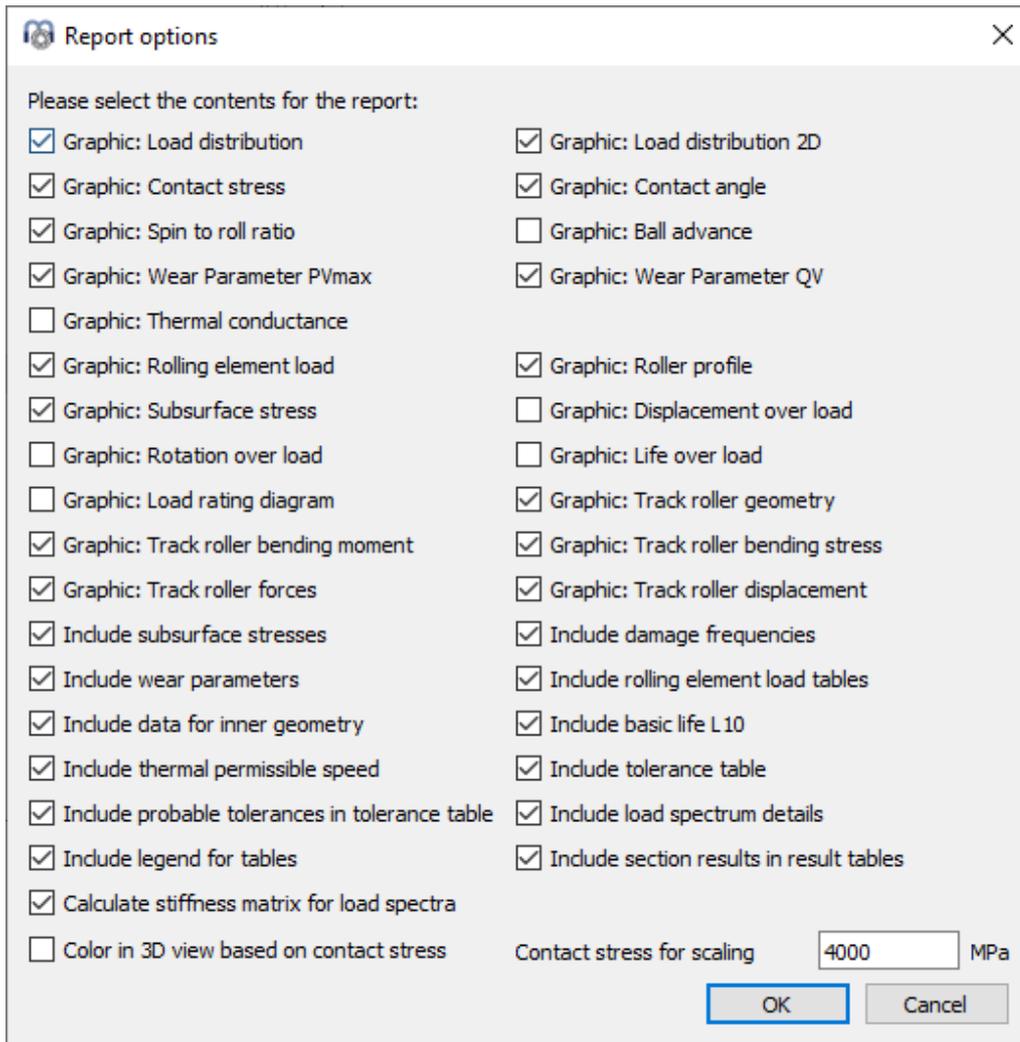
In the menu "Report->Options", the contents of the report can be configured. The graphics to be included can be selected and some sections of the report could be discarded if not of interest. The legend for all the table parameters can be shown in the report.

3.5.2.1 Saving main report

The main report can be saved using the menu "Report->Save Report As" in different formats. It can be saved as PDF, DOC, DOCX, XLSX or ODT (Open Office) format.

3.5.2.2 Customizing report logo

As default, a MESYS logo is shown on top of the report. This can be customized by defining a logo in the "mesys.ini" file. The logo should be created using a size, so that it fits into the header table. The DPI setting in the logo is considered, so that a high-resolution image could also be used. Additionally, it also exists the possibility to create an own template of the supported file formats (DOC, DOCX and ODT) and generate the report from it, thus having the chance to customize the logo.

3.5.3 Tolerances Report

In the menu "Report->Tolerance" report a special report for tolerances can be generated. While the tolerances for the selected calculation case are included in the main report, the additional report shows tolerances and pressure for maximum, medium and minimum clearance. If selected under options, the minimum probable and maximum probable clearances are included too.

3.5.4 Custom Report Templates

Custom report templates can be added in subdirectory "templates/rbc". Reports based on these templates can be generated by selecting the report template under "Report->Report templates".

The report templates are word files with DOCVARIABLE fields. These fields are filled on generation of the report.

Possible fields are:

- DOCVARIABLE VAR name [unit]: the variable with the name 'name' is printed. Optionally, a unit can be provided in brackets, for example [mm]. Additionally, a format could be added like %6.3g or %6.3f, where 'f' always generates a fixed point output and 'g' generates an exponential output for large numbers.

- DOCVARIABLE VARU name [unit]: the variable with the name 'name' plus the unit are printed. Optionally a unit can be provided in brackets, for example [mm].

- DOCVARIABLE TABLE tablename: A table is inserted.

- DOCVARIABLE GRAPHIC graphicName width=130 height=70: A graphic is inserted. Optionally, height and width can be given in millimeters.

Regarding details for variable names, table names and graphic names, please ask support@mesys.ch for additional documentation.

3.5.5 Result Graphics

Several results can be shown as graphics. All graphics are also appended to the report, so that a full documentation is provided in one document.

Using the context menu (right mouse button) in a graphics window, the graphic can be exported as file or be printed. For diagrams, "Diagram options" allow to adjust units and range of axes.

3.5.5.1 Bearing configuration

The bearing configuration shows the bearing rows and their position. It is mainly used together with the bearing configuration. The bearing inner and outer diameter have to be defined for generating this image.
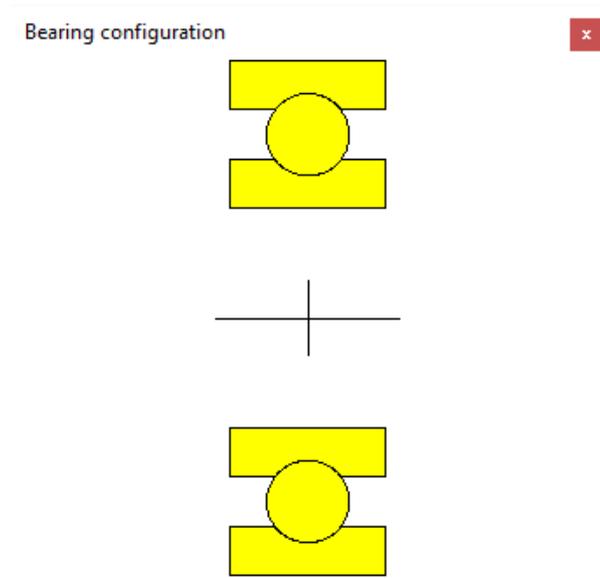
3.5.5.2 Load distribution

Here the contact stress of an angular contact bearing with radial and axial load and relatively high speed is shown. The contact stress is larger on the inner race (red) in the load zone but slightly larger on the outer race on the opposite side because of centrifugal loads.
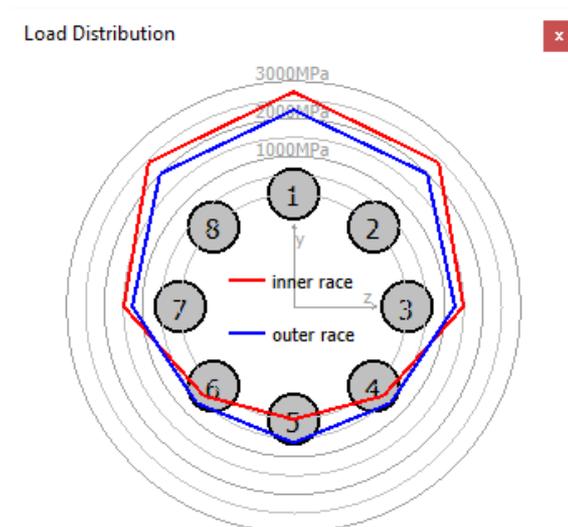
3.5.5.3   Load distribution 2D

The 2D load distribution shows the pressure distribution on inner and outer race as color strip. The maximum stress for the scale can be defined under report options. The graphic is available for a single load case and for the whole load spectrum.

This diagram also gives an impression of contact angle variation.
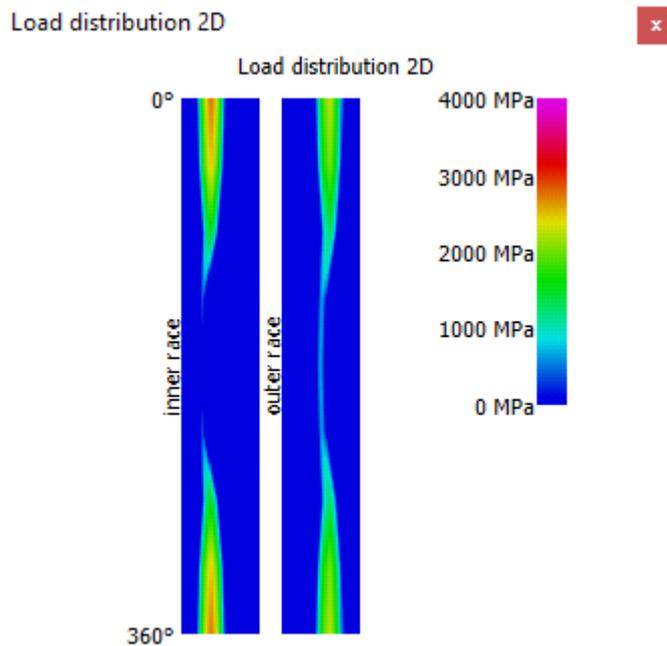
3.5.5.4   Load distribution 3D

This diagram shows the load distribution on a 3D visualization of the bearing. The stress level is indicated by the color of the rolling element and the contact ellipse. The visualization can be accurately rendered in any perspective.

The 3D views can be rotated with the left mouse button. Panning is activated by pressing "Shift", zooming by pressing "Ctrl".
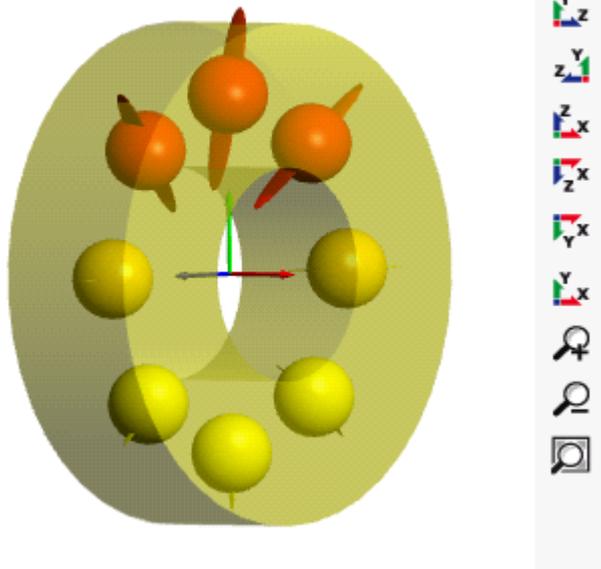
3.5.5.5 Contact angle

The contact angle is shown for an angular contact bearing with nominal contact angle of 20°. In the load zone the contact angles of inner and outer race are almost equal, but there is a big difference opposite to the load zone because of the centrifugal load.
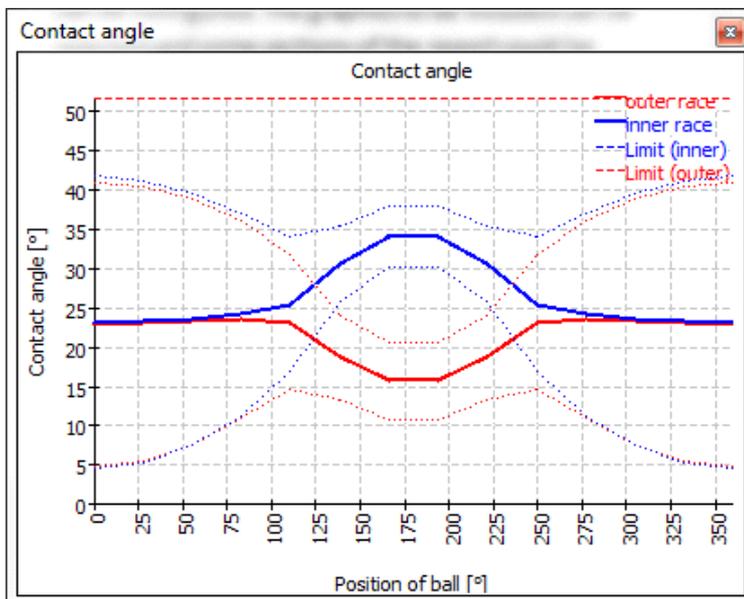
The extension of the contact ellipsis is shown with the dotted lines, so we have a wide contact ellipsis at 0° and a smaller one at 180°. The contact angle corresponding to the shoulder is shown as dashed straight line at the top, so the shoulder is high enough in this case.

3.5.5.6   Spin to roll ratio

The spin to roll ratio can be shown also. The calculation considers the setting for gyroscopic moment under settings. If no inner or outer raceway control is defined, then the distribution of spinning is based on coulomb friction.
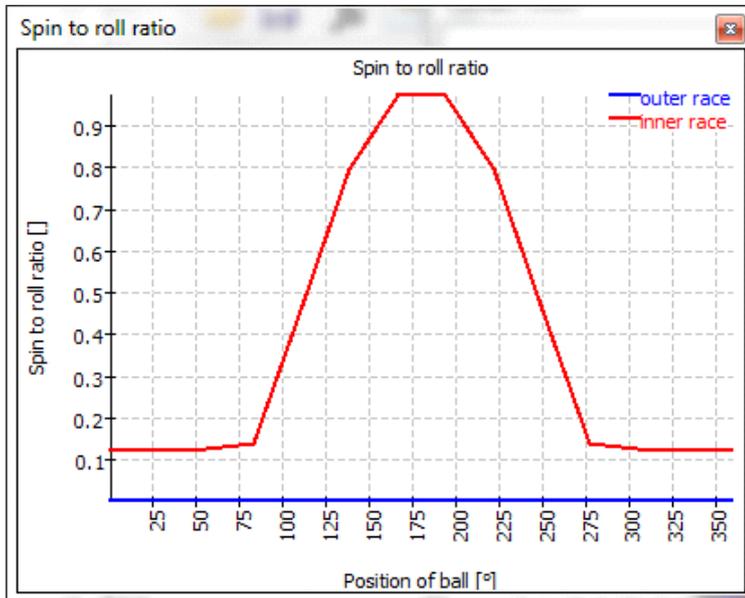
3.5.5.7   Circumferential ball advance

The circumferential ball advance is shown as graphics. It is calculated based on the assumption that the orbit speed of each ball is constant in one section. Because of variation of contact angles the ball orbit speed varies between the rolling element. This leads to variations in the distance between the rolling elements. The ball advance should be smaller than the clearance in the cage pocket in case of high speed.

3.5.5.8   Wear parameter QV

For ball bearings spinning and rolling speed are calculated in the contact ellipsis.

The wear parameter QV is calculated for each contact zone as $QV = \int (p \cdot v) dA$
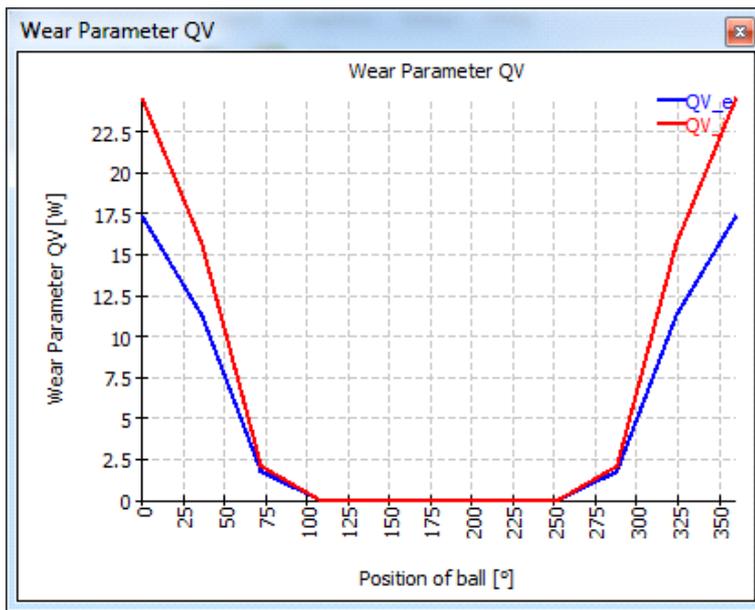
3.5.5.9 Wear Parameter PVmax

The wear parameter PVmax is the maximum value of the product pressure times sliding velocity in the contact ellipsis.
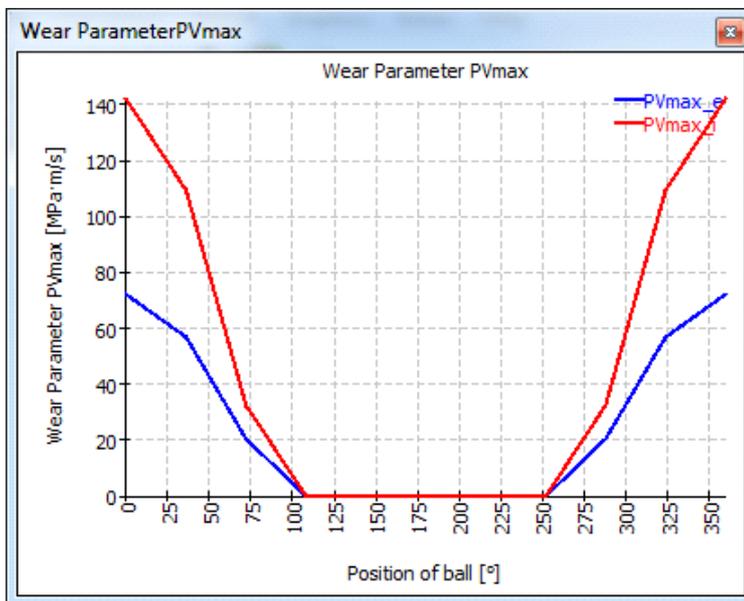
3.5.5.10 Contact stress for roller bearings

The contact stress for roller bearings is shown over the axial position or the length of the roller. This example is for a cylindrical roller bearing under radial and axial load and without misalignment between inner and outer race.

Because of the tilting of rollers under axial load the radial load is not distributed equally over the length of the roller.
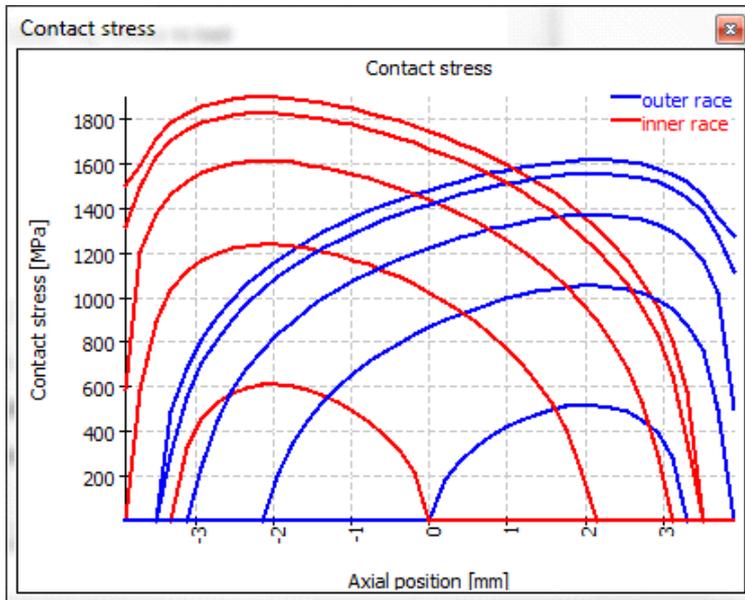
For axial cylindrical roller bearings and axial spherical roller bearings the roller orientations is inside to outside, while it is left to right for the other bearing types.

3.5.5.11 Roller profile

The roller profile and the profile of the races can be shown graphically. The profile can only be user-defined if the extended method for pressure distribution is activated.
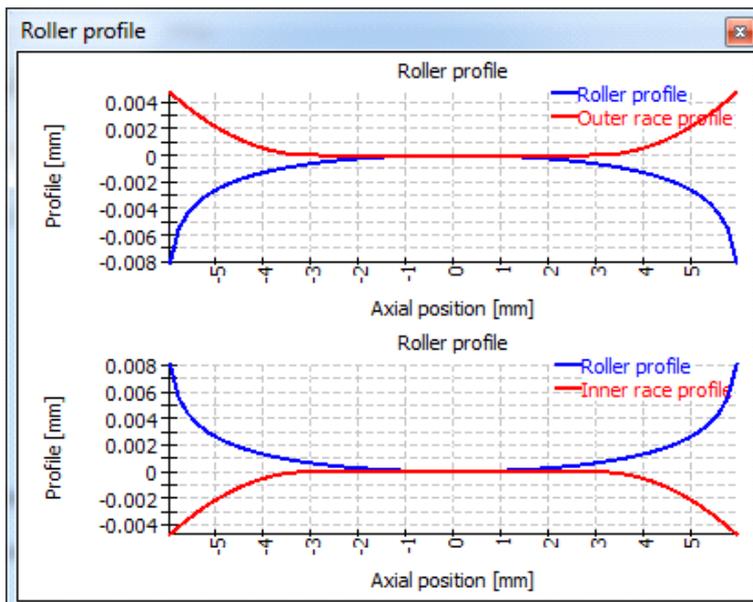
3.5.5.12 Subsurface stress

The subsurface shear stress is shown for the contact with the highest surface pressure. The required hardness depth is shown in the diagram dependent on the fatigue strength of inner and outer race.

The red curve is for the shear stress of the inner race, the blue curve for the outer race.
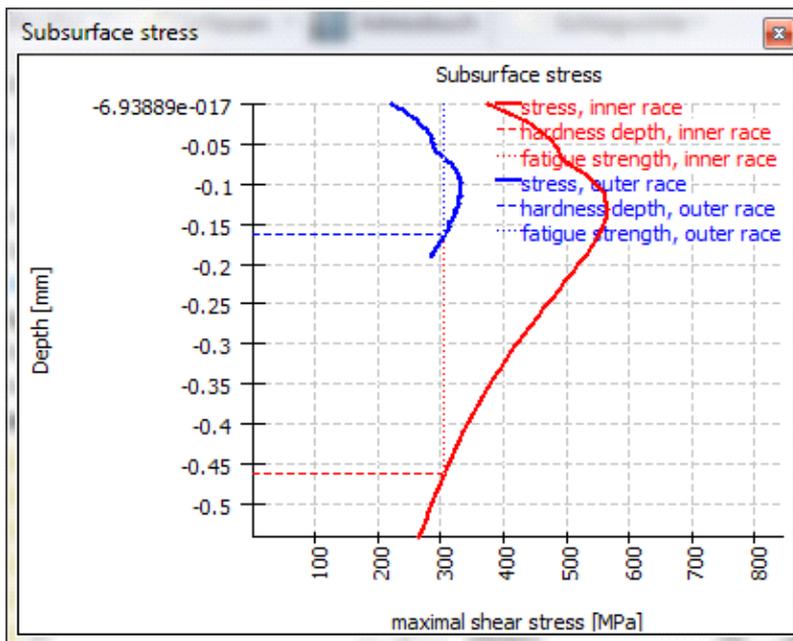
3.5.5.13 Load rating diagram

The load rating diagram shows the permissible range for tilting moment and axial load for different criteria which can be configured under 'Calculation'->'Load rating diagram'.
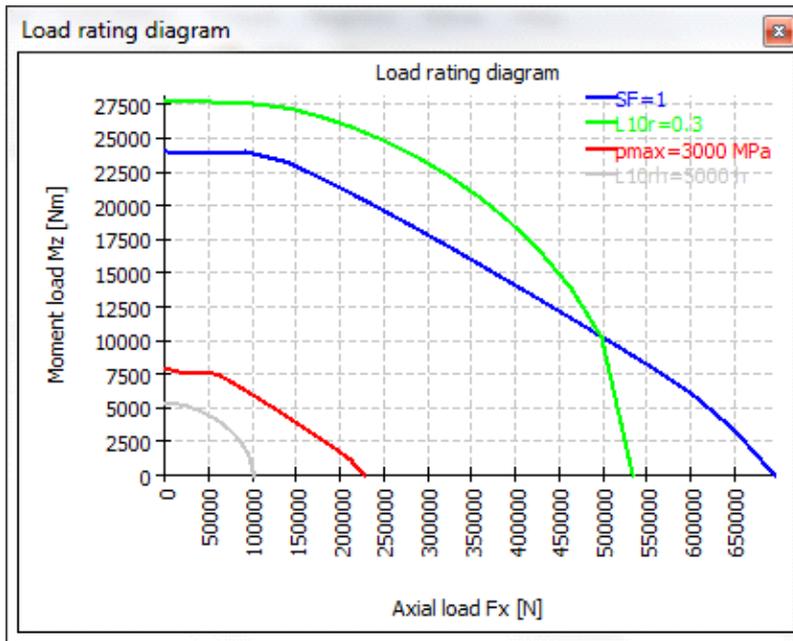
3.5.5.14 Track roller geometry

The track roller geometry is shown if defined using point data. For symmetric cross sections, the mirrored is shown in grey and the critical point for bending is marked by a red circle.
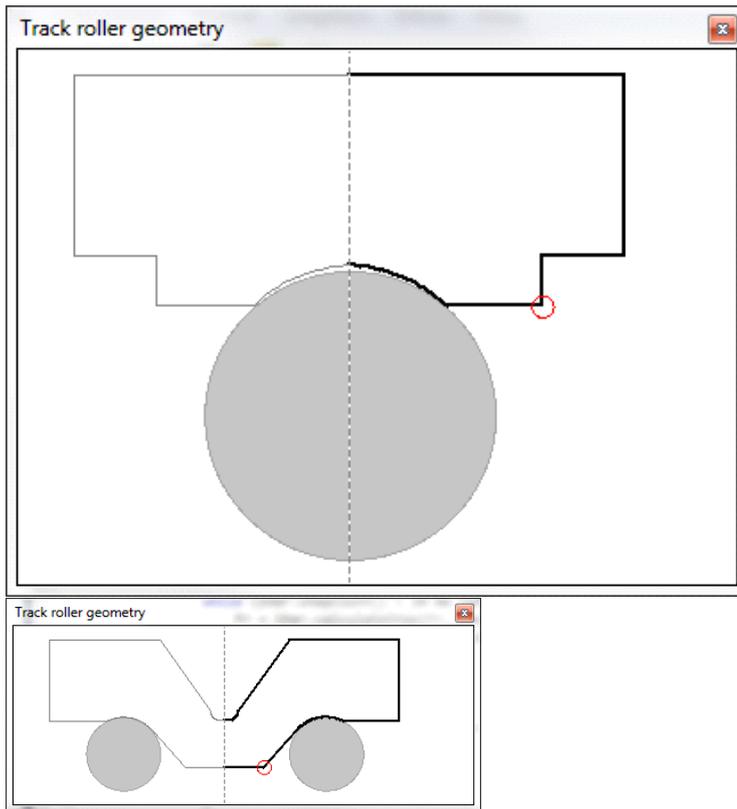
3.5.5.15 Track roller bending moment

The track roller bending moment is shown using the local coordinate system. For a pure radial load on a symmetric cross section, only the Moment around the bearing axis M_a occurs. The moment M_r is a tilting around a radial axis and the moment M_t is a torsional moment in the beam describing the outer ring.
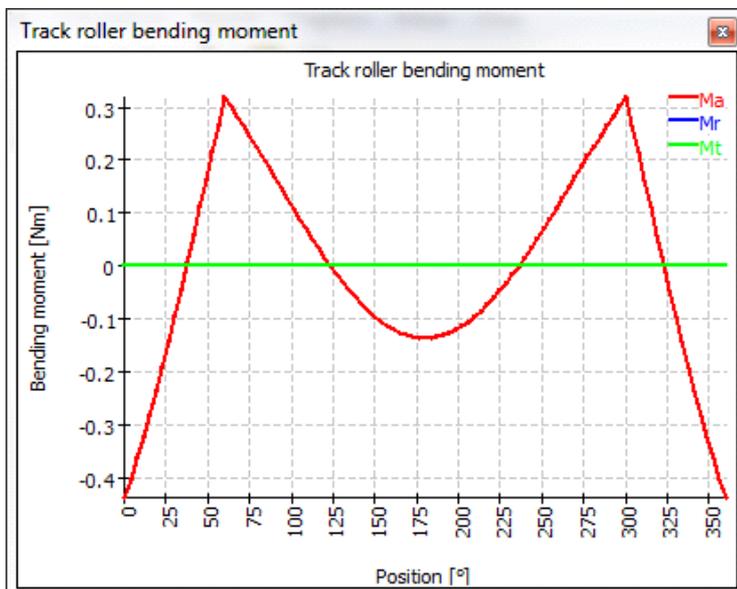
3.5.5.16 Track roller bending stress

The bending stress and the stress due to tension are shown in the diagram. The bending stress is calculated for a critical point either defined manually or automatically by the software. These stresses are used to calculate a static and dynamic safety factor for the outer ring.
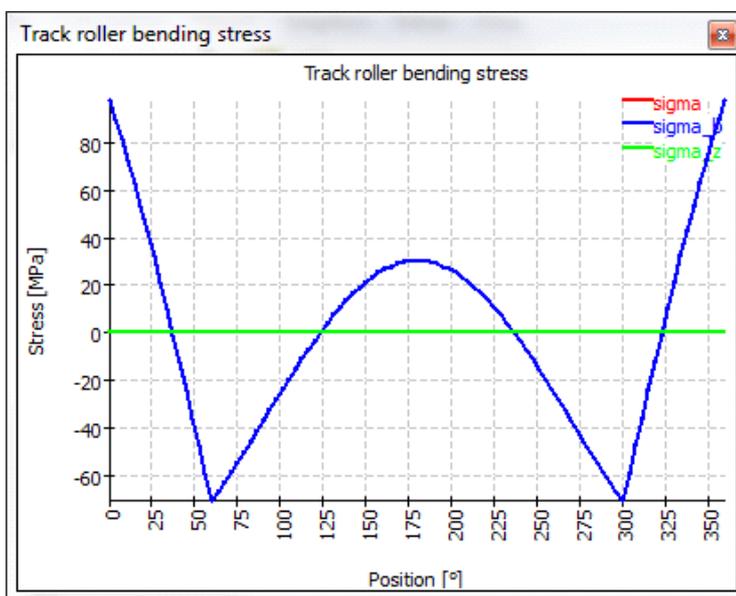
3.5.5.17 Track roller forces

The forces in the beam modeling the outer ring are shown. The force F_a is in the direction of the bearing axis, therefore it is a shear force in the beam. The force F_r is a shear force in radial direction and F_t is the tangential force, so it is tension (positive) or pressure (negative) in the beam.

We can also see that loads are only introduced at 0°, 60° and 300°, so only three rolling elements are under load.
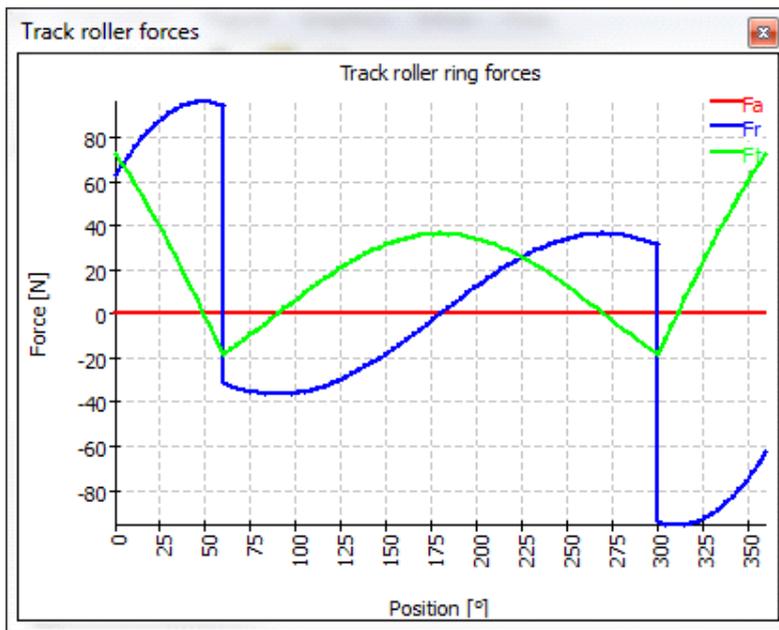
3.5.5.18 Track roller displacements

The displacements are shown in local coordinates and in global coordinated. The global coordinates allow seeing quickly that this example makes mainly a rigid body movement of the outer ring in negative y-direction. In other example,s the deformations of the ring might be different, so the values in local coordinates are more important.
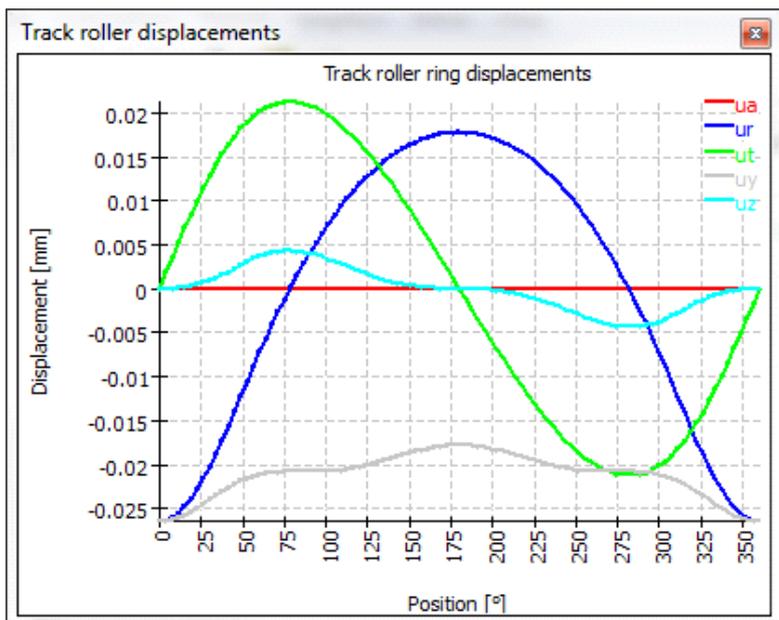
3.5.5.19 Displacement over load

The displacement over load can be shown as graphics. All load components are multiplied with a load factor and the displacements are shown. Only loads are multiplied with the load factor, displacements or rotations entered are not modified.

This diagram can be used as information about stiffness. Dependent on the type of loading (axial or radial), different stiffness can be calculated.
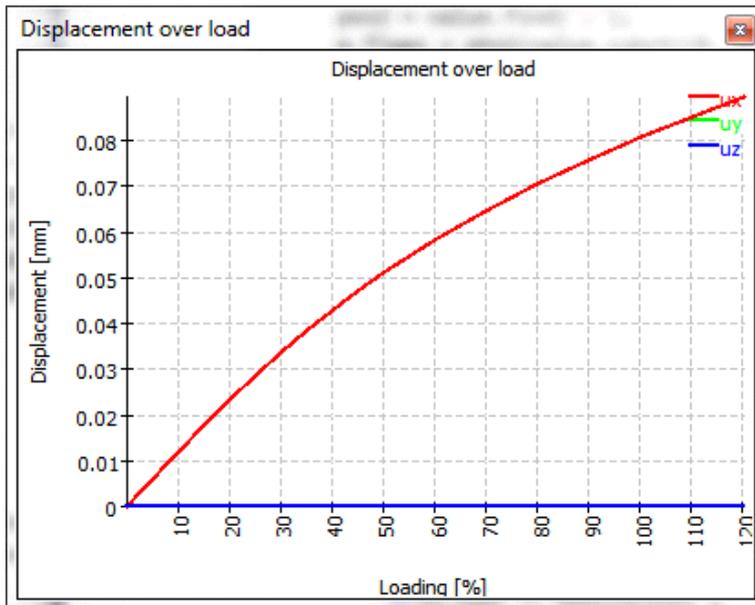
3.5.5.20 Rotation over load

Like in the graphics before, the rotations are shown dependent on a load factor. This graphic can only be shown if the moment is an input not for the tilting angle as an input. All loads are multiplied with the load factor.
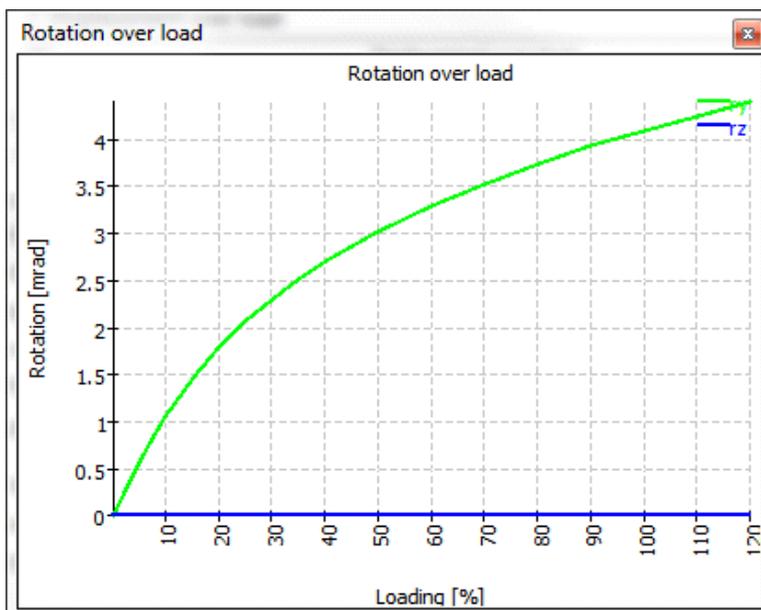
3.5.5.21   Life over load

The life over a load factor can be shown as a diagram. All loads and moments are multiplied by a load factor; all inputs for displacements/rotations stay constant. As reference the loading is shown on top of the diagram.
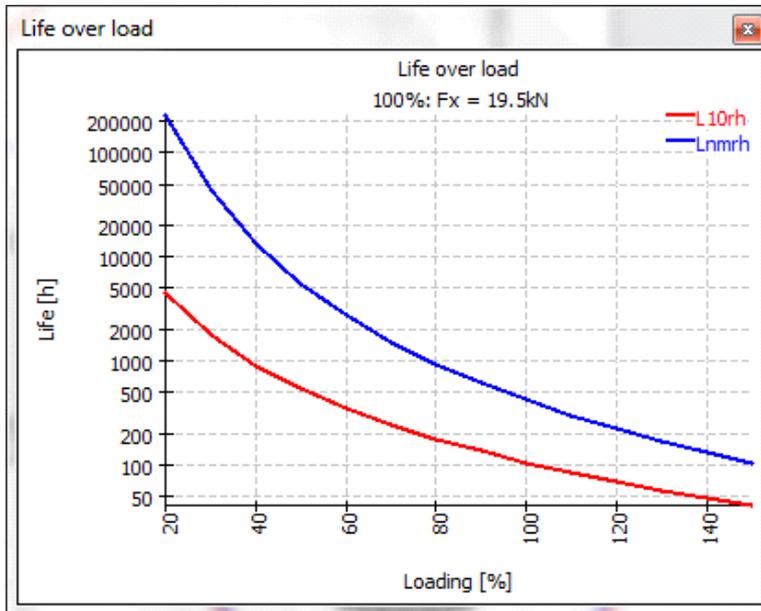
3.5.5.22   Radial expansion of races

The radial expansion of races is calculated using axisymmetric FEA considering fitting, temperatures and centrifugal forces. As comparison the radial expansion according to thick ring geometry as used for the calculation of fitting is shown. The dashed line shows the default calculation of fitting, the dotted line the calculation using equivalent cross section. This graphic can be used to check which approach is suited better for the actual conditions.

Radial expansion of races
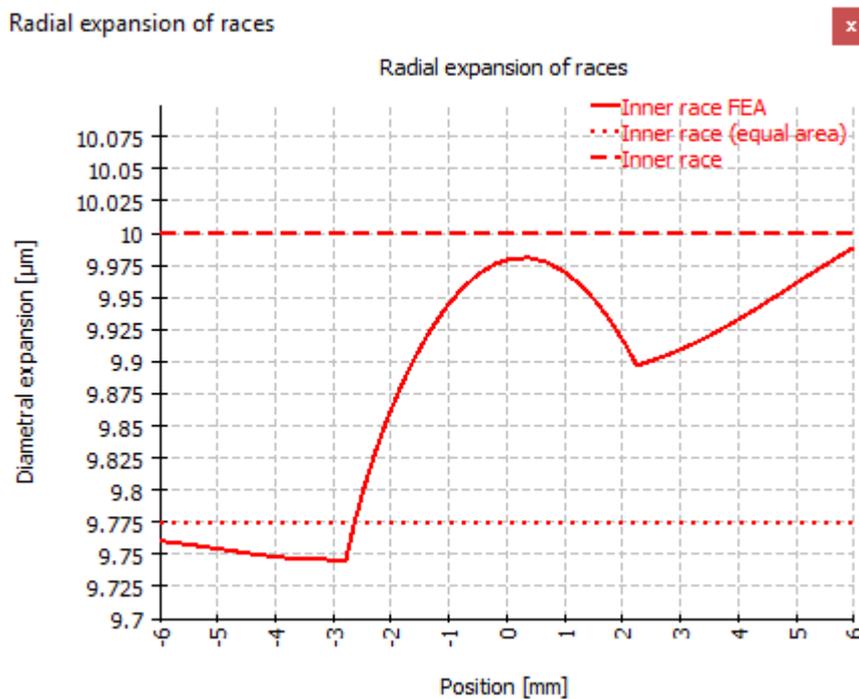
3.6   Bibliography

ISO/TR 10657. 1991. *Explanatory notes on ISO 76.* s.l. : ISO, 1991.

ISO/TR 1281-1. 2008. *Rolling bearings - Explanatory notes on ISO 281 - Part 1: Basic dynamic load rating and basic rating life.* s.l. : ISO, 2008.

AGMA 925. 2003. *Effect of Lubrication on Gear Surface Distress.* 2003.

Baalmann, K. 1994. Gleichung für die Sollviskosität nach DIN ISO 281. *Tribologie und Schmierungstechnik.* 1994.

Baly, Hatem. 2005. *Reibung fettgeschmierter Wälzlager.* Hannover : Universität Hannover, 2005.

Barnsby, Roger. 2003. *Life ratings for modern rolling bearings.* NewYork : ASME International, 2003.

Boresi, Arthur P. and Schmidt, Richard J. 2005. *Advanced Mechanics of materials.* s.l. : John Wiley&Sons, 2005.

Correns, Martin. 2015. Berechnung der Ermüdungslebensdauer von Wälzkörpern. *11. VDI Fachtagung Gleit- und Wälzlagerungen.* 2015.

de Mul, J. M., Kalker, J. J. and Fredrikson, B. 1986. The ContactBetween Arbitrary Curved Bodies of Finite Dimensions. *Journal of Tribology.* 1986, Vol. 108.

DIN 7190-1. 2017. *Pressverbände - Teil 1: Berechnungsgrundlagen und Gestaltungsregeln für zylindrische Pressverbände.* 2017.

DIN 732. 2010. *Wälzlager - Thermisch zulässige Betriebsdrehzahl - Berechnung und Beiwerte.* s.l. : DIN, 2010.

Fujiwara, Hiroki and Yamauchi, Kazuto. 2010. Tolerance Design of Logarithmic Roller Profiles in Cylindrical Roller Bearings. *Journal of Advanced Mechanical Design, Systems, and Manufacturing.* 4, 2010, Vol. 4.

Gohar, Ramsey. 2001. *Elastohydrodynamics.* London : Imperial College Press, 2001.

Harnoy, Avraham. 2003. *Bearing Design in Machinery.* New York, Basel : Marcel Dekker, 2003.

Harris, T., Rumbarger, J. H. and Butterfield, C. P. 2009. *Wind Turbine Dsign Guideline DG03: Yaw and Pitch Rolling Bearing Life.* Golden, Colorado : National Renewable Energy Laboratory, 2009.

Harris, Tedric A. and Kotzalas, Michael N. 2007. *Rolling Bearing Analysis: Advanced Concepts of Bearing Technology.* s.l. : CRC Press, 2007.

Harris, Tedric A. and Kotzalas, Michael N. 2007. *Rolling Bearing Analysis: Essential Concepts of Bearing Technology.* CRC Press : s.n., 2007.

Heemskerk, R. 1980. EHD lubrication in rolling bearings, review of theory and influence on fatigue life. *TRIBOLOGIA E LUBRICATIONE.* 1980.

Houpert, L. 1999. Bearing Life Calculation in Oscillatory Applications. *Tribology Transactions.* 1999, Bd. 42.

ISO 1002. 1983. *Rolling Bearings - Airframe bearings - Characteristics, boundary dimensions, tolerances, static load ratings.* 1983.

ISO 15312. 2003. *Rolling bearings - Thermal speed rating - Calculation and coefficients.* 2003.

ISO 281. 2007. *Rolling bearings - Dynamic load ratings and rating life.* 2007.

ISO 286-1. 2010. *ISO code sytem for tolerances of linear sizes - Basis of tolerances, deviations and fits.* 2010.

ISO 5753. 2009. *Rolling bearings - Internal clearance - Part 1: Radial internal clearance for radial bearings .* 2009.

ISO 76. 2006. *Rolling bearings - Static load ratings.* s.l. : ISO, 2006.

ISO/DIS 20056-1. 2016. *Rolling bearings - Load ratings for hybrid bearings will rolling elements made of ceramic - Part 1: Dynamic load ratings.* 2016.

ISO/DIS 20056-2. 2017. *Rolling bearings - Load ratings for hybrid bearings will rolling elements made of ceramic - Part 2: Static load ratings.* 2017.

ISO/TR 1281-2. 2008. *Rolling bearings - Explanatory notes on ISO 281 - Part 2: Modified rating life calculation, based on a systems approach to fatigue stresses.* s.l. : ISO, 2008.

ISO/TS 16281. 2008. *Rolling Bearings - Methods for calculating the modified reference rating life for universally loaded bearings.* s.l. : ISO, 2008.

Koch, Asmus. 1996. *Steigerung der Höchstdrehzahl von Schrägkugellagern bei Ölminimalmengenschmierung.* Aachen : Shaker Verlag, 1996.

Nakajima, Katsuhiko. 1995. Thermal Contact Resistance Between Balls and Rings of a Bearing Under Axial, Radial, and Combined Loads. *Journal of Thermophysics and Hear Transfer.* 9, 1995, Bd. 1.

Niemann, G. und Winter, H. 2003. *Maschinenelemente Band 2: Getriebe allgemein, Zahnradgetriebe - Grundlagen, Stirnradgetriebe.* Heidelberg : Springer, 2003.

—. 2004. *Maschinenelemente Band 3: Schraubrad-, Kegelrad-, Schnecken-, Ketten-, Riemen-, Reibradgetriebe, Kupplungen, Bremsen, Freiläufe.* Heidelberg : Springer, 2004.

Niemann, G., Winter, H. und Höhn, B.-R. 2005. *Maschinenelemente Band 1: Konstruktion und Berechnung von Verbindungen, Lagern, Wellen.* Heidelberg : Springer, 2005.

Paland, E.-G. 1968. Die Lastverteilung in einem auf der Ebene abgestützten Zylinderrollenlager. *Maschinenbautechnik (Schmierungstechnik).* 17, 1968, Bd. 8.

Schlecht, Berthold. 2007. *Maschinenelemente 1.* München : Pearson Studium, 2007.

—. 2010. *Maschinenelemente 2.* München : Pearson Studium, 2010.

Teutsch, Roman and Sauer, Bernd. 2004. An Alternative Slicing Technique to Consider Pressure Concentrations in Non-Hertzian Line Contacts. *Journal of Tribology.* 2004, Vol. 126.

MESYS Ball Screw Calculation

4 MESYS Ball Screw Calculation

This Ball Screw calculation (Version 06/2024) calculates the load distribution and the reference rating life according to ISO/TS 16281 (DIN 26281).

The inner geometry of the ball screws must be provided by the user. The operating clearance can be specified. The loading can be specified as force/moment or displacement/rotation independently for each of the five degrees of freedom. Bearing sets can be defined by using a configuration with multiple rows.

The following results can be found in the report:

- Reference rating life according ISO/TS 16281

- Load distribution between rolling elements

- Reaction forces/moments and displacements/rotations

- Contact pressure for each contact

- Static safety factor

In addition to the report, the results are shown in several graphics. Report and user interface are both available in metric or US customary units. Supported languages for user interface and report are English, German, French, Spanish, Turkish, Japanese, Korean and Chinese.

4.1 Input Parameters

4.1.1 General

On the first input page in addition to a project name, several settings can be done.
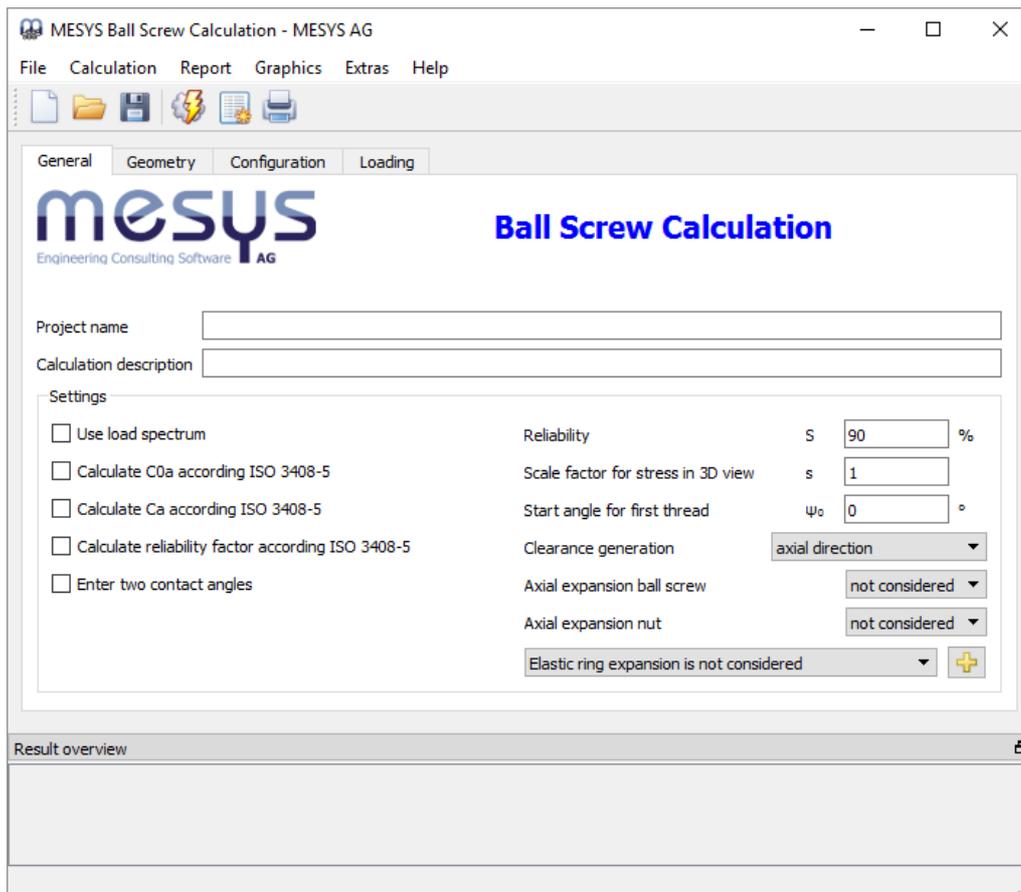
4.1.1.1   Project name and calculation description

The project name and the calculation description are just inputs which are shown in the report header. They can be used to enter information about the purpose of the calculation for documentation.

4.1.1.2   Use load spectrum

A load spectrum can be used instead of a single load case. This option can be activated here. For each load case, a full calculation using all factors is made. The resulting life is calculated using the life of each element (load case). Result graphics are only shown in the report for the selected result element of the load spectrum. However, the result graphics corresponding to any element number (case number) of the load spectrum can be shown under the menu 'Graphics'->'Load spectrum'.

 All other graphics available under the menu 'Graphics' are only valid for the selected element.

Additionally, under 'Graphics'->'Load distribution (Load spectrum)', all the load distribution cases are superimposed on the chart at the same time. This is also possible for the roller graphics.

Please note that more intermediate results are printed in the report if a calculation is done for a single load case.

4.1.1.3 Calculate COa according to ISO 3408-5

If this option is selected, the static load capacity is calculated according to (ISO 3408-5, 2006). In this case, material properties like Youngs modulus have no effect on the static load capacity.

If the option is not selected the static load capacity is calculated according (ISO 3408-5, 2006), but using f0 according (ISO/TR 10657, 1991) instead of k0. In this case, the static load capacity is based on a permissible contact stress of 4200 MPa as for axial ball bearings according (ISO 76, 2006).

4.1.1.4 Calculate Ca according to ISO 3408-5

If this option is set, the dynamic load capacity is calculated using the exponent 0.86 in equation 8 of (ISO 3408-5, 2006). If the option is not set, the exponent 0.7 is used as in (ISO 281, 2007), (ISO/TR 1281-1, 2008) or (Lundberg, et al., 1947).

The dynamic load capacity is calculated according

$$C_a = f_c \cdot (i \cdot \cos \alpha)^{0.86} \cdot \tan \alpha \cdot z_l^{2/3} \cdot D_w^{1.8} \cdot (\cos \alpha)^{1.3}$$

with either an exponent 0.86 or an exponent of 0.7.

4.1.1.5 Calculate reliability factor according to ISO 3408-5

The modification factor for reliability is calculated according to the two parameter Weibull distribution according (ISO 3408-5, 2006) if this option is set. If the option is not selected, the three parameter Weibull distribution as in (ISO 281, 2007), (ISO/TR 1281-1, 2008) is used.

4.1.1.6 Enter two contact angles

Either one or two contact angles can be defined. Different contact angles can be used for screw and nut. The contact angle is defined by radial contact of the ball to either screw or nut.

4.1.1.7 Reliability S

As default, the bearing life is calculated for a reliability of 90%. The required reliability can be changed here with limits 90% and <100%. The life modification factor for reliability 'fac' is calculated according to this input using either the three parameter Weibull relationship as given as formula in (ISO/TR 1281-2, 2008) or the two parameter Weibull relationship used in (ISO 3408-5, 2006).

4.1.1.8 Scale factor for stress in 3D view s

The stress distribution is scaled to s*Dw for a contact stress of 4000MPa in the 3D view.

4.1.1.9 Start angle for first thread Ψ_0

The start angle defines the angular position of the left end of the first thread. With an angle of 0°, the start is on the y-axis. The angle defines the position of the middle of the gap before the first ball.

4.1.1.10 Clearance generation

The nominal contact angle, ball diameter and radii of races are used to define the positions for centers of curvature in the case of zero clearance.

There are four possibilities to define clearance:

1. Centers of curvature are moved axially. This is only possible for gothic profiles with contact angle larger zero. The maximum clearance that can be achieved is limited.

2. Centers of curvature are moved radially.

3. Centers of curvature are moved in direction of contact angle. This cannot be used in case of contact angle zero.

4. Clearance is generated by reduction of ball diameter. This slightly reduces load capacities as the reduced ball diameter is used for calculation of load capacity.

4.1.1.11 Axial expansion of ball screw and nut

The axial expansion of the ball screw and the nut three options are available: "Not considered", "fixed left", "fixed right". In case the load is introduced on one side, the screw will axially expand or compress because of the axial stresses. "Fixed left" means that the load is introduced on the left of the element and the axial stress will reduce to the right.

4.1.1.12 Elastic ring expansion

An elastic expansion/compression because of radial forces can be considered. The calculation is based on thick ring theory with constant pressure at the outside or inside. Either the minimum or the medium contact force of the balls is used to calculate this elastic deformation. The deformation is constant over the whole length of the nut. Therefore, the method is not suited for a pure moment load.

4.1.2 Ball screw geometry

4.1.2.1 Load capacities

The load capacities can be an input or an output. If they are given by the user they will not be changed because of surface hardness of the material. The surface hardness is only considered if the values are calculated by the software.

The load capacity will be calculated using the free contact angle $\alpha 0$, not the nominal contact angle α. This mainly makes a difference for nominal contact angle of zero in case of a full radius in races.

4.1.2.1.1 Dynamic load number

The dynamic load number is used for the calculation of bearing life. It is calculated according (ISO 3408-5, 2006) with factor fc according (ISO/TR 1281-1, 2008). Either 0.86 or 0.7 according (ISO 281, 2007) is used as an exponent.

The dynamic load number can be influenced by a modification factor available at material properties.

4.1.2.1.2 Static load number

The static load number is calculated according (ISO 76, 2006) and (ISO/TR 10657, 1991) or according (ISO 3408-5, 2006).

4.1.2.2　Number of starts

The number of starts defines the number the ratio between lead and pitch or the number of parallel ball tracks.

4.1.2.3　Number of turns

Each ball track has this number of turns. It does not have to be an integer value.

4.1.2.4　Lead

The lead is the axial travel of the nut for a 360° rotation.

4.1.2.5　Lead angle

Instead of the lead, the lead angle can be used as input. The lead is
$$P_h = D_{pw} \cdot \pi \tan \varphi$$

4.1.2.5.1 Direction

The lead can either be right or left handed. The sense of rotation is defined by this input.

4.1.2.6　Number of loaded balls per turn

The number of loaded ball per turn us usually calculated according (ISO 3408-5, 2006). It can be overwritten in case spacers are used.

4.1.2.7　Ball diameter

The diameter of the balls is specified here.

4.1.2.8 Pitch diameter

The pitch diameter is the diameter of the centers of rolling elements when they have the same amount of clearance to both races. The diameters of inner and outer race are calculated by this value, the diameter of rolling element, the contact angle and the clearance.

4.1.2.9 Contact angle

Either one or two contact angles can be defined. For contact angle larger zero a gothic profile is used, for contact angle of zero a full radius is used.

If a full radius should be used, the clearance has to be generated either radially or by reduction of ball diameter.

4.1.2.10 Conformity of ball screw and ball nut

The conformity is the ratio between radius of curvature of a race and the ball diameter. For geometric reasons, the value has to be larger than 0.5. The values used for the calculation of tables in ISO 281 are $f_i = f_e = 0.52$ for radial ball bearings and $f_i = f_e = 0.535$ for thrust ball bearings as stated in (ISO/TR 1281-1, 2008).

Using the ⊞-button behind the input field, the conformity can be calculated from a given radius. Ensure that you input the correct roller and pitch diameter before.

4.1.2.11 Shoulder diameter of inner and outer ring

The shoulder diameter of inner and outer ring can be defined.

The shoulder diameter is compared to the maximum extension of the contact ellipsis. A warning is shown if the ellipsis would extend above the shoulder. The Hertzian stress is not valid in that case.

The required shoulder diameter is shown in the report together with a length ratio eLR_i, eLR_e which is showing a safety of shoulder length. The length ratio is defined as the length from lower end of contact ellipsis to shoulder (the green line the graphics) divided by the length of the contact ellipsis 2a (red line in the graphics). The value should therefore be larger than 1 or 100%.

Instead of using an absolute value, the shoulder height can be defined using a percentage of ball diameters. A factor of 50% would be a shoulder up to the pitch diameter, so the factor should be between 10% and 40%. Using this factor allows to have default geometry on changing ball diameter or pitch diameter.

4.1.2.12 Axial clearance

The axial clearance is the difference between the possible axial displacement in positive and negative direction. It can be negative for gothic profiles, but it has to be positive for full radius profiles.

A pretension can be entered by using a negative value for the axial clearance Pa. A ▤-button allows the input of radial clearance instead of axial clearance or a free contact angle can be used for full radius profiles.

4.1.2.13 Tolerance class

The tolerance class will be considered in calculation of static and dynamic load capacity.

4.1.2.14 Material

The material of the ball screw the nut and the balls can be defined here. Using the ➕-button, the hardness and material treatment can be defined, too.

The material of the ball does only affect the contact stiffness in calculation of load distribution, it is not considered for the calculation of life.

4.1.2.15 Ball screw inner diameter

The inner diameter in case of a hollow screw. This input is only used in case axial or radial expansion is considered.

4.1.2.16 Nut outer diameter

The outer diameter of the nut. This input is only used in case axial or radial expansion is considered. In case the input value is zero, a diameter of Dpw + 20*Dw is used internally.

4.1.2.17 Ball screw lead error

An optional lead error for the ball screw. This will affect the load distribution.

4.1.3 Bearing configuration

A group of nuts can be considered in the calculation. The nuts are assumed to be connected to a rigid shaft and housing. The loads or displacements are given for the origin of the bearing group instead for each single bearing.

A relative position can be entered for each nut. An axial offset allows the definition of pretension in the system. The offset is an additional movement of the ball nut. The displacements, forces and moments under loading are applied at the reference position 0. So in most cases the positions should be chosen symmetrically around zero.
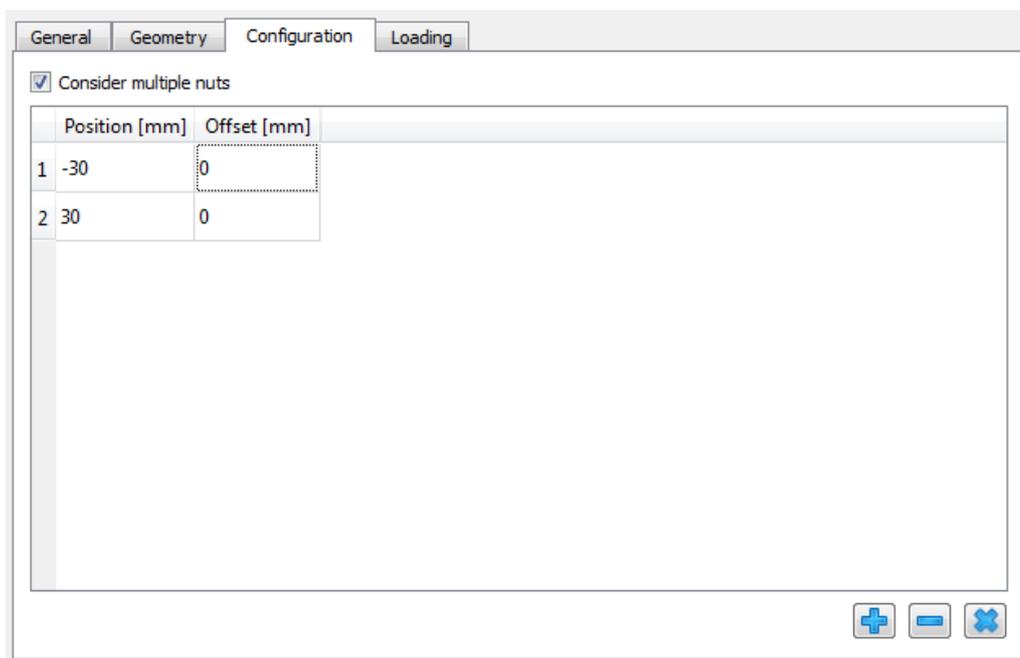
Additional nuts can be added using the ➕-button. A row can be deleted using the ➖-button and the ✖-button clears all inputs.

With the context menu (right mouse button) in the table, several results values can be shown in the table. This allows seeing quickly how load or life is distributed among the nuts.

4.1.4   Loading

For each coordinate direction, you can select if a force or a displacement should be entered.

A moment load can be entered for two directions only, because the torque around the axis is connected to the axial force. The rotation angle rx will affect the axial displacement only.

Use the radio buttons to select the elements you want to enter.

4.1.4.1   Coordinate system

The coordinate system is defined with x as the axial direction. The y-axis (in green) points up to the middle of the gap before the first ball and the angle ψ is positive around the x-axis or clockwise in the diagram on the right (looking in the direction of the x-axis). The angle starts with zero on the y-axis; the start angle of the ball screw can be changed under settings. Moments are positive if acting around the corresponding axis.
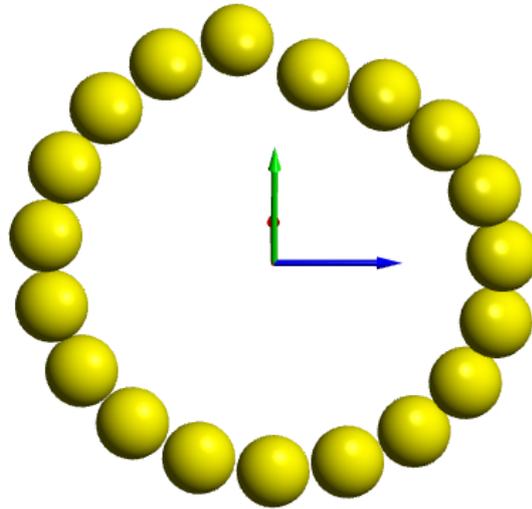
The load is acting on the inner ring, so a positive load in y-direction leads to a loading of the rolling elements on the top.

4.1.4.2   Inner ring rotates to load

The resulting life is slightly different if the load distribution on a race is stationary or rotating. For many bearings, the inner ring rotates relative to the load. This is the case if the inner ring is connected to a rotating shaft and the load is stationary in space.

What is the correct input if only an axial load occurs? In this case, the input doesn't matter since the results are the same for both settings if all the rollers have the same loading.

4.1.4.3   Outer ring rotates to load

For the outer ring, the same is valid as for the inner ring. In many applications the outer ring is stationary to the load.

4.1.4.4   Temperature of screw and nut

The temperatures of the shaft and the nut are considered in axial elongation of the screw and the nut and they will affect the radial clearance due to radial thermal expansion.

4.1.5 Load spectrum

If at the tab-page 'General' the flag for the calculation with load spectrum is set, instead of for a single load case, the calculation will be done for a load spectrum. The load or displacement, the rotation speed and the temperatures can be specified for each element.

	Frequency	Fx [N]	Fy [N]	Fz [N]	rx [mrad]	ry [mrad]	rz [mrad]	ni [rpm]	ne [rpm]	Ti [°C]	Te [°C]
1	0.45	8000	0	0	0	0	0	1000	0	20	20
2	0.35	5000	0	0	0	0	0	50	0	20	20
3	0.1	2000	0	0	0	0	0	100	0	20	20

General | Geometry | Configuration | **Loading**

☑ inner Ring rotates to load   ☐ Outer ring rotates to load   Results for No 1
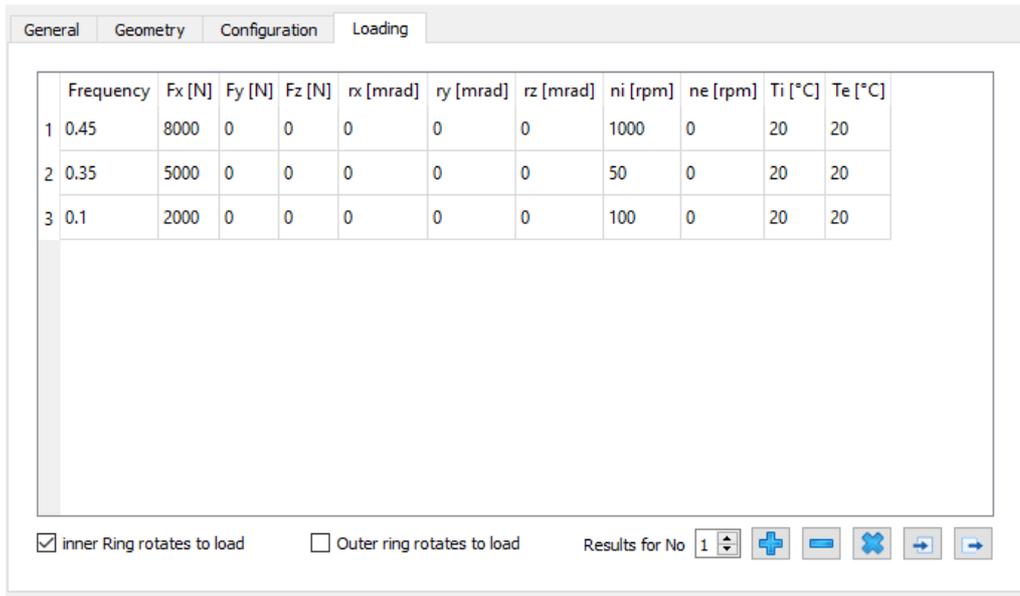
Using the context menu that opens clicking the right mouse button, you can select which forces or moments should be used as input values. If a value is not selected, the corresponding displacement/rotation can be entered.

Rows can be added using the ➕-button or a selected row can be deleted using the ➖-button. The ✖-button clears all the inputs.

Using the ➡-button, the load spectrum is read from a file. The columns that are currently shown with its units have to be used for the input file. As the data from the file is added to the inputs, you must clear the inputs first to define a full load spectrum. Likewise, a created load spectrum table can be exported with the corresponding ➡-button.

As for the single load case, a selection is available if the ring is rotating relative to the load or not. A ring not rotating to the load would be on the safe side.

The load distribution is calculated for each load case. The resulting life for the whole load spectrum is shown as result and the minimum static safety of all elements. The result graphics are only shown for the element of the load spectrum that is chosen as results on this page.

4.2   Results

Results are provided as a small result overview directly in the software, a main PDF text report and separate graphic windows. Graphic windows can be docked onto the main program interface and are automatically updated on each calculation.

4.2.1   Result Overview

The result overview shows the reference life L_{10r} in 10^6 rotations and the reference life L_{10rh} in hours. The maximum pressure and the static safety provide information about the maximum loading.

The results overview can be configured under Extras->Results overview.

The list on the left shows all available items for the result overview. Using the arrows selected items can be moved to the list on the right which is used for the result overview. The order of the selected results can be changed by the up/down arrows on the right. Note that only valid results values are actually shown in the result overview. So you will see fewer values than selected in the settings.

The settings for the result overview are saved in user settings not in the calculation file. So they will be the same for different calculations.

4.2.2 Main Result Report

The text report is provided in PDF format.

In the menu "Report->Options" the contents of the report can be configured. The graphics to be included can be selected and some sections of the report could be discarded if not of interest. The legend for all the table parameters can be shown in the report.

4.2.2.1 Saving main report

The main report can be saved using the menu "Report->Save Report As" in different formats. It can be saved as PDF, DOC, DOCX or ODT (Open Office) format.

4.2.2.2 Customizing report logo

As default, a MESYS logo is shown on top of the report. This can be customized by defining a logo in the "mesys.ini" file. The logo should be created using a size, so that it fits into the header table. The DPI setting in the logo is considered, so that a high-resolution image could also be used. Additionally, it also exists the possibility to create an own template of the supported file formats (DOC, DOCX and ODT) and generate the report from it, thus having the chance to customize the logo.

4.2.3 Result Graphics

Several results can be shown as graphics. All graphics are also appended to the report, so that a full documentation is provided in one document.

Using the context menu (right mouse button), in a graphics window the graphic can be exported as file or be printed.

4.2.3.1 Load distribution 3D

Here the contact stress of a ball screw is shown as 3D-view.
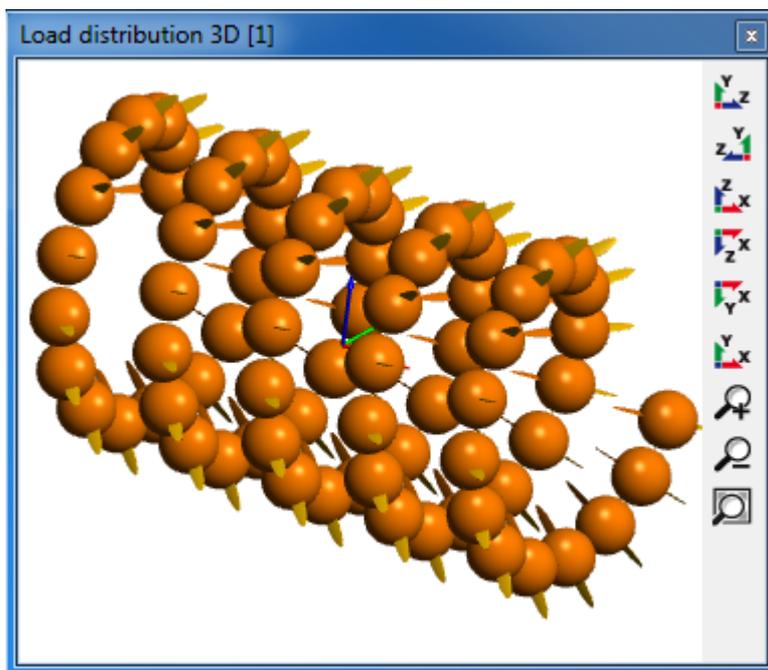
4.2.3.2 Contact stress

The contact stress is shown for each ball and each of the four possible contacts.
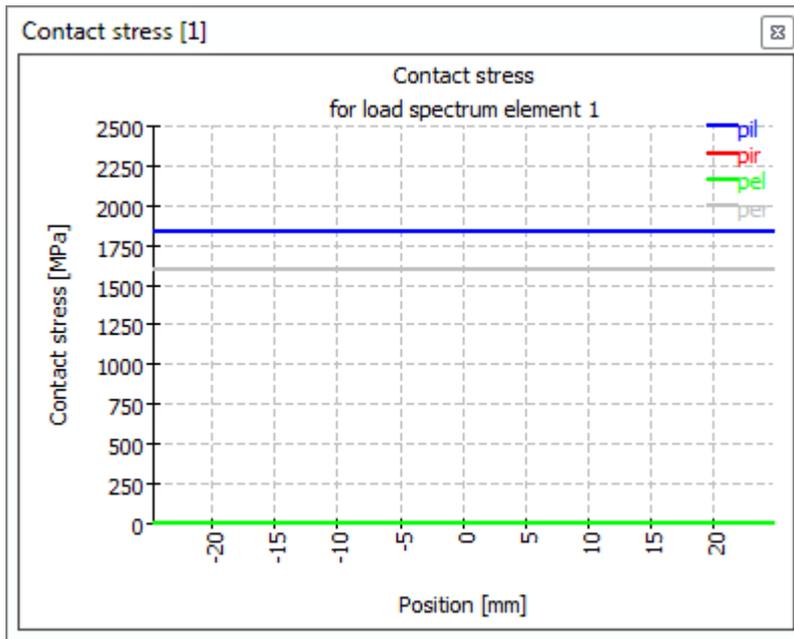
4.2.3.3 Contact angle

The contact angle is shown for a nominal contact angle of 40°. As for typical axially loaded ball screws, both the load level for all balls and the contact angles of inner and outer race are almost equal along the axial position.

The extension of the contact ellipsis is shown with the dotted lines. The contact angle corresponding to the shoulder is shown as dashed straight line at the top, so the shoulder is high enough in this case.
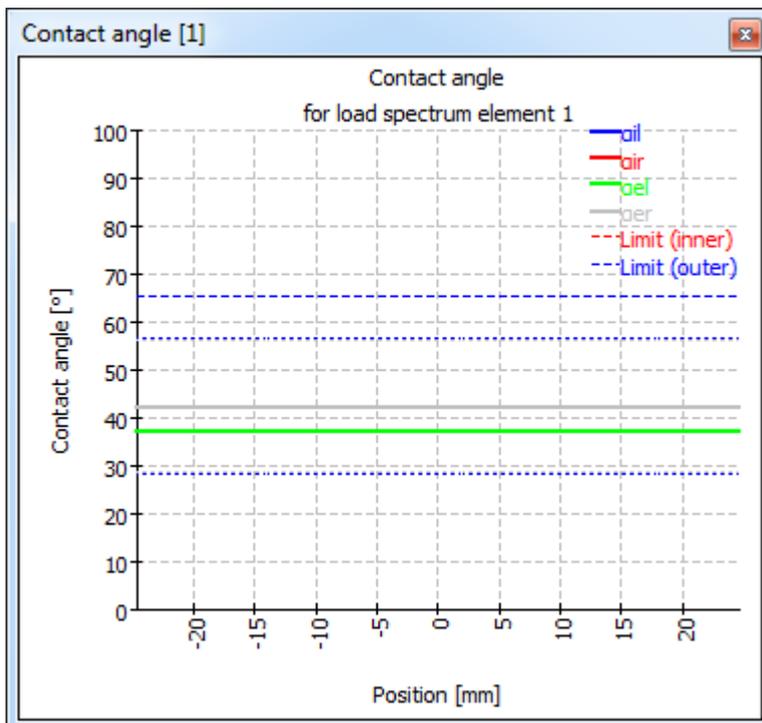
4.2.3.4 Ball loads

This diagram shows the forces on each ball for the up to four contacts.
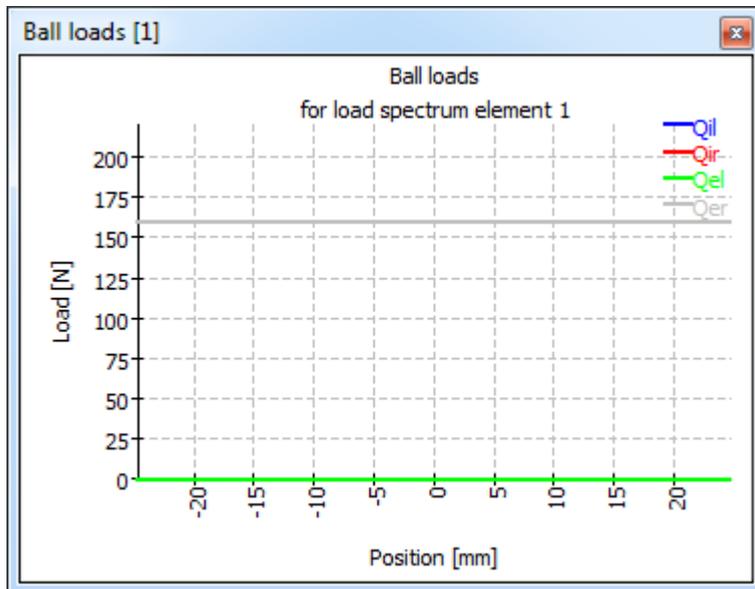
4.2.3.5 Circumferential ball advance

The circumferential ball advance is shown as graphics. It is calculated based on the assumption that the orbit speed of each ball is constant in one section. Because of variation of contact angles the ball orbit speed varies between the rolling element. For calculation is done in the same way as for ball bearings, the lead is not taken into account. A decrease in ball advance means that the next ball is slower and this will lead to forces between the balls. The calculation of orbit speed does not take into account a contact between the balls.

Circumferential ball advance
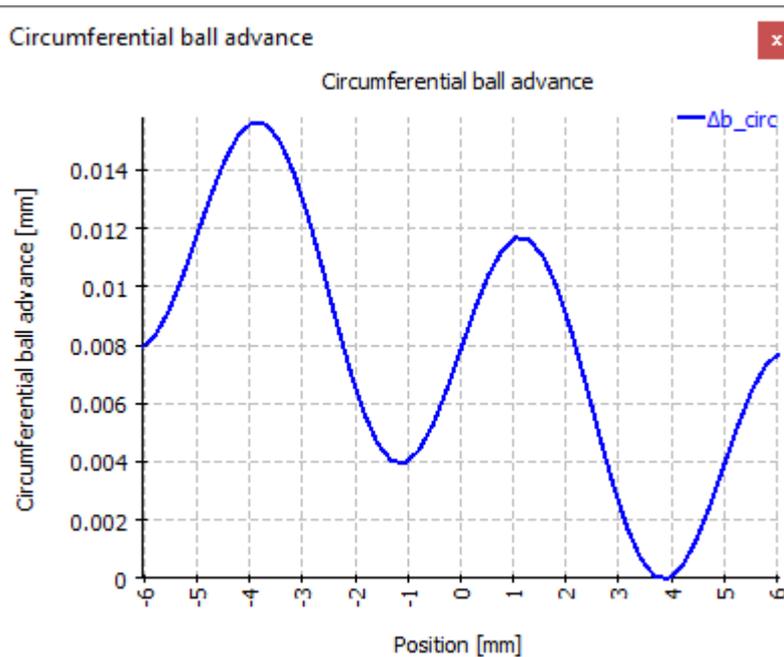
4.3 Bibliography

ISO/TR 10657. 1991. *Explanatory notes on ISO 76.* s.l. : ISO, 1991.

ISO/TR 1281-1. 2008. *Rolling bearings - Explanatory notes on ISO 281 - Part 1: Basic dynamic load rating and basic rating life.* s.l. : ISO, 2008.

AGMA 925. 2003. *Effect of Lubrication on Gear Surface Distress.* 2003.

Baalmann, K. 1994. Gleichung für die Sollviskosität nach DIN ISO 281. *Tribologie und Schmierungstechnik.* 1994.

Barnsby, Roger. 2003. *Life ratings for modern rolling bearings.* NewYork : ASME International, 2003.

Boresi, Arthur P. and Schmidt, Richard J. 2005. *Advanced Mechanics of materials.* s.l. : John Wiley&Sons, 2005.

de Mul, J. M., Kalker, J. J. and Fredrikson, B. 1986. The ContactBetween Arbitrary Curved Bodies of Finite Dimensions. *Journal of Tribology.* 1986, Vol. 108.

DIN 7190. 2001. *Pressverbände - Berechnungsgrundlagen und Gestaltungsregeln.* 2001.

DIN 732. 2010. *Wälzlager - Thermisch zulässige Betriebsdrehzahl - Berechnung und Beiwerte.* s.l. : DIN, 2010.

Fujiwara, Hiroki and Yamauchi, Kazuto. 2010. Tolerance Design of Logarithmic Roller Profiles in Cylindrical Roller Bearings. *Journal of Advanced Mechanical Design, Systems, and Manufacturing.* 4, 2010, Vol. 4.

Harnoy, Avraham. 2003. *Bearing Design in Machinery.* New York, Basel : Marcel Dekker, 2003.

Harris, T., Rumbarger, J. H. and Butterfield, C. P. 2009. *Wind Turbine Dsign Guideline DG03: Yaw and Pitch Rolling Bearing Life.* Golden, Colorado : National Renewable Energy Laboratory, 2009.

Harris, Tedric A. and Kotzalas, Michael N. 2007. *Rolling Bearing Analysis: Advanced Concepts of Bearing Technology.* s.l. : CRC Press, 2007.

Harris, Tedric A. and Kotzalas, Michael N. 2007. *Rolling Bearing Analysis: Essential Concepts of Bearing Technology.* CRC Press : s.n., 2007.

Heemskerk, R. 1980. EHD lubrication in rolling bearings, review of theory and influence on fatigue life. *TRIBOLOGIA E LUBRICATIONE.* 1980.

Houpert, L. 1999. Bearing Life Calculation in Oscillatory Applications. *Tribology Transactions.* 1999, Bd. 42.

ISO 15312. 2003. *Rolling bearings - Thermal speed rating - Calculation and coefficients.* 2003.

ISO 281. 2007. *Rolling bearings - Dynamic load ratings and rating life.* 2007.

ISO 286-1. 2010. *ISO code sytem for tolerances of linear sizes - Basis of tolerances, deviations and fits.* 2010.

ISO 3408-5. 2006. *Static and dynamic axial load ratings and operational life.* s.l. : ISO, 2006.

ISO 5753. 2009. *Rolling bearings - Internal clearance - Part 1: Radial internal clearance for radial bearings .* 2009.

ISO 76. 2006. *Rolling bearings - Static load ratings.* s.l. : ISO, 2006.

ISO/TR 1281-2. 2008. *Rolling bearings - Explanatory notes on ISO 281 - Part 2: Modified rating life calculation, based on a systems approach to fatigue stresses.* s.l. : ISO, 2008.

ISO/TS 16281. 2008. *Rolling Bearings - Methods for calculating the modified reference rating life for universally loaded bearings.* s.l. : ISO, 2008.

Nakajima, Katsuhiko. 1995. Thermal Contact Resistance Between Balls and Rings of a Bearing Under Axial, Radial, and Combined Loads. *Journal of Thermophysics and Hear Transfer.* 9, 1995, Bd. 1.

Niemann, G. und Winter, H. 2003. *Maschinenelemente Band 2: Getriebe allgemein, Zahnradgetriebe - Grundlagen, Stirnradgetriebe.* Heidelberg : Springer, 2003.

—. 2004. *Maschinenelemente Band 3: Schraubrad-, Kegelrad-, Schnecken-, Ketten-, Riemen-, Reibradgetriebe, Kupplungen, Bremsen, Freiläufe.* Heidelberg : Springer, 2004.

Niemann, G., Winter, H. und Höhn, B.-R. 2005. *Maschinenelemente Band 1: Konstruktion und Berechnung von Verbindungen, Lagern, Wellen.* Heidelberg : Springer, 2005.

Paland, E.-G. 1968. Die Lastverteilung in einem auf der Ebene abgestützten Zylinderrollenlager. *Maschinenbautechnik (Schmierungstechnik).* 17, 1968, Bd. 8.

Schlecht, Berthold. 2007. *Maschinenelemente 1.* München : Pearson Studium, 2007.

—. 2010. *Maschinenelemente 2.* München : Pearson Studium, 2010.

Teutsch, Roman and Sauer, Bernd. 2004. An Alternative Slicing Technique to Consider Pressure Concentrations in Non-Hertzian Line Contacts. *Journal of Tribology.* 2004, Vol. 126.

MESYS Hertz Calculation

5 MESYS Hertz Calculation

The MESYS Hertz Calculation (Version 06/2024) calculates the Hertzian stresses and subsurface stresses for point or line contact.

The following results can be found in the report:

- Major half axis of contact ellipsis a

- Minor half axis of contact ellipsis b

- Approach of both bodies

- Contact stiffness R

- Hertzian stress

- Maximal shear stress

- Maximal octahedral shear stress

- Depth for max. shear stress

- Depth for max. octahedral shear stress

- Equivalent stress (Tresca)

- Equivalent stress (Mises)

In addition to the report, the results are shown in several graphics. Report and user interface are both available in metric or US customary units. Supported languages for user interface and report are English, German, French, Spanish, Turkish, Japanese, Chinese and Korean.

5.1 Input Parameters

The input parameters are all shown at a unique page:
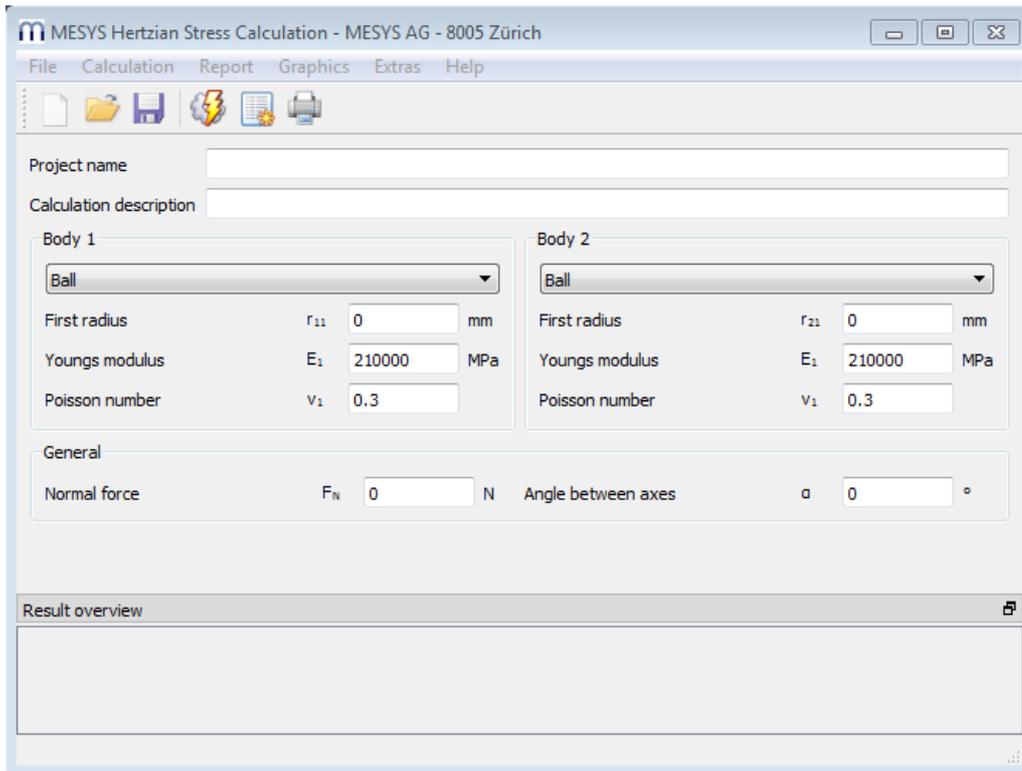
5.1.1 Project name and calculation description

The project name and the calculation description are just inputs which are shown in the report header. They can be used to enter information about the purpose of the calculation for documentation.

5.1.2 Bodies in contact

Body types such as ball, cylinder, ellipsoid and plane are available on the drop-down lists corresponding to the 'Body 1' and 'Body 2' sections. All combinations are possible, apart from the plane-plane contact for obvious reasons.
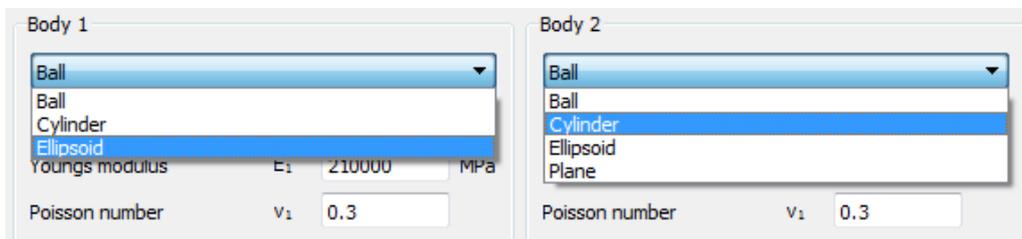
5.1.2.1 Radii of curvature
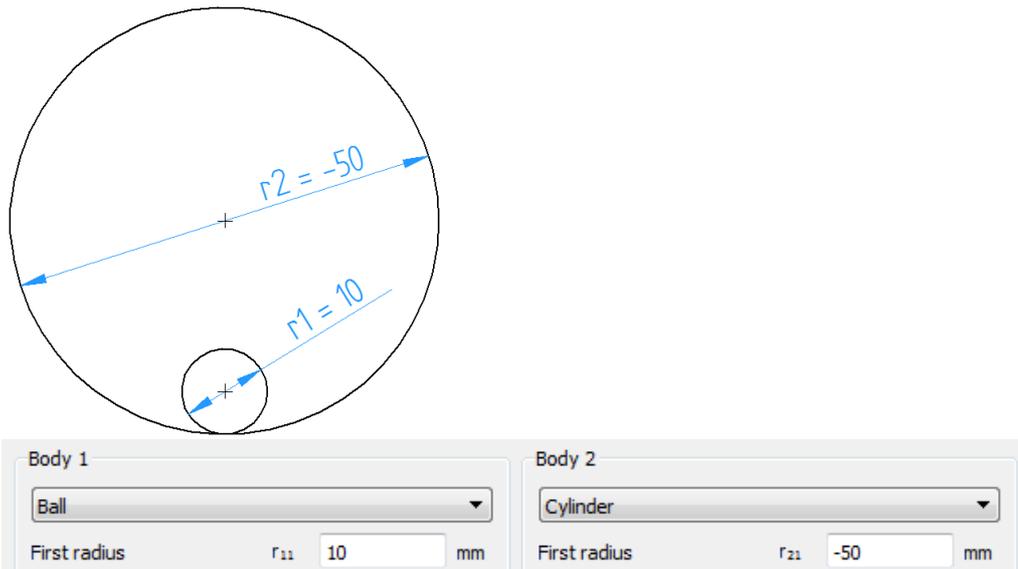
Only one radius is needed for balls and cylinders. Note that concave surfaces have to be defined by means of negative values.

It will be necessary to specify the two radii of an ellipsoid, so this input is activated when selected:
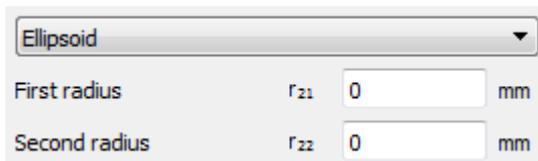
5.1.2.2   Material properties

Only young modulus and poison's ratio are needed for the bodies in contact. The stresses for both bodies are the same for equal materials, for different materials they can differ.

5.1.3   General

5.1.3.1   Normal force

Here the contact load is specified as normal force.

5.1.3.2   Angle between axes

The radii of curvature for two bodies can be given in two planes and an angle between these planes can be specified. The angle is defined between the lines

which are normal to the planes where the radii r11 and r21 lie, as shown in the picture:
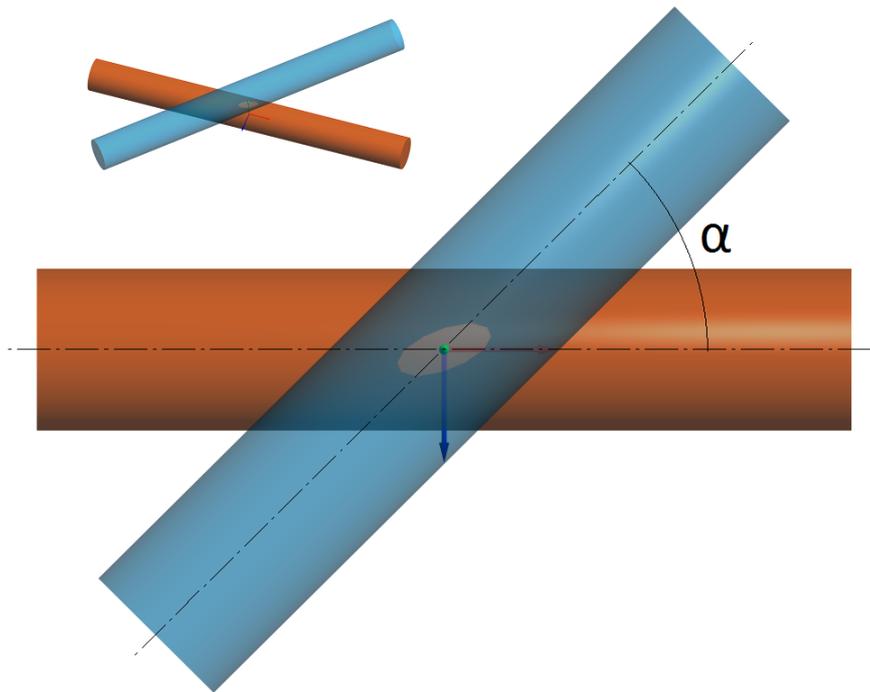
5.1.3.3 Length for line contact

For the case of two cylinders in contact, it is required to define the length for line contact, so the corresponding input field will be activated under 'General' as shown in the picture above.

5.2 Results

Results are provided as a small result overview directly in the software, a main PDF text report and separate graphic windows. Graphic windows can be docked onto the main program interface and are automatically updated on each calculation.

5.2.1   Result Overview

The results overview on the bottom of the window shows the most important results. The results overview can be configured under Extras->Results overview.

The list on the left shows all available items for the result overview. Using the arrows selected items can be moved to the list on the right which is used for the

result overview. The order of the selected results can be changed by the up/down arrows on the right. Note that only valid results values are actually shown in the result overview. So you will see fewer values than selected in the settings.

The settings for the result overview are saved in user settings not in the calculation file. So they will be the same for different calculations.

5.2.2 Main Result Report

The text report is provided in PDF format.

In the menu 'Report->Options' the contents of the report can be configured. The graphics to be included can be selected and some sections of the report could be discarded if not of interest. The legend for all the table parameters can be shown in the report.

5.2.2.1 Saving main report

The main report can be saved using the menu "Report->Save Report As" in different formats. It can be saved as PDF, DOC, DOCX or ODT (Open Office) format.

5.2.2.2 Customizing report logo

As default, a MESYS logo is shown on top of the report. This can be customized by defining a logo in the "mesys.ini" file. The logo should be created using a size, so that it fits into the header table. The DPI setting in the logo is considered, so that a high resolution image could also be used. Additionally, it also exists the possibility to create an own template of the supported file formats (DOC, DOCX and ODT) and generate the report from it, thus having the chance to customize the logo.

5.2.3 Result Graphics

All the available charts are under the menu 'Graphics'. For the current version, the stress curves for both bodies as well as displacement over load or stress over load can be analyzed.
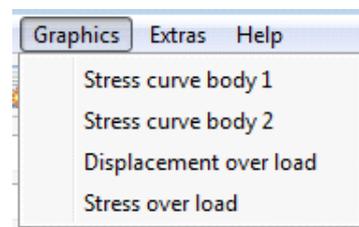
5.2.3.1 Stress curves for body 1 and body 2

Maximum principal stress (σ_{xx} σ_{yy} σ_{zz}), maximum shear stress (τ) and maximum octahedral shear stress (τ_{Oct}) are represented for at the line of contact under the surfaces.
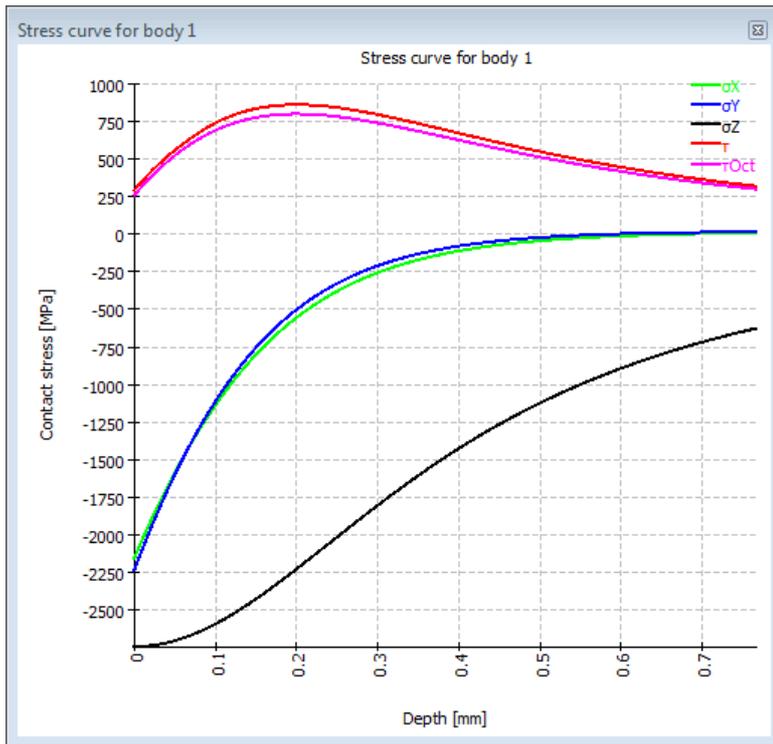
5.2.3.2   Displacement over load

This chart represents how the bodies (1 and 2) move towards each other by amounts of δ1 and δ2 as the load increases. Therefore, delta is the total decrease of distance by an amount δ=δ1+δ2 between points on the bodies not affected by the local deformation near 0.

Displacement over load
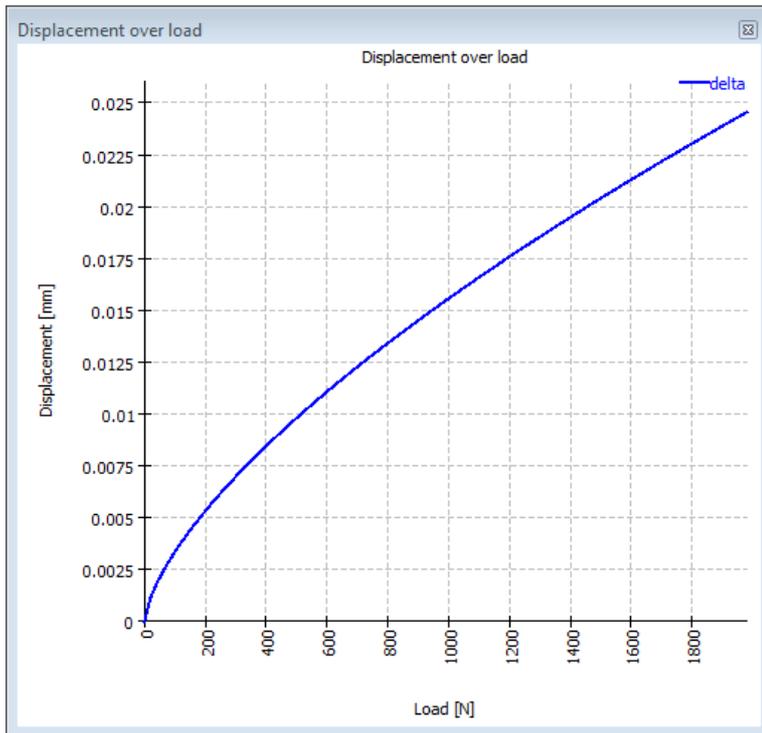
5.2.3.3 Stress over load

Here the Hertzian stress, maximum shear stress and maximum octahedral stress are represented against the load.

Stresses over load
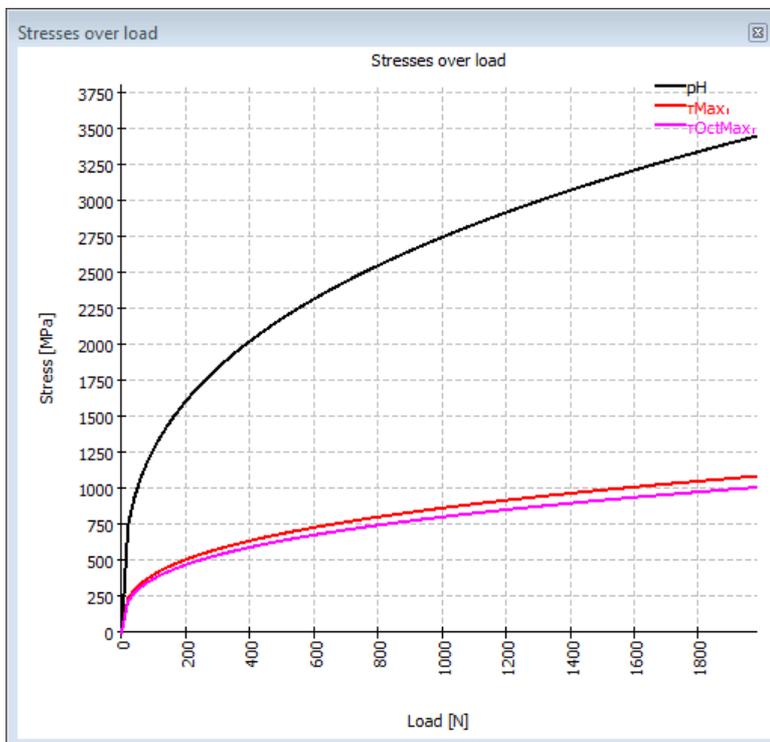
5.3   Bibliography

Boresi, Arthur P. and Schmidt, Richard J. 2005. *Advanced Mechanics of materials.* s.l. : John Wiley&Sons, 2005.

MESYS Axial-Radial-Rollerbearings

6 MESYS Axial-Radial-Rollerbearings

The calculation software for axial-/radial-roller-bearings or three row slewing rings (Version06/2024) is considering deformations of bearing rings using finite element analysis. The bearing geometry of axial symmetric rings can be defined by the user by a polygon in the cross section. An arbitrary number of axial or radial cylindrical roller bearings or cross roller bearings can be defined and the pretension of bolts is considered. Loads can be applied on a surface or on single points. For the definition of stiffness a calculation of a series with variation of load is possible.

As results the following data is available in several graphics and a PDF report:

- The pressure distribution on the rollers

- The bearing life for each row and the system according ISO/TS 16281

- The deformation and the stiffness defined by two points

- The maximal and minimal load on the bolts

6.1 General usage

The software can be called with a file name on the command line. The results can be saved using the parameter "-save". Example: „MesysAxRaRBC64.exe file1.xml -save"

For larger models, the calculation time can be up to an hour. Therefore the following approach is recommended: Choose 'Only preprocessing' and build the model interactively. Save the model and run the calculation using the command line. Afterwards, the results can be loaded into the program to evaluate the results.

The number of processor cores used in FEA can be changed by changing following entry in mesys.ini:

```
[axrarbc]
numberofthreads=2
```

Using more than four threads will lead only to small improvements.

6.2 Inputs

The input data is provided on several input pages which are described on the following pages. The input data is for the following geometry:

6.2.1 General

6.2.1.1 Number of parts

The bearing can be build using several parts. In usual cases two parts are used. There is no restriction on the number of parts.

6.2.1.2 Number of bearings

The number of radial, axial and cross roller bearings can be defined here. In usual cases one radial and two axial bearings are needed.

6.2.1.3   Maximum angle for mesh

For the meshing in circumferential direction, a maximum angle can be defined. Using a value of zero, the angle results from number of bolts and rollers. The angle can only be reduced but not be increased.

A large angle can lead to increased stiffness of the parts. To estimate the influence of the angle, some calculations for comparison can be made.

6.2.1.4   Number of bolted joints

One or more rings of bolts can optionally be considered.

6.2.1.5   Consider load spectrum

A load spectrum can be considered. If a calculation of series is active, only multiple speeds can be defined. Without calculation of series, also different loads can be defined per load spectrum element.

6.2.1.6   Only preprocessing

If this option is activated only the model is created but no FEA is performed on running the calculation.

6.2.1.7   Allow general geometry input

If this option is not set, the geometry is defined by horizontal and vertical lines only. Then it is sufficient to enter one coordinate only per geometry line, the second will be set automatically.

If the option is activated, all values for the geometry coordinates have to be defined, but a general polygon is permitted as geometry. A meshing algorithm is used then, that will lead to less elements than with horizontal and vertical lines only.

6.2.1.8   Number of surface loads

The number of loads acting on surfaces can be defined using this input value.

6.2.1.9   Number of contacts

A contact between parts or between a part and the rigid environment can be defined if 'Allow general geometry input' is activated. The number of contacts is the number of contact definitions. Each definition can be used multiple times.

The ➕-button allows to define the properties for each contact definition:

The first column defines if the contact is between a part and the rigid environment (the 'housing') or between two parts.
The second column allows to define the clearance in the contact.

All contacts lead to a normal force in case of contact, but there are different options for the tangential force, which can be defined in the third column:

- "Fixed on contact" will constrain the tangential movement in case of contact.

- "No constraint" will lead to no constraint in tangential direction.

- "Welded" will lead to a tangential constraint always. In addition, also the normal forces can act in both directions.

The stiffness input in the last column should be zero in most cases. In case a value larger zero is used, it will be used for the normal stiffness. This can be useful for fixing a part in space with low stiffness. As the stiffness is used per node, its effect depends on the mesh size.

6.2.1.10   Reliability of rolling bearings

The bearing reliability is 90% as default. The input is used for the calculation of life modification factor a1.

The reliability is set for all bearings to the same value.

6.2.1.11   Series calculation

A series of load cases can be calculated by multiplying the load with a factor.

The load factor for the first calculation is the 'start factor' for the last calculation the 'end factor' is used. A given number of steps is calculated in between. For the results, a selection is available for the load step shown in graphics.

6.2.1.12 Lubrication

For the calculation of modified life, the definition of the lubrication is needed. The lubricant viscosity is calculated for the lubricant temperature using the reference values at 40°C and 100°C.

The selection 'Consider lubrication' defines if the modified life Lnmrh and the data for lubrication is shown in the report. For a small rotation speed, there will be no effective lubrication and the a_{ISO} factor approaches 0.1.

The lubricant cleanliness is considered by the factor e_c. It can be selected if the lubricant contains effective EP additives which can have an influence on a_{ISO} factor for small speed.

The e_c Factor can either be defined by lubricant cleanliness or it can be input by the user. An overview of recommended e_c factors is shown in the following table:

Contamination level	e_C	
	$D_{pw} < 100mm$	$D_{pw} \geq 100mm$
Extreme cleanliness Particle size of order of lubricant film thickness, laboratory conditions	1	1
High cleanliness Oil filtered through extremely fine filter; conditions typical for bearings greased for life and sealed	0.8 to 0.6	0.9 to 0.8
Normal cleanliness Oil filtered through fine filter; conditions typical for bearings greased for life and shielded	0.6 to 0.5	0.8 to 0.6
Slight contamination	0.5 to 0.3	0.6 to 0.4
Typical contamination Conditions typical of bearings without seals; course filtering; wear particles from surroundings	0.3 to 0.1	0.4 to 0.2
Severe contamination Bearing environment heavily contaminated and bearing arrangement with inadequate sealing	0.1 to 0	0.1 to 0
Very severe contamination	0	0

6.2.2 Geometry of parts

The geometry of parts is defined in a table. The points (radius and y-coordinate) have to be entered clockwise.

If the general geometry input is not activated, the geometry can be defined using horizontal and vertical lines. A coordinate which is not changed to the previous point can be set to zero. So the number of input values can be reduced by factor two.

If the general geometry input is activated, the geometry can be defined by lines in arbitrary direction. All coordinate values have to be defined in this case.

The ⊞-button behind the name of the part allows to define the material properties for this part. The material data is only used for the FEA calculation of this part, not for the stiffness of the contact between roller and races.

Using the buttons on the lower right additional lines can be added or deleted. The ⊞-button copies the selected line or appends a new line if nothing is selected. The ⊟-button deletes the selected line and the ✖-button clears the whole table.
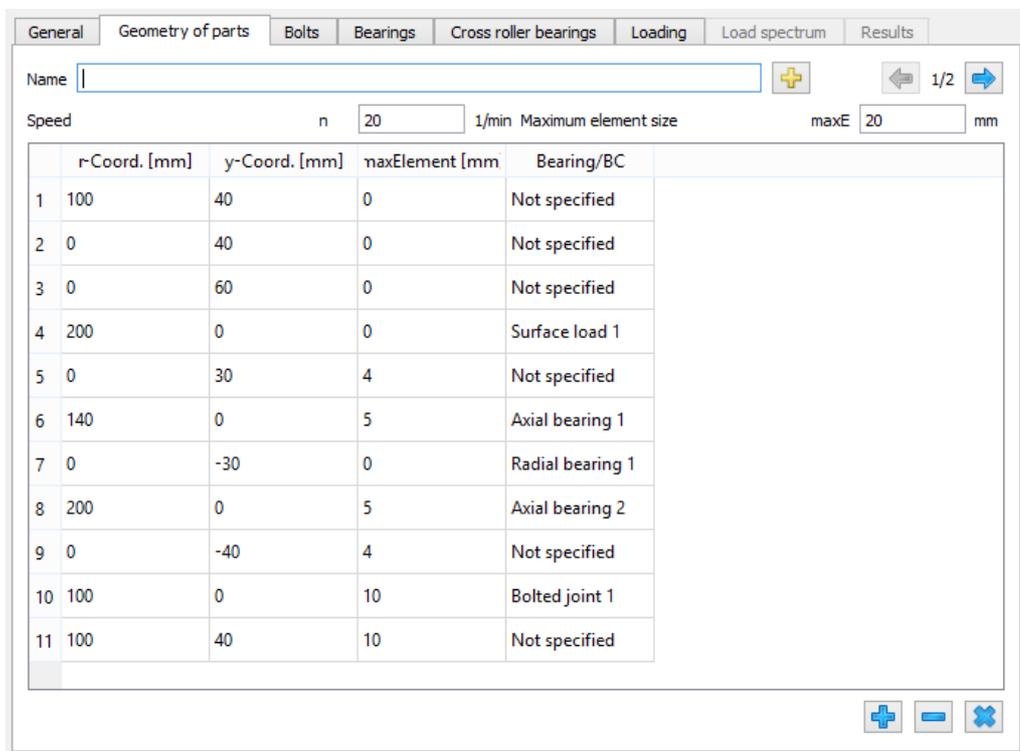
	r-Coord. [mm]	y-Coord. [mm]	maxElement [mm]	Bearing/BC
1	100	40	0	Not specified
2	0	40	0	Not specified
3	0	60	0	Not specified
4	200	0	0	Surface load 1
5	0	30	4	Not specified
6	140	0	5	Axial bearing 1
7	0	-30	0	Radial bearing 1
8	200	0	5	Axial bearing 2
9	0	-40	4	Not specified
10	100	0	10	Bolted joint 1
11	100	40	10	Not specified

6.2.2.1 Element size

The size of elements can be defined as a maximal size per part or in the third column of the table for a single line. An input of zero leads to the standard value.

If bending is expected in a part, more elements are required. One element can model a constant stress state correctly. For a linear stress distribution, several elements are necessary.

6.2.2.2 Speed

The rotation speed for the bearing life calculation can be defined for each part. The sign is considered, so one part can have a positive and one part can have a negative speed.

6.2.2.3 Boundary conditions

Boundary conditions can be defined line by line. The following selections are available:

- Radial fixed: All nodes for this surface are fixed radially.

- Axial fixed: All nodes for this surface are fixed axially.

- Radial and axial fixed: All nodes for this surface are fixed radially and axially.

- Bolted joint: The head of the bolts is on this plane.

- Radial bearing: A radial bearing is connected to this plane .

- Axial bearing: An axial bearing is connected to this plane. This input is used for axial positioning.

- Cross roller bearing: A cross roller bearing is connected to this surface.

- Surface load: A load is applied to this surface.

- Contact: A contact to a second part or to the rigid environment is considered.

6.2.3 Bolted joint

The data for the bolted joint can be defined on this page.

The bore for the bolts and the thread can be in the same part or in case of contact definitions in different parts. The depth of the bore is measured from the surface; the thread in the part is starting below.

If no counterbore exists, the diameter and the depth of the counterbore can be set to zero. The cylinder with the diameter of the counterbore may not intersect and edges of the part. If necessary, the diameter has to be reduced for the calculation.

The pretension is defined per bolt. All bolts have the same pretension. Either the effective pretension can be defined or a mounting torque and a subsidence value. According to VDI 2230, the subsidence value for surface roughness between $10 < Rz < 40$ is $3\mu m$ for the head and the thread and $2\mu m$ for each layer. For most cases, it will be $fz = 3+3+2 = 8\mu m$.

The pitch of the thread is needed for the calculation of the core diameter of the bolt needed for its stiffness. Also, it is needed for the calculation of pretension force from mounting torque.

If the mounting torque is entered, the mean diameter of head support and the friction values for thread and head support are required.

6.2.4 Bearings

The geometry of the bearings can be defined on this input page.

6.2.4.1   Number of rolling elements

The number of rollers has to be defined here.

6.2.4.2   Diameter of rollers

The diameter of the rollers is defined here.

6.2.4.3   Pitch diameter

The pitch diameter through the centers of the rollers has to be defined for both bearing types. For radial bearings a warning is shown if the pitch diameter is not between the selected planes.

6.2.4.4   Effective length of rollers

The effective length of rollers Lwe is the length which is the basis of the calculation of the load capacity. It is the total length of the roller reduced by its radii.

6.2.4.5   Radial-/axial clearance

The clearance is not defined by the dimensions of the parts, so that there is one input for clearance and it is not needed to define the geometry with many digits. The clearance is the nominal clearance before tightening the bolts.

A pretension is defined as negative clearance.

6.2.4.6   Axial position

The axial position of the center of the roller is entered here. For axial bearings the axial position is determined by the position of the selected planes of the parts.

6.2.4.7   Profile

Profiles can be defined for inner race, outer race and roller.

Define roller profiles

Difference between effective and total roller length	ΔLw	2 mm
Number of roller sections used	nSec	41

Roller

Type of profile: Standard profile according ISO/TS 16281

Profile definition over effective roller length

Inner race

Type of profile: Tangential crowning

Profile definition over total roller length

Profile	pr	5 µm
Unmodified cylindrical part	Lwu	80 %

Outer race

Type of profile: User defined

Profile definition over total roller length

Profile	pr	5 µm

OK   Cancel

As general inputs, the difference between effective and total roller length can be defined. As it increases the stiffness of the roller, it has an influence on the calculation.

The number of sections for the calculation has to be larger 30 and has to be an odd number. The calculation accuracy and the calculation time increase with a larger number of sections.

Several options are available for the definition of profiles:

1. Standard profile according ISO/TS 16281:
   This is using the defaults of the standard, which is logarithmic profile of the roller and no profile on the races.

2. Logarithmic profile:
   The amount of relief can be defined and a cylindrical part that should not be modified. The following formula is used:

$$P(x) = pr \cdot \ln\left(\frac{1}{1 - \left\{\frac{2|x|-L_{wu}}{L_{we}-L_{wu}}\right\}^2}\right)$$

For the roller, a sizing button provides the default relief according ISO/TS 16281. The default relief of 0.00035*Dw for short cylindrical roller bearings corresponds to a contact stress of 3000MPa.

3. Modified logarithmic profile:
   The amount of relief is calculated according following formula derived from (Fujiwara, et al., 2010):

$$P(x) = A \cdot \ln\left(\frac{1}{1 - \left\{1 - e^{-\frac{pr}{A}}\right\}\left(\frac{2|x|-L_{wu}}{L_{we}-L_{wu}}\right)^2}\right)$$

   using

$$A = 2D_w \cdot \left(\frac{p \cdot (1-v^2)}{E}\right)^2$$

   The maximum relief is not infinity at the edge of the roller like in the logarithmic profile according ISO/TS 16281 but limited to the maximum value of pr.
   Using p = 3000MPa and pr=Infinity, the results are the same as for the logarithmic profile according ISO/TS 16281 above.
   The material data for this profile is always taken from the contact roller/inner race.

4. Tangential crowning:
   A circular arc tangential to the cylinder is used to generate the profile. The amount of relief and a percentage of unmodified cylinder length can be defined.

5. Crowning:
   A circular arc with center positioned axially on the center of the roller is used to generate the profile. The amount of relief and a percentage of unmodified cylinder length and a transition radius can be defined.

6. Two tangential radii:
   A first cylindrical part with width "Lwu" is extended by two tangential radii. The width of the first radius is defined as percentage "Lwr1". A requirement is "Lwr1" > "Lwu". "Lwu" can be set to zero to have two radii only.

7. Exponent
   An exponent "e" and an unmodified length "Lwu" for the profile can be specified. The exponent has to be $e \geq 2$. The following formula is used

for the profile:

$$P(x) = pr \cdot \left(\frac{2|x| - L_{wu}}{L_{we} - L_{wu}} \right)^{e}$$

8. Read file:
   The profile can be read from a file. The data has to be specified with two values on each line, an axial position and the profile relief. The axial position is scaled with the roller length and the relief is scaled with the input for the profile relief. So a range from -1 to +1 can be used for the axial position and 0 to 1 for the relief.
   A very simple file is shown here. It was selected for the inner race profile on the right:

   The unsymmetrical profile is used from left to right for most bearings. For taper roller bearings and axial spherical roller bearings, the profiles is used from left to right if direction of contact angle is left. For contact angle to the right, the profile is mirrored. So mounting conditions do not influence the profile.

9. Read file without scaling:
   Like option "Read file" the data is read from a file. Using this option no

scaling will be done. The axial extension has to be defined from -Lwe/2 to +Lwe/2 or from 0 to Lwe. The profile is defined in "mm" and won't be scaled too.

10. User Input:

Instead of defining a file name, the points for the profile can be defined in a table. Again as above, use a range of -1 to +1 for the axial position and a value between 0 and 1 for the profile relief.
The table in the diagram on the right shows the same data as the file above.
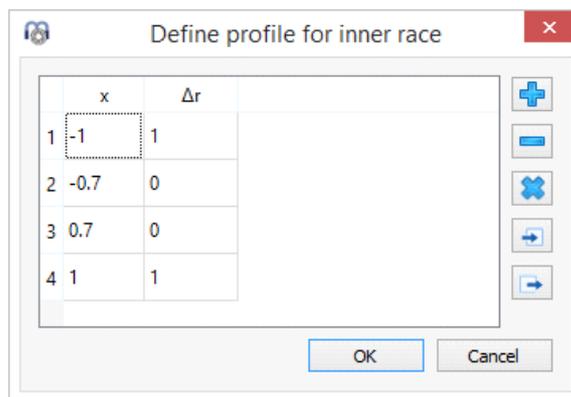
11. User defined equation:

A user defined equation can be defined in ECMAScript language. The variable 'x' has the range $-1 < x < +1$ and variables 'Dw', 'Lwe', 'Lw' are defined and in 'mm'. The result of the equation is considered in 'mm' and a positive value increases the gap.
Examples are "return 0.01*x**4;" or "return -0.00035*Dw*Math.log(1-x*x);" or using conditions like

```
if(x < -0.5) {
        return 0.01*(-0.5-x);
}
if(x > 0.5) {
        return 0.01*(x-0.5);
}
return 0;
```

6.2.4.8   Surface hardness

The surface hardness is used to reduce the load capacity of the bearings according to NREL guideline DG03. Only a hardness less than 58 HRC leads to a change of load capacity. The hardness is only considered if the load capacity is calculated by the software.

6.2.4.9   Dynamic load capacity

The dynamic load capacity is calculated according ISO 281. Its value can be overwritten by the user. The hardness is considered as follows for values less than 58HRC:

$$C' = C\left(\frac{HRC}{58}\right)^{3.6}$$

If the load capacity is entered by the user, no further reduction by hardness is considered.

6.2.4.10 Static load capacity

The static load capacity is calculated according ISO 76. Its value can be overwritten by the user. The load capacity is multiplied by the following value f_s for reduced hardness:

$$f_s = 2\left(\frac{HV}{800}\right)^2 \text{ with } f_s \leq 1$$

If the load capacity is entered by the user, no further reduction by hardness is considered.

6.2.4.11 Fatigue load

The fatigue load is calculated according ISO 281 appendix B.3.2.1.3. The value can be overwritten by the user.

6.2.5 Cross roller bearings

For cross roller bearings, three additional inputs compared to the other bearing types are necessary:

Configuration

> In addition to cross roller bearings also a calculation using angular roller bearings is available. All rollers have the same orientation in this case.

The number of rollers is the total number in each case. For cross roller bearings, it has to be an even number.

Calculate as

The bearing can be calculated as axial or radial bearing. Dependent on the selection, either Ca, C0a, Pa or Cr, C0r, Pd are shown as inputs.

Contact angle

The contact angle can be chosen between 40°-50°. Standard is 45°. For angular roller bearings, the permissible range is 20°-70°.

6.2.6 Loading

The load can be defined as point or surface load. Surface loads should be preferred because of smoother load introduction. Point loads can only be defined inside of a part.
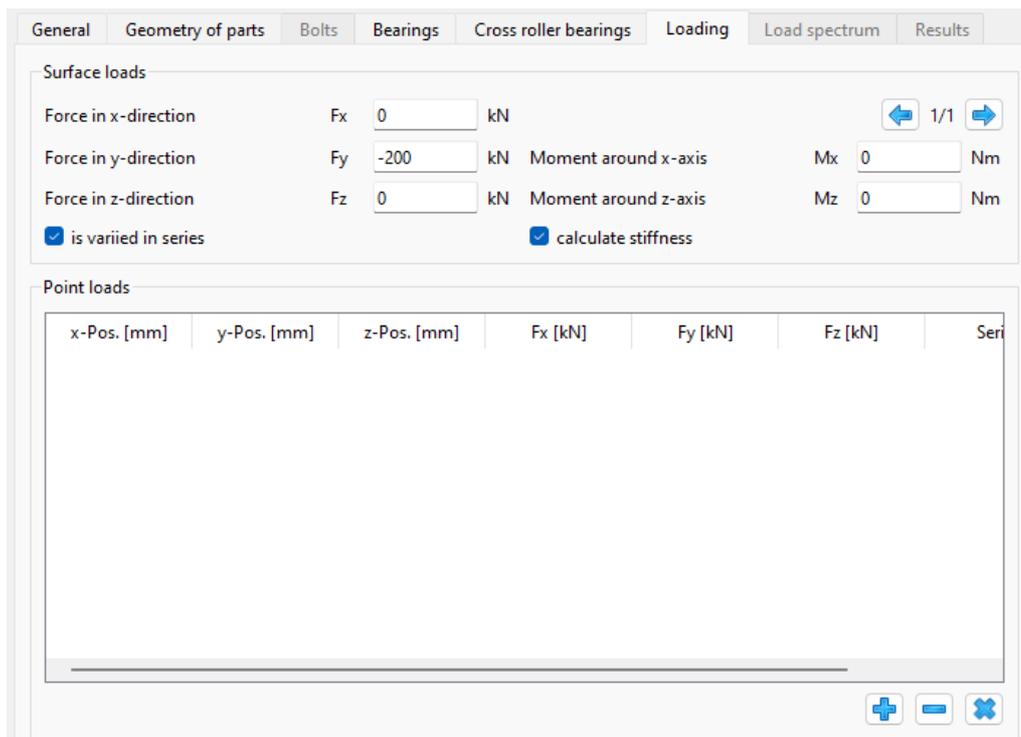
If "is variied in series" is selected, the force is multiplied with the load factor on active calculation of series. The option is also used to add the force into the load spectrum, if this is active.

If the option "calculate stiffness" is active the stiffness for the selected face is calculated and added to the report. A central node is connected to the surface by averaging and then the stiffness for this node is calculated. The values in the report are the reciprocal values of the main diagonal of the compliance matrix.

6.2.6.1 Surface loads

Surface loads can be defined by an input of three force components and two moment components. A torque around the y-axis cannot be taken by the bearing, therefore it is not available as input.

For each load, a selection can be made if the load should be multiplied by a load factor in a calculation for series. If not selected, the load is considered constant for all load steps.

6.2.6.2 Point loads

Point loads are defined by a force vector with three components and a position. The position has to be inside of a part.

For point loads it can be defined, too if the load should be multiplied with a load factor in calculation for series or taken as constant.

The point load is introduced using several nodes in the neighborhood of the given position, so the load center is on the selected position.

6.2.7 Load spectrum

If the load spectrum calculation was activated, it can be defined on page "Load spectrum". Only loads are shown where the flag "is varied in series" is set.

	Frequency	P1-n [rpm]	P2-n [rpm]	SL1-Fx [kN]	SL1-Fy [kN]	SL1-Fz [kN]	SL1-Mx [Nm]	SL1-Mz [Nm]
1	0.5	20	0	0	-200	0	0	0
2	0.5	30	0	100	-300	0	0	0

6.3   Results

If the results page is active, evaluation of results is possible without a new FEA calculation. Pressing the calculate button just evaluates the results.

6.3.1   Factor for displacements

The factor for displacements defines the exaggeration of deformations used in graphics. It is used in 3D view and 2D view for deformation of a section.

6.3.2   Angle for deformation graphics

The angle for the section used in 2d view for deformations can be selected here. The same angle is used in the diagrams for stresses within the parts.

6.3.3   Minimal and maximal stress for graphics

For the stress diagrams, the range for the color scale can be set here.

6.3.4   Calculation of stiffness

For the calculation of stiffness and the diagrams for deflection and tilting angle, two points have to be defined. The software averages the deflections around the circumference for these points and the result is used to calculate a stiffness value.

6.3.5   Graphics

6.3.5.1   3D-View

The 3D-view can be rotated using the left mouse button. Using the Ctrl-button, the size can be changed.
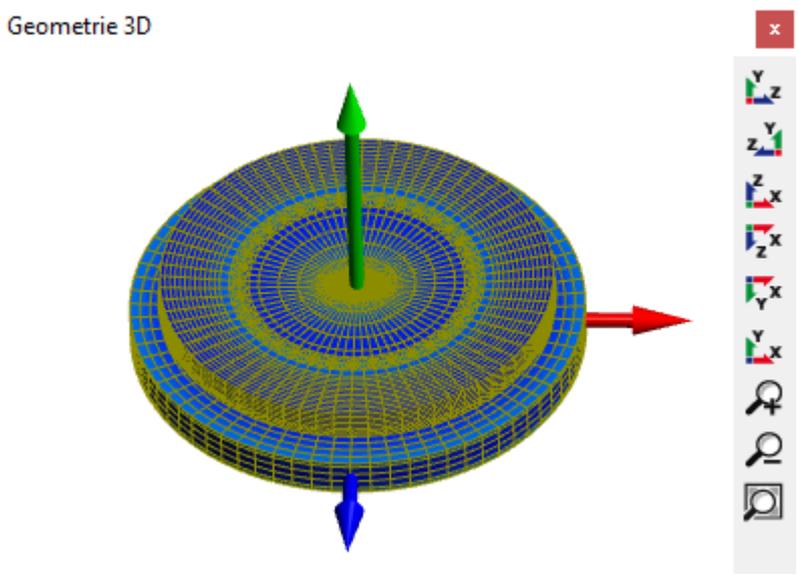
6.3.5.2   Geometry 3D Cut view

The cut view is a cut of the 3D geometry using the angle defined on the result page.
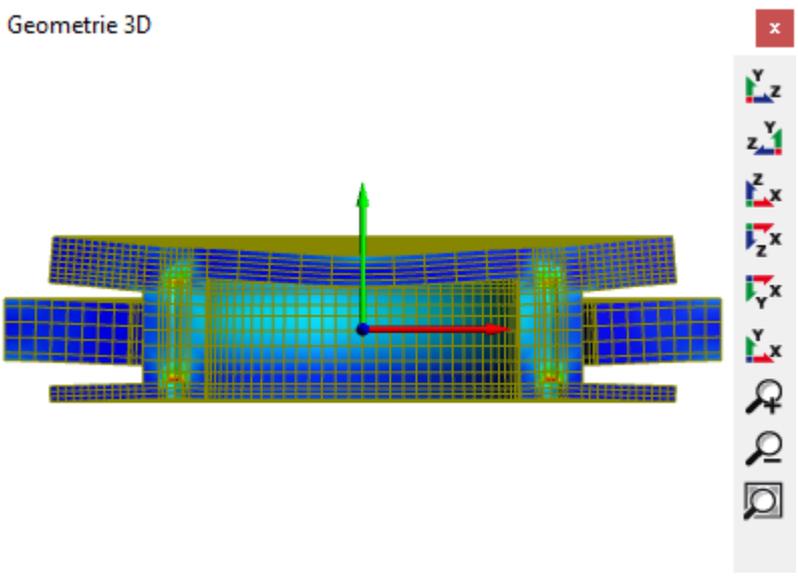
6.3.5.3   Bearings 3D

The load distribution can be shown on the 3D bearing set.
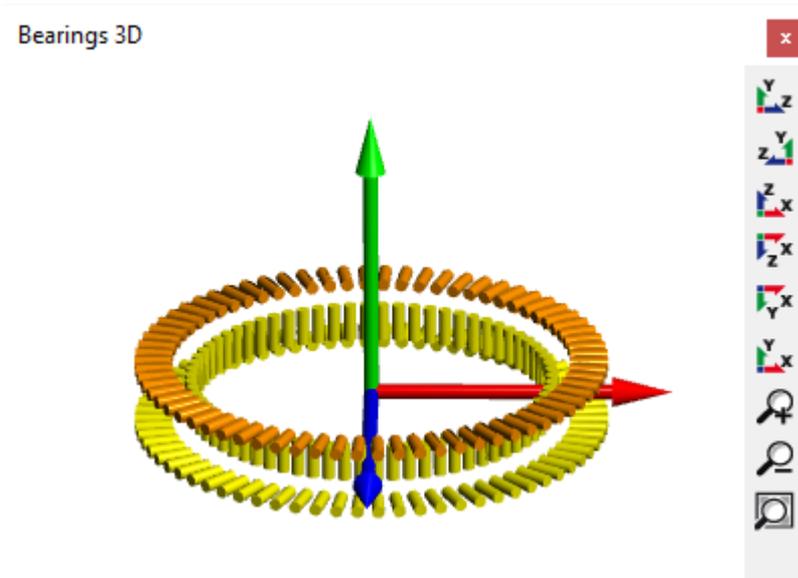
6.3.5.4 Geometry

The graphics 'Geometry' is available after preprocessing. It can be used to check meshing and the position of bolts and bearing rows.
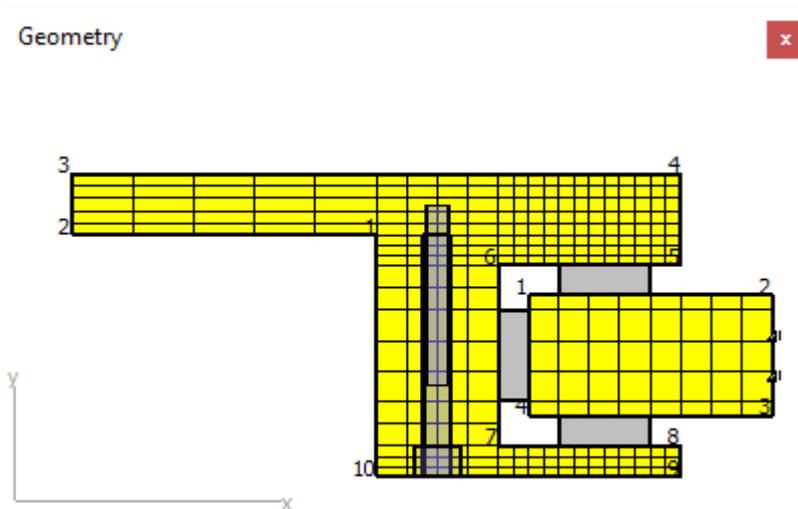
6.3.5.5 Deformation

The graphics 'Deformation' shows the deformed mesh for a given angle. The two points for evaluating the displacements are marked with color.

The first point is shown in red, the second in blue.

Deformation
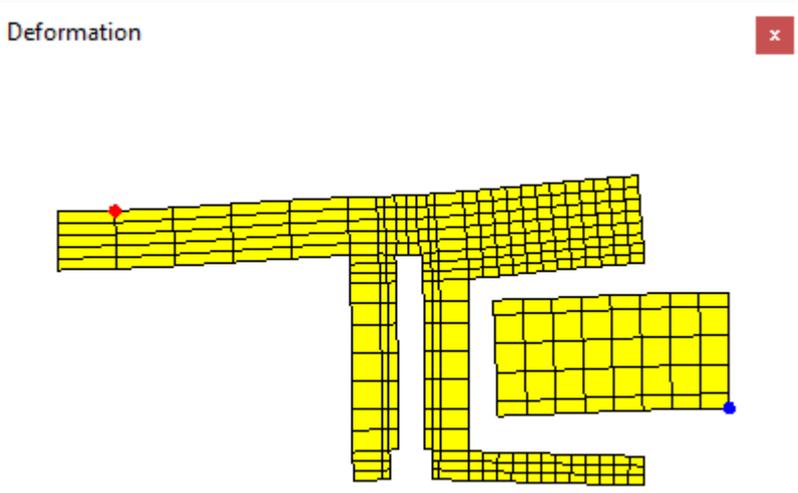
6.3.5.6 Displacements

The displacements for the selected points are shown over the circumference.
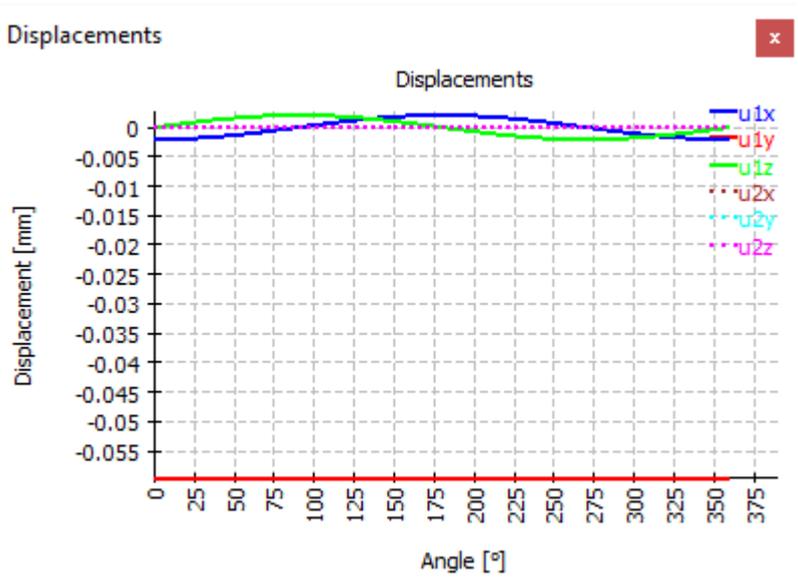
6.3.5.7 Displacements over load factor

In case of a series calculation, the mean displacements are shown over load factor.

6.3.5.8   Tilting angle over load factor

Like the mean displacement, also the mean tilting angle can be shown dependent on the load factor in a diagram.
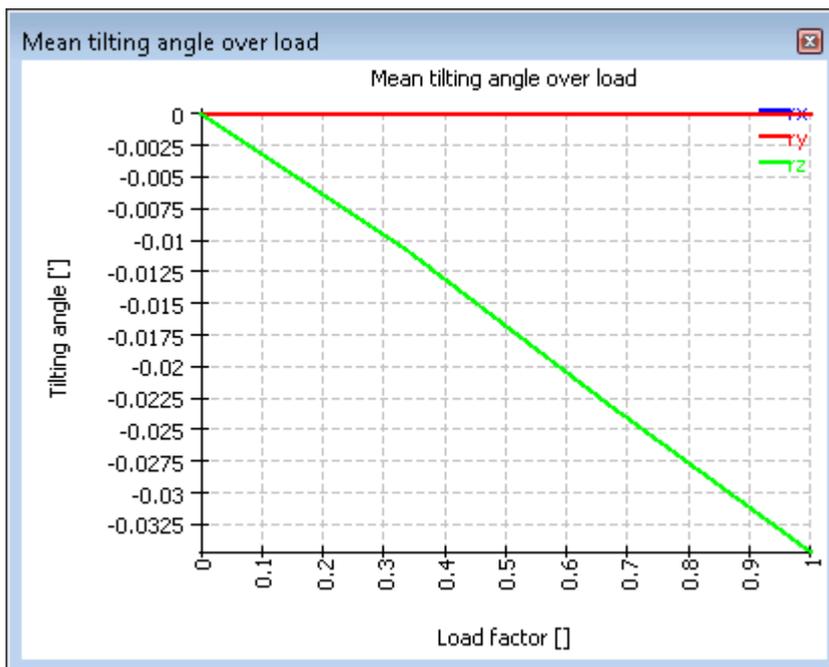
6.3.5.9   Equivalent stress in parts

The equivalent stress in part is shown in this diagram.

Equivalent stress
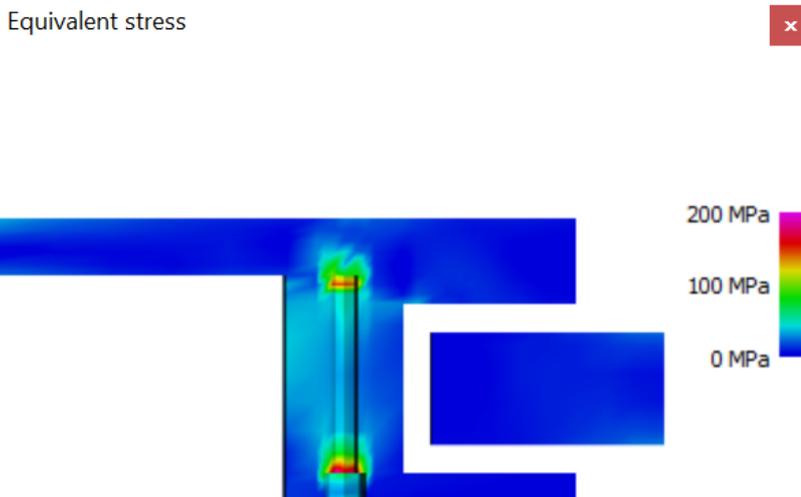
6.3.5.10 Axial, radial and tangential stress in parts

Further diagrams show the axial, radial and tangential stress in the parts. The range for the color scale can be defined on the results page.

Axial stress
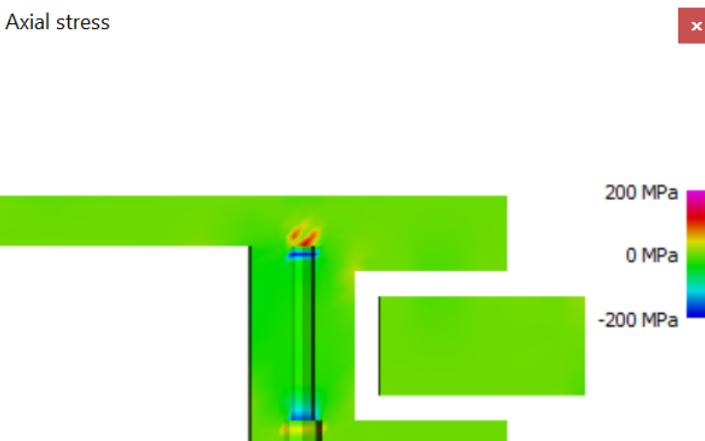
6.3.5.11 Contact stress for bearing

The contact stress is shown over the length of the rollers. In the user interface, there is one graphic for all bearings of one type, in the report, one diagram is shown for each bearing.

The position in the diagram is either the axial or radial coordinate with the origin at the center of the roller. For radial bearings, the orientation is like the

orientation of the y-axis. The position is running from bottom to the top of the bearing. For axial bearings, the orientation is the radius; the position is running from inside to outside.
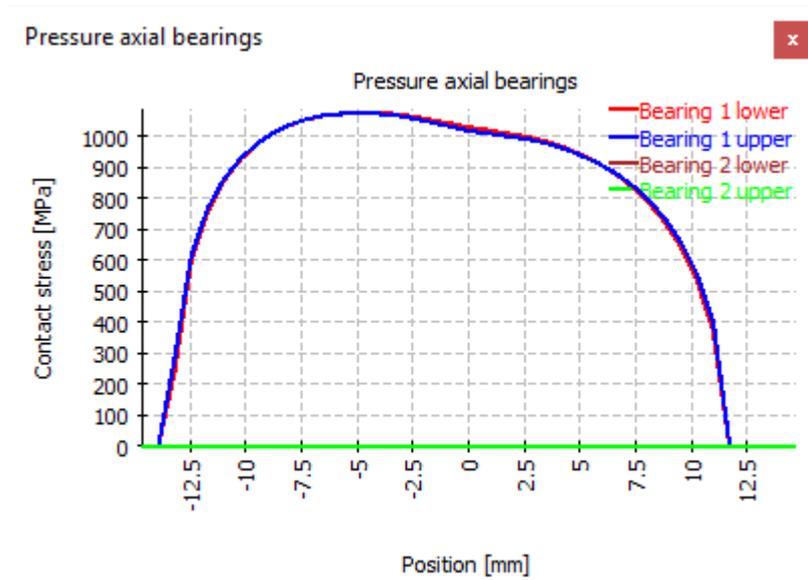
6.3.5.12 Contact stress over angle

The contact stress can also be shown over the angular position. The maximal contact stress per roller is shown. Angle zero is in the x-axis, 270° on the z-axis.
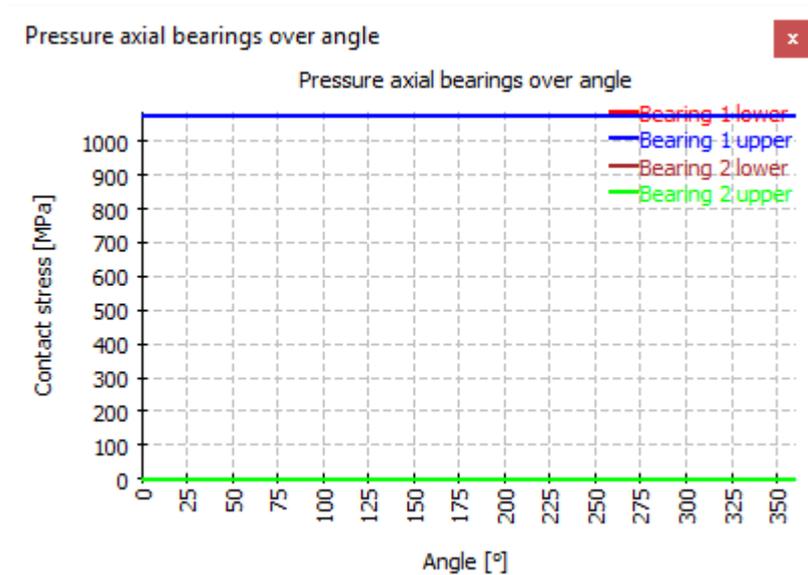
6.3.5.13 Forces over angle

The distribution of roller forces over angle is shown in this diagram. For force is the total force between roller and race.
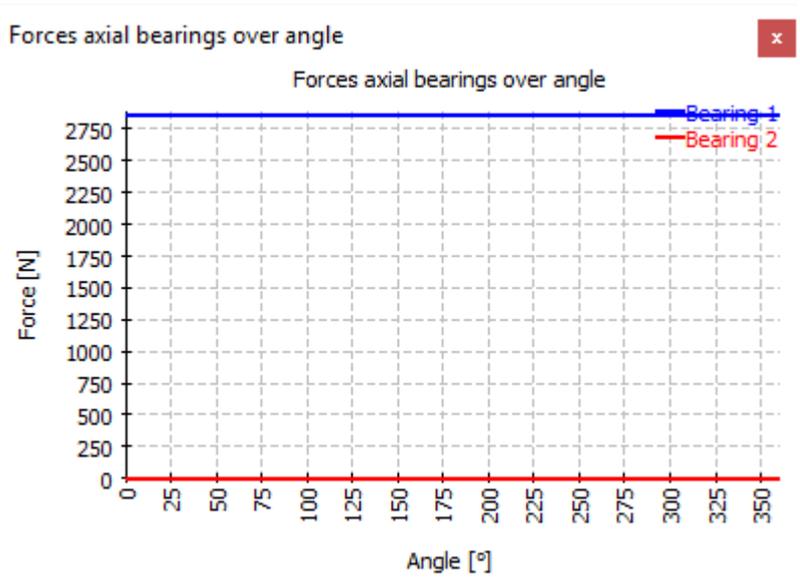
6.3.5.14 Roller profile

The profiling of the rollers is shown in this diagram. It is mainly used for check of inputs.
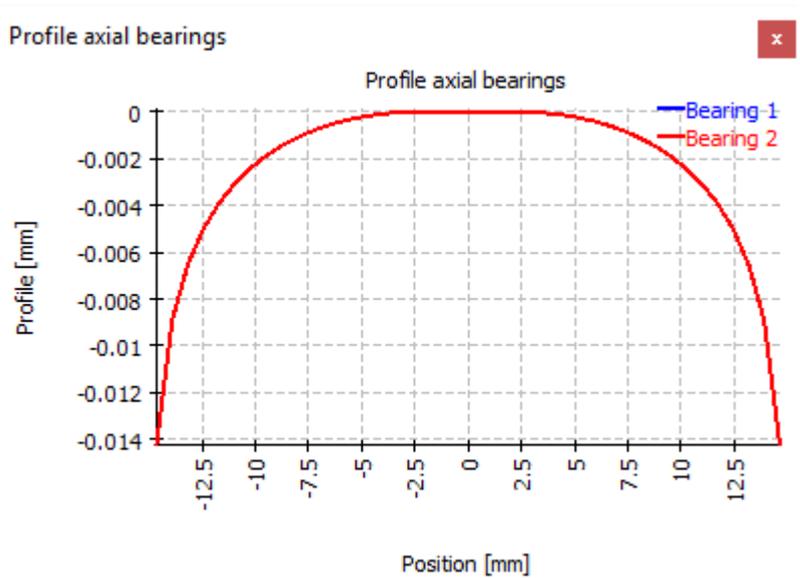
6.3.5.15 Bolt forces over angle

This diagram shows the bolt forces over the angle.

Bolt forces over angle

Bolt forces over angle
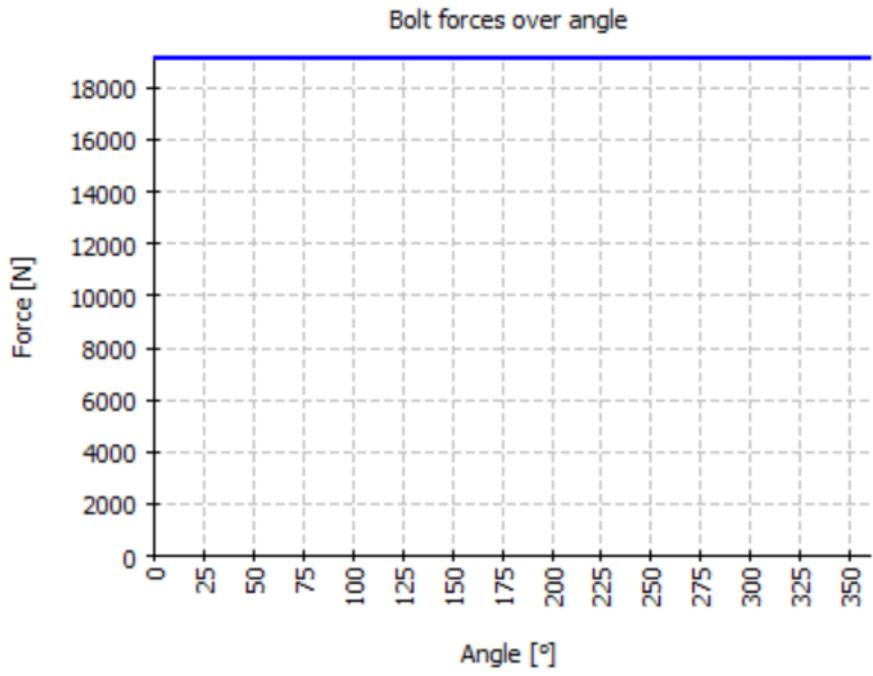
MESYS Cylindrical Gear Pair

7 MESYS Cylindrical Gear Pair

The cylindrical gear pair calculation (Version 06/2024) calculates the gear geometry according to ISO 21771-1 and strength according to ISO 6336. The following features are supported:

- Gear geometry calculation according to ISO 21771-1. For internal gears all input and output diameters are used as positive values in the software as they are used for manufacturing and documentation. Only the number of teeth for internal gears is set to a negative value.

- Gear strength calculation according to ISO 6336 parts 1, 2, 3, 5, 6. Different options are available for the tooth thickness and contact ratio to be used.

- Support for load spectra. In addition to speed and torque also mesh deviations, face load coefficients or mean stress influence factors can be defined in the load spectrum.

- Tooth form graphics for single teeth, full gear, manufacturing or engagement

- Graphics for specific sliding, safeties over life, permissible torque over life

- FEA based tooth form stress using 2D-FEA. For helical gears either the virtual spur gear can be used or the normal section.

- 3D-Gear geometry available as STEP export. The transverse sections is generated with a accuracy better than $10^{-4} m_n \cos(\beta)$ and then extruded along a helix. Accuracy along the helix depends on the CAD system. Currently no profile or flank modifications are included.

7.1 Input parameters

The input parameters are shown on four or five tab pages.

7.1.1 General

On the first input page in addition to a project name and description, several settings can be done.
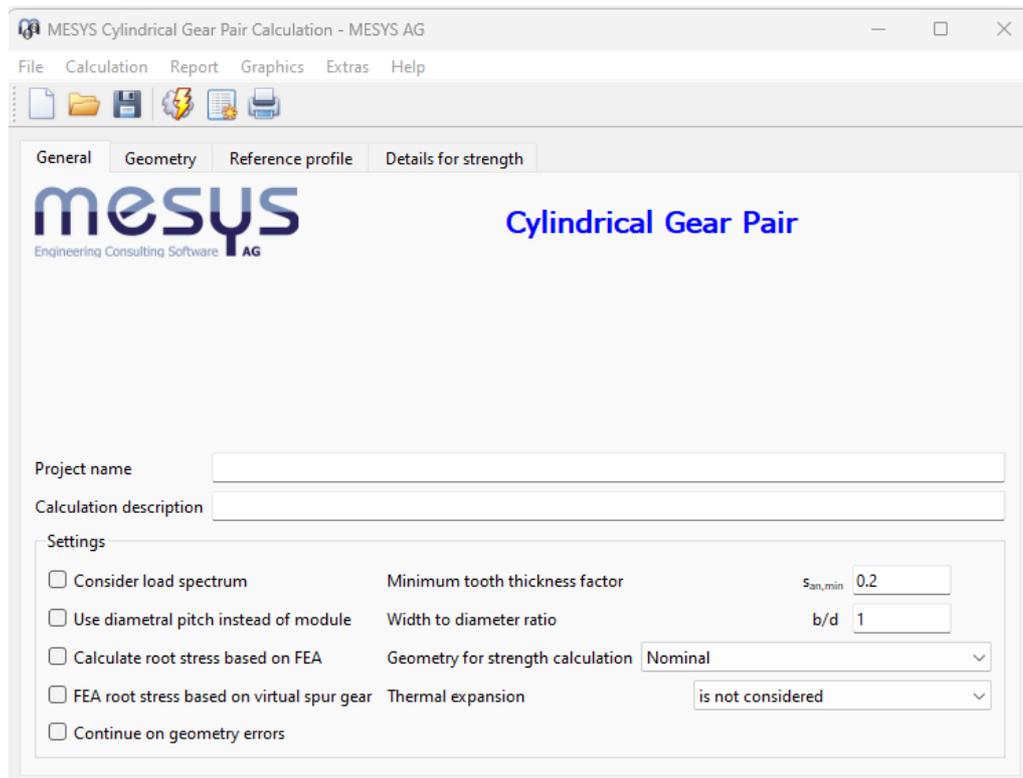
- Select "Consider load spectrum" if a calculation with load spectrum should be done.

- Select "Use diametral pitch instead of module" if the diametral pitch should be an input.

- The calculation of FEA based tooth root stresses can be activated. For helical gears it can be based on the normal section or on the virtual spur gear.

- If "Continue on geometry errors" is set, the software will not stop on geometry errors. Some results might be invalid in this case.

- The minimum tooth thickness factor is used for warnings in case of small top land.

- The width to diameter ratio is used for sizings of the geometry.

- For the strength calculation several options are available for the geometry to be considered within the tolerance field. ISO/TR 6336-30 tells it would use nominal dimension with minimal tooth thickness for factors YS/YF, but this does not fit to all examples in the report. In case of large tolerance fields either the average position in the tolerance field or the minimum tooth thickness should be used. The minimal tooth thickness will be the critical one for bending strength. For the contact ratio either the maximum or the minimum could be the critical one, dependent on the actual application.

- The thermal expansion of the gears and the housing could be considered on the contact ratio.

7.1.2 Geometry

The geometry of the gears is defined on this page.
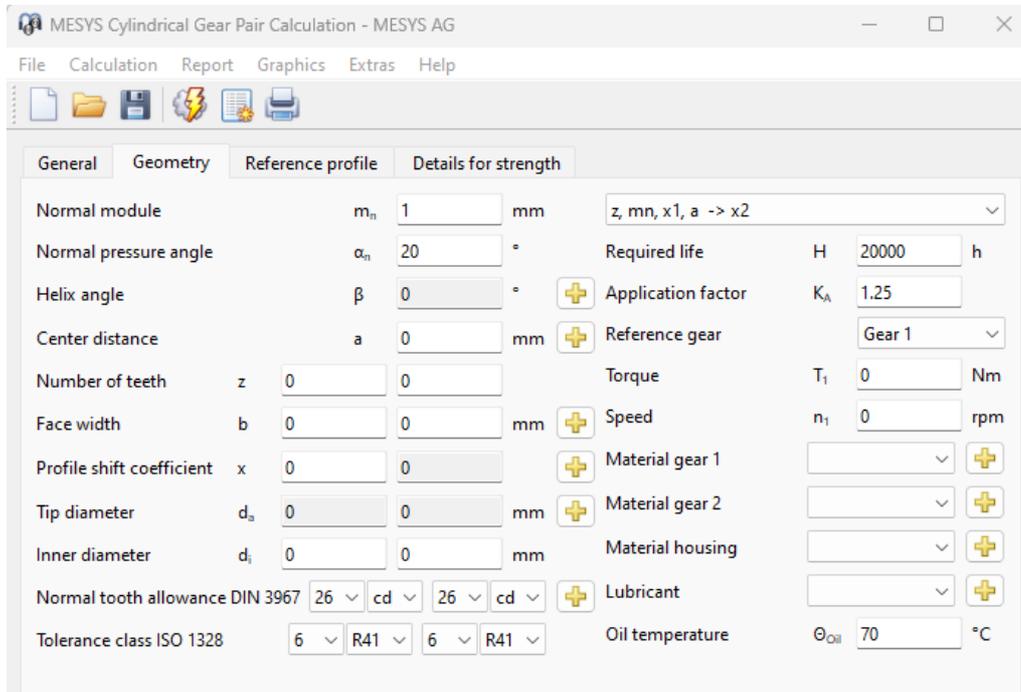
For the selection of active input values, the list on the upper right can be selected. It defines, which values are inputs and which are outputs. The last two options also include a strength calculation using the torque as an input for geometry.

7.1.3 Reference profile

The reference profile of the gear can be defined by three options. Either using a reference profile a hob or a shaper cutter. The hob is not supported for internal gears.
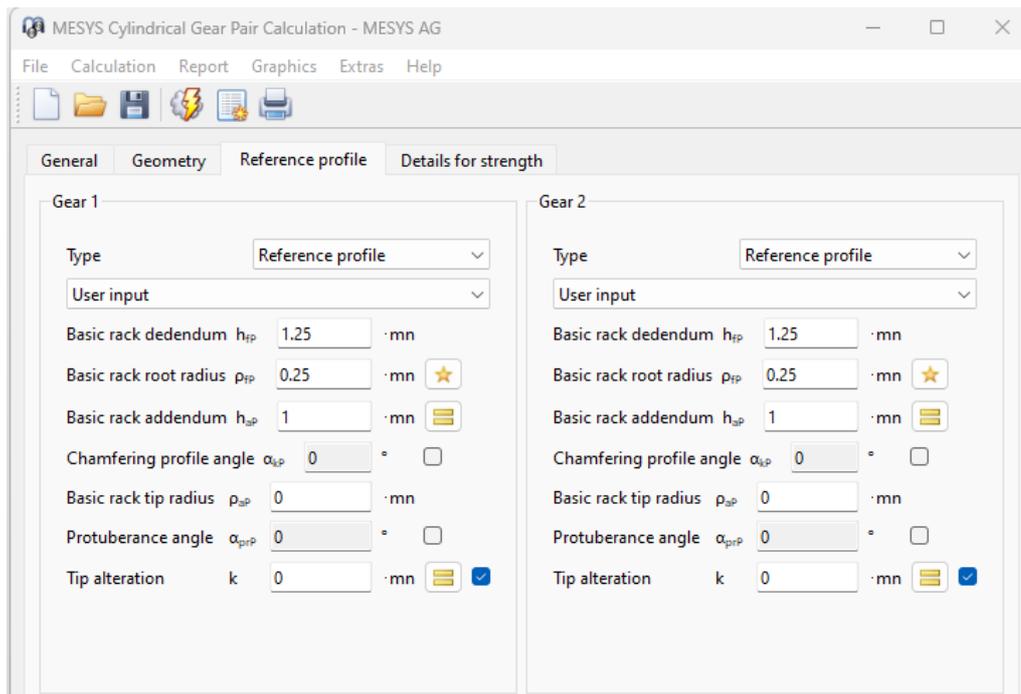
If a hob or a shaper cutter is selected, both the dedendum of the tool hfP0 and the addendum of the reference profile can be defined, if its not a topping tool. The dedendum of the tool needs to be larger than the addendum of the basic rack, else the tool would cut the tip diameter. A warning is shown if this is not the case, but the tip diameter is still calculated by the basic rack addendum. Please select a topping tool in this case.

7.1.4 Details for strength

Details for the strength calculation according to ISO 6336 can be defined on this page.
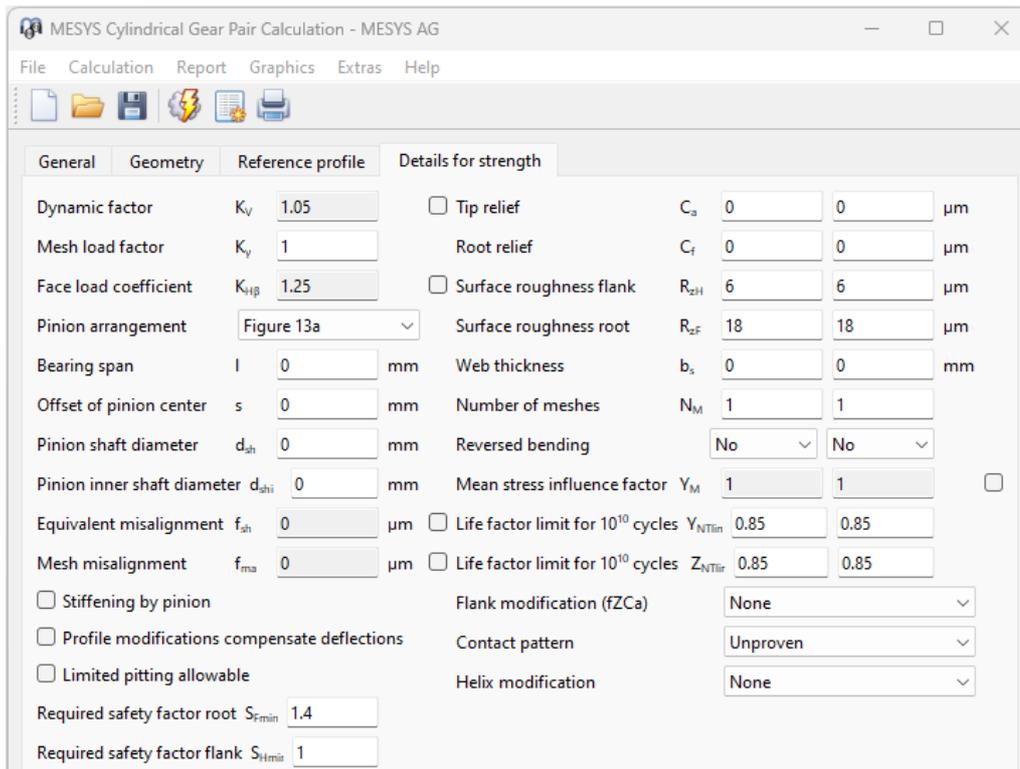
7.1.5   Load spectrum

If activated under General an additional input page for the load spectrum is available.
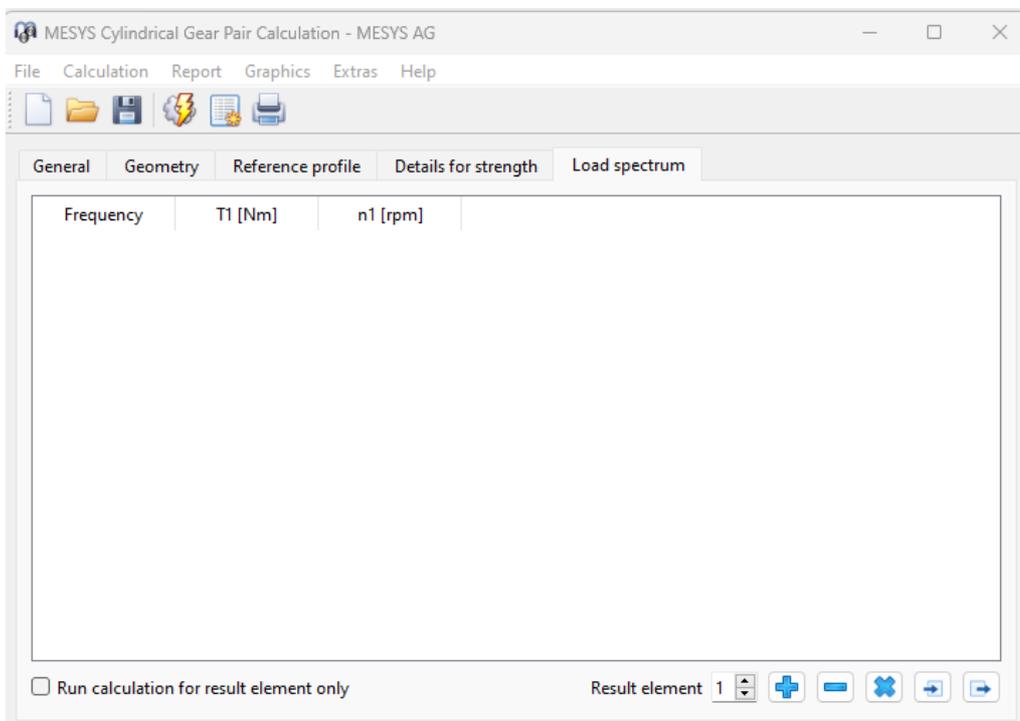
The first column 'Frequency' needs to sum up to one (or less) for all load spectrum elements. Additional columns can be added using the context menu with the right mouse button.

Rows can be added using the -button or a selected row can be deleted using the -button. The -button clears all the inputs.

Using the -button, the load spectrum is read from a file. The columns that are currently shown with its units have to be used for the input file. As the data from the file is added to the inputs, you must clear the inputs first to define a full load spectrum. Likewise, a created load spectrum table can be exported with the corresponding -button.

7.1.5.1   Calculation for a single load spectrum element only

It is possible to run the calculation for one element (load case) only if the user sets the flag 'Run calculation for result element only'. The desired load case is chosen by either using the arrows of the small box next to the -button or typing the element number into it directly. The graphics will be only shown for the selected element.

This allows using the load spectrum calculation as input interface for independent load cases, which are calculated independently.

7.2   Results

Results are provided as a small result overview directly in the software, a main PDF text report and separate graphic windows. Graphic windows can be docked onto the main program interface and are automatically updated on each calculation.

7.2.1   CAD export

Several graphics can be selected in menu Graphics->CAD. The tooth form in the transverse section is exported using Bezier curves of accuracy better than $10^{-4} m_n \cdot \cos(\beta)$. This transverse section is then extruded along the helix for a helical gear. The accuracy of the extrusion depends on the CAD system.

Currently no profile corrections and flank line corrections are considered.

MESYS Gear Positions

8 MESYS Gear Positions

MESYS Gear Positions (Version 06/2024) calculates the positioning of the gears in a system that must meet specific criteria related to gear design data and geometric constraints between the gears.

The following results can be found in the report:

- Center coordinates of all gears of the system

- Graphical representations of the feasible gear configurations

In addition to the report, the results are shown in several graphics. Report and user interface are both available in metric or US customary units. Supported languages for user interface and report are English, German, French, Spanish and Korean.

8.1 Input Parameters

The input parameters are all shown at three different tab pages.

8.1.1 General
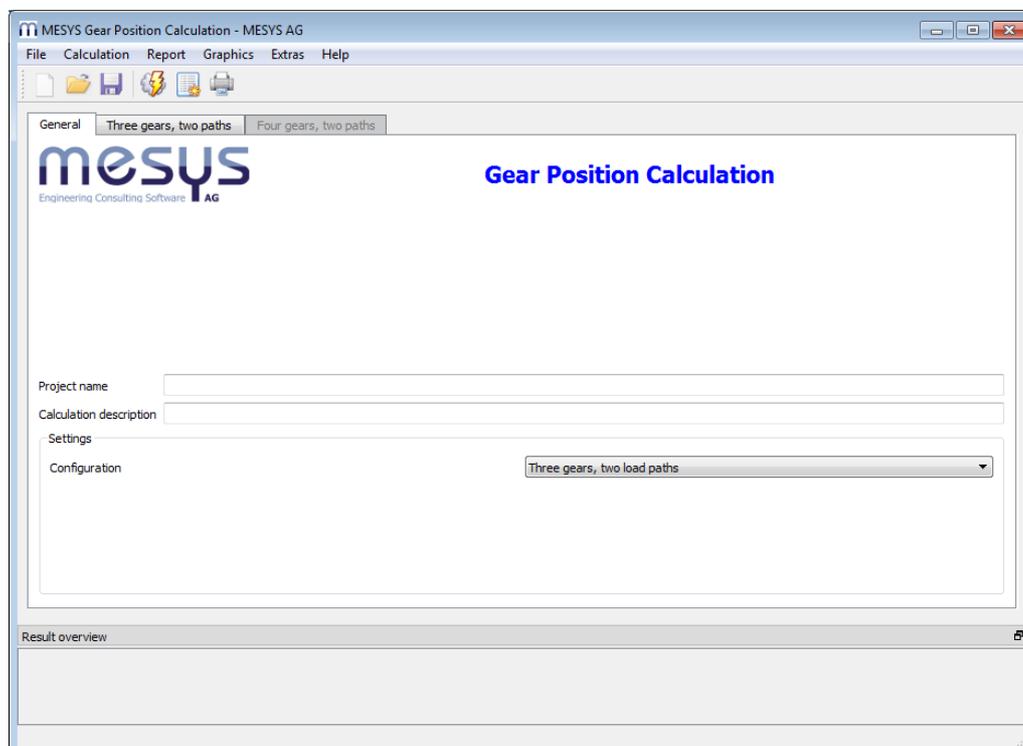
Project name and calculation description

The project name and the calculation description are just inputs which are shown in the report header. They can be used to enter information about the purpose of the calculation for documentation.

Settings

Two configurations are available:

- Three gears, two load paths

- Four gears, two load paths

8.1.2 Three gears, two paths

This module of the program allows a configuration of a symmetric system with two different paths for transmitting the power composed of three gears each.
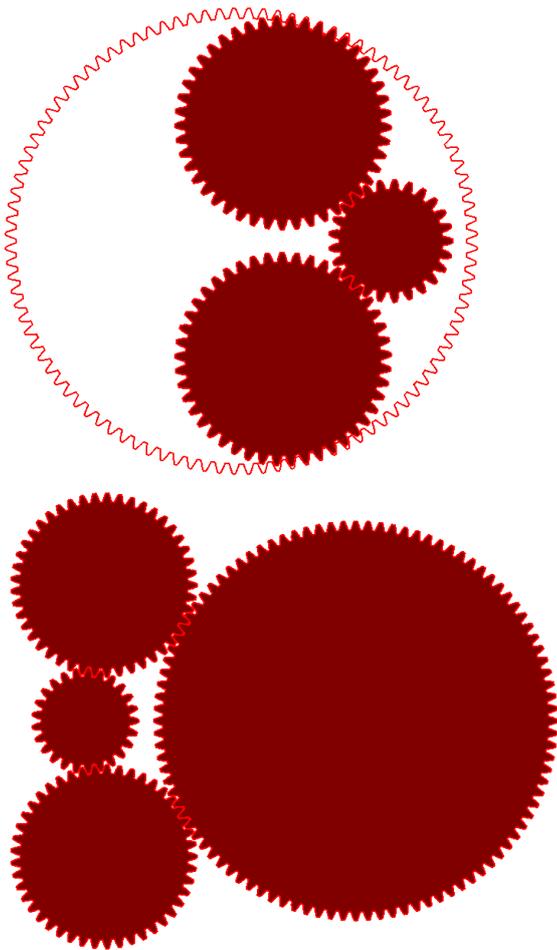
8.1.2.1   System data

At this section, common general data and settings for the whole gear system can be defined, such as:

- Normal module

- Normal pressure angle

- Helix angle

- Basic rack addendum: h_{aP} is the coefficient to define the addendum of the standard basic rack tooth profile defined as addendum = $h_{aP} * m_n$. This value is normally equal to 1, thus being the addendum = m_n.

- Required clearance: The required clearance is a restriction that sets a minimum permissible distance between the tooth tips of unconnected gears to prevent them from colliding with each other.

8.1.2.2 Gears

For each of the three gears, the number of teeth and profile shift can be specified as design parameters. A positive value for the teeth will be understood as an external gear by default, thus being negative values used for modeling internal gear teeth. Due to assembly reasons, only the third gear can accept negative values for the teeth.

8.1.3 Four gears, two paths
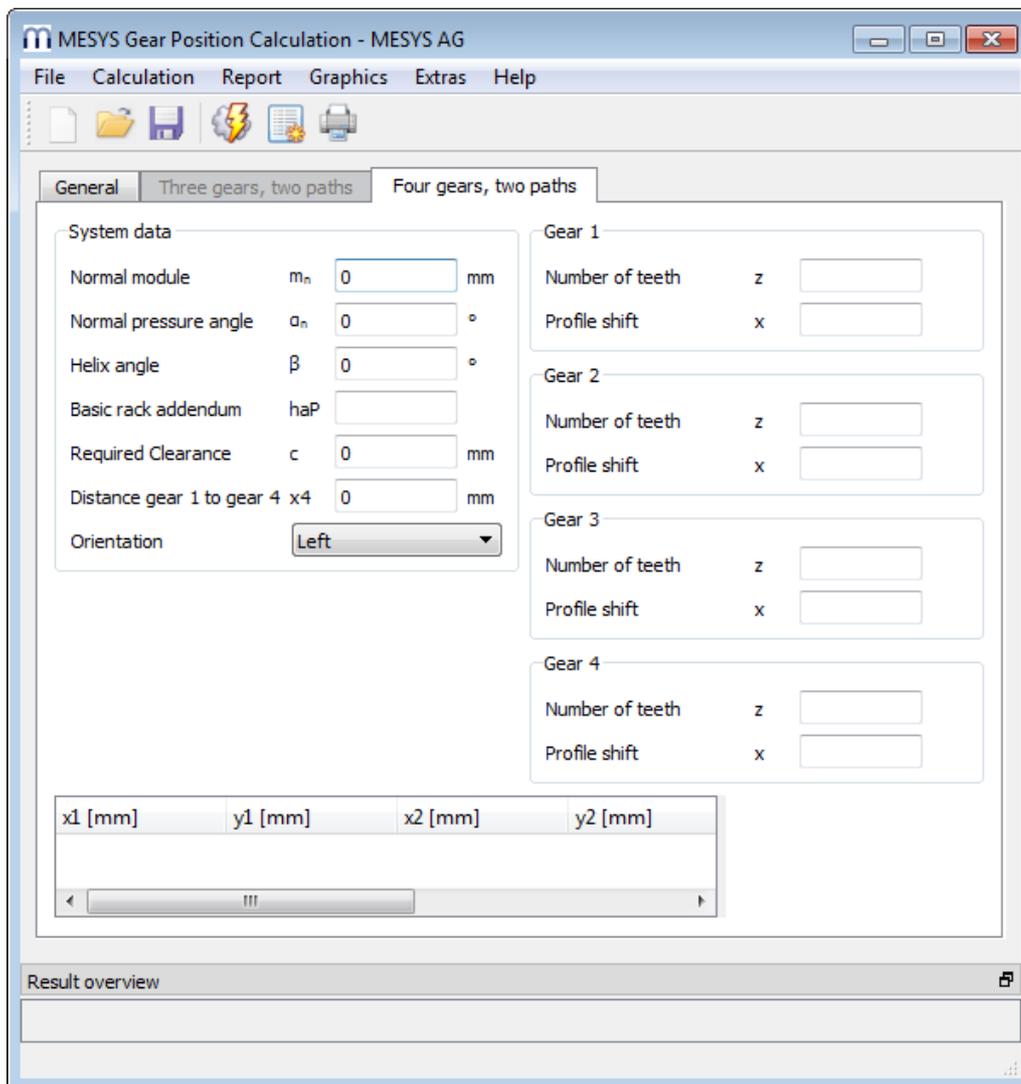
This second module of the program allows a configuration of a symmetric system with two different paths for transmitting the power composed of four gears each.

8.1.3.1   System data

As for before at this section, common general data and settings for the whole gear system can be introduced, such as:

- Normal module

- Normal pressure angle

- Helix angle

- Basic rack addendum: h_{aP} is the coefficient to define the addendum of the standard basic rack tooth profile defined as addendum = $h_{aP} * m_n$. This value is normally equal to 1, thus being the addendum = m_n.

- Required clearance: The required clearance is a restriction that sets a minimum permissible distance between the tooth tips of unconnected gears to prevent them from colliding with each other.

- Distance gear 1 to gear 4: This distance is defined as the distance between the gear centers of the gear 1 and gear 4. Positive values will drag the center of gear 4 rightwards with respect to the center of gear 1.

- Orientation: The orientation can be selected as 'Right' or 'Left', which will constrain the possible results to those configurations where the second gear is restricted to maintaining its position either on the left or the right side of the center line between the first and third gear, as shown below:

Orientation: Left   Orientation: Right

8.2   Results

Results are provided as a small result overview directly in the software, a main PDF text report and separate graphic windows. Graphic windows can be docked onto the main program interface and are automatically updated on each calculation.

x1 [mm]	y1 [mm]	x2 [mm]	y2 [mm]	x3 [mm]	y3 [mm]	x4 [mm]	y4 [mm]	cMin [mm]
0	0	-17,1074	14,6146	3,96374	39,3585	20	0	2,22924
0	0	-15,3923	16,4111	6,60614	40,3343	20	0	5,82229
0	0	-13,4067	18,0696	9,45352	41,1706	20	0	9,13916
0	0	-11,1271	19,556	12,5316	41,8387	20	0	11,2395
0	0	-8,51245	20,8276	15,8838	42,3002	20	0	12,6907
0	0	-5,4801	21,8224	19,5951	42,4981	20	0	14,4522
0	0	-1,82268	22,4261	23,8748	42,323	20	0	16,5926
0	0	3,40671	22,2406	29,6693	41,3854	20	0	17,4812
0	0	10,32	19,9937	36,5901	39,1282	20	0	12,9874
0	0	13,1398	18,2646	38,9175	38,0576	20	0	9,5292

Result 1

Result overview

Gear ratio	i	6	Number of results	n	16

Result 5

Result 10

Result 15