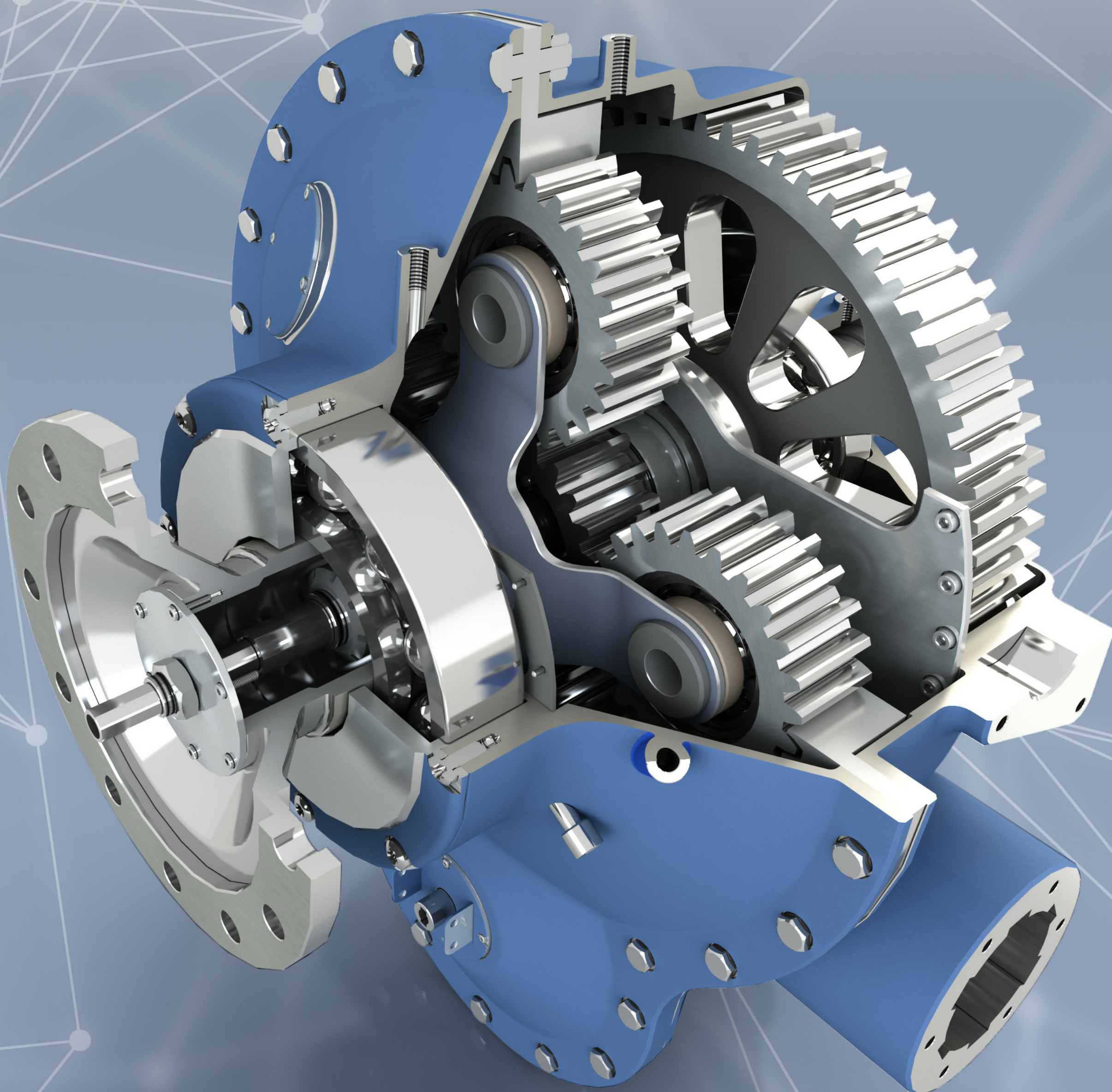
T-FLEX Gears 17

USER MANUAL

Gears Design



Content

Main Concepts

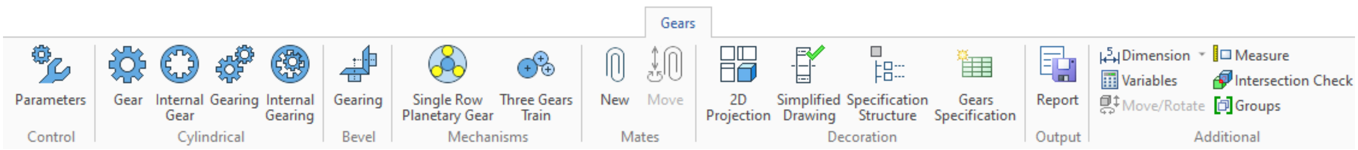
T-FLEX Gears is an integrated add-on in T-FLEX CAD. The application supports such system-wide functionality for T-FLEX CAD as parameterization and work with units of measurement.
Add-on commands are called through the Ribbon. The main add-on commands for creating and analyzing gear objects have a dialog standard for all T-FLEX CAD commands, as well as an alternative modal dialog.

Topics in this section:

Installation and Launch

To install and run the add-on, you must perform the following sequence of actions:

- Install T-FLEX Prerequisites 17. To install, run the Setup.exe file from the "T-FLEX Prerequisites 17" directory and follow the instructions of the installation program.
- Install CAD-system T-FLEX CAD 17. To install, run the .msi file and follow the instructions of the installation program.
- Install the "T-FLEX Gears" plug-in add-on. To install, run the .msi file from the "T-FLEX Gears 17" directory and follow the instructions of the installation program.
- To start the Gear add-on, you need to open an existing 3D document or create a new one and save it to disk.
- Then go to the Gears tab and select the required command for creating and calculating gears.



When choosing commands for creating gears and mechanisms, if the current document has not yet been saved to disk, the system will promptly offer to save a new document.

Add-on Commands

Parameters. Setting default parameters when running add-on commands.
Cylindrical > **Gear**. Creation of a single, non-connected, cylindrical gear with external teeth according to the specified parameters.
Cylindrical > **Internal Gear**. Creation of a single, non-connected, cylindrical gear with internal teeth according to the specified parameters.
Cylindrical > **Gearing**. Command to create a cylindrical gearing. Allows you to perform geometrical and strength analysis of the engagement. This is the main spur gear dialog divided into thematic expandable tabs.
Cylindrical > **Internal Gearing.** An additional way to enter the **Gearing** command, in which the starting settings differ: instead of the external one, the internal gearing is active.
Bevel > **Gearing.** Command to create a bevel gearing.
Mechanisms > **Single Row Planetary Gear**. Dialog of automatic arrangement of gearing into a single-row planetary mechanism according to the given parameters of the mechanism.

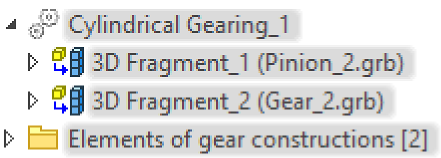
Mechanisms > **Three Gears Train**. Dialog of automatic arrangement of gearing into a chain of three gears according to the specified parameters of the mechanism.
Decoration > **Simplified Drawing**. Command for creating a simplified drawing of gears on the projections of the add-on objects.
Decoration > **Specification Structure**. Dialog for setting a typical table of parameters for the design of the gear drawing.
Decoration > **Gears Specification**. Dialog for creating a design table on the selected drawings of gears.
Output > **Report**. Dialog for creating reports on the performed calculations.

Working with Add-on Objects

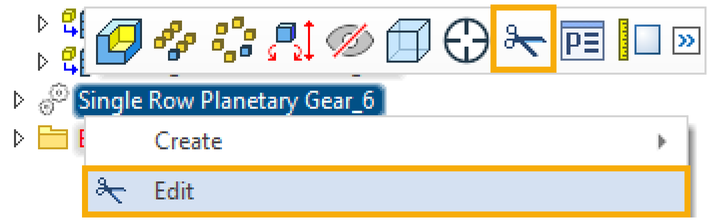
To start working with the add-on, an existing 3D document or a new 3D document (assembly or part) must be opened. Any object created using the add-on in the current 3D document will be a fragment or a group of fragments of the first level. The current document will be an assembly of such fragments. The current document can be either empty, or already contain bodies and fragments.
Upon completion with confirmation of the commands to create single gears or gearings, 3D models of gearing or single gears will be created. Optionally, you can also create 2D models of gearing objects simultaneously with 3D models. Each element of a gear pair or mechanism is a fragment of the first level inserted into the assembly: the current 3D document in which the gearing object was created. A fragment, which is a gearing element, is saved by the add-on from the gear template to the folder where the working project is saved. When editing a gearing using the add-on commands, the original body geometry inside the gearing fragment is edited. The body within the gearing fragment defines a ring gear. When editing gear models using add-on commands, all modifications made on the gear model will be saved. The only exceptions are modifications made directly on the tooth profile, because the number of teeth during editing can be changed.
When inserted into an assembly, all gearing fragments receive transformations that ensure their correct position in the gearing: if you manually edit the position of the gearing fragments, their position in gearing will be violated.
In the **Gears** add-on, to distinguish gears in a pair of gearing, the lower (on the model and animation of the gearing) gear is called **Pinion**, and the upper gear is called **Gear**.

Gears in Model Tree

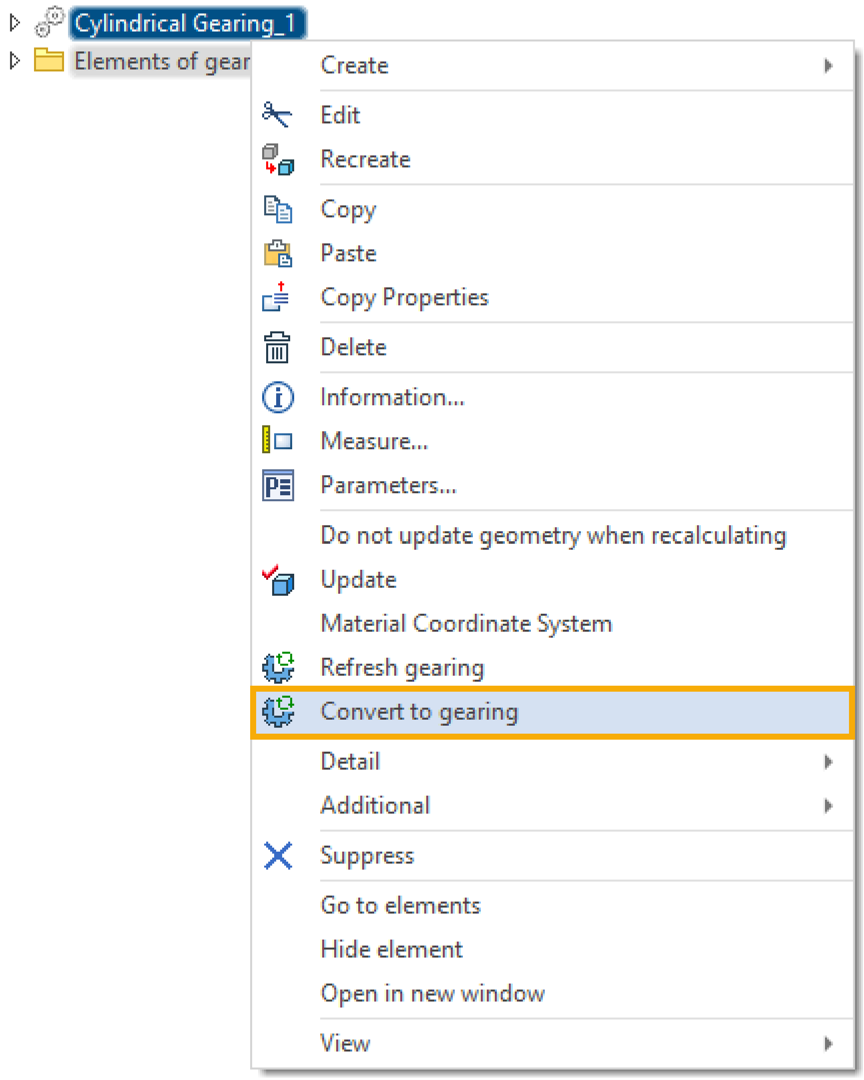
Each pair of gearing fragments is combined into an object with a corresponding name: when such an object is called for editing, the dialog of the Gearing command will open.



Each mechanism is united into an object of the corresponding mechanism containing gearing objects, each of which contains fragments of gears: when you call the mechanism object for editing, the mechanism dialog will open, when you call the gearing object for editing, the dialog of the **Gearing** command will open.

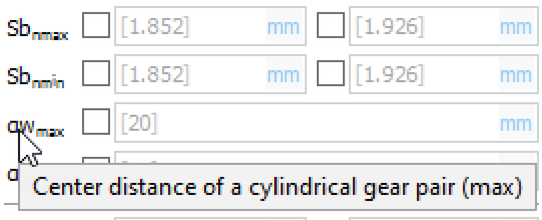


When creating a single gear (or internal gear), an object containing only one fragment of a gear will also be created: when such an object is called for editing, the gear dialog will be called.
The option of converting a single gear to a gearing is available through a special command of the context menu: in this case, the **Gearing** command dialog will open.

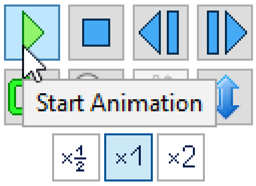


Hints

The add-on uses a large number of symbols to indicate parameters. The use of symbols is only applicable in cases where there is a standard for the designation with a symbol. For all parameters that have a symbolic designation in the add-on interface, hints with the full name of the parameter are provided. To call the hint window, which is the system-wide mechanism of T-FLEX CAD, you need to point and hold the cursor on the object of interest, in this case, on the parameter symbol.

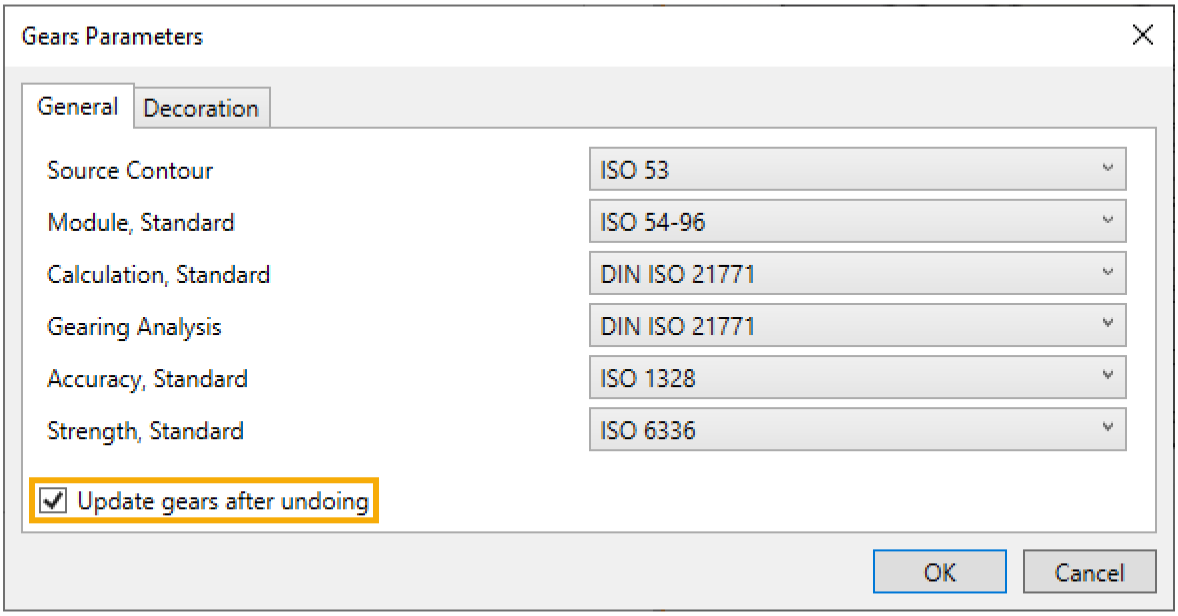


There are similar hints on the icons of buttons and switches.



Undo

As mentioned earlier, each element of a gear pair or mechanism is a fragment of the first level inserted into the assembly. If the **Undo** command is used to cancel actions in the add-on command, then you need to set the **Update gears after undoing** in the **Parameters** command dialog box, in this case, all the parameters of the gears will be returned one step back, and the fragments of the gears themselves are recalculated and updated in the assembly.



By default, this flag is set so that canceling actions will lead to the expected result. However, regeneration of 3D fragments of gears can take a long time. At the same time, if changes in the document do not concern gears, or do not lead to final values for the formation of gears, then it is more rational to remove this flag. If the undoing nevertheless leads to changes in the parameters of the gears, they can always be manually updated using the **Full Regeneration** command, the Gears item.

If the document was regenerated, but the gears themselves were not regenerated, the gearing object will be marked with a special icon.

After updating the gearing, the icon will disappear.

Manual Position Editing

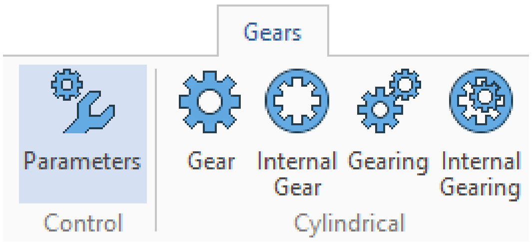
For the correct position of the gears that are part of the add-on objects, fragment transformations are used. Calculated transformations of fragments are automatically assigned in special groups at each regeneration by the add-on commands. All custom additions to transformation groups, automatically created by the add-on, will be deleted during subsequent regeneration.

> You cannot edit or delete transformation groups created by the add-on.

If you need to add a manual transformation for a fragment included in the add-on objects, you need to create a custom transformation group of a fragment (or several groups) and add the necessary transformations to them.

Parameters and Parametrization

The **Parameters** command is intended for managing the general settings of the add-on.



In the T-FLEX Gear add-on, parameterization is available for all fields of the main dialogs.

Topics in this section:

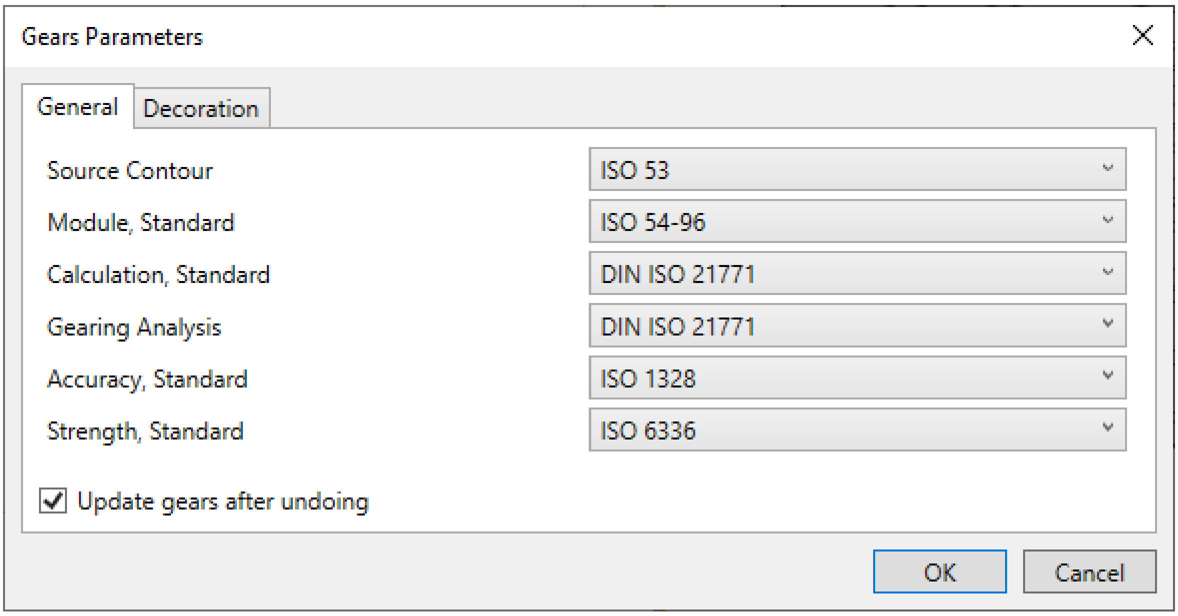
- Parameters
- Parametrization and Updating

Parameters

The command allows you to set parameters for the add-on. The dialog is divided into 2 tabs:

- General;
- Decoration.

The first tab indicates the standards that are automatically selected in all commands of the add-on. This allows the user to customize the work of the add-on to the standards that he uses.

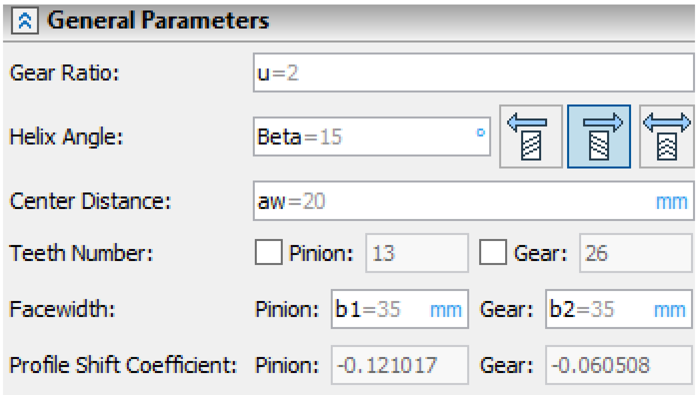


The option **Update gears after undoing** determines how the add-on works in the **Undo** command mode.

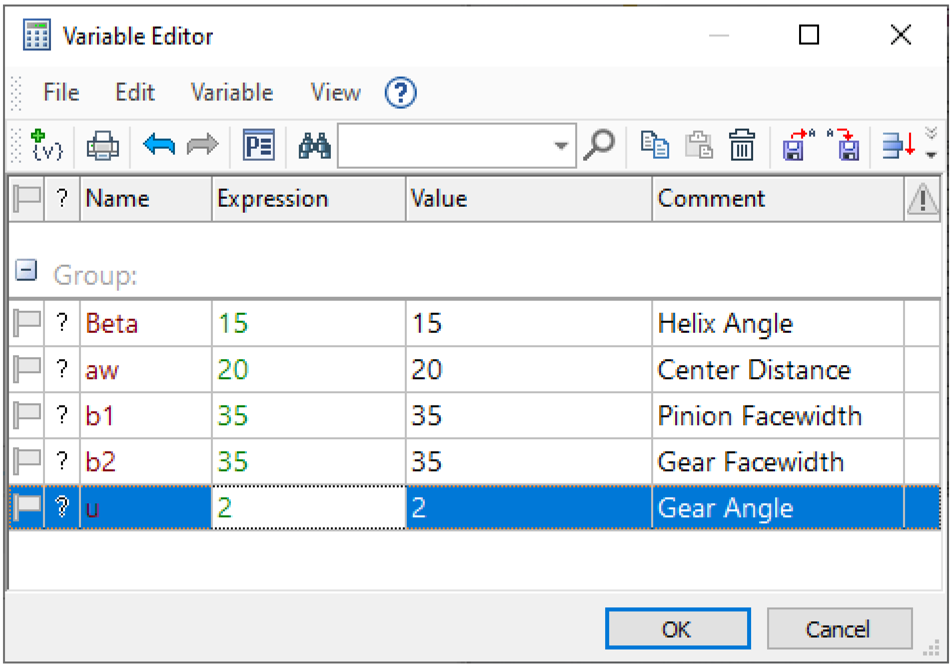
On the second tab, you can set up a typical version of the gear drawing design table for the add-on as a whole, i.e. customize the initial view of the table for the Specification Structure command. The dialog on the **Decoration** tab duplicates the dialog of the **Specification Structure** command. The customization is not a one-time, and, if necessary, it is always possible to make changes to the typical prototype of the specification.

Parametrization and Updating

All fields of the main command dialog support parameterization.

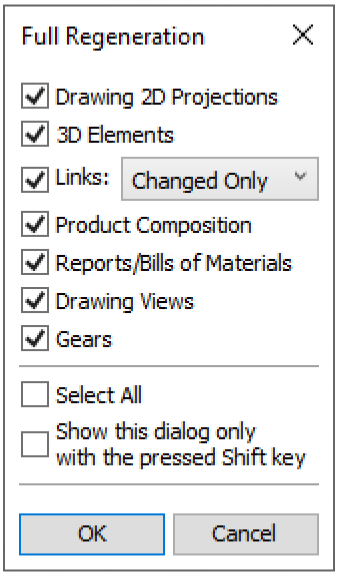


As with parameterization in T-FLEX CAD commands, all variables are available in the variable editor. All the possibilities of working with variables are available for variables in the editor.

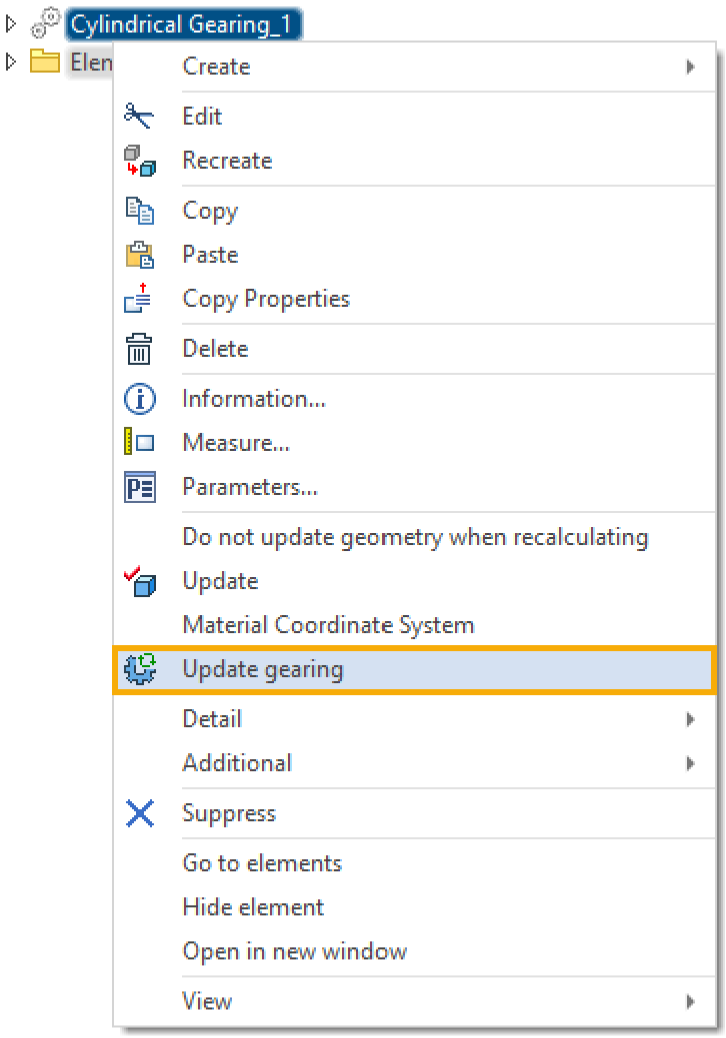


When changing the values of variables, as a result of calculation by formulas or simply by manual input, the new values will be displayed in the link editing dialog. When you exit the gearing edit with confirmation, the gearing fragments will be regenerated.

To regenerate all the gearings according to the new values of the variables and not enter the editing of the gearing, the command **Full Regeneration** provides a special option **Gears**.



Also, any gearing can be updated through the context menu of the gearing object.



Solving Typical Tasks

This section shows basic examples and explanations to familiarize yourself with the add-on. The Quick Start section describes an introductory lesson on working with the T-FLEX Gears add-on.

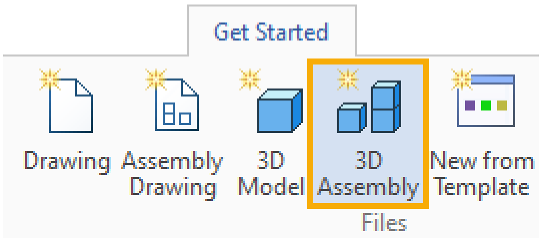
Topics in this section:

- Quick Start
- Gearing Creation
- Spur and Helical Gears
- Double Helical Gear
- External and Internal Gearing
- Separate Gear
- Geometry Modification
- Binding and Mates
- Control Dimensions Calculation
- Calculation and Creation of Backlash

Quick Start

In this lesson, we will create a cylindrical gearing and define basic geometrical and kinematic parameters, perform standardized gearing analysis, strength analysis, create a gears specification, and much more.

To get started, open the 3D assembly prototype by running the command from the ribbon:



In this 3D assembly, we will be creating gears.

> Each gear is created from a 3D model prototype. To work with the add-on, you need to save the main assembly file, since all created gears are placed in the folder with the assembly file.

Save your document to disk. After saving, go to the **Gears** ribbon tab .

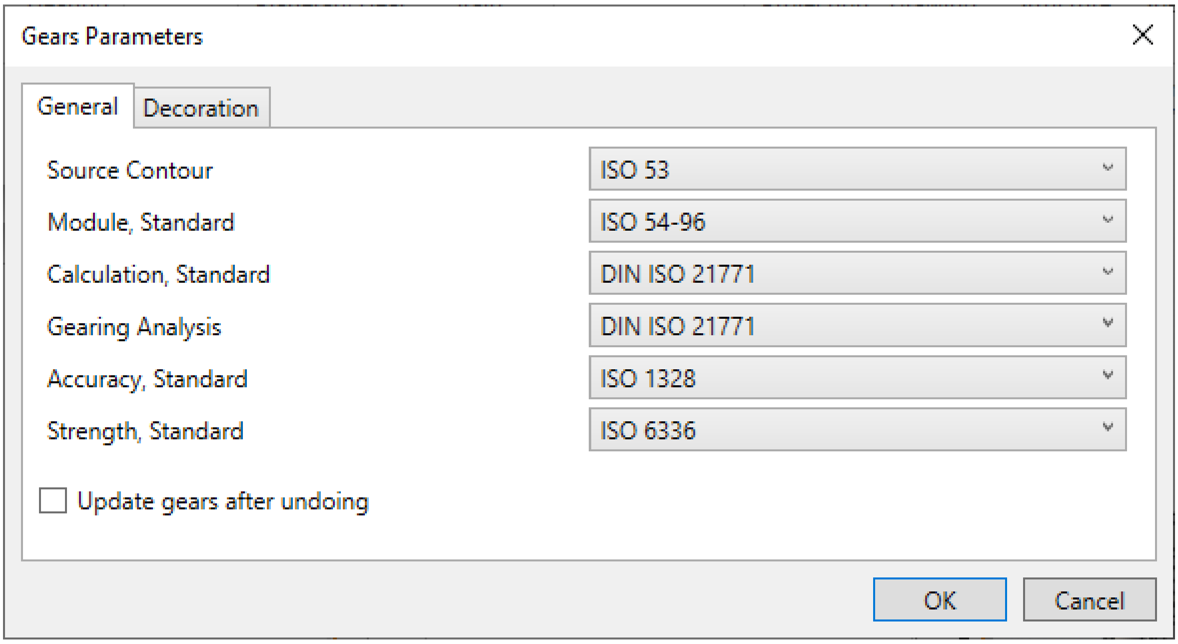
The tab is divided into 6 groups - **Control**, **Cylindrical**, **Bevel**, **Mechanisms**, **Mates**, **Decoration**, **Output** and **Additional**.

The **Control** tab contains the **Parameters** command, which allows you to specify general settings for the add-on. The dialog is divided into 2 tabs:

- General;

- Decoration.

The first tab contains standards that are automatically selected in all commands of the add-on. This allows the user to customize the work of the add-on to the standards that he uses.



In this tutorial, we will use the standard parameters. In addition, you can configure the **Update gears after undoing** option. When this option is enabled, the gearing will be regenerated each time an action is **Undo**. When the option is disabled, the gear will not be regenerate when the action is canceled and it can only be regenerated using the **Full Regeneration**, the **Gears** option.

> The use of this option is recommended when the gears do not have a large number of teeth, because otherwise, each undo will take a long time.

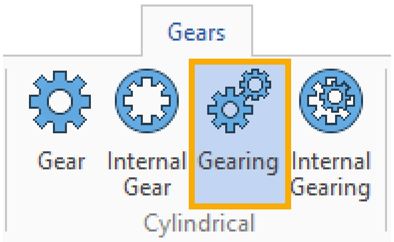
On the second tab, you can set up a typical version of the gear drawing design table for the add-on as a whole, i.e. customize the initial view of the table, but we'll talk about that later.
We will use the table below as initial parameters.

Parameter Name		Designation	Value
1. Teeth number	pinion	z_1	32
	gear	z_2	64
2. Normal module, mm		m	5
3. Facewidth, mm	pinion	b_1	60
	gear	b_1	60
4. Profile shift coefficient	pinion	x_1	0
	gear	x_2	0
5. Helix angle		β	16°25'
6. Quality according to ISO 1328		—	6
7. Flank roughness, μm		*Rz*	2,0
8. Fillet surface roughness, μm		*Rz*	2,0

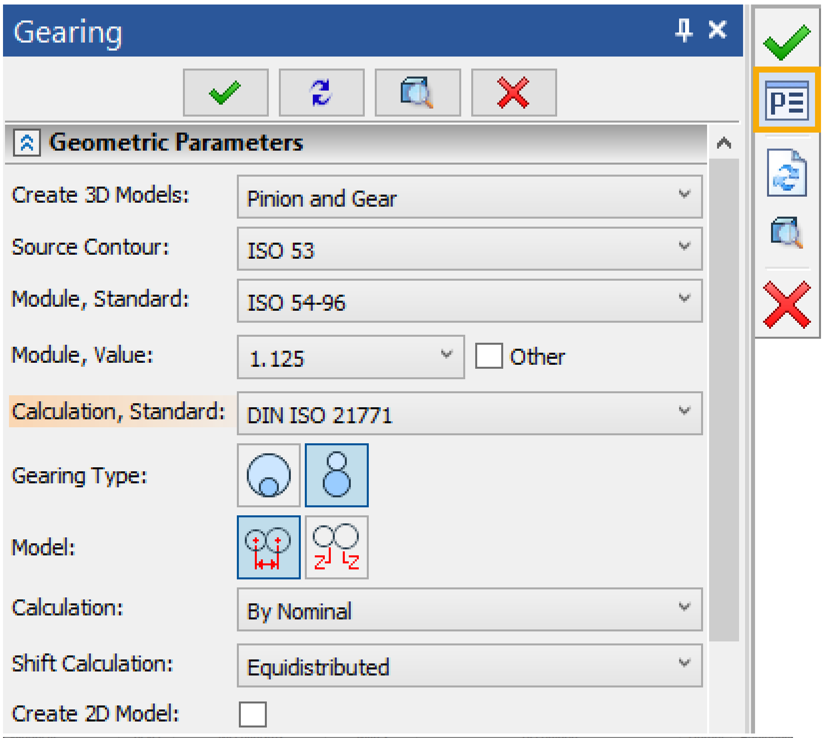
9. Load cycle		—	Constant load $T_1 = 170\ N * m$
10. Drive gear rotation frequency (speed), revolution		n_1	600
11. Required Service Life, h		L_h	1000
12. Material	pinion		Case Hardened Steels (Wrought) (Eh)
	gear		Through Hardened Steels Carburized (NV nitrocar.)
13. Yield Stress, MPa	pinion	H_{O1}	1000
	gear	H_{O2}	900

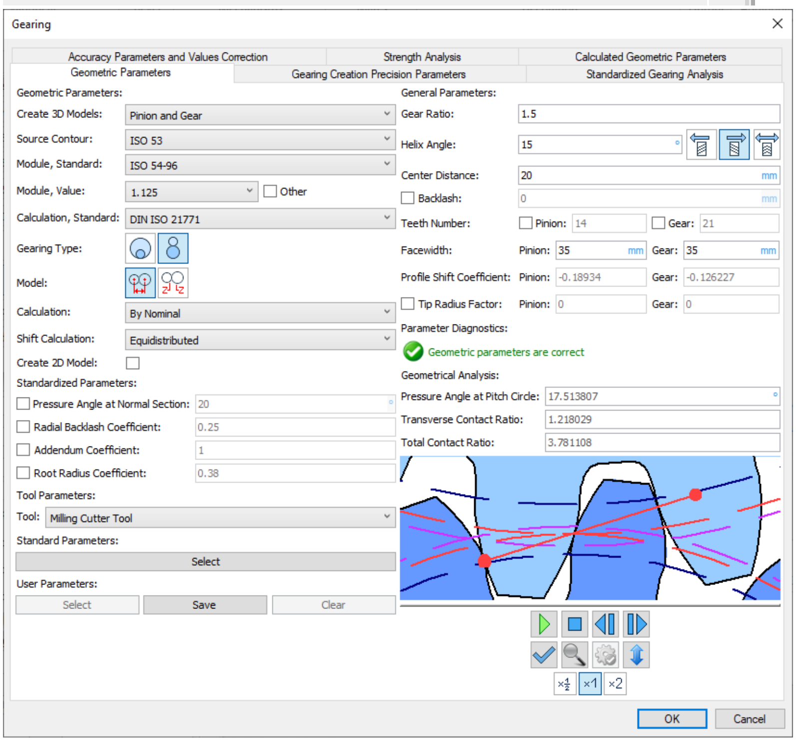
Set the general calculation parameters and create the gearing

To get started, run the **Gearing** command from the **Gears** ribbon tab:

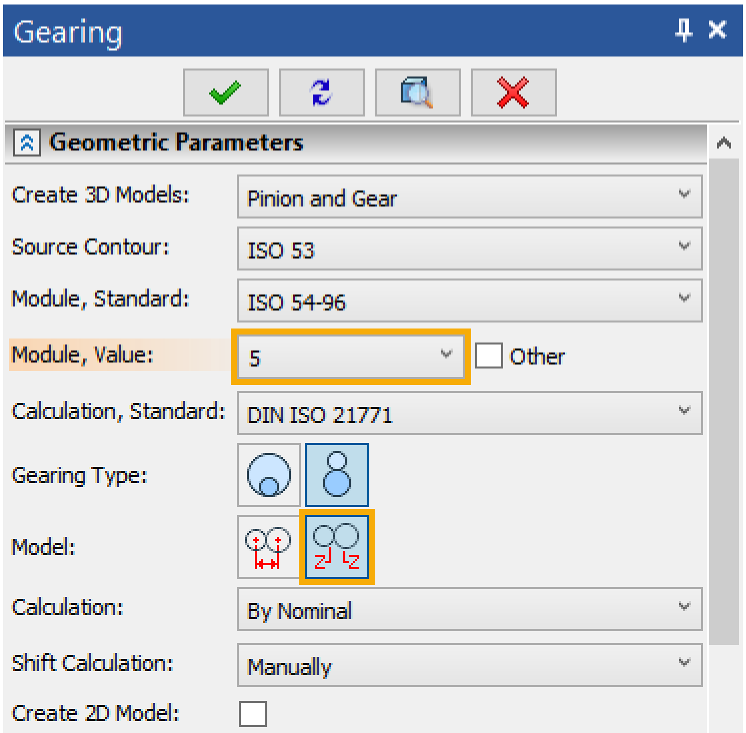


> You can work with the **Gears** add-on both in the system **Parameters** window and in a special modal window. The modal window is called from a special automenu command:





In the **Geometric Parameters** tab, set the tooth modulus equal to **5** and select the model type **Teeth Number**, since in the original table we know the value of the number of teeth on both gears:

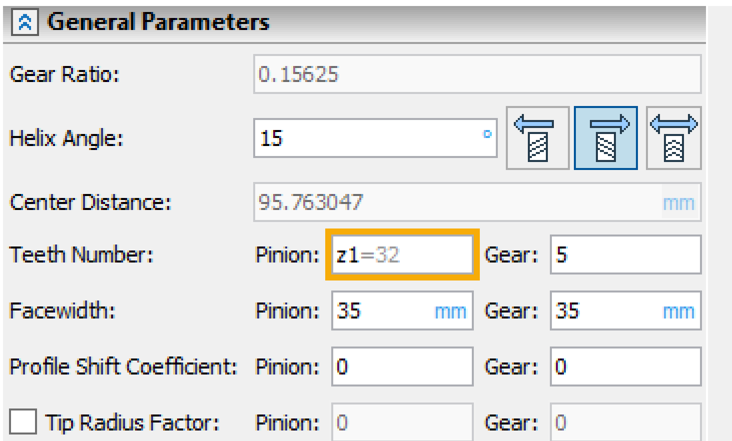


We will leave the other parameters by default, since we will create a gearing according to the standard general parameters with an external gearing type.

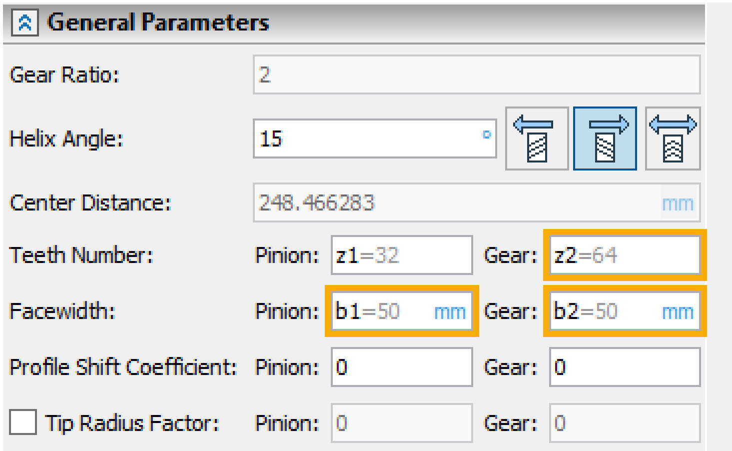
Go to the **General Parameters** tab.

> All fields of the main command dialog support parameterization. As with parameterization in T-FLEX CAD commands, all variables are available in the variable editor. All the possibilities of working with variables are available for variables in the editor.

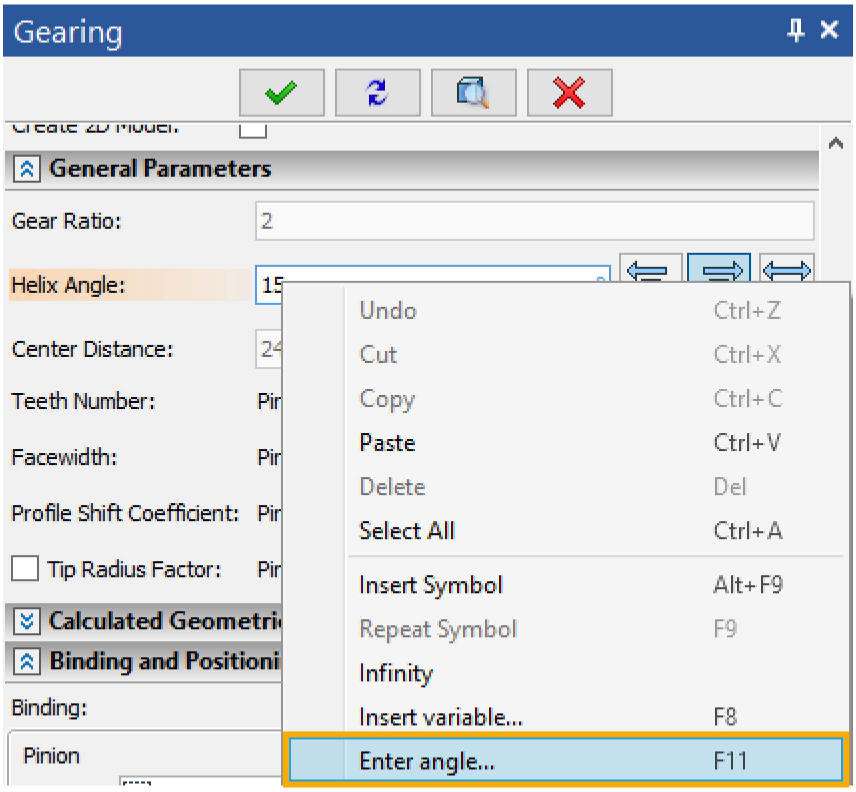
In this tab, we will set variables for some of the general parameters. Enter the variable **z1** in the **Teeth Number** on pinion field and set the value **32**:



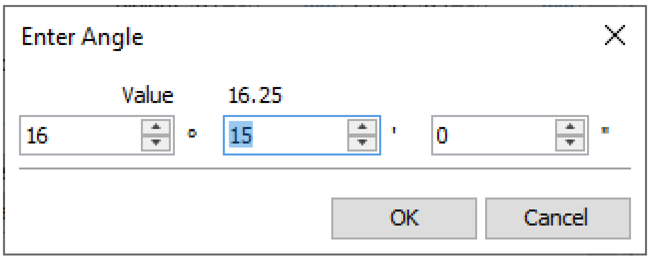
Similarly, set the number of teeth on the gear equal to **z2=64** and set the thicknesses **b1** and **b2** equal to **50** mm for each element:



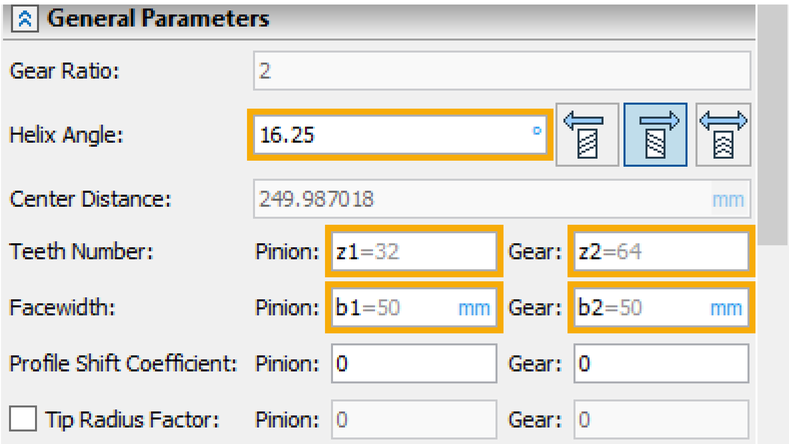
Now you need to set the helix angle of the teeth. By condition, the helix angle is **16°15'**. Click  in the **Helix Angle** field and select **Enter Angle**...:



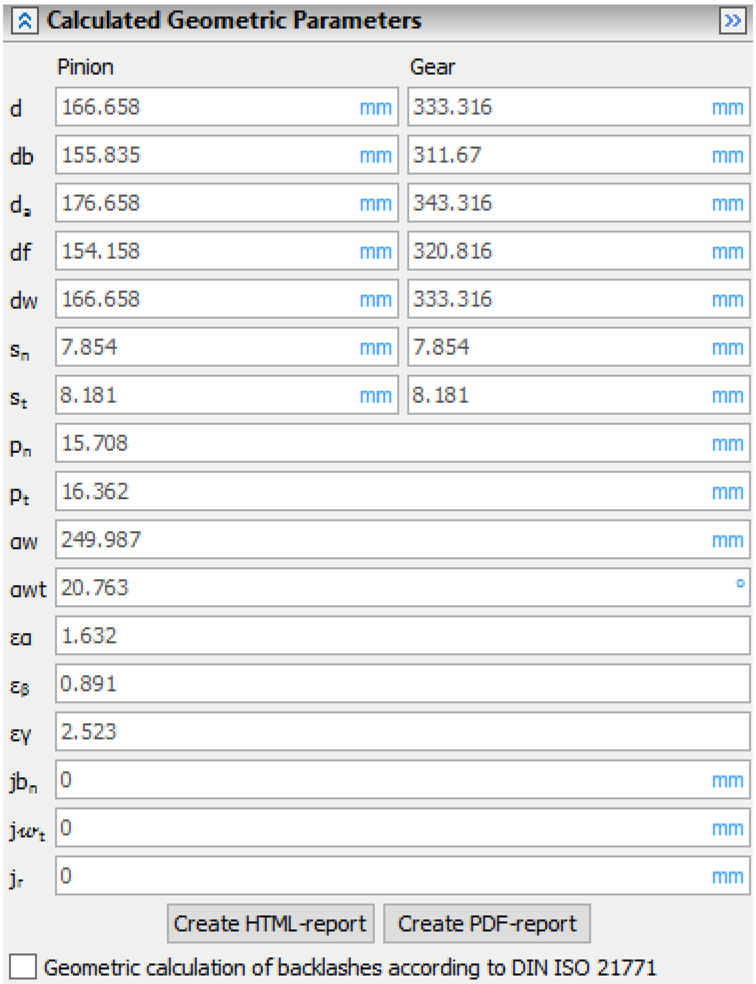
In the window that appears, enter the helix angle values and click **OK**:



Thus, we have set all the general gearing parameters:

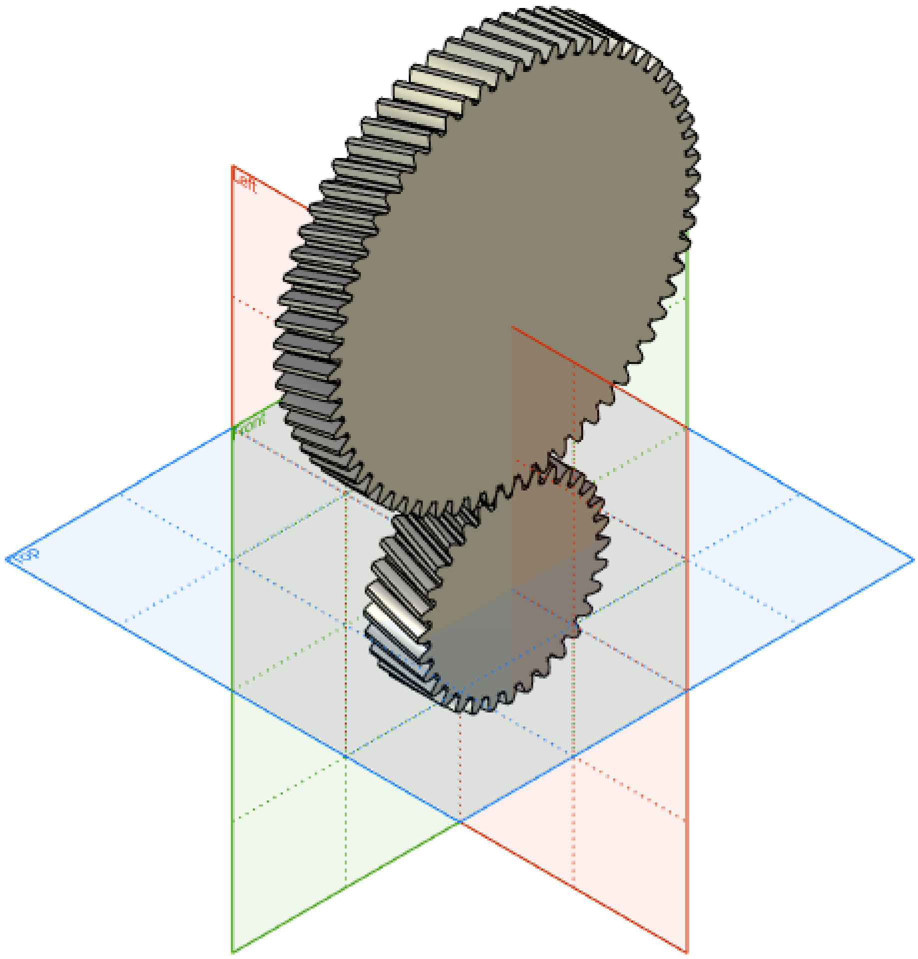


The following result should appear on the **Calculated Geometric Parameters** tab:

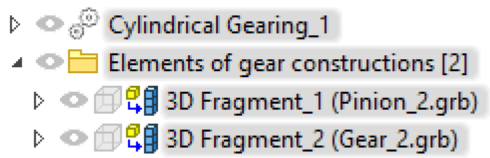


The button for outputting all geometric and kinematic parameters to a separate HTML or PDF file is located on the **Calculated Geometric Parameters** tab.

Now you can create a gearing. Click ✓. As a result, a cylindrical helical gear will appear in the 3D scene:

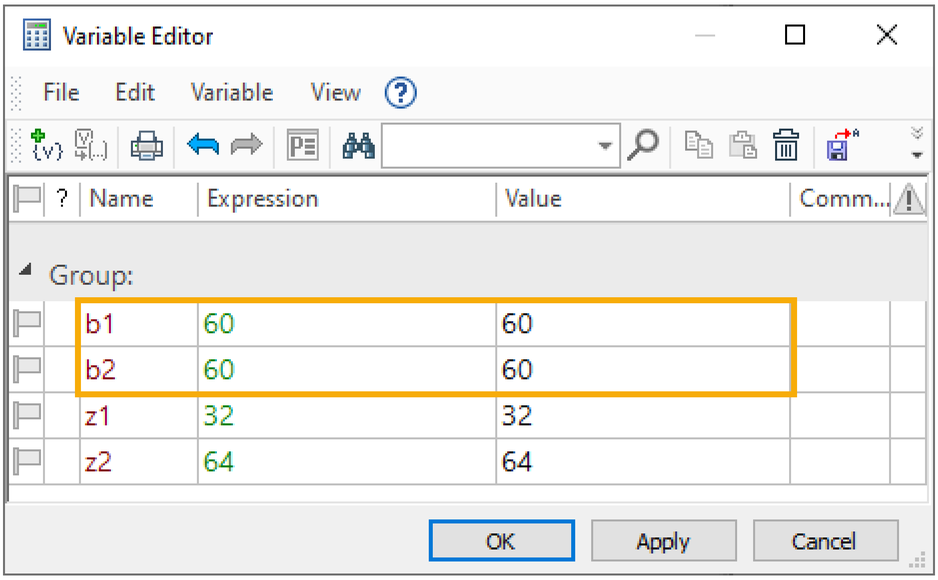


Gears are marked with a special icon in the model tree, and its elements are placed in the **Elements of gear constructions** folder:

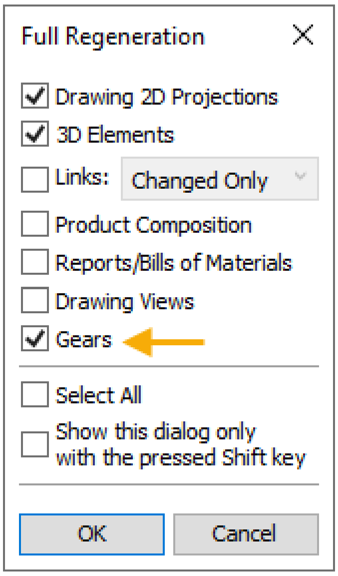


After creating the gearing, you can create drawings or modify the geometry of the gear elements (create holes, hubs, etc.). To do this, go to the required fragment and start working using standard T-FLEX CAD tools. For this tutorial, we will focus only on working with the **Gears** add-on.

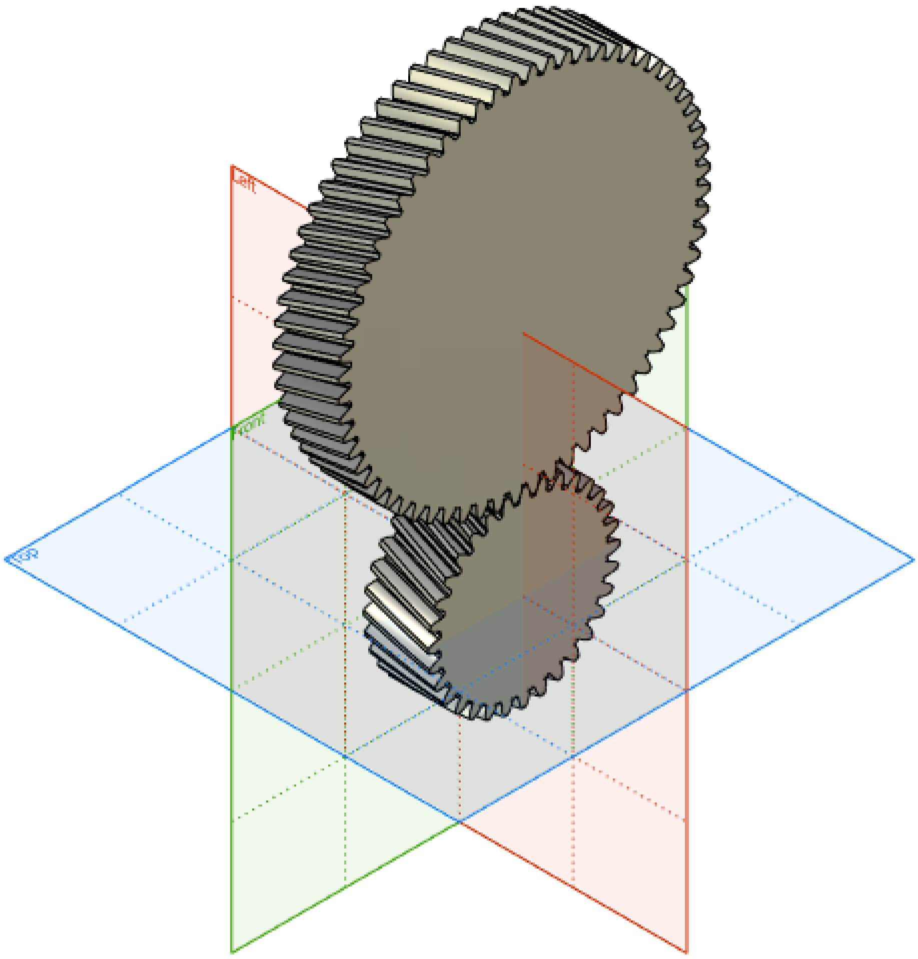
Now change the value of the thicknesses **b1** and **b2** to **60** mm from the variable editor:



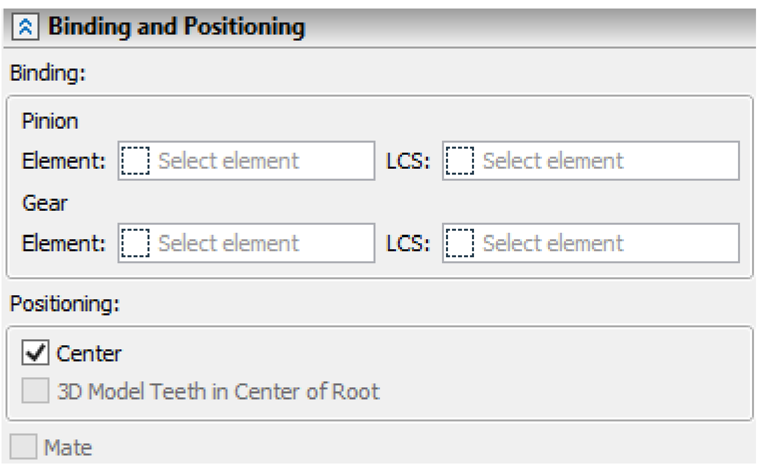
Click **OK**. In the **View** toolbar, select **Full Regeneration**, activate the Gears checkbox and click the OK button.



As a result, the gears geometry will be updated:



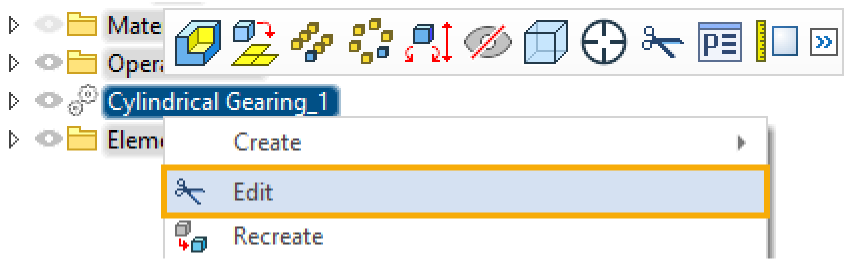
Gears can also be created along cylindrical faces, 3D nodes and LCS, center the position of gears if they have different facewidth, and also position the teeth in the center of the root (when calculating in the middle of the tolerance field). The **Binding and Positioning** tab is intended for this.



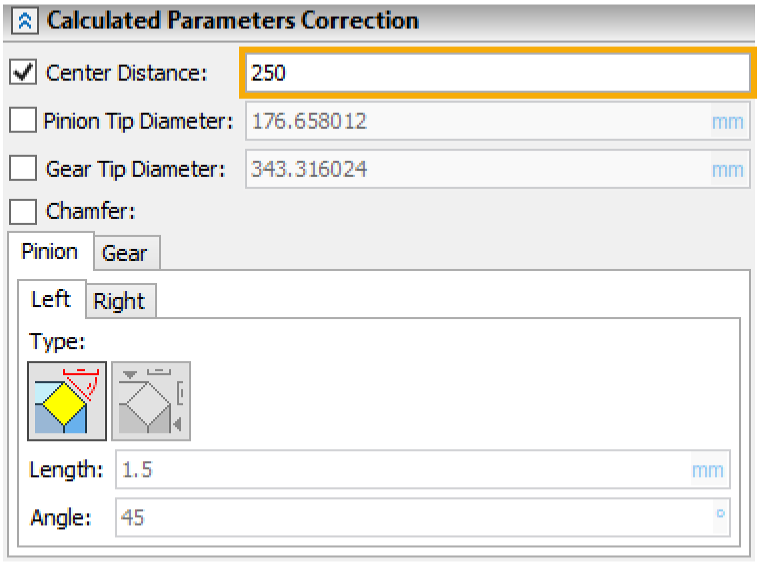
Edit the gearing and perform geometric analysis

Suppose we need to round off the center distance from **249.987** to **250**.

Select in the model tree entry **Cylindrical Gearing_1** and click **Edit**:

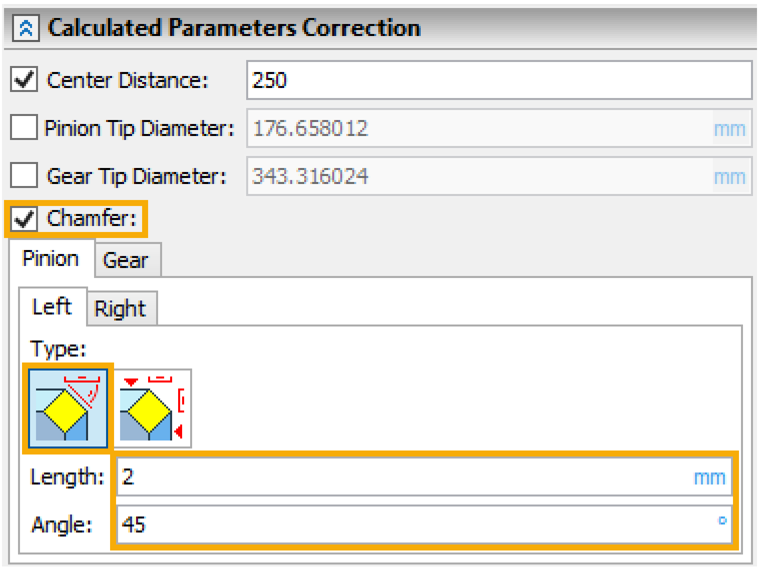


Go to the **Calculated Parameters Correction** tab. Check the box next to **Center Distance** and enter a value of **250** mm. Thus, you will enter the correction for the calculated value of the center distance:

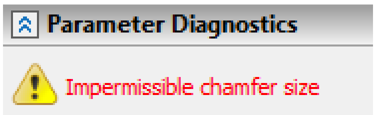


If you change any parameter related to the center distance (for example, the number of teeth), then the **Center Distance** option will take priority and the center distance will remain 250 mm, although the number of teeth (respectively, the calculated center distance) will be recalculated.

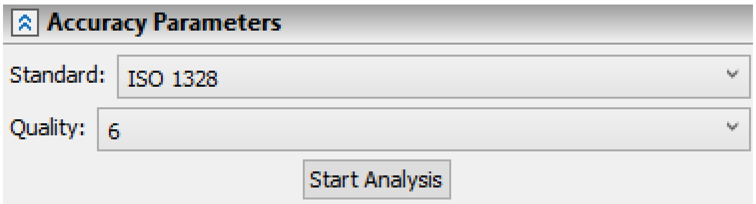
Next, specify the **Length - Angle** chamfer for the pinion and gear with **2x45°** parameters:



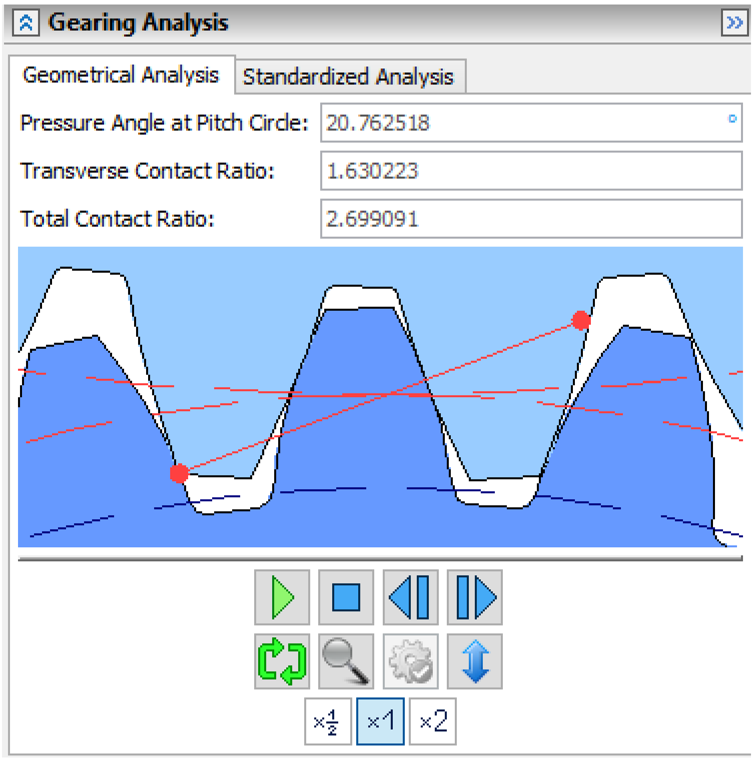
If you enter incorrect parameters for gearing, this will be displayed in the special **Parameter Diagnostics** tab :

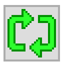


In the **Accuracy Parameters** tab, leave the default parameters.

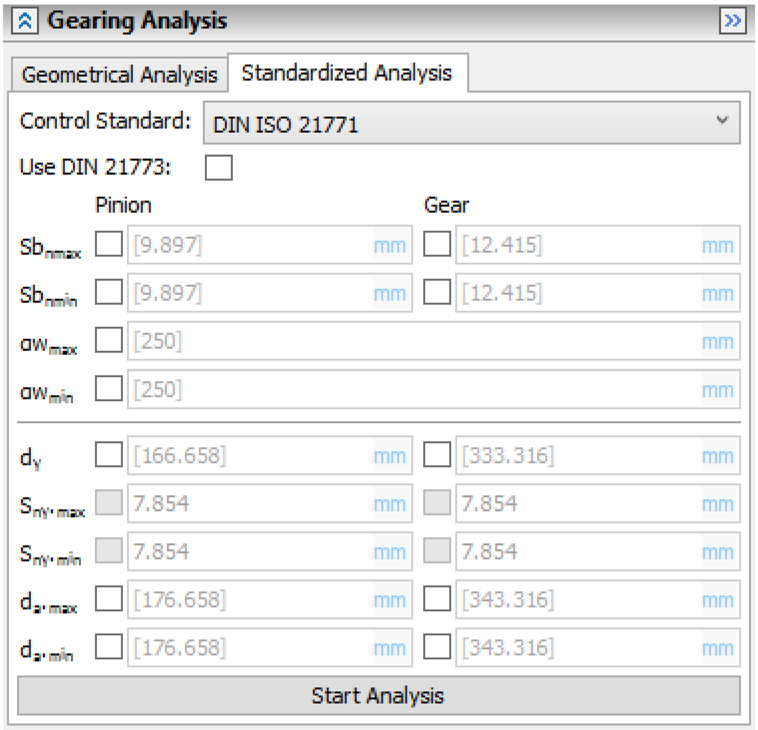


Now go to the **Gearing Analysis** tab:

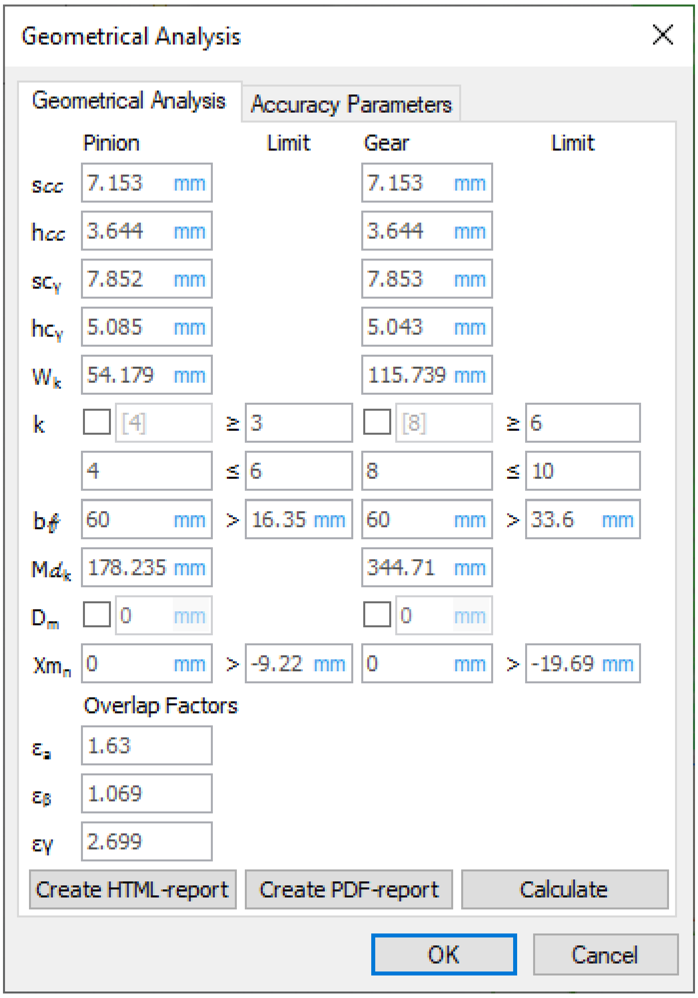


In this tab, you can see an animated gearing scheme. You can zoom in and out, start animation, change its speed, go one step forward or one step back, etc. The  button is for updating the gear animation. After pressing the button, it changes to  and shows the relevance of the animation and graphs window. The  button shows no intersections. If the gearing is not relevant or there is some kind of intersection, then the buttons will change.

Go to the **Standardized Analysis** tab:



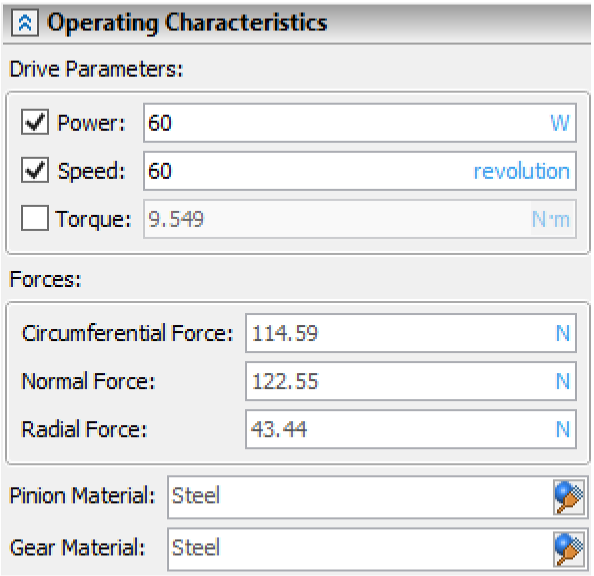
Here you can select the standard by which the gearing analysis will be performed and set the additional parameters. Leave the default values and click the **Start Analysis** button. A window will appear in which the geometrical analysis of the gearing will be displayed in detail:



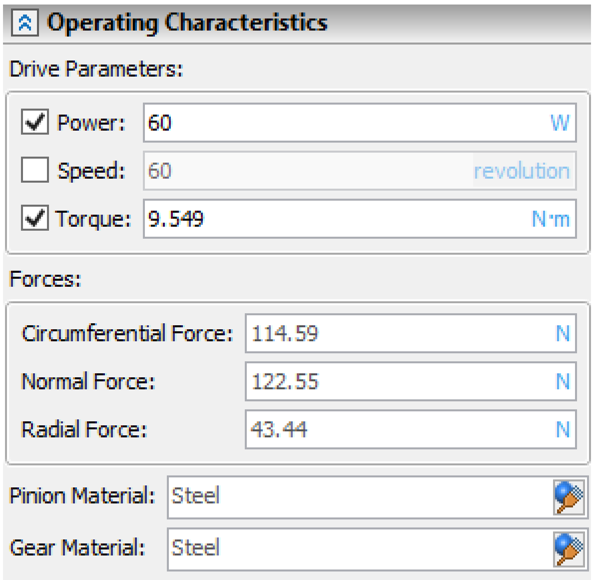
The number of teeth can be specified for the base tangent length. In this lesson, we are satisfied with the calculated value of this parameter. If you need to enter the values of these parameters manually, then you must activate the checkbox, enter the values and click the **Calculate** button. In this case, the window will be updated and the value of the base tangent length will change.
As with gearing analysis, you can create HTML and PDF geometric analysis reports here.
Click **OK**. This completes the geometric analysis.

Perform strength analysis

For such a calculation, it is necessary, first of all, to specify the parameters of the drive. Go to the **Operating Parameters** tab:

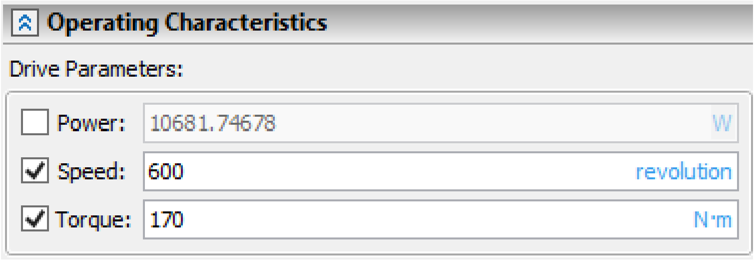


By condition, we know the speed and torque. Activate the **Torque** flag:

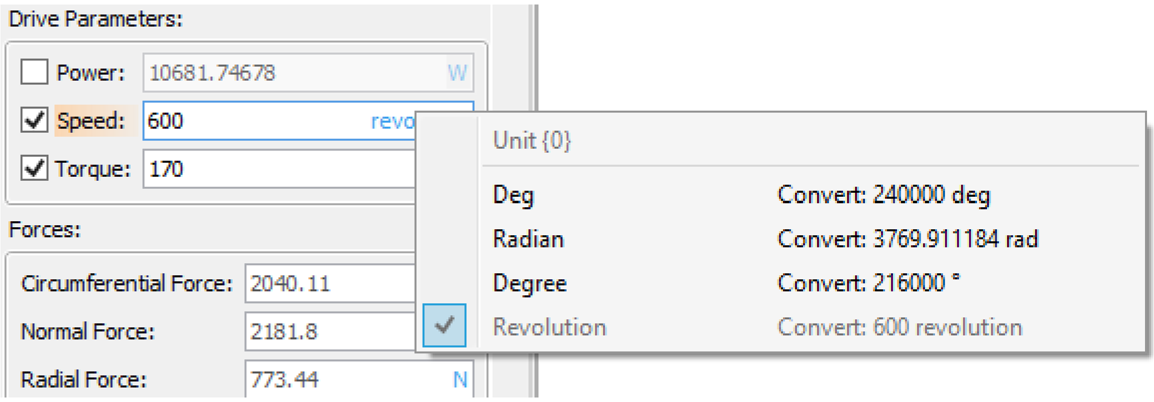


> At least two drive parameter values must be set in the **Gears** add-on.

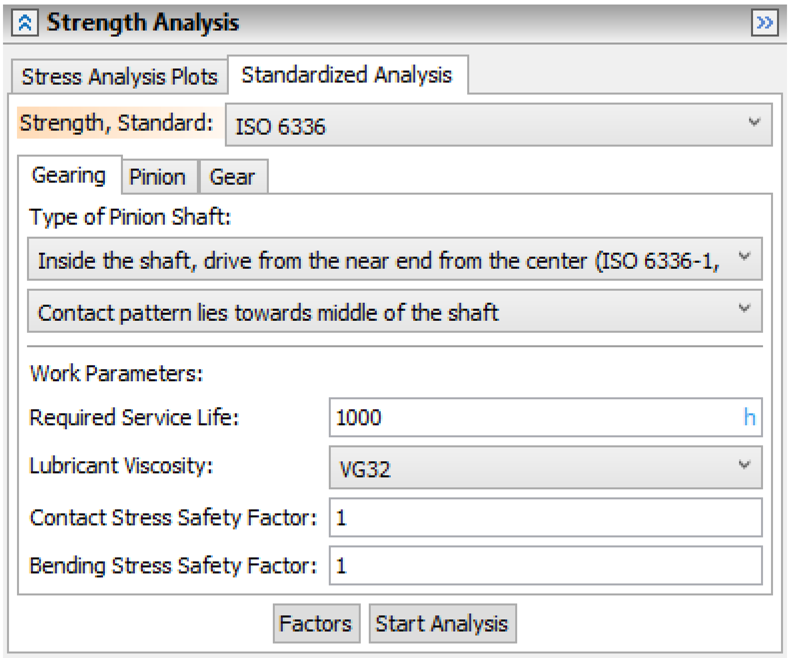
Now activate the **Speed** flag and set the number of revolutions equal to **600** and the value of the torque **170** N * m:



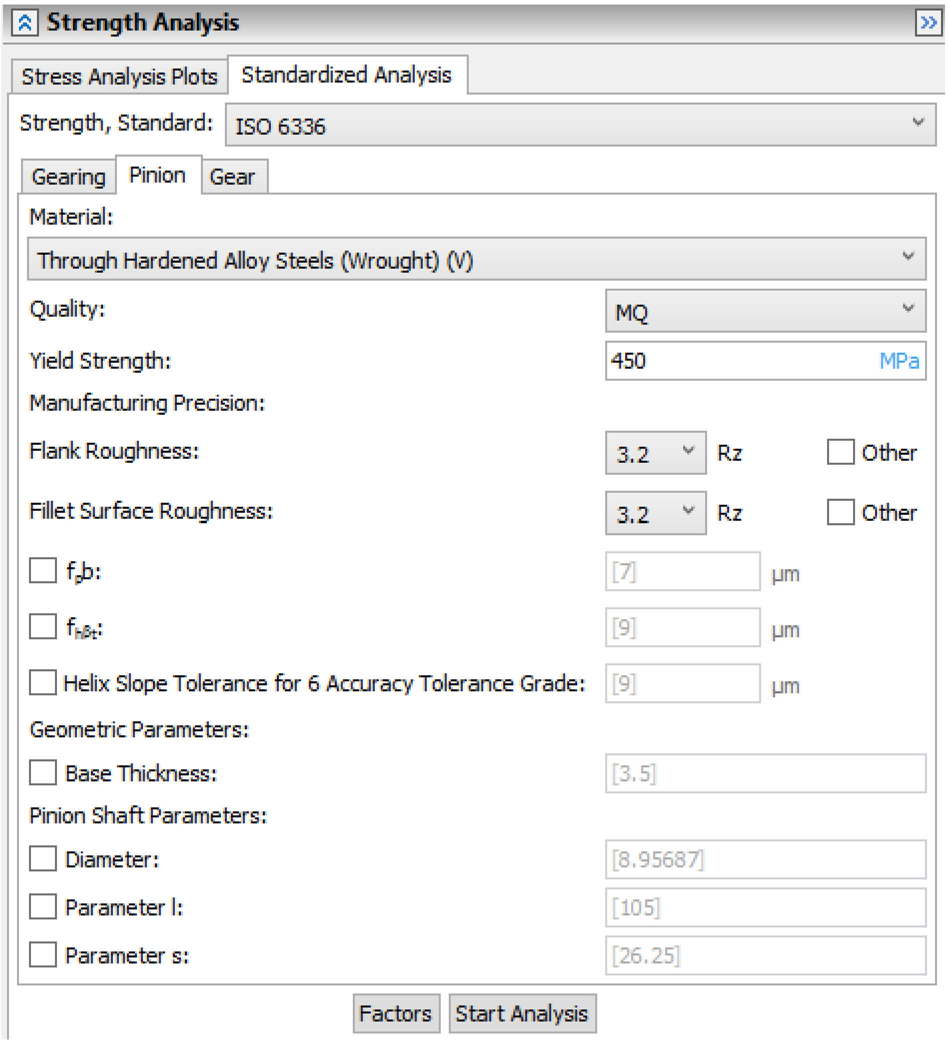
For almost every field, you can choose a system of units of measurement:



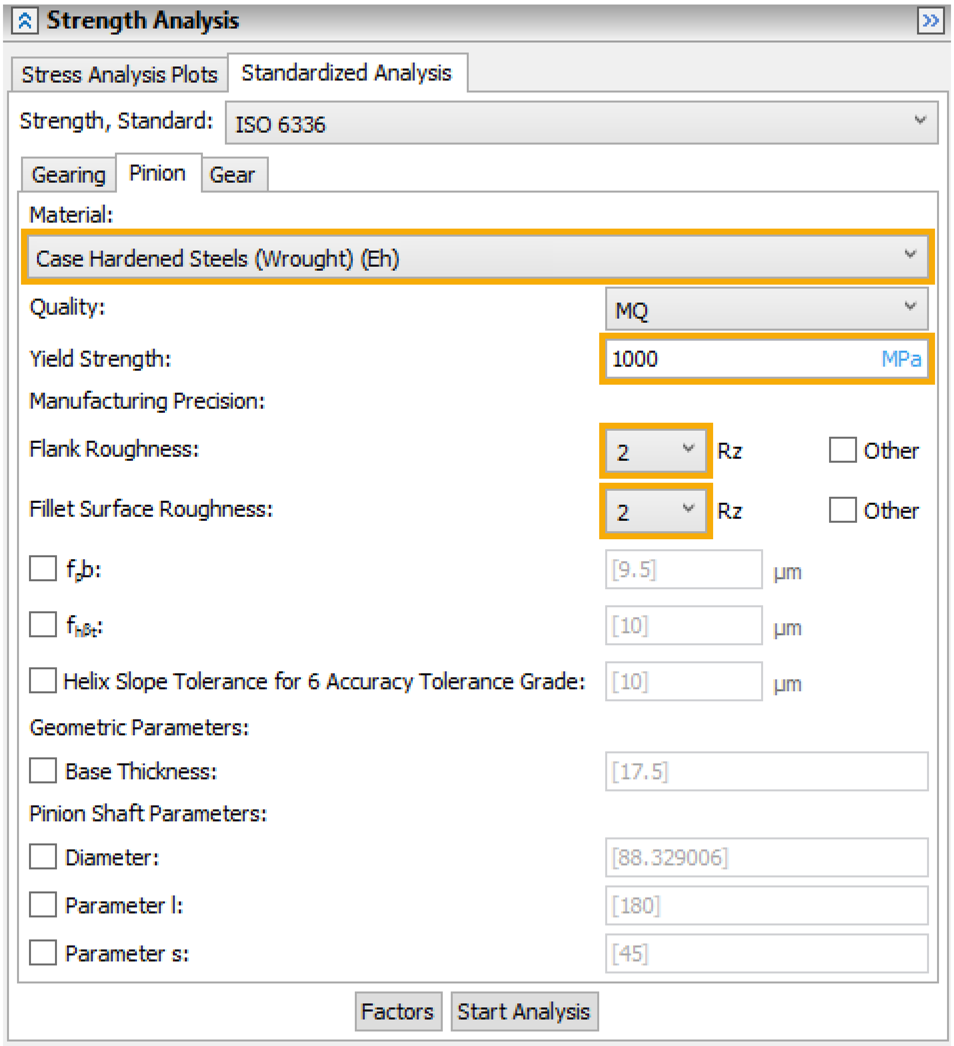
Note that power is automatically calculated from speed and torque.
This completes the assignment of operating characteristics.
Go to the **Strength Analysis** tab, the **Standardized Analysis** tab:



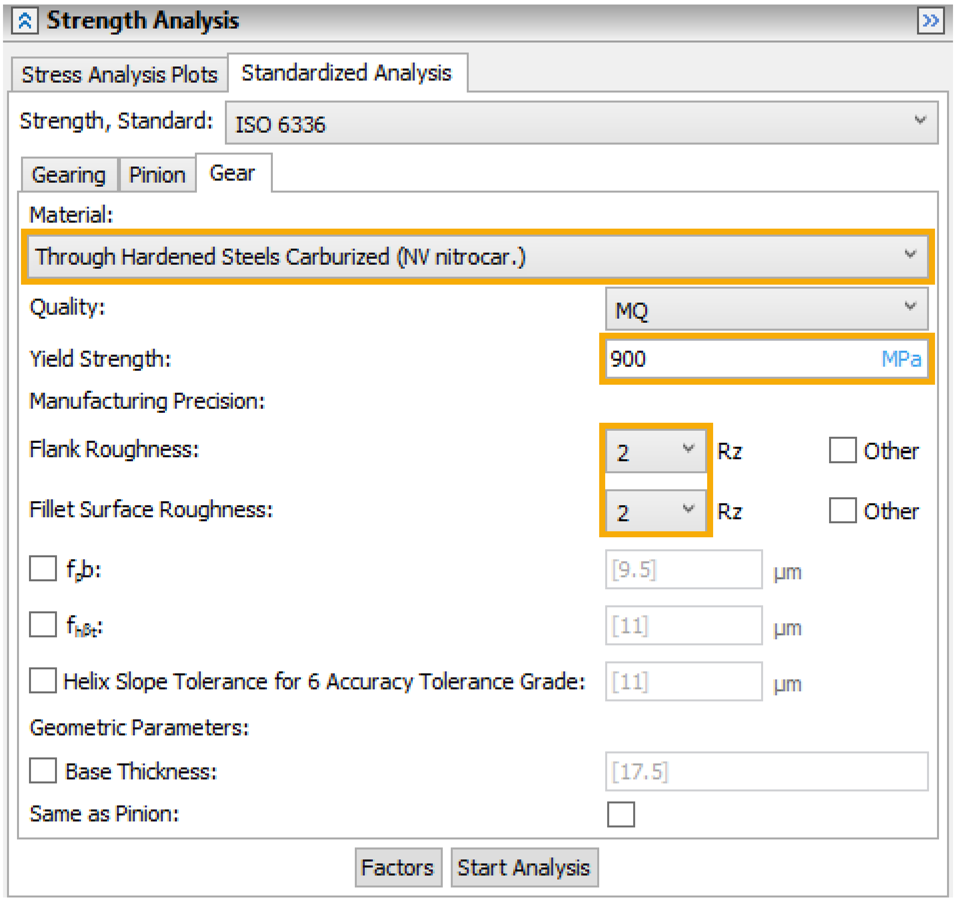
On this tab, you can select the standard by which you can perform the strength analysis and set the general strength parameters for the gearing.
According to the condition, the required service life is 1000 hours. It was the same as the default resource.
Go to the **Pinion** tab:



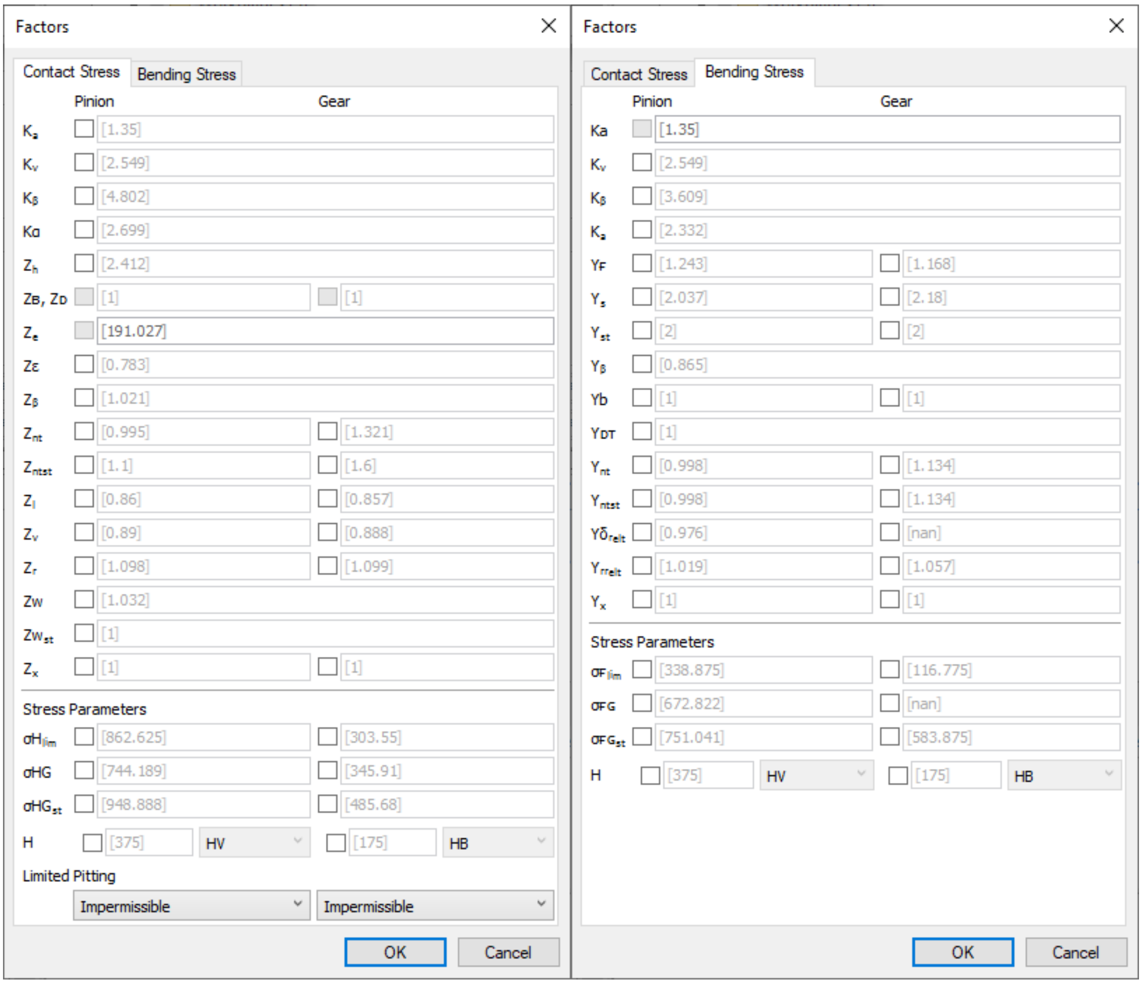
Here we can set the individual strength parameters for the pinion. Select the material type **Through Hardened Steels Carburized (NV nitrocar.)**, set the **Yield Strength** to **1000 MPa** and the **Flank Roughness** and **Fillet Surface Roughness** to **2 Rz**.



Here we can set the individual strength parameters for the gear. Select the material type **Normalized Low Carbon Steels (Cast) (St)**, set the **Yield Strength** to **900 MPa** and the **Flank Roughness** and **Fillet Surface Roughness** to **2 Rz**.

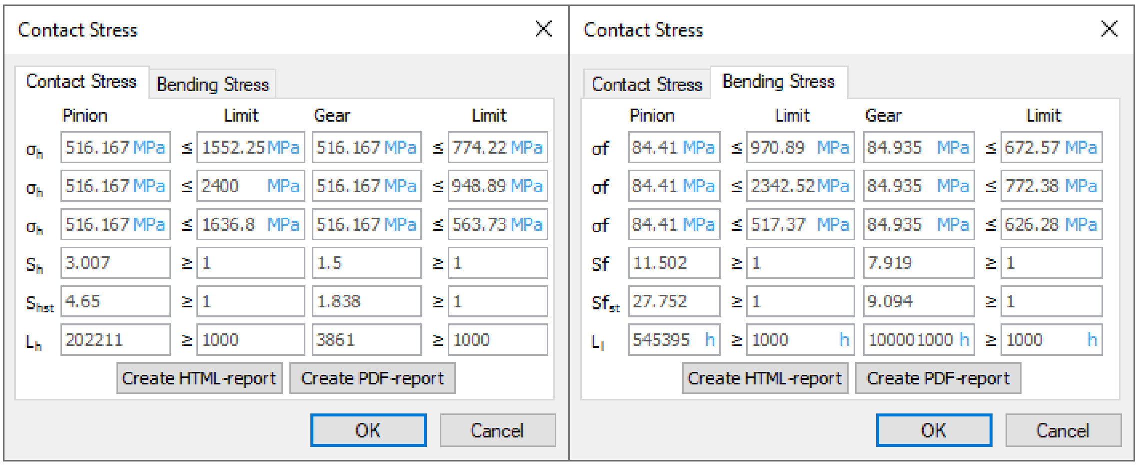


There is a separate setting for the factors for correcting contact and bending stresses:



Here you can see the values of some factors and, if necessary, correct them. In this lesson, we will not make additional corrections.

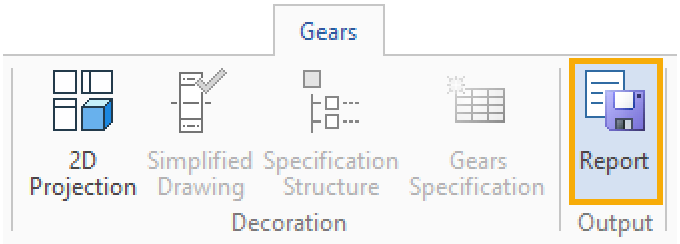
Click the **Start Analysis** button. In the window that appears, the parameters of the contact and bending stresses will appear.



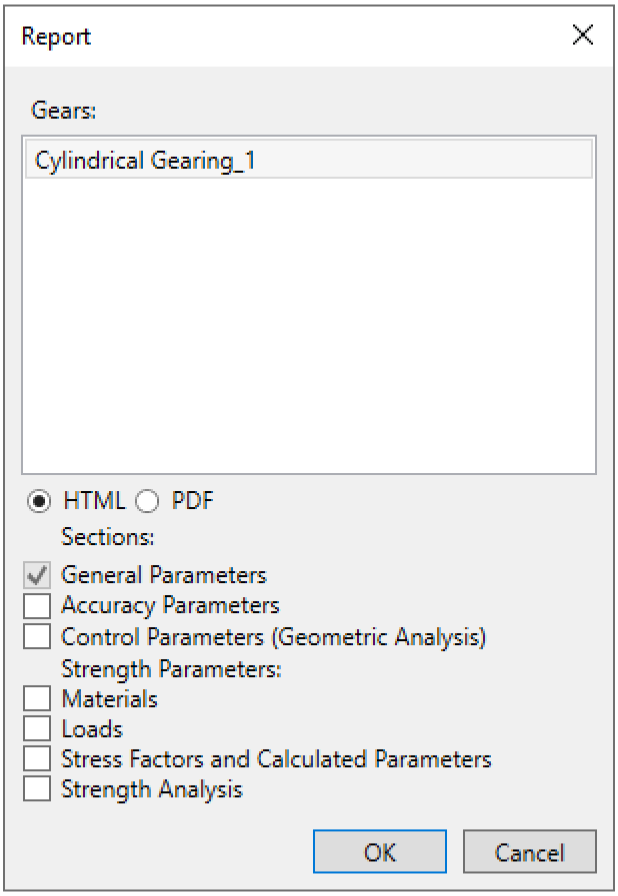
As can be seen from the result, all parameters are within their limits. This completes the strength analysis. Click  to save all changes.

Create report

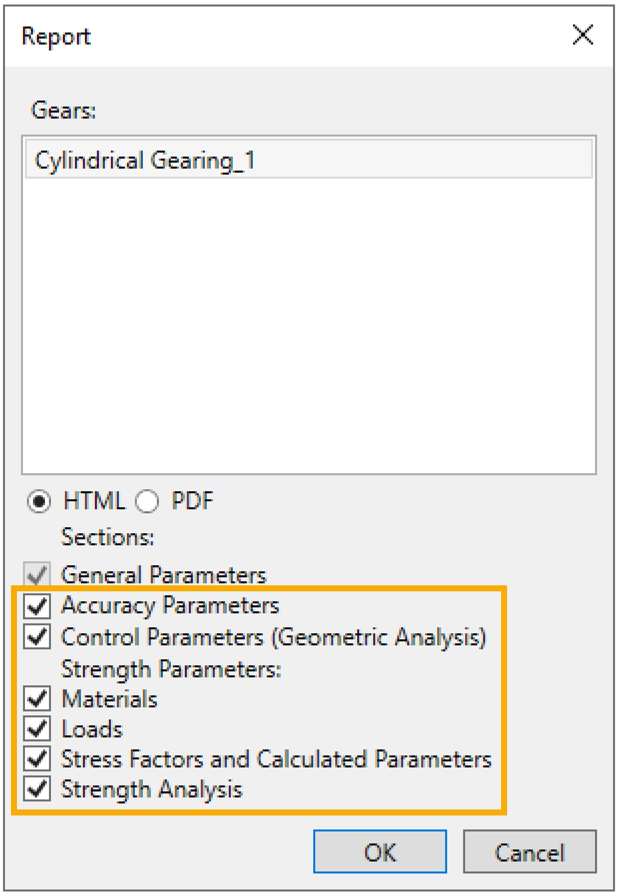
Now you can create a report. Run the appropriate command from the ribbon:



In the report window, you can select the calculation category that should be displayed in the report:



Select all categories.



Next, select your preferred format and click **OK**.
As a result, a detailed report will open, which can be saved, transferred to another workstation, or printed.

> The selected category flags will be automatically selected in subsequent calculations.

Congratulations!

You have created your first gearing in the T-FLEX CAD system.

1) General Parameters

1.1) Gearing Type

External
Twist direction: Right, Left

1.2) Source Contour

Source Contour: ISO 53

Standardized Parameters:

Pressure Angle at Normal Section, °	20
Radial Backlash Coefficient	0.25
Addendum Coefficient	1
Root Radius Coefficient	0.38
Dedendum Coefficient	1.25

1.3) Module

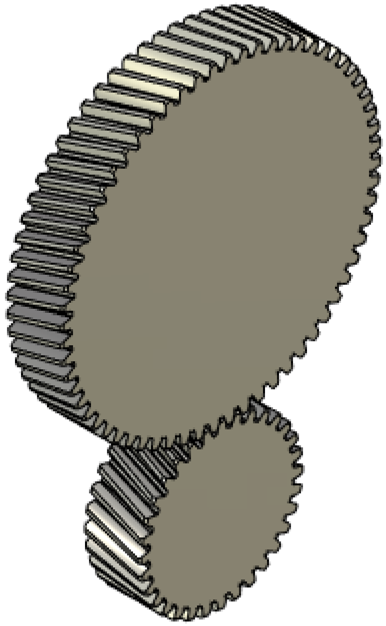
Module, Standard: ISO 54-96
Module, Value: 5

1.4) General Parameters

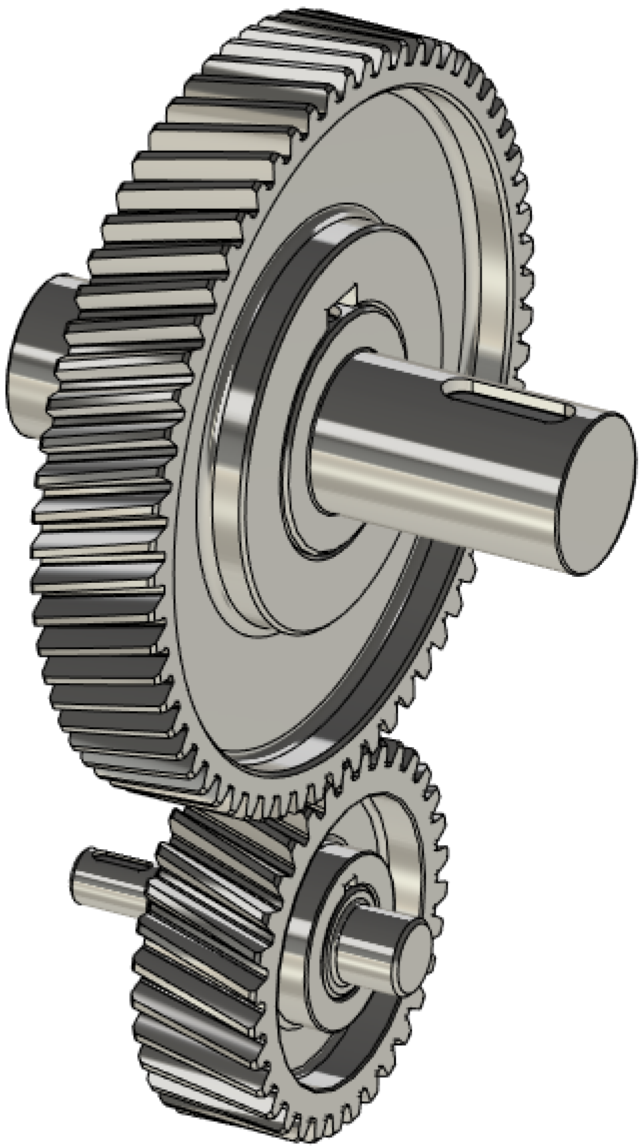
	Pinion	Gearing	Gear
Gear Ratio		2	
Helix Angle, °		16.25	
Center Distance, mm		249.99	
Teeth Number	32		64
Facewidth	60, mm		60, mm
Profile Shift Coefficient	0		0

1.5) Calculated Geometric Parameters

		Pinion	Gearing	Gear
Pitch diameter, mm	d	166.66		333.32
Base diameter, mm	db	155.83		311.67
Tip diameter, mm	d_a	176.66		343.32
Root diameter, mm	df	154.16		320.82
Working pitch diameter, mm	dw	166.67		333.33
Normal pitch tooth thickness, mm	s_n	7.85		7.85
Transverse pitch tooth thickness, mm	s_t	8.18		8.18
Normal pitch, mm	p_n		15.71	
Transverse pitch, mm	p_t		16.36	
Center distance, mm	αw		250	
Pressure angle at tranverse section, °	αwt		20.76	
Transverse contact ratio	εα		1.63	
Overlap ratio	εβ		1.07	
Total contact ratio	εγ		2.7	



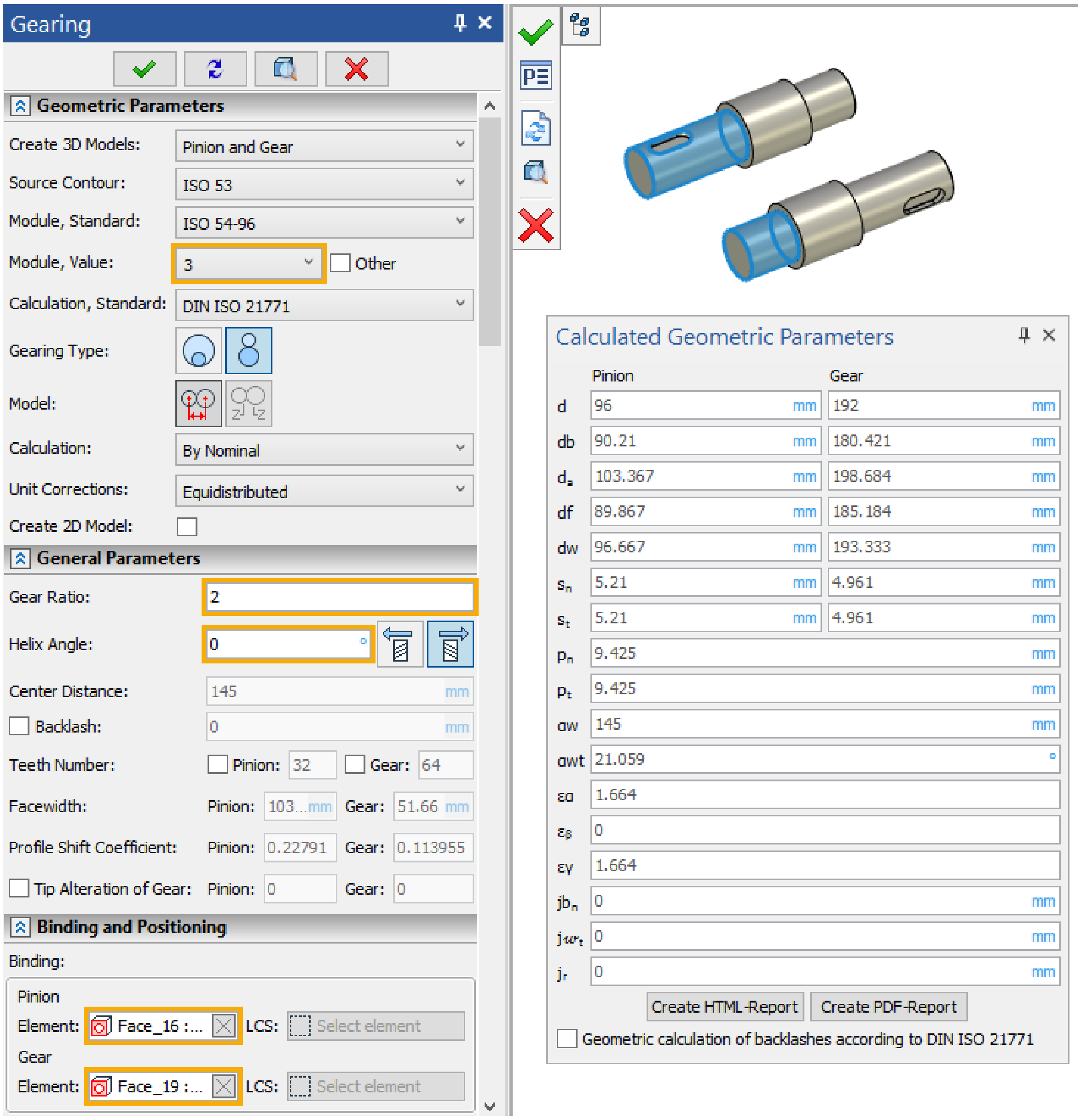
From each resulting gear, you can make a final part using T-FLEX CAD 3D and 2D modeling tools and continue with the design of the entire assembly.



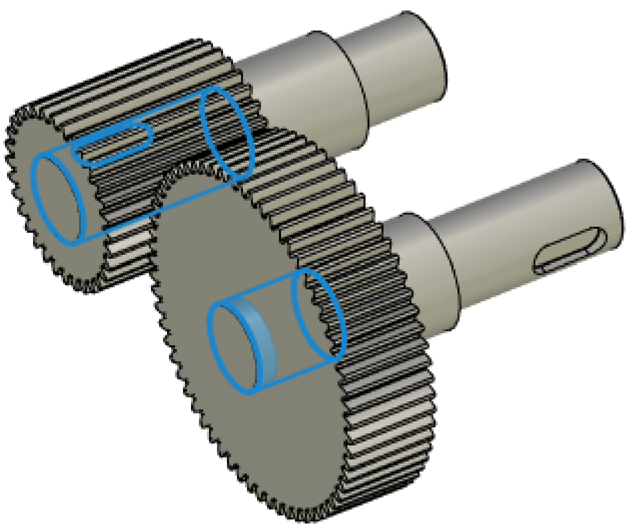
In addition to gearing, you can create separate gears, internal gears, bevel gears, single-row planetary mechanisms, and gear trains. The sequence of work discussed in this lesson is also suitable for working with the above mechanisms.
At the moment, it is possible to calculate only cylindrical and bevel gears, however, other types of gearing will be added to the module, such as hypoid, crown, screw and others.

Gearing Creation

To create a gearing, it is enough to select the module, enter the gear ratio, helix angle (or set zero if the gear is spur) in Gearing command, and also specify two cylindrical surfaces that define the shaft on which the gearing is placed.

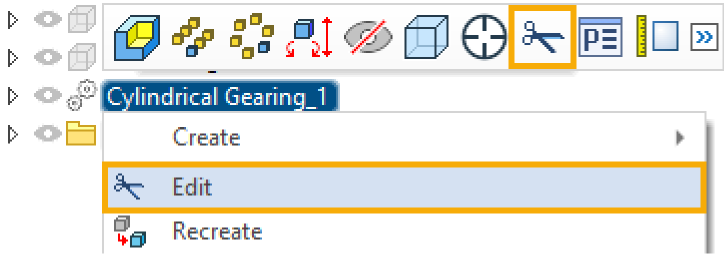


After clicking , the model of the gearing will be created, consisting of two fragments.



This is the easiest and fastest way to create a gearing. There are also other scenarios of work, which are determined by the selected model, the type of gearing, the method of calculating the profile shift

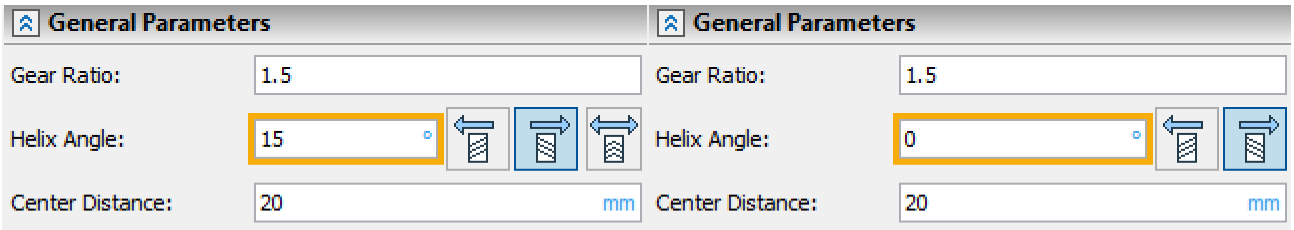
coefficient, the method of binding the gears and other options about which you can read in more detail in the corresponding section.
By default, the Geometric Parameters, General Parameters, and Binding and Positioning tabs are open. The parameters set on these tabs determine the minimum data for creating a gearing.
If the user knows in advance the standards with which he works, they can be set as default standards.
Any gearing can be changed, corrected, or additional calculation can be made for it using the available method. To do this, you need to call the gearing object for editing via the context menu.

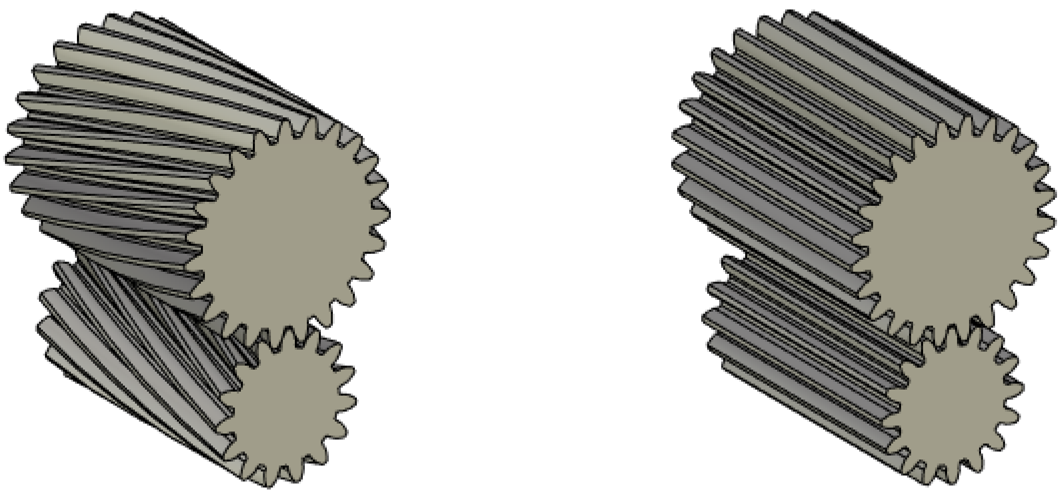


Selected gearing will be opened via the add-on dialog. There are limitations in changing the gear after its creation: you cannot change the type of gearing and change the double helical gear to normal and normal to double helical gear. Any changes are allowed before the gearing is created.
As a result of the add-on work, 3D models of the gear rim are always created. The necessary modifications to the geometry, such as creating a hub, grooves, etc., are performed by the user using T-FLEX CAD tools.

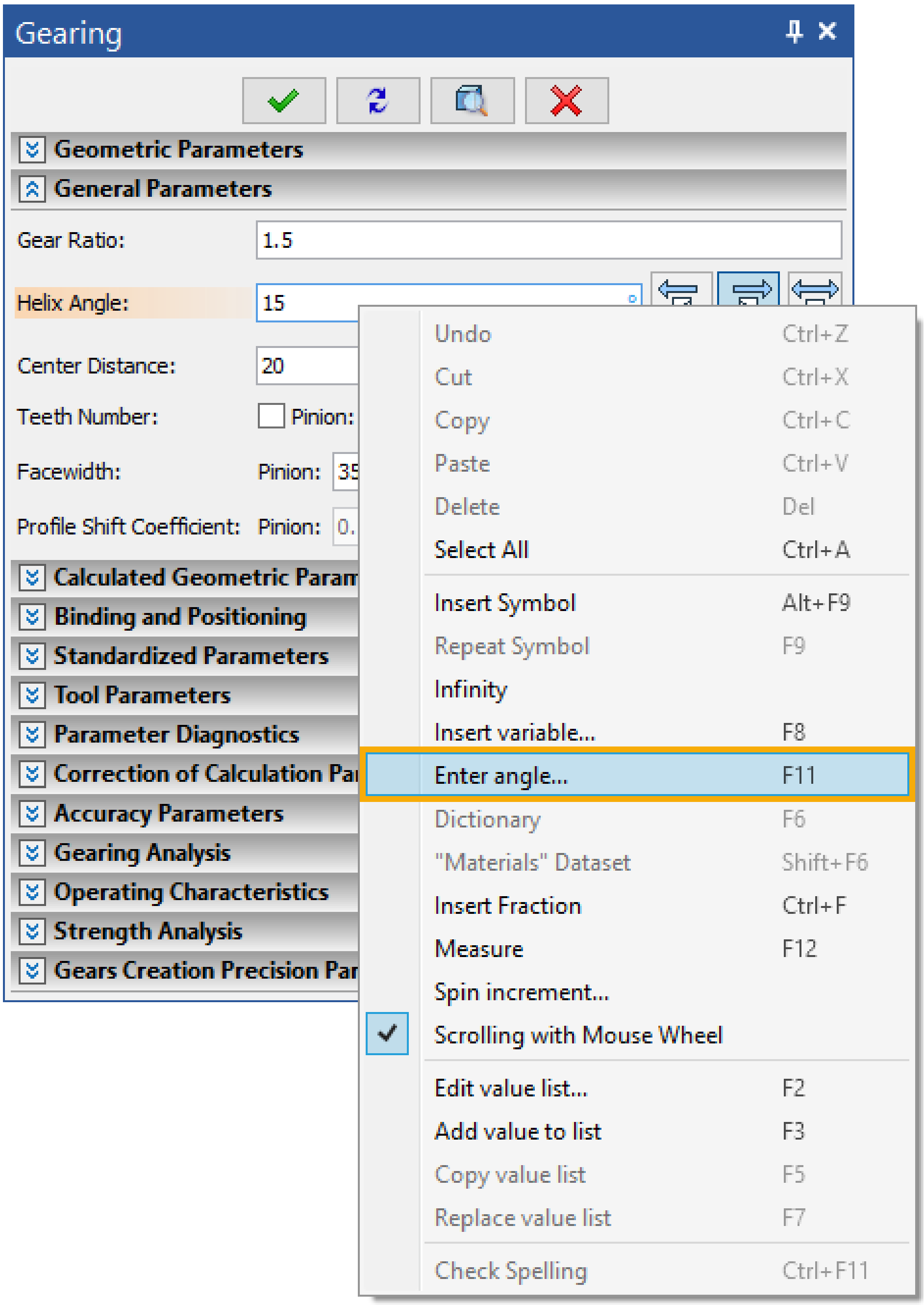
Spur and Helical Gears

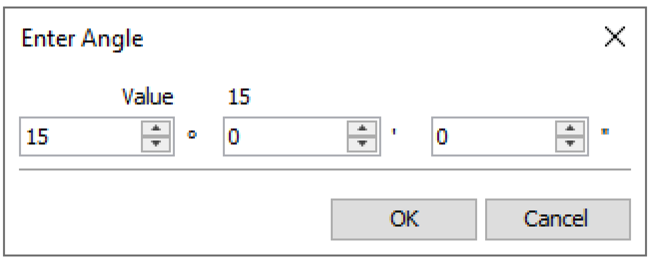
The order of creating a spur or helical gear differs only in the value of the helix angle. If the value is zero, the gear is spur gear, if it is nonzero, the gear is helical.





The angle can be entered in minutes and seconds through a special command of the context menu of the input field.





Negative helix angles are not allowed. To change the direction of the helix angle, use the special switches: left hand , and right hand . The helix angle is defined for the pinion. On the gear with external gearing, the helix angle will be opposite, with internal gearing, the helix angle on the gear will be like on a pinion.

Information about the helix angle on the gear and pinion is available in the report and in the gears specification.

In further calculations of the geometry and control dimensions, deviations or strength, it is not necessary to additionally indicate that the gearing is helical or spur: in the add-on, the calculation methods are automatically applied depending on the entered value of the helix angle.

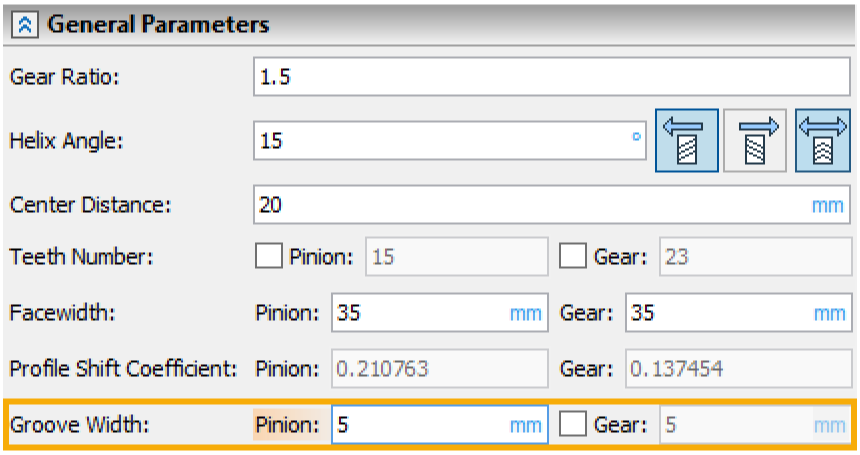
Double Helical Gear

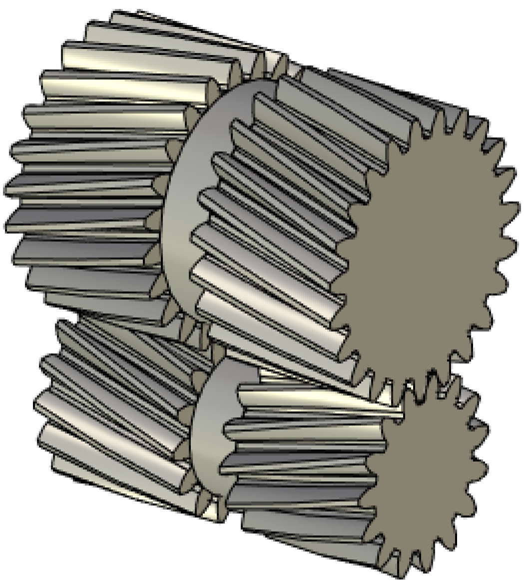
To create a double helical gearing, you need to activate the button. To cancel the creation of the double helical gearing, deactivate the button by pressing the button again. When creating a double helical gearing, the slope option "Left Hand - Right Hand" or "Right Hand - Left Hand" is determined by the left slope and right slope switches.

If the left slope is selected, the double helical gearing will be "Left Hand - Right Hand", if the right slope is selected, then vice versa. On the gear with external gearing, the slope of the double helical gearing will be opposite, with internal gearing, the slope of the double helical gearing on the gear will be like on a pinion.

Information about the helix angle direction on the gear and pinion is available in the report and in the gears specification.

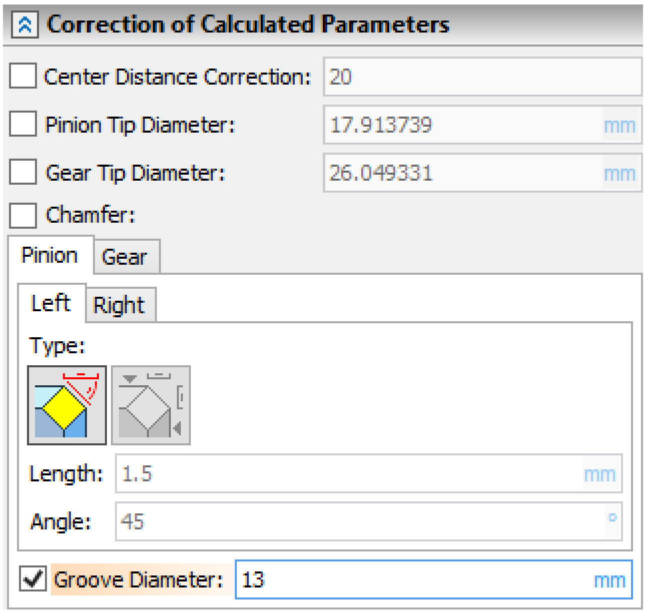
When the double helical gear button is activated, the input fields for the groove width become available.





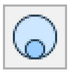
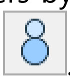
By default, the groove width on the gear is the same as the groove width on the pinion. You can enter a different value for the gear, for this you need to set the flag next to the field.

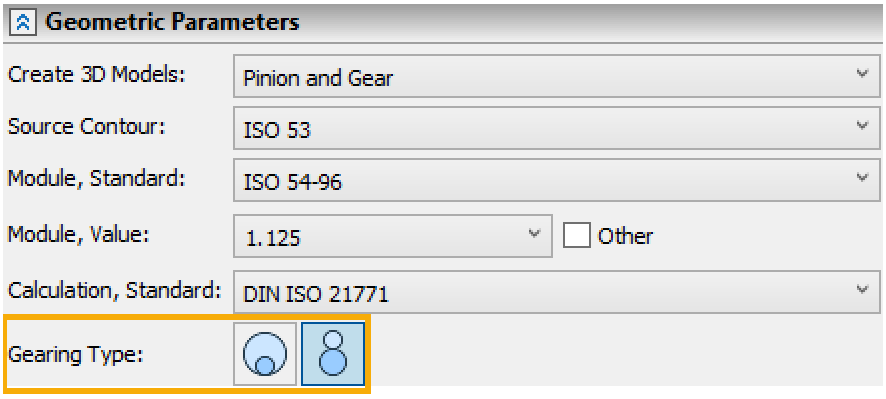
The groove diameter is automatically calculated to be slightly less than the tooth root diameter. The value of the root diameter can be changed on the Calculated Parameters Correction tab: for a gear and a wheel on the corresponding tabs.

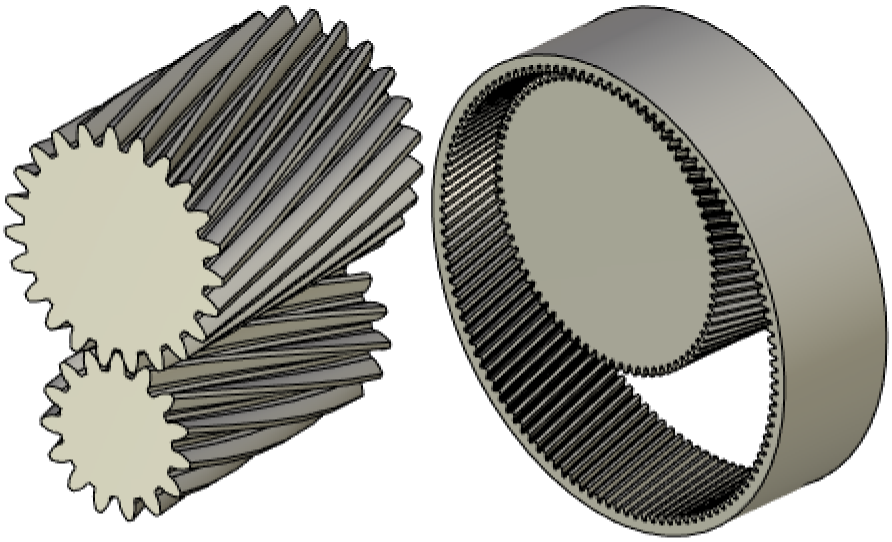


In further calculations of the geometry and control dimensions, deviations or strength, it is not necessary to additionally indicate that the gearing is double helical: in the add-on, the calculation methods are automatically applied depending on the activation of the double helical gear button.

External and Internal Gearing

The procedure for creating an external or internal gearing differs by the choice of a switch in the **Gearing Type** option: internal gearing  and external gearing .





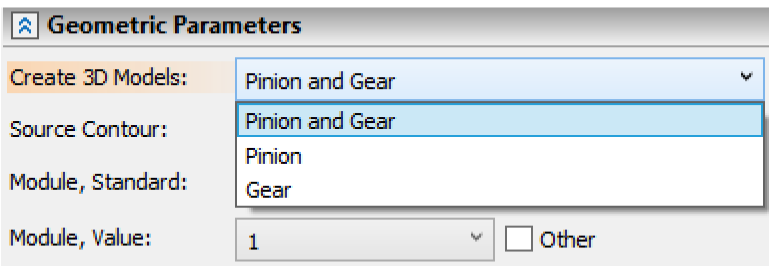
In further calculations of the [geometry and control dimensions](), [deviations]() or [strength](), it is not necessary to additionally indicate that the gearing is internal or external: in the add-on, the calculation methods are automatically applied depending on the entered gearing type on the **Geometric Parameters** tab.

For a gear with internal teeth, the outer diameter is automatically calculated. This size is not regulated by anything, it is relative and cannot be changed through the add-on dialog. The add-on creates a pinion rim, as for the external gearing, and the user carries out all other geometry [modifications]() using T-FLEX CAD tools.

Separate Gear

Separate gears can be created in the add-on. In this case, you can create a gear for which the parameters of the mating gear are known, or you can create a gear as a separate model that is not associated with another gear.

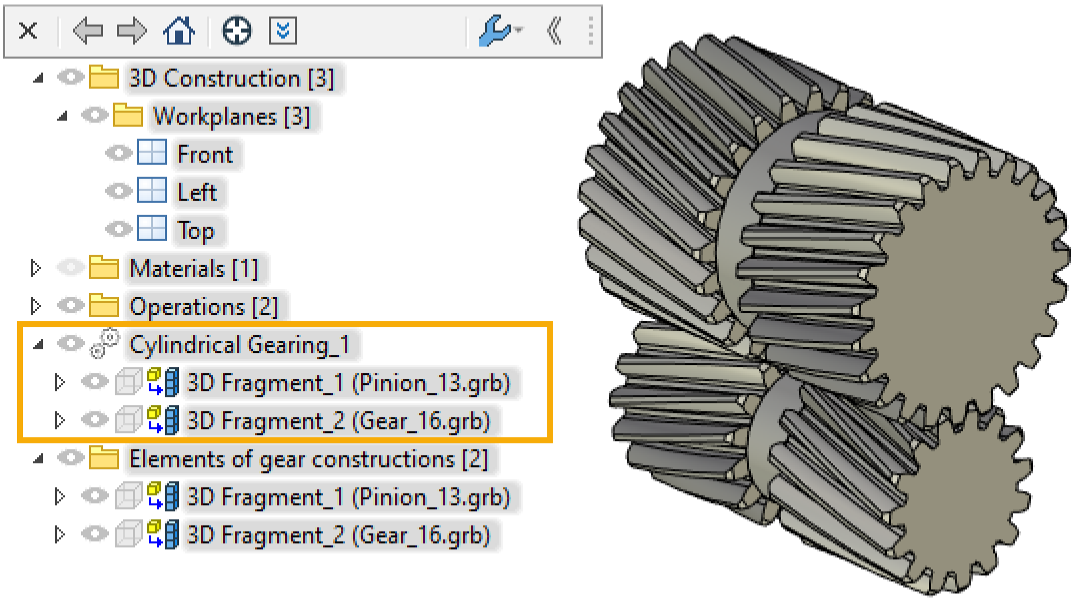
If you need to create a 3D model of a pinion or gear when all the parameters of the gearing are known, then you need to use the [Gearing]() command and select the **Pinion** or **Gear** option in the **Create 3D Models** list.



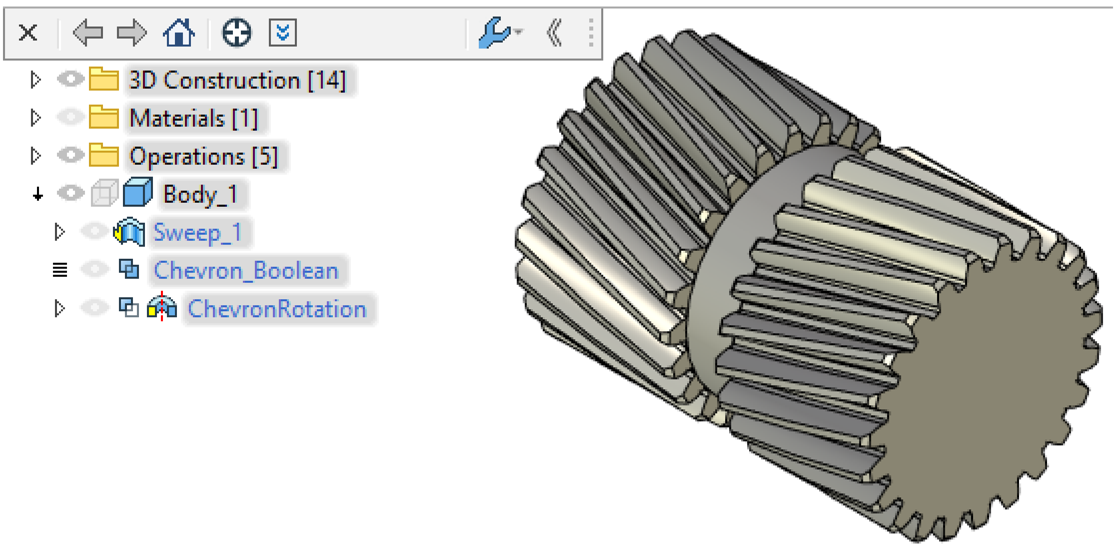
With this method of creating a gear, working with the add-on is completely analogous to working when creating a gearing, it will simply create only one specified model of the two.
If the parameters of the gearing and the mating gear are not known and are not needed, then you can use the Gear command if you need a gear with external teeth, or Internal Gear command if you need a gear with internal teeth. In this case, strength analysis or geometric analysis will not be possible.

Geometry Modification

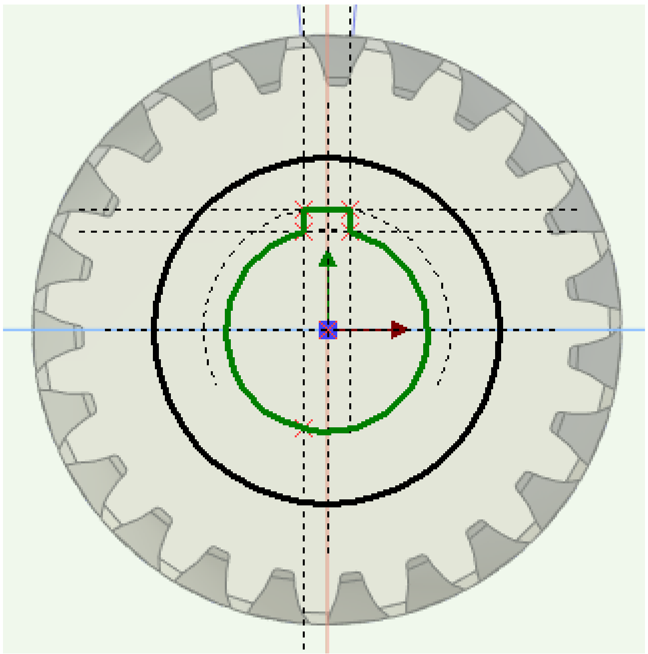
As a result of working with the add-on, 3D models (fragments) of the gear rim are created. For example, the toothed rims of the external double helical gearing.



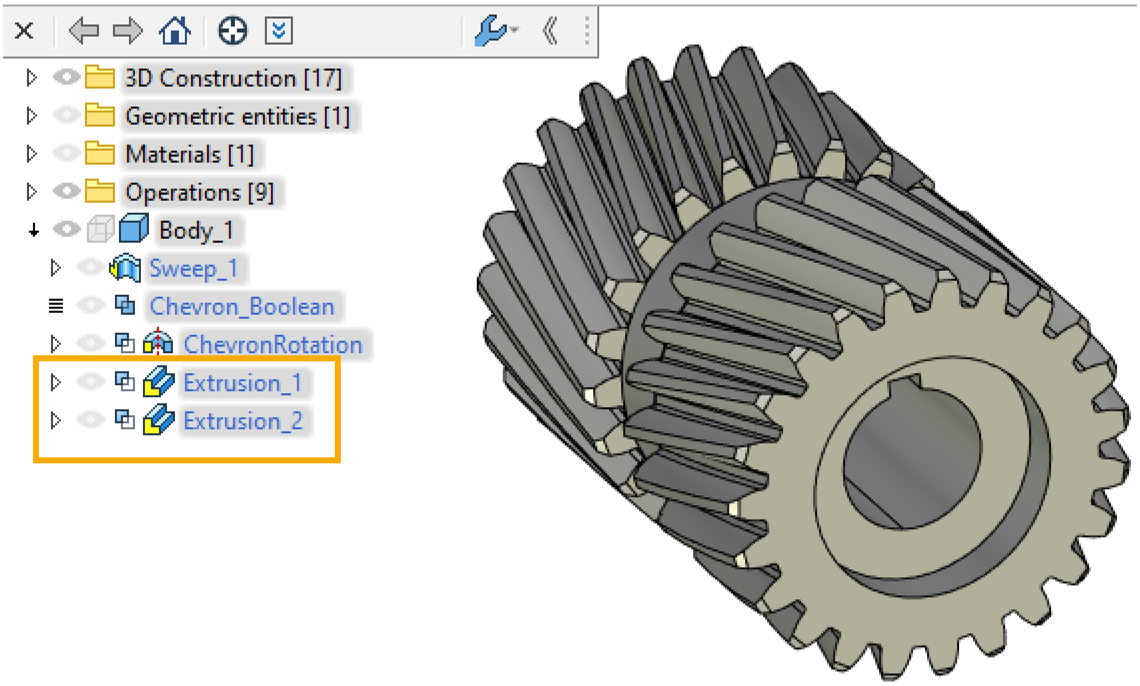
In order to make a model of a real gear from a crown model, the user needs to modify the model using T-FLEX CAD. In this case, all the improvements made will be saved when editing gears through the add-on dialog. For example, let's create a hub on a gear. To do this, you need to open the gear fragment, for example, through the context menu of the fragment. The fragment already contains the body of the double helical gear rim created by the add-on.



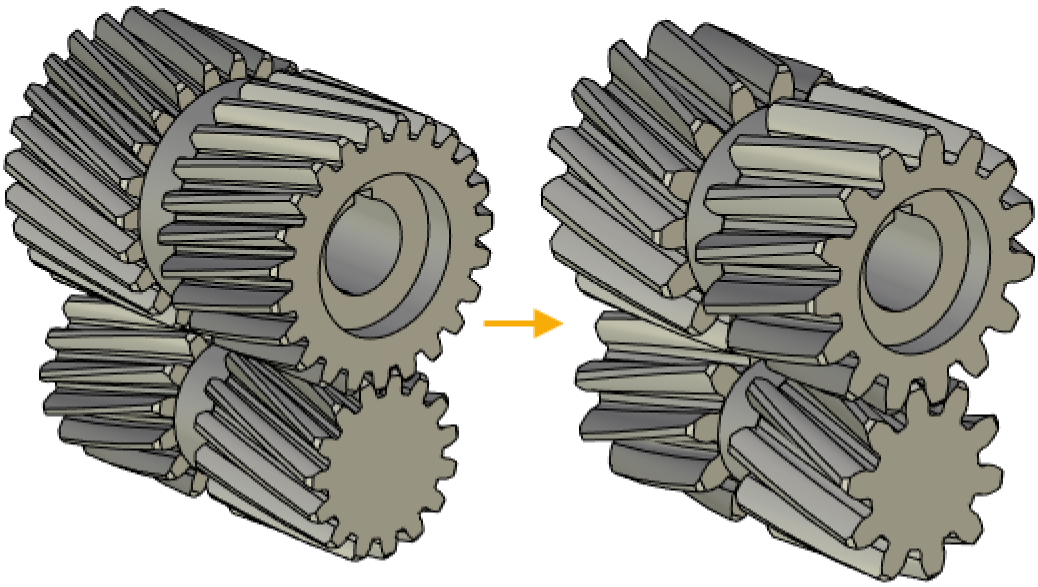
Create profiles on the body of the gear for the subsequent creation of the hub.



Using 3D modeling operations, we will create a hub with a groove.



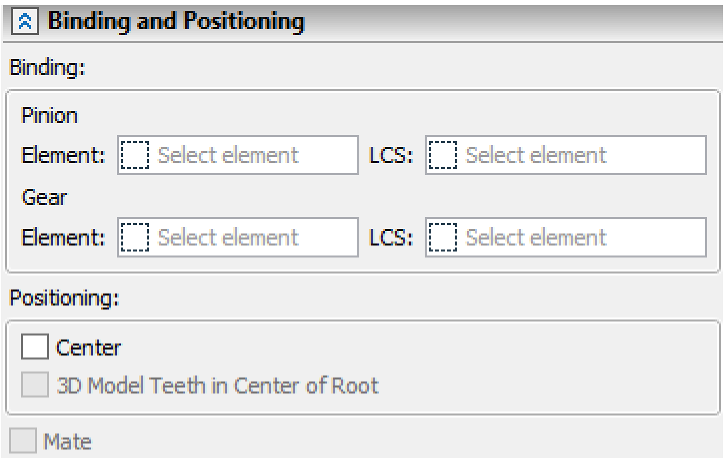
Open the source document and use the module to change the gearing parameters.



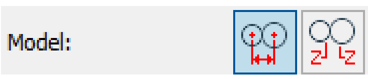
The made hub remained unchanged, while there were fewer teeth on the gear. At the same time, the gears remained in gearing.

Binding and Mates

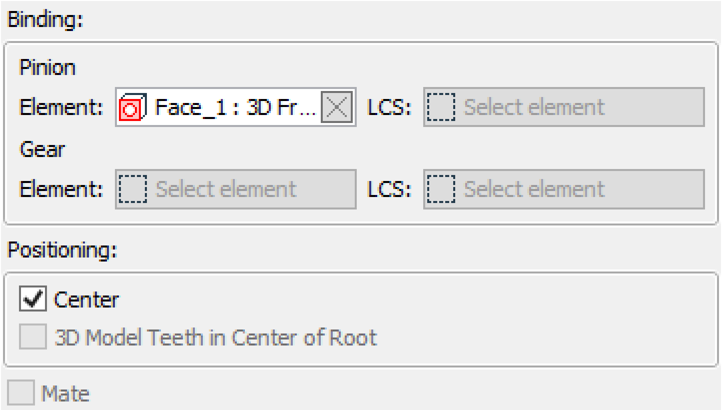
To position the gearing, use the **Binding and Positioning** tab.



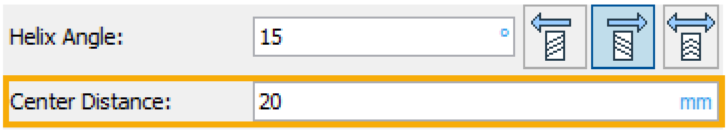
If the model by center distance is selected, then binding will be available for both the pinion and the gear.



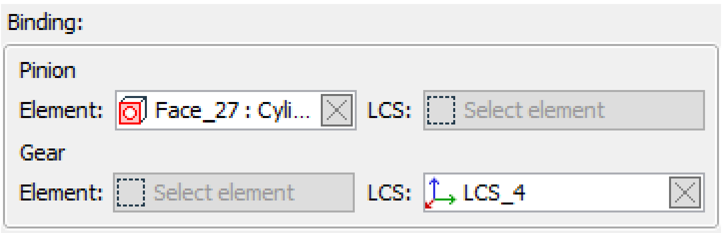
If the model by teeth number is selected, then the binding will be available only for the pinion or only for the gear. As soon as one of the binding options is selected, all the others will be closed.

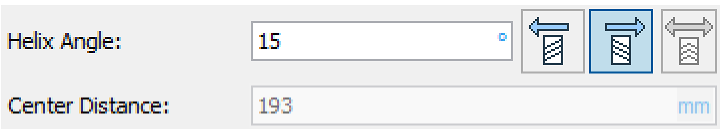


This is due to the fact that for the calculation model by teeth number, the center distance is calculated. When calculating by the model with a given center distance, if the binding is selected only for a pinion or only for a gear, the center distance is determined by the value in the **Center Distance** field.



If snapping is selected for both the pinion and the gear at the same time, the center distance will be calculated as the distance between the axes of the selected elements for binding, or as the distance between the axis of the selected element and the Y-axis of the snap LCS, or as the distance between the Y-axes of the snap LCS.





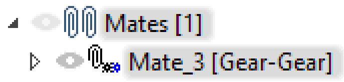
When selecting elements for binding, you must observe the rule that the axes of the selected elements must be parallel. If an LCS is selected, similar requirements apply to the Y-axis of this LCS. The Y-axis of the LCS coincides with the axis of the gear. If the specified rule is not followed, the system will issue a warning and the element for binding will not be selected.

To select a binding element, select the corresponding field, then select the binding element in the 3D scene. When the **LCS** field is selected, only LCS can be selected in the 3D scene. When you select the **Element** box, you can select a cylindrical face or a circular face in the 3D scene. The circular face will correspond to the end of the shaft from which the gear should fall. Cylindrical face will define the axis of the shaft and the facewidth of the gear. If a cylindrical face is selected, the gear facewidth input field is closed and is automatically calculated from the length of the cylindrical face along the axis.

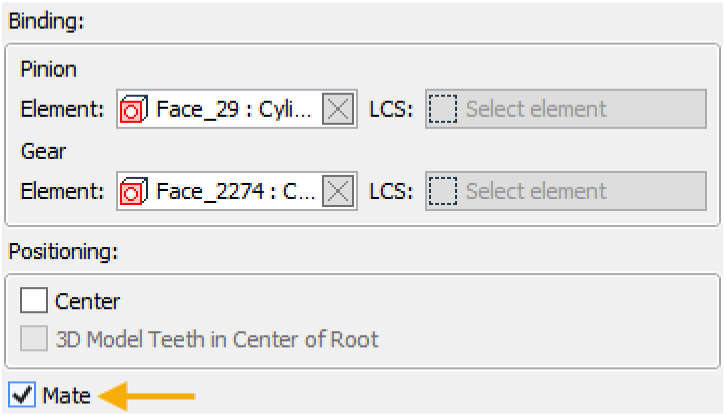
The **Center** option allows you to align the pinion with the gear.

The description of the **3D Model Teeth in Center of Root** option is given in the corresponding section.

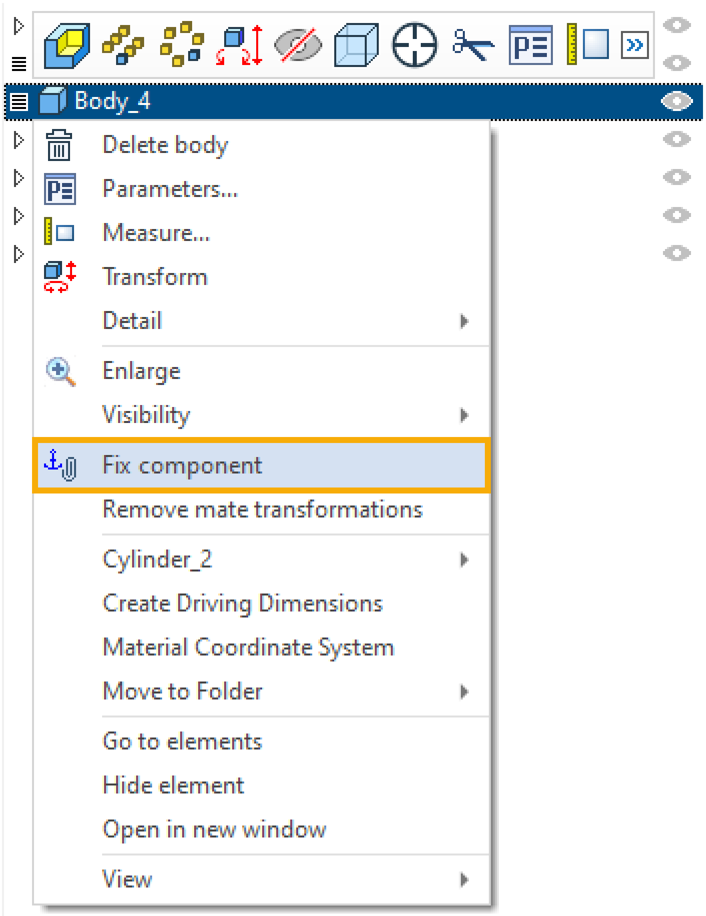
The **Mate** option allows you to automatically mate **Gear - Gear**. In this case, the ratio of angular velocities is automatically determined from the gear ratio.



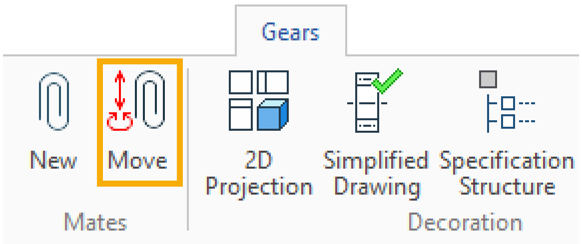
The option is available if both the pinion and the gear are attached to the selected objects, therefore, for the calculation option using the **Teeth Number** model, the **Mate** option will not be available.



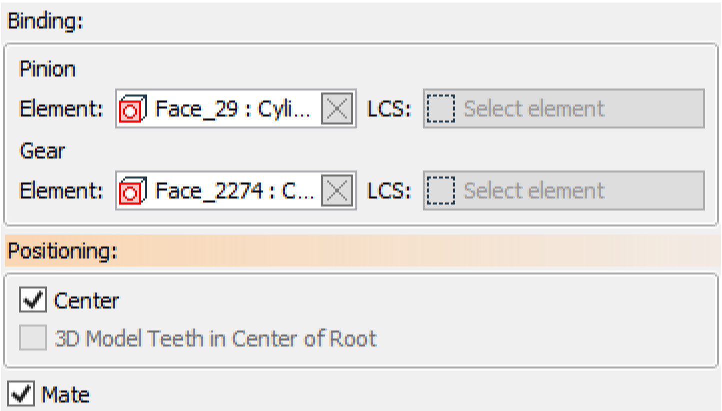
If the **Element** or the LCS created on the basis of the body is selected as the binding object, it is necessary, after creating the gearing, to fix the body to which the selected element belongs, or the body to which the LCS is anchored. Otherwise, the **Move** command will not work correctly.
To fix the body, you can use the command in the body context menu.

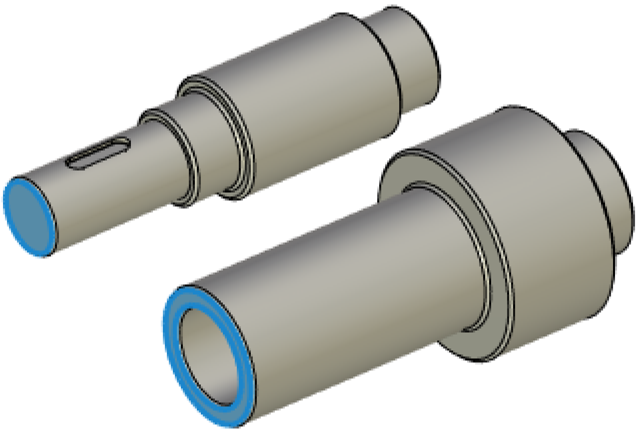


Further, to visualize the gearing, you can use the **Move** command.

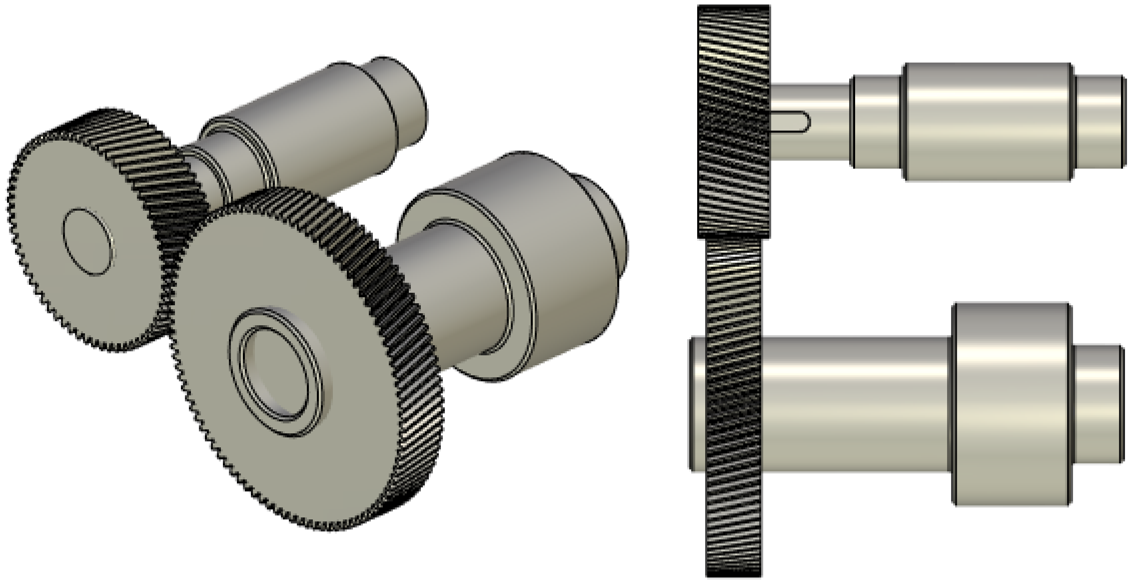


One of the typical options for work may be the following scenario. We set all the geometric and basic gearing parameters. We select the ends of the shaft for binding, specify the centering and creation of mating.

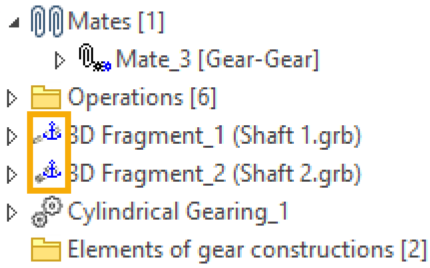




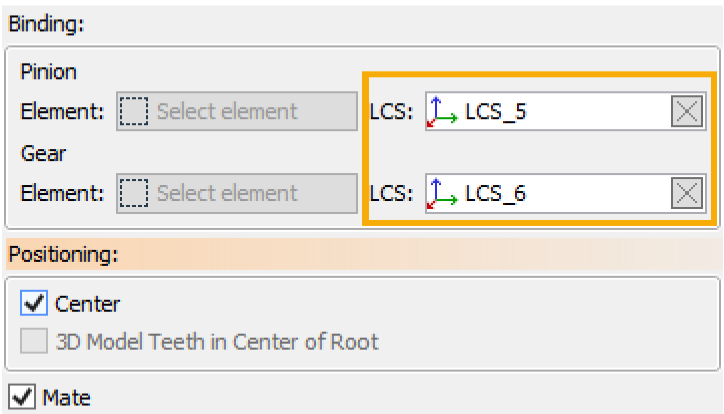
After completing the gearing creation, we obtain a 3D model of the gears on the shafts..



Next, we fix the shafts.



Now you can rotate the gearing using the **Move** command, simulating its operation.
When editing a gearing, it is important to pay attention to the fact that the binding of gears is created through the LCS, the position of which is determined by the selected end faces of the shafts.



If it is necessary to edit the binding, the created LCS can be removed from the binding field using ☒, and the binding can be set again.

Control Dimensions Calculation

Generally accepted inspection dimensions and their tolerances can be calculated on the **Gearing Analysis** tab - Standardized Analysis tab and Accuracy Parameters tab. The generally accepted dimensions that are indicated on the drawings and in the documentation include the center distance, the size along the base tangent length, the size along the rollers (balls) and the thickness of the tooth along a constant chord.

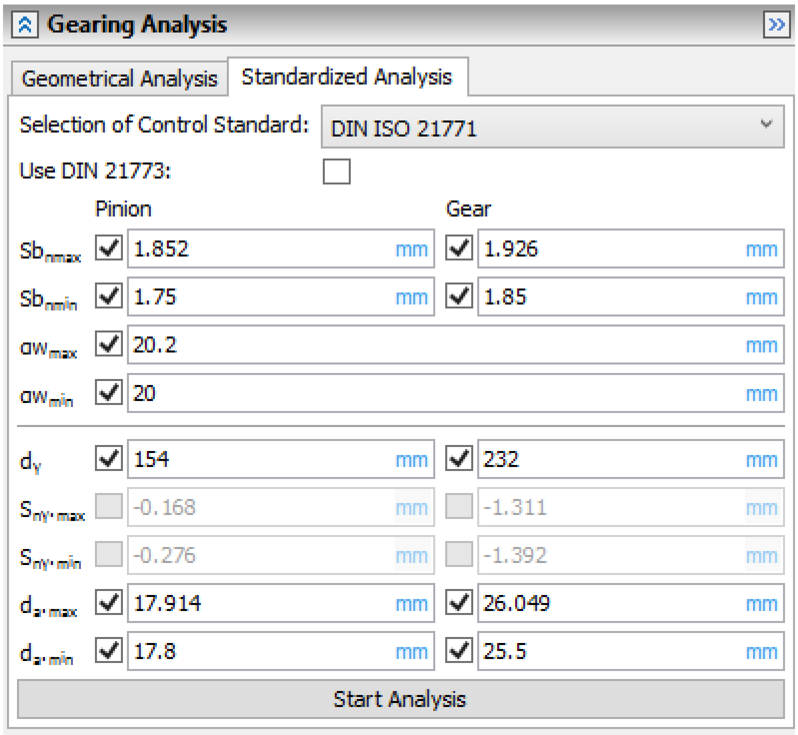
Depending on the standard, the method for calculating the tolerances for the dimensions indicated differs. So, for example, to calculate dimensions and tolerances in accordance with GOST, you will need to use the tab for standardized geometry analysis, where the nominal values of these dimensions will be calculated, and then calculate the accuracy parameters - where the deviations will be calculated. If you do the calculation according to ISO, then the nominal dimensions and deviations will be calculated on the standardized geometry analysis tab.

Topics in this section:

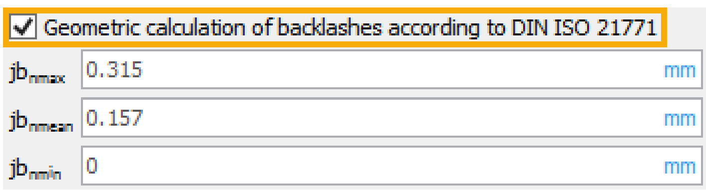
- Control Dimensions According to DIN ISO
- Control Dimensions According to DIN
- Control Dimensions According to GOST

Control Dimensions According to DIN ISO

On the **Gearing Analysis** tab, on the Standardized Analysis tab, you must specify from the **DIN ISO 21771** list and enter the deviations in tooth width, center distance, tip diameter. You also need to indicate the diameter on which the thickness of the tooth will be measured.

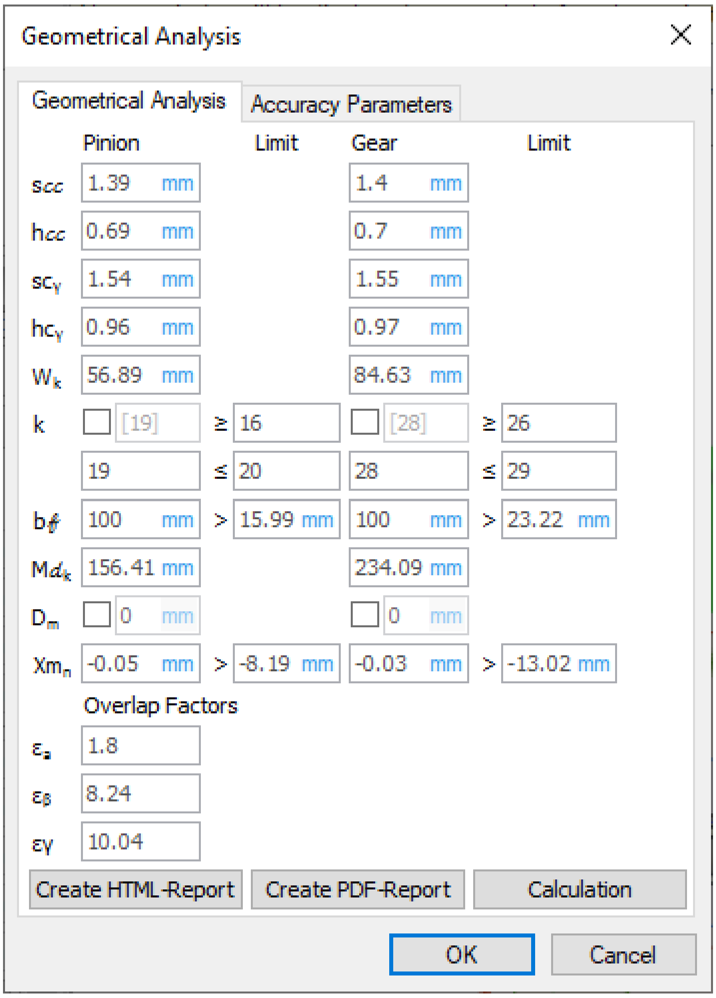


When calculated according to DIN ISO 21771, the calculated backlashes allow the user to understand the extent to which the entered deviations influence the theoretical maximum and minimum possible backlashes. The backlashes should not be negative, if the backlash is negative - the field will be highlighted in red. Backlashes values are available on the Calculated Geometric Parameters tab when the option **Geometric calculation of backlashes according to DIN ISO 21771** is checked.

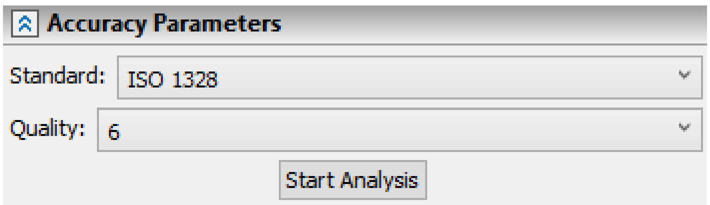


At the same time, in the analysis results, the values of the backlashes will slightly differ from those calculated by the geometric parameters of the thickness of the teeth, since deviations according to ISO 1328 will also be taken into account.

The button **Start Analysis** allows you to calculate the nominal values of the control dimensions, their deviations, and also calculate the backlases [taking into account ISO 1328](#).
There are 2 tabs in the modal window: **Geometric Analysis** and **Accuracy Parameters**. On the **Geometric Analysis** tab, the nominal values of the control dimensions are given.

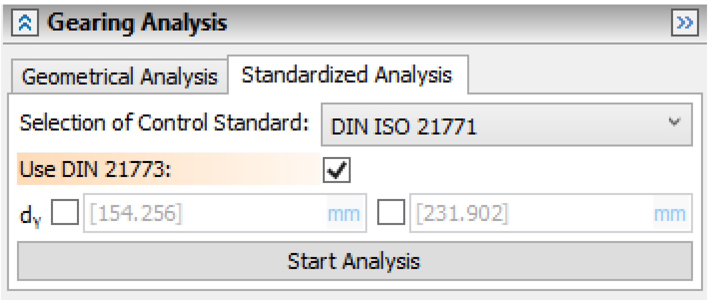


For the number of teeth in the length of the base tangent length and for the thickness of the gears, limits are given that limit the possibility of measurement.
On the **Accuracy Parameters** tab, deviations of control dimensions and backlashes are calculated taking into account ISO 1328.
To calculate according to **ISO 1328**, it is sufficient to indicate the quality.



Control Dimensions According to DIN

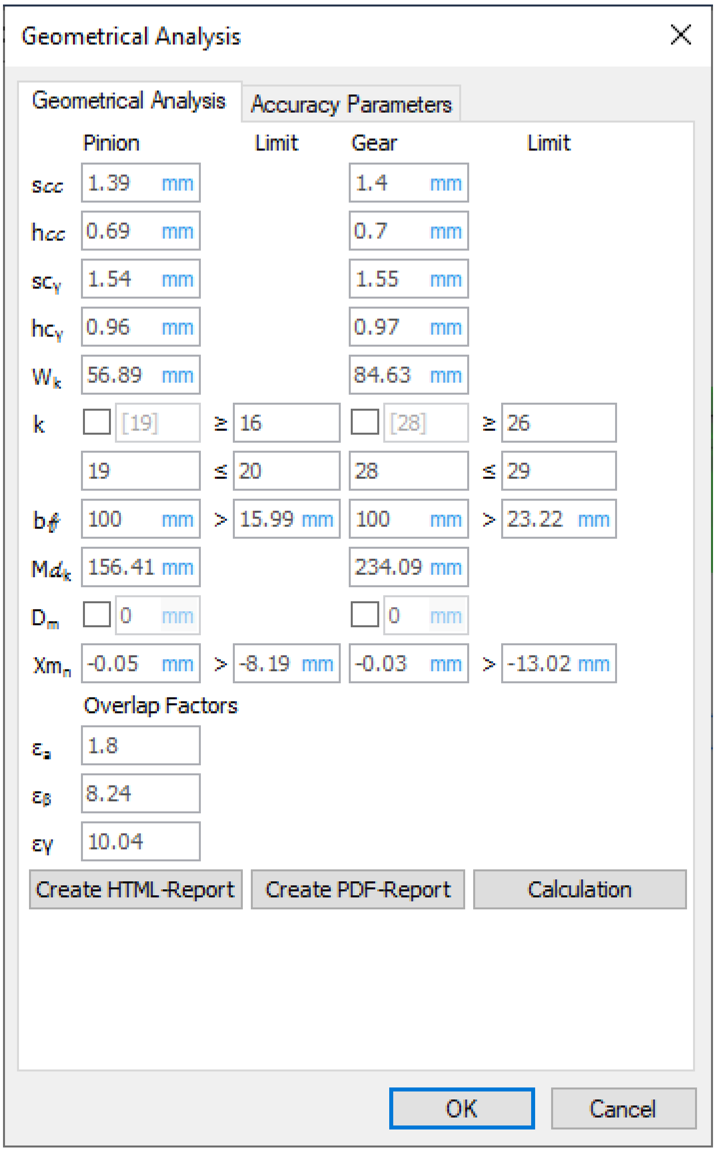
Control dimensions and control tolerances are regulated by DIN 21773, which in turn is a supplement to DIN ISO 21771. Therefore, to calculate control dimensions according to DIN, select **DIN ISO 21771** in the **Gearing Analysis** tab, [Standardized Analysis](#) tab and check the check box **Use DIN 21773**.



It is necessary to enter the diameter of the measurement of the thickness of the tooth (Y-cylinder diameter) or leave the default values.

The button **Start Analysis** allows you to calculate the nominal values of control dimensions, their deviations, as well as backlashes.

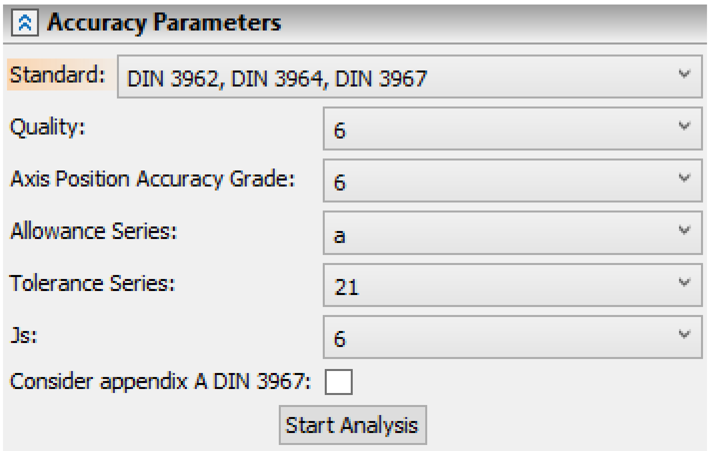
There are 2 tabs in the modal window: **Geometric Analysis** and **Accuracy Parameters**. On the **Geometric Analysis** tab, the nominal values of the control dimensions are given.



For the number of teeth in the length of the base tangent length and for the thickness of the gears, limits are given that limit the possibility of measurement.

On the **Accuracy Parameters** tab, control dimension deviations and backlashes are calculated taking into account DIN 3962 and DIN 3967.

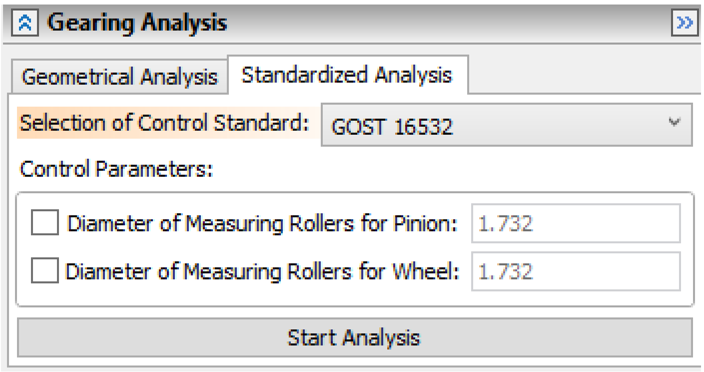
The calculation of deviations according to DIN 3962 and DIN 3967 is performed on the Accuracy Parameters tab.



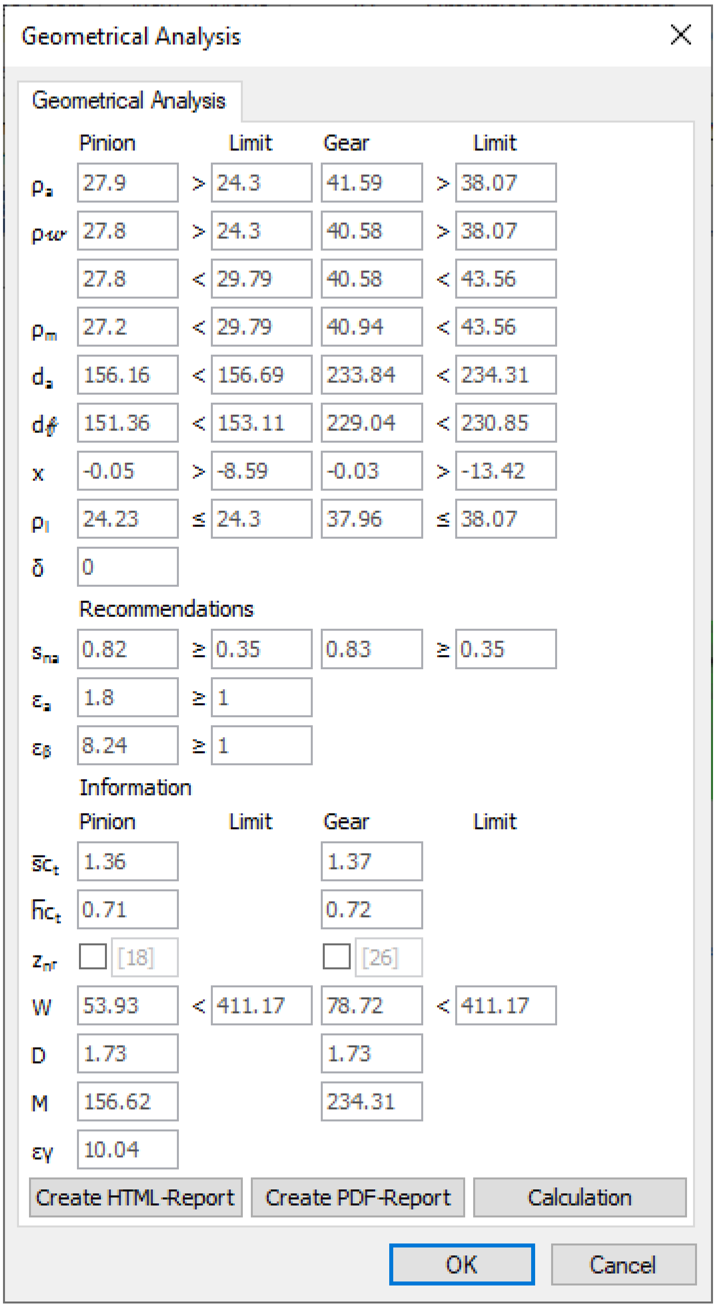
The checkbox **Consider appendix A DIN 3967** makes it possible to calculate the backlashes using an alternative method. The results of the calculation of backlashes taking into account Appendix A DIN 3967 will be displayed on the **Backlashes** tab of the model window for calculating the accuracy parameters.

Control Dimensions According to GOST

On the **Gearing Analysis** tab, on the Standardized Analysis tab, you must specify the diameter of the measuring rollers (balls) from the **GOST 16532** list. Or use the default value of diameters taken according to the recommendations of GOST 16532 from the list of GOST 2475.

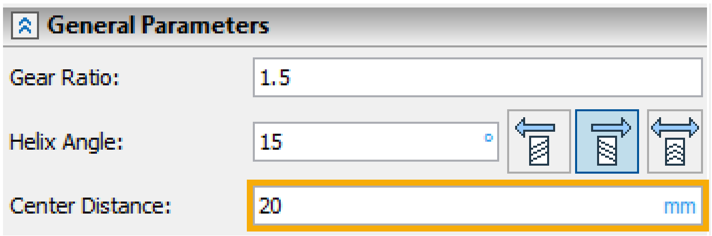


Next, you need to click the **Start Analysis** button. In the appeared modal window, the nominal values of the control dimensions will be shown.

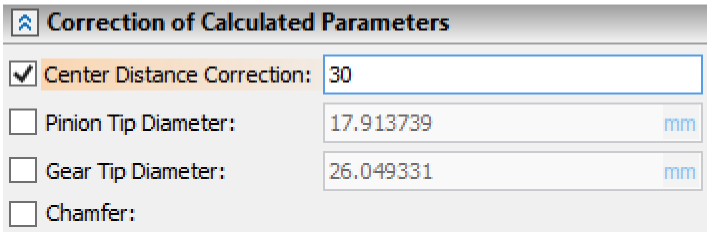


For the base tangent length size, a limit value is indicated that is permissible for a given crown width. For narrow coarse-modular rims, a situation may arise when the measurement cannot be made. In this case, you can change the calculated teeth number in the measurement. If you change the calculated number of teeth in the measurement, you need to use the **Calculate** button at the bottom of the window.

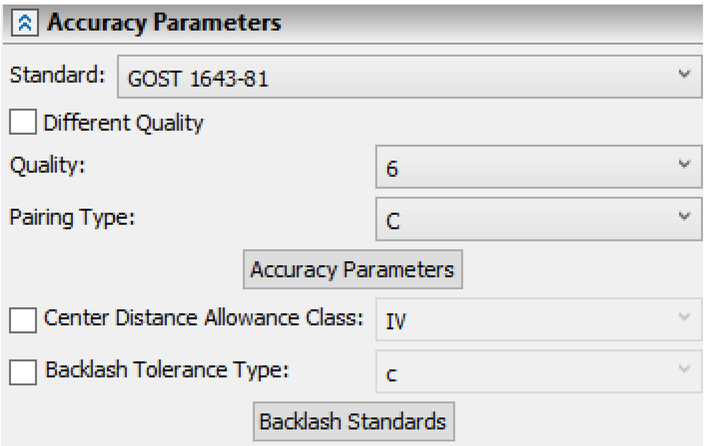
The user sets the value of the center distance by himself ...



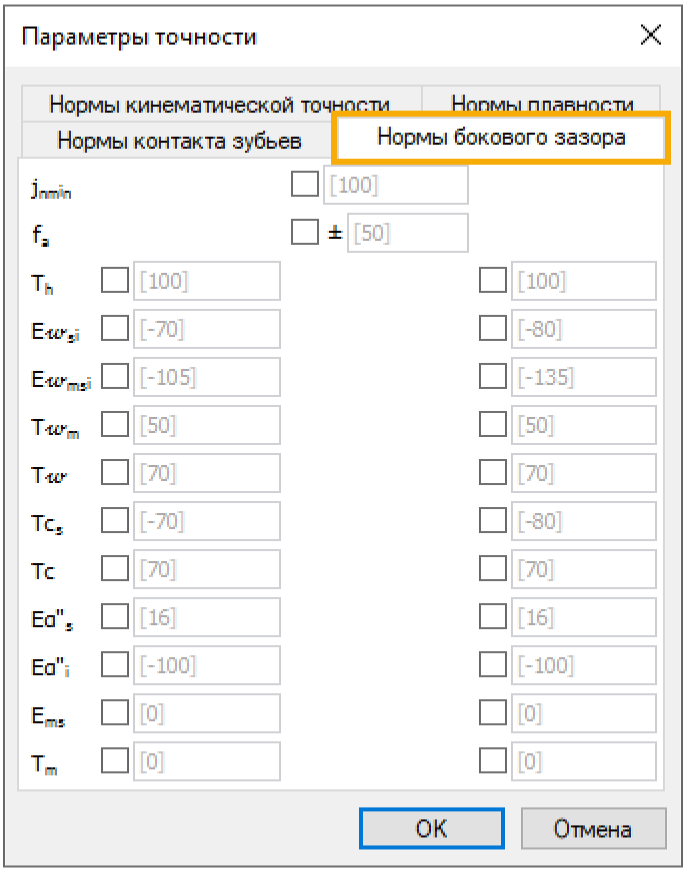
... or adjusts the calculated one.



Next, you need to calculate the tolerances on the Accuracy Parameters tab.

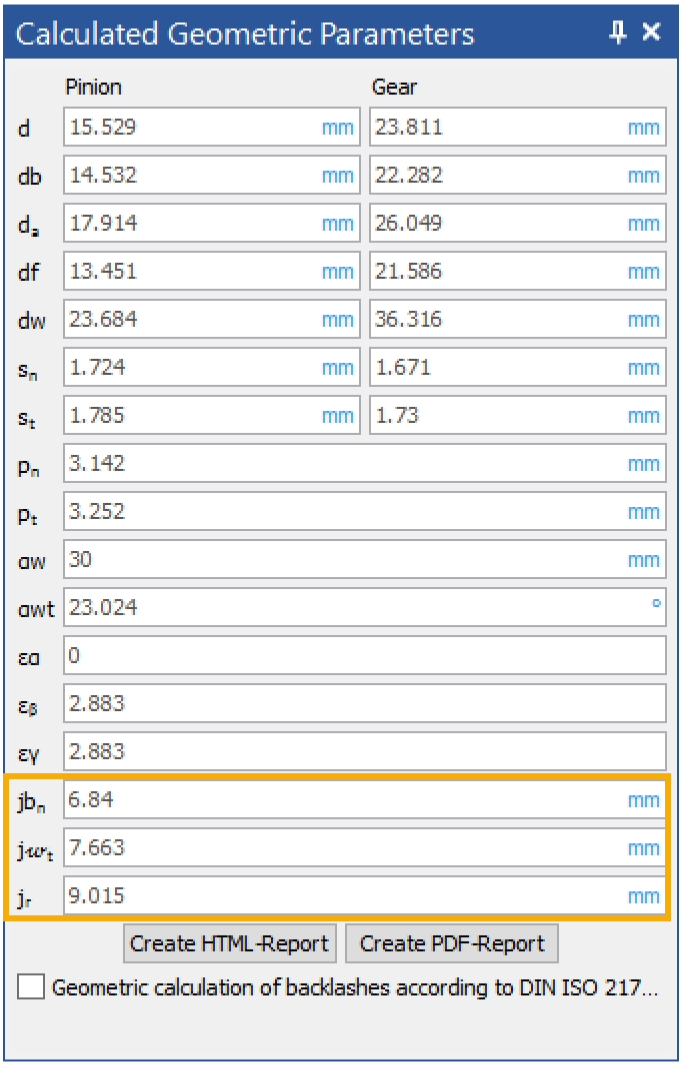


After clicking the **Start Analysis** button, a modal window will appear, where all the tolerances in accordance with GOST 1643 will be displayed, including the tolerances for the considered control dimensions and the center distance.



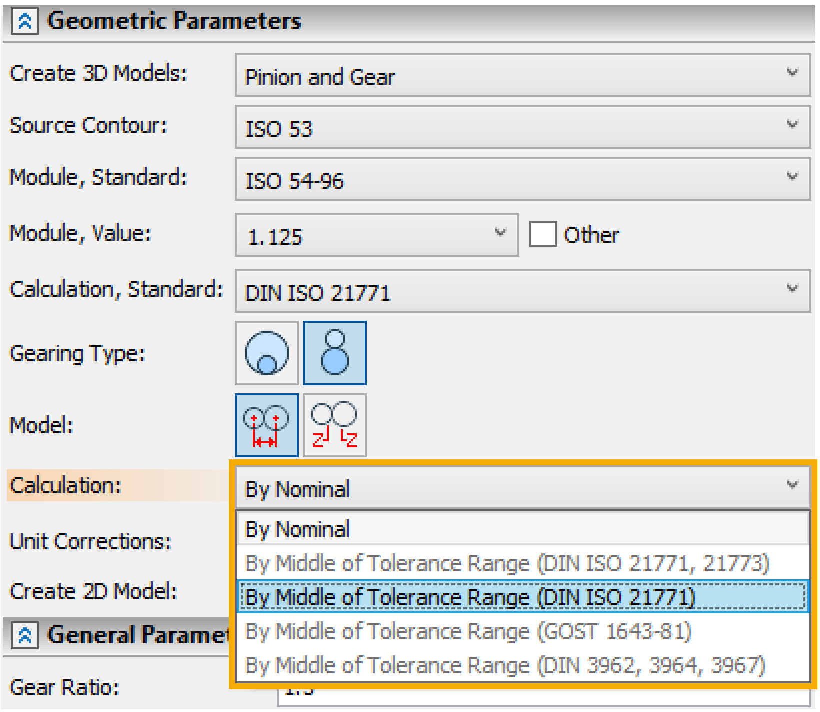
Calculation and Creation of Backlash

Typically, nominal gear dimensions bring the gears into contact with no backlash. The geometry of the gears without backlash is idealized and theoretical. This geometry is suitable for creating documentation, standardized calculations, and assigning tolerances. The actual geometry of the gears in the assembly implies a backlash. Generally, the backlash is determined by manufacturing tolerances. However, it is possible to set the backlash in the nominal geometry values, or combine both approaches. The add-on provides for the possibility of creating a nominal gearing geometry without a backlash, and a nominal geometry with a backlash, as well as calculating the geometry in the middle of the tolerances range with an analysis of the resulting backlash. If the nominal values of the gearing geometry already form a backlash, then the calculation in the middle of the tolerances range will take this into account, and the resulting geometry will form a backlash as the sum of two factors.
Backlash values in different directions can be controlled using special fields on the Calculated Geometric Parameters tab.



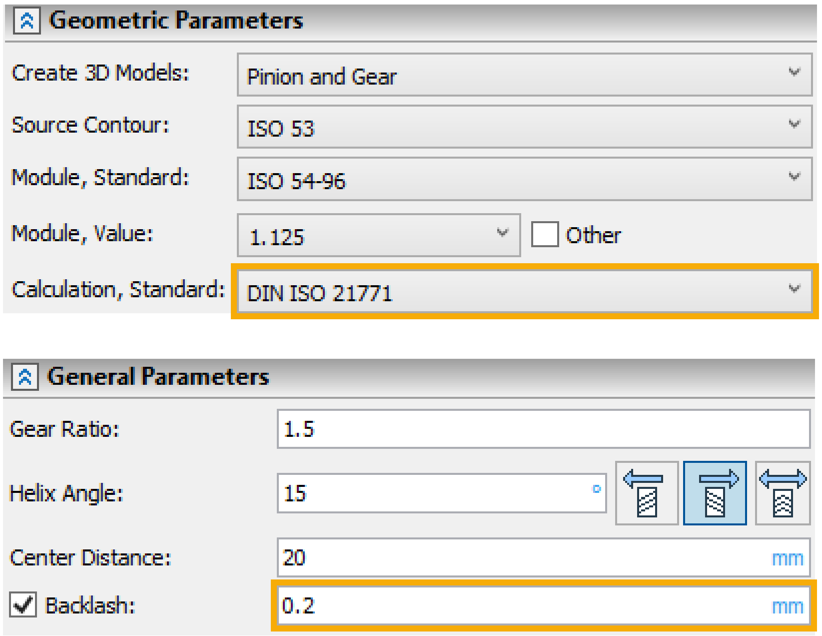
On this tab, all fields, including the highlighted ones, related to the dimensions of the backlash, always show the dimensions of the current calculated geometry, which can be both nominal and functional, calculated in the middle of the tolerance range. The model in the animation window and the created 3D model will always correspond to the geometric parameters specified on this tab. It is important to remember this when calculating the model in the middle of the tolerance range, because nominal parameters that determine the initial data for the calculation and which the user sets may differ from the final ones after the calculation in the middle of the tolerance range.

The option for calculating geometry in nominal value or in the middle of the tolerance range is specified in the special Calculation list. The availability of calculation according to the standards specified in the list depends on the selected standards for calculating the accuracy and the standard for calculating the control parameters.



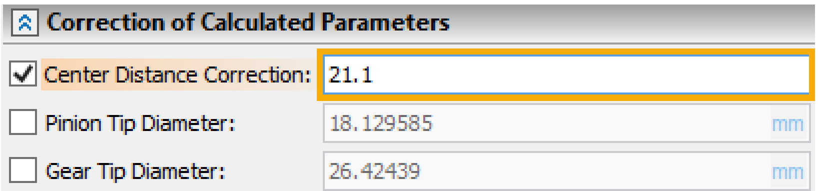
The following tools are available to create backlash on nominal geometry.

Setting the nominal backlash. When using the calculation standard DIN ISO 21771, a value for the nominal backlash can be set.



Correction the center distance. With external gearing, an increase in the center distance will lead to an increase in the backlash, with an internal backlash, it will increase with a decrease in the center distance. You can combine the nominal backlash value and the center distance value to obtain the final nominal backlash.

The resulting value will be calculated in the above fields on the Calculated Geometric Parameters tab.



To create a backlash taking into account manufacturing and assembly tolerances, switch the **Calculation** list to one of the available calculation options in the middle of the tolerance range.

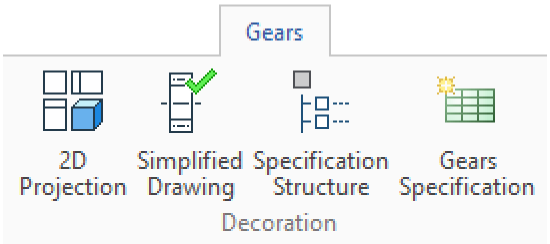
> The calculation of the tolerances depends on the nominal geometric parameters. Therefore, it is necessary to change and analyze the nominal dimensions in the **By Nominal** mode. The add-on allows you to edit the parameters of the geometry calculation and the main gearing parameters, which are nominal, also in the mode of recalculation to the middle of the tolerance range. But the methodologically correct work, which is easier to understand, is the preliminary creation and analysis of nominal geometry in the **By Nominal** mode, after which you can switch to the calculation mode in the middle of the tolerance range for analysis and creation of functional geometry. It is also recommended to edit the nominal values also in the **By Nominal** mode.

The resulting backlash in the calculation mode in the middle of the tolerance range will depend on the accuracy parameters and geometric control parameters.

> The work on the **Accuracy Parameters** tab and the **Standardized Analysis** tab in the calculation mode in the middle of the tolerance range is limited. This is due to the fact that geometric control parameters and accuracy parameters depend on the nominal dimensions. To change the accuracy parameters and geometric control parameters, you need to switch to the **By Nominal** mode. After the specified parameters are changed, you can again switch to the calculation mode in the middle of the tolerance range.

Drawing Settings

The decoration commands allow you to configure and create a gears specification, as well as quickly create a projection and configure a simplified drawing for it. Commands are available in the ribbon.



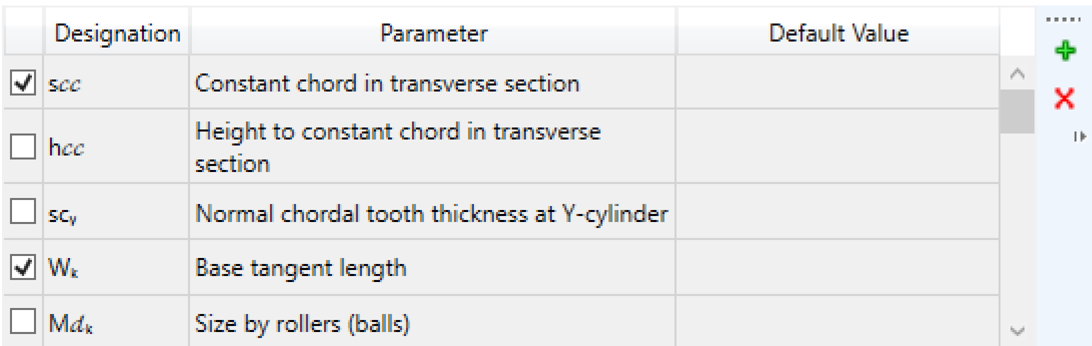
Topics in this section:

- Specification Structure
- Gears Specification
- Simplified Drawing

Add specification data

The **Add specification data** command allows you to set up a typical version of the table for the design of a gear drawing for a document, i.e. for all gears of all gearings that are included in the current assembly.

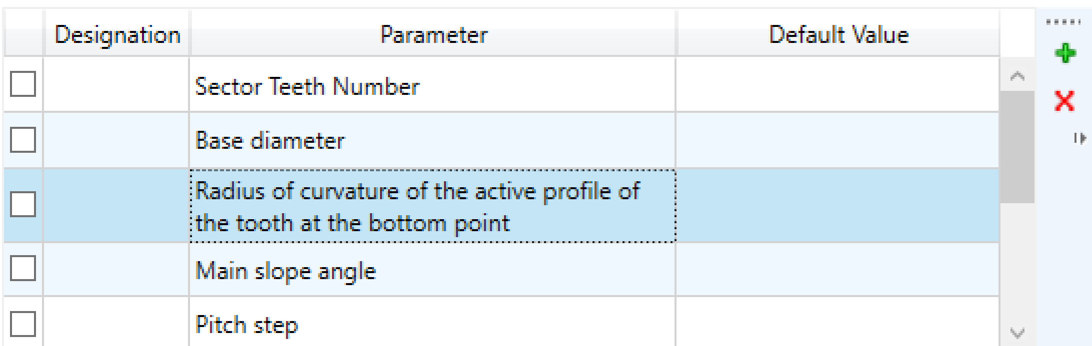
In the upper part, you select which of the standard control dimensions you want to bring to the table.



Control Data:

	Designation	Parameter	Default Value
☑	*scc*	Constant chord in transverse section	
☐	*hcc*	Height to constant chord in transverse section	
☐	sc_y	Normal chordal tooth thickness at Y-cylinder	
☑	W_k	Base tangent length	
☐	Md_k	Size by rollers (balls)	

Additional reference parameters are indicated at the bottom.



Other Reference Data:

	Designation	Parameter	Default Value
☐		Sector Teeth Number	
☐		Base diameter	
☐		Radius of curvature of the active profile of the tooth at the bottom point	
☐		Main slope angle	
☐		Pitch step	

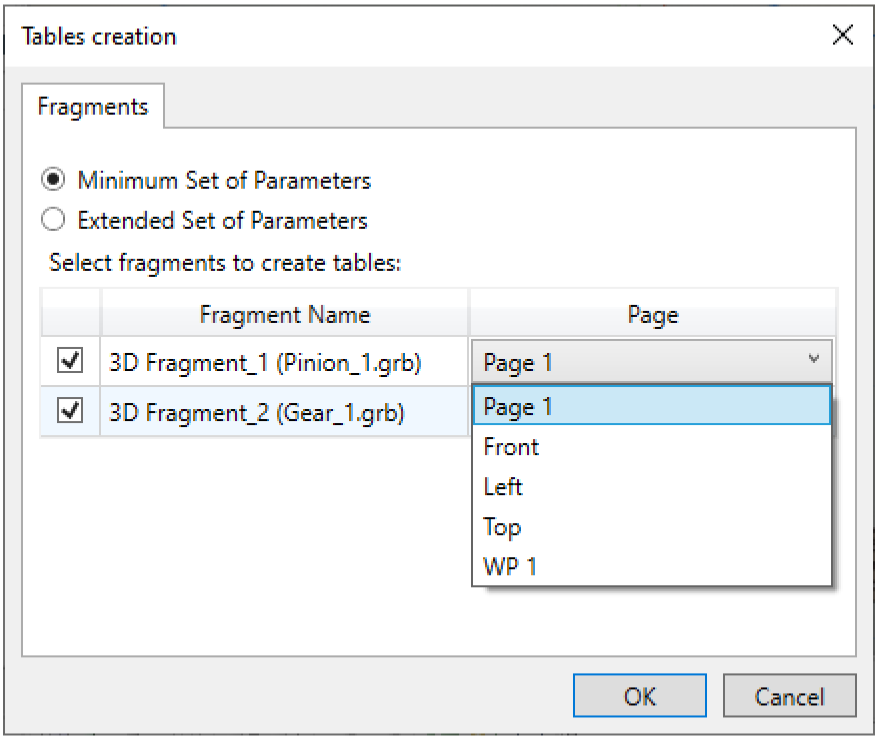
If the proposed list is not sufficient, it is possible to create an additional parameter using the icon ✚.

Setting the flag next to the parameter name means its inclusion in the design table of the gear drawing.

By Default. The button returns the settings of the table prototype to the default state.

Create specification table

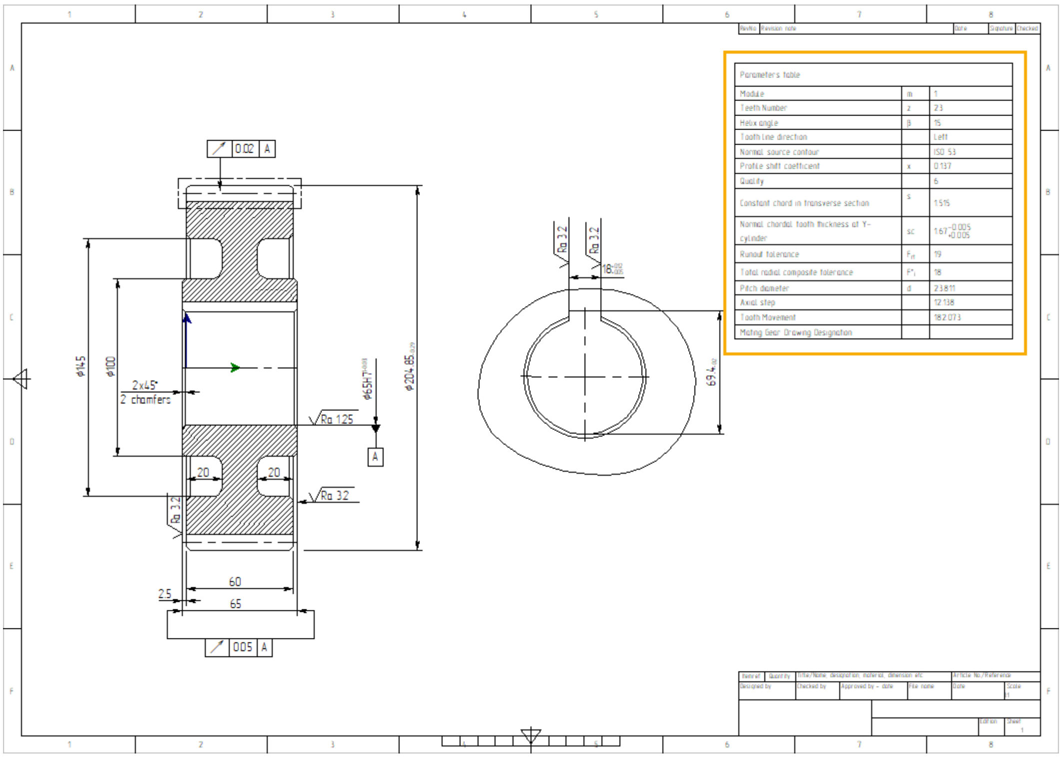
The dialog allows you to select for which gears and on which page of the document each of the gears needs to create a table of parameters. Setting the flag next to the name of the gear fragment means that a drawing design table will be created for this model. For each fragment, all available pages are displayed in the list.



Minimum Set of Parameters. Selecting this option means that a minimal table will be created.
Extended Set of Parameters. Selecting this option means that the table will be created in the form specified by the user in the **Specification Structure** command.
Setting the flag next to the name of the gear fragment means that a parameter table will be created for this gear.
If the specification structure for this document has not yet been defined by the user, before calling the **Gears Specification** command dialog, the Add specification data command dialog will be called. After confirming the specification structure, in the **Add specification data** command the dialog of the **Gears Specification** command will be called.
After clicking the **OK** button, a gear drawing design table will be created for all the specified gears.

Parameters table		
Module	m	1
Teeth Number	z	23
Helix angle	β	15
Tooth line direction		Left
Normal source contour		ISO 53
Profile shift coefficient	x	0.137
Quality		6
Constant chord in transverse section	s	1.515
Normal chordal tooth thickness at Y-cylinder	sc	$1.67^{-0.005}_{+0.005}$
Runout tolerance	F_{rt}	19
Total radial composite tolerance	F''_i	18
Pitch diameter	d	23.811
Axial step		12.138
Tooth Movement		182.073
Mating Gear Drawing Designation		

The table will automatically be placed in the upper right corner of the drawing format. All fragments in which the design tables are created will be automatically opened, and the page on which the table is created will be active.

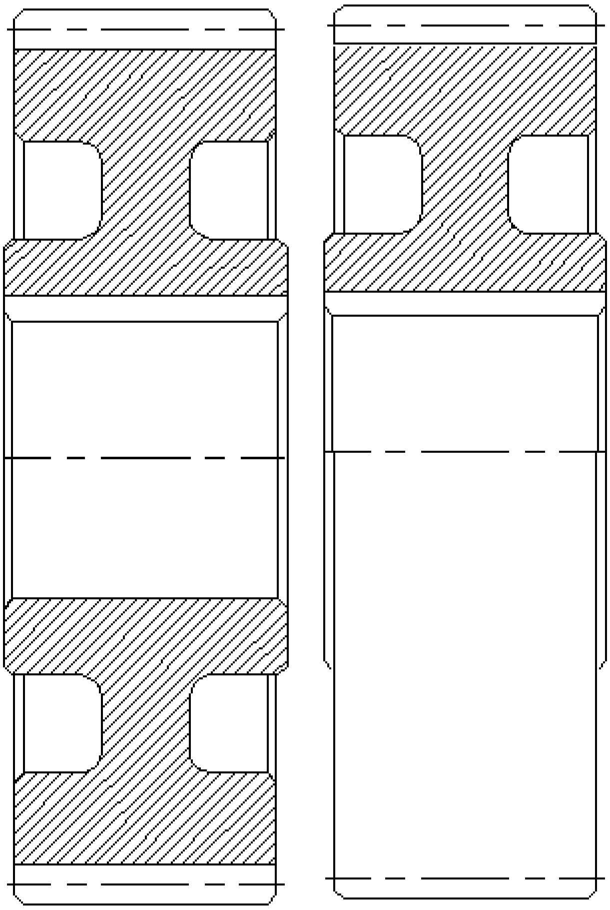


Simplified Drawing

The command allows you to create a simplified image of a gears and gearings. The command puts a simplified drawing on top of the projections of the add-on's objects. Projections for which it is possible to create a simplified drawing must meet a number of requirements. Projections of individual gears or gear shafts when creating part drawings:

- the gear must be created in the add-on commands;
- the cut must pass through the axis of the gear rim;
- the cut can be full or up to the axis.

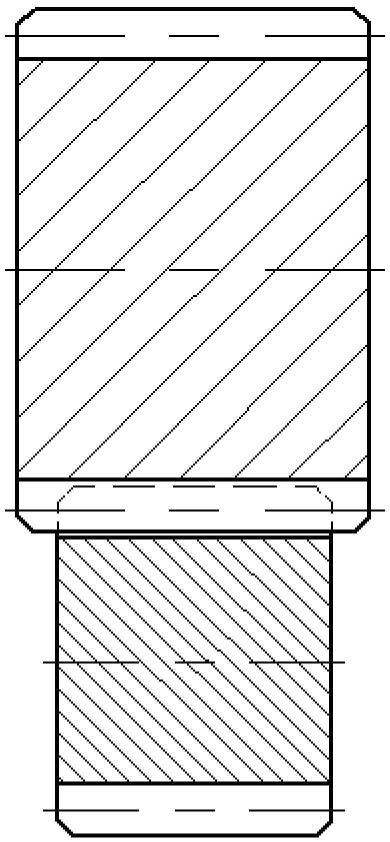
Examples of cuts with an applied simplified drawing.



Gearings projections or gearings sequences when creating assembly drawings:

- gearing must be performed in the Gearing, Three Gears Train or Single Row Planetary Gear commands;
- the projection is made on the document where the gearing is created;
- the cut of the gearing passes through the axes of the gears;

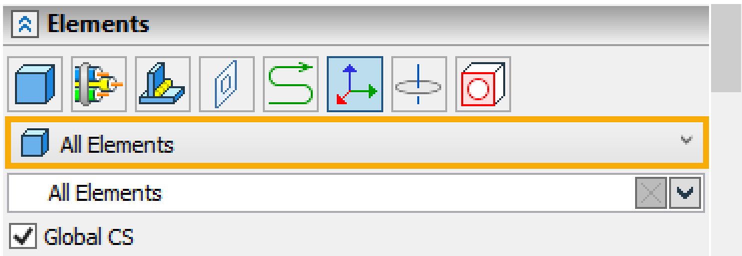
An example of gearing with the applied simplified drawing:



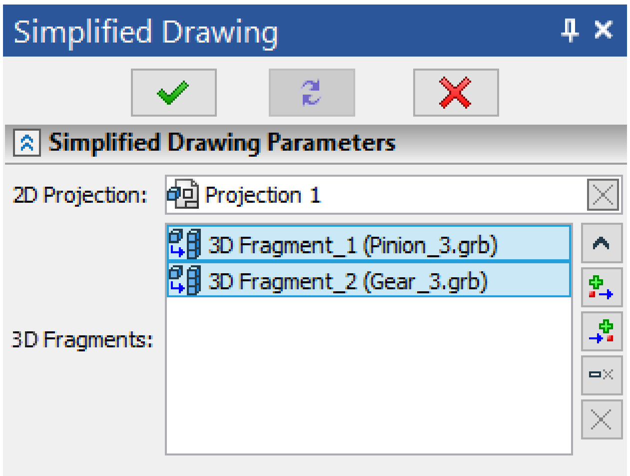
For the correct positioning of the simplified drawing on the projection, it is necessary to project the coordinate systems of the gears.

> The projections, on which it is later planned to apply a simplified drawing, must contain the projections of the gear coordinate systems. Coordinate system projections can be made when creating or editing a projection.

To do this, in the **2D Projection** command on the **Elements** tab, select the Coordinate system option, then select All elements from the list.

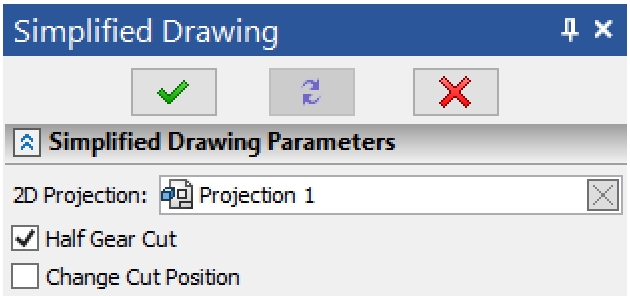


For convenience, the **2D Projection** command has been moved to the add-on ribbon in the **Decoration** section.
After a projection has been obtained that satisfies all the above conditions, you can use the **Simplified Drawing** command. After calling the command, in the **2D Projection** field, specify the projection on which you want to create a simplified drawing. If you are drawing a gearing, then in the **3D Fragment** field, all gears that meet the requirements for creating gear projections will be added.



The list of fragments selected automatically can be edited.

If you are drawing a part, then when you select a projection, instead of the **3D Fragment** field, the **Half Gear Cut** option appears, which is necessary if the cut is made only up to the axis. When this option is checked, an additional option appears to change the position of the cut part relative to the axis - **Change Cut Position.**



At the end of the command with confirmation, simplified drawings of gears will be created on the selected projection.

Report

The following types of reports are available in the add-on.

- General Parameters.
- Accuracy Parameters.
- Control Parameters (Geometric Analysis).
- Strength Parameters.

Each of the report types is available for creating in the corresponding section of the **Gearing** command.

The report on the general parameters is available on the General Calculated Parameterstab.

Accuracy parameters report is available in the accuracy parameters results window according to the selected standard. The results window is available when working on the Accuracy Parameters tab.

The control parameters report (geometric analysis report) is available in the gearing analysis results window according to the selected standard. The results window is available when working on the Gearing Analysis tab.

The report on strength parameters consists of the following parts.

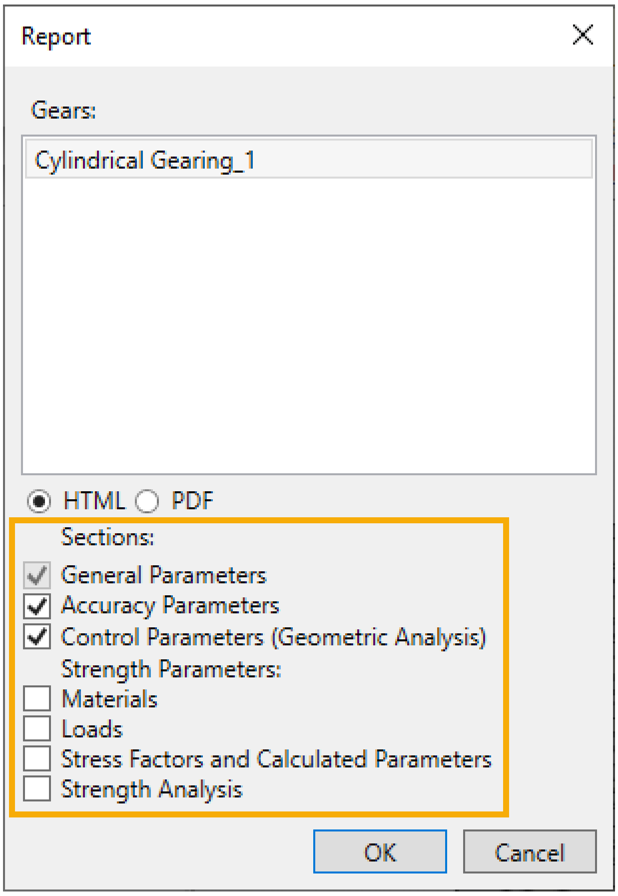
- Materials.
- Loads.
- Stress Factors and Calculated Parameters.
- Strength Analysis.

The report on the strength parameters is available in the strength analysis results window according to the selected standard. The results window is available when working on the Strength Analysis tab.

When creating the above types of reports, two formats are available to choose from: *.html and *.pdf. The selection is made by pressing a button. When creating a report in *.pdf format, you will be prompted to enter a name and specify the directory for saving.

The report on the general parameters is included in the first part of all types of reports.

The **Report** command allows you to create a general report with a choice of report types that are included in it. Setting the flag next to the name of the report type displays it in the general report. If, while working with the command, the user did not perform any of the types of analysis, the flag of inclusion in the general report will be unavailable.



The report flag for the general parameters is set by default, because without this data, the gearing information is meaningless.

Cylindrical Gearing

Creating, calculation and analysis of cylindrical gears and gearings is possible according to the following standards at the user's choice.
ISO 53, DIN 867, GOST 13755 - regulate the source contour.
ISO 54, DIN 780, GOST 9563 - regulate the module values.
DIN ISO 21771 and DIN 21773, GOST 16523, GOST 19274, - regulate the calculation of geometry and control dimensions.
ISO 1328, DIN 3962, DIN 3964 and DIN 3967, GOST 1643 - regulate the calculation of deviations.
ISO 6336, DIN 3990, GOST 21354 - regulate the calculation of strength characteristics.
Non-standard options are available for calculating geometry, control dimensions and strength. An animation of the gearing is also available with the analysis of intersections, taking into account the method of making the gears.
Thus, the following options are available for cylindrical gears.

- Creation of 3D models of gears, gear pairs, gear mechanisms;
- Creation of 2D models of gears, gear pairs;
- Animation of gearing with the display of the most important parameters;
- Graphic display of stresses, slip rate and length of the contact line throughout the entire gearing;
- Ability to create geometry of gears according to ISO, DIN, GOST standards, as well as custom parameters;
- The ability to analyze the geometry according to the selected type of manufacturing tool;
- Automatic diagnostics of gear pair jamming, as well as inadmissible values of gearing parameters;
- Calculation of tool shifts based on optimization with specified limit parameters;
- Analysis of geometric parameters according to ISO, DIN, GOST standards;
- Analysis of strength parameters according to ISO, DIN, GOST standards;
- Calculation of manufacturing accuracy parameters according to ISO, DIN, GOST standards;
- The ability to automatically generate reports, including all calculated parameters grouped into tables;
- The ability to automatically generate decoration elements for drawings of gears.

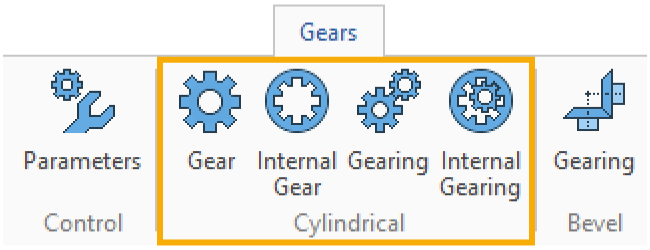
To create a pair of cylindrical gears in gearing, use the **Gearing** command. To create a single, unconnected, cylindrical gear, use the **Gear** command. The **Internal Gear** command allows you to create a single internal gear. For single gears, a gearing command is available.
The **Gearing** command allows you to create both and only one object of a gear pair while retaining the ability to analyze the gearing. In the mode of creating only one gearing object, you must also indicate a paired gear, the model of which will not be created. The specified mode is necessary for creating a sequence of gearing and is automatically used in the commands for creating mechanisms.
The **Gear** or **Internal Gear** commands allow you to create a single gear according to the specified parameters without the possibility of geometrical analysis of the gearing and strength analysis of the gearing. The command can be used if it is necessary to obtain a 3D or 2D model of a gear without specifying additional parameters of the paired gearing.

Topics in this section:

- Standardized Parameters
- Tool Parameters
- Parameter Diagnostics
- Calculated Parameters Correction
- Accuracy Parameters
- Gearing Analysis
- Operating Characteristics
- Strength Analysis
- Gearing Creation Precision Parameters

Commands for Creating Cylindrical Gears

The add-on has four commands for creating cylindrical gears and gearing. The commands have a uniform interface in style and functionality. You can call commands from the ribbon.

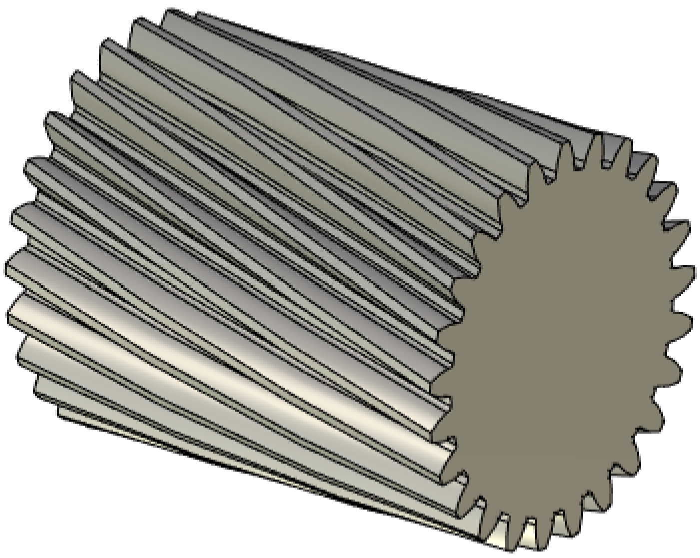


Topics in this section:

- Gear
- Internal Gear
- Gearing
- Internal Gearing

Gear

The command allows you to create a 3D model (optional 2D model) of a single cylindrical gear with external teeth.



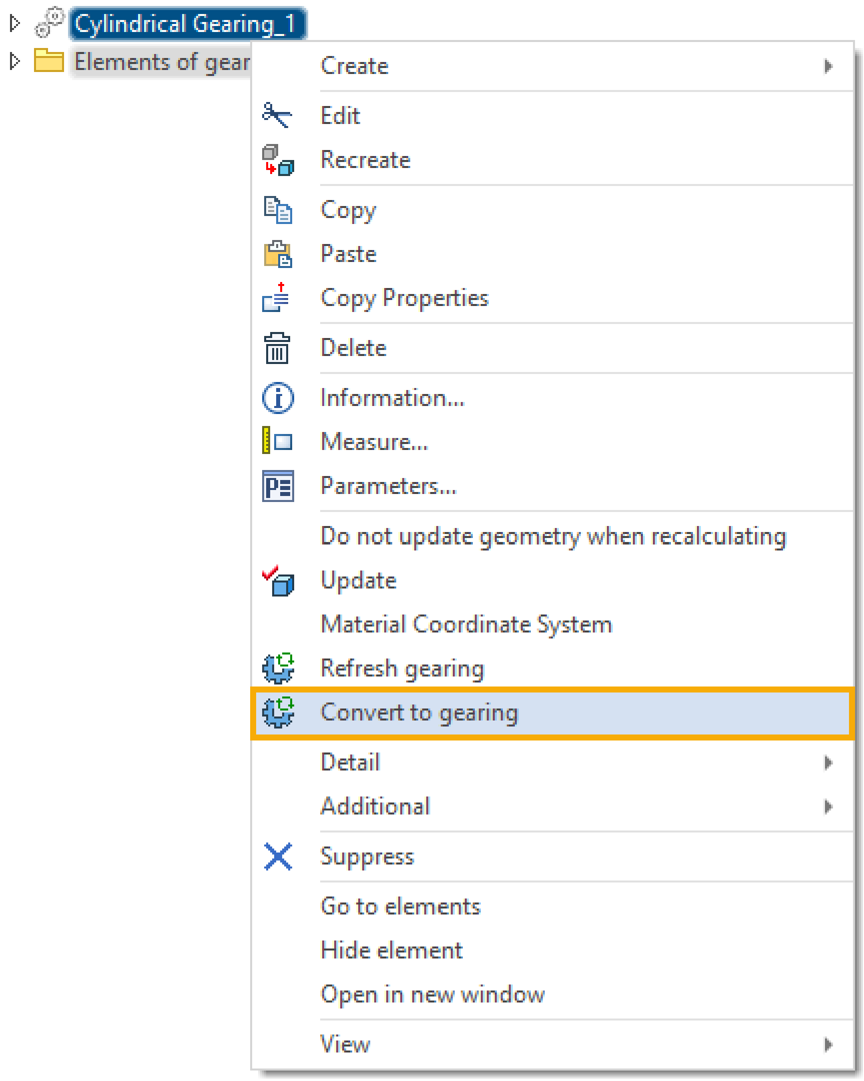
The following tabs are available in the command.

- Geometric Parameters;

- General Parameters;
- Standardized Parameters;
- Binding;
- Chamfer;
- Accuracy Parameters;
- Creation Precision Parameters.

All tabs work in the same way as the main command of cylindrical gears **Gearing**. In this case, the teeth number and the profile shift coefficient can only be specified, without the possibility of calculation.

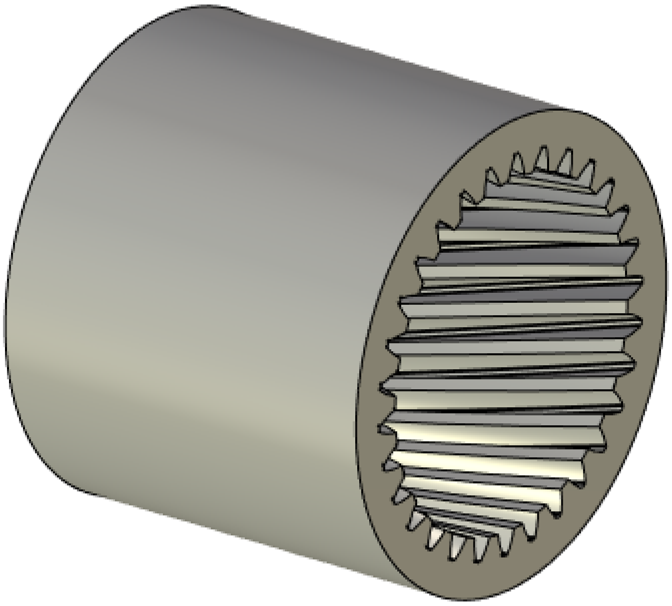
The created 3D model of the gear is a special object of the module and is displayed accordingly in the model tree. In the context menu of the module object, the command for converting a gear to a gearing is available.



In this case, the created gear, while keeping its parameters, will be opened through the **Gearing** command dialog. This will set the default parameters for the second gear. All parameters of both the original gear and the second gear will be editable. The previously created, original, gear becomes an object of gearing for which all types of analysis provided by the functionality of the module are available.

Internal Gear

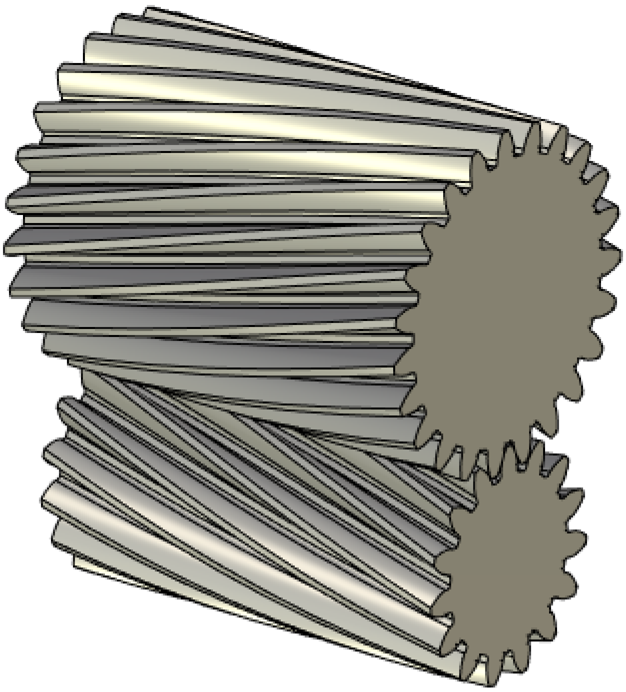
The command allows you to create a 3D model (optional 2D model) of a single cylindrical gear with internal teeth.



In terms of functionality, the **Internal Gear** command is similar to the Gear command.

Gearing

The command allows you to create a 3D model (optional 2D model) of the external or internal gearing of two cylindrical gears.



Allows you to make the strength calculation of the gearing, determine the tolerances and nominal values of the control dimensions, select or calculate the backlashes in the gearing.
The following tabs are available in the command.

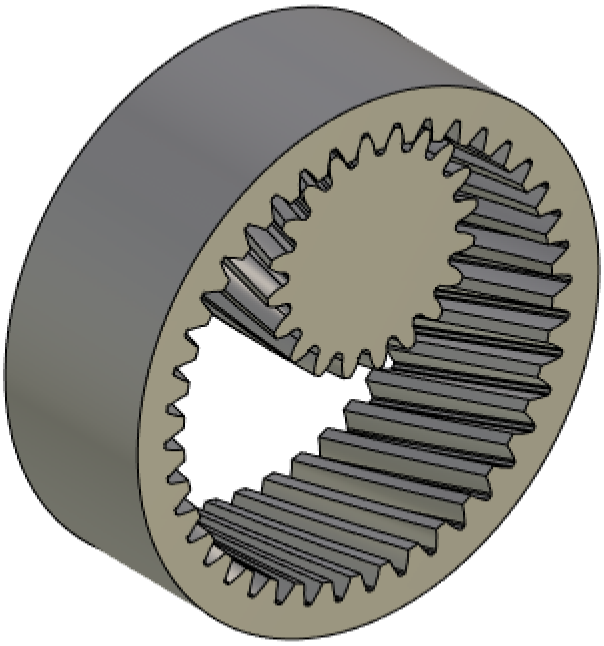
- Geometric Parameters;
- General Parameters;
- Calculated Geometric Parameters;
- Binding and Positioning;

- Standardized Parameters;
- Tool Parameters;
- Parameter Diagnostics;
- Calculated Parameters Correction;
- Accuracy Parameters;
- Gearing Analysis;
- Operating Characteristics;
- Strength Analysis;
- Creation Precision Parameters.

The **Gearing** command is the main command for creating cylindrical gears and gearings.

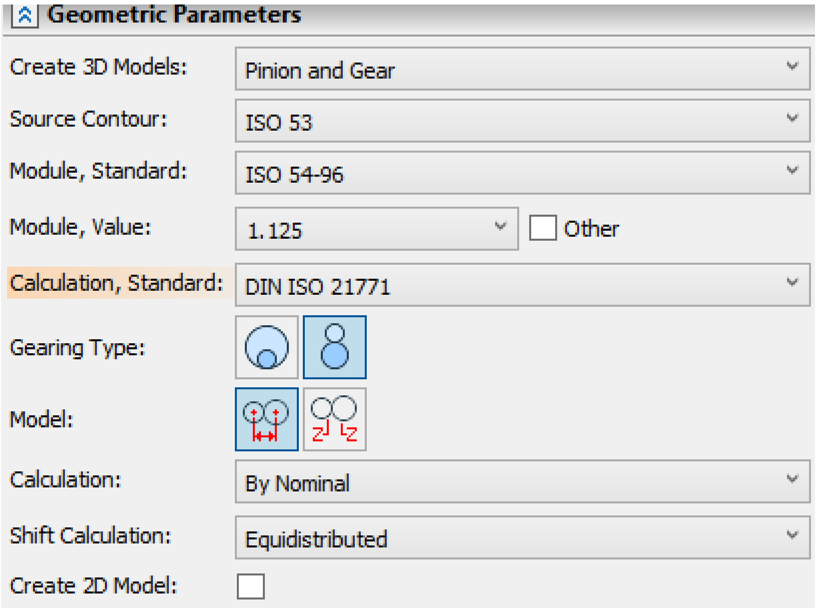
Internal Gearing

The command allows you to create a 3D model (optional 2D model) of the external or internal gearing of two cylindrical gears. This is an additional way to call the Gearing command. Unlike calling the command via the **Gearing** command, calling the command via the **Internal Gearing** icon by default starts the command with the selected internal gearing type and with the **Teeth Number** model.



Geometric Parameters

Geometric parameters determine the contour and size of the teeth, the type of gear (internal or external), and the method of data entry.



Create 3D Models. This option allows you to select from the list which 3D objects to add to the scene:

- **Pinion and Gear;**
- **Pinion;**
- **Gear.**

This option is needed when creating a chain of gearing, where one of the objects is common to two gearing and should not be duplicated.

Source Contour. This option allows you to select the following standards from the list:

- **ISO 53**;
- **DIN 867;**
- **GOST 13755-2015**;
- **Manually.**

The contour standard defines the input availability and relationship of the parameters in the **Standardized Parameters** group. The **Manually** option allows you to freely change all parameters of the original contour without reference to standards.

Module, Standard. Defines a list of module values allowed by the specified standard.

Module, value. The option allows you to select from the list the module value that is allowed by the selected standard. **Other**. Setting the flag opens a field for entering the value of the module that is not provided for by the standard.

Calculation, Standard. The option allows you to select the calculation of the geometry of gears according to the standards:

- **DIN ISO 21771.**
- **GOST 16532**;

Calculation standard defines calculation formulas, interconnection and availability of gearing parameters input.

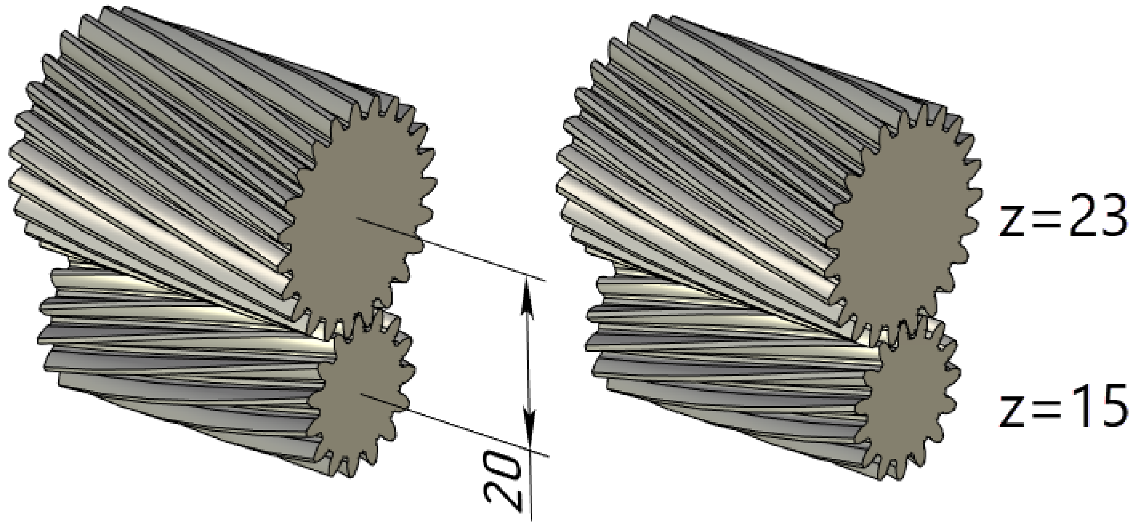
Gearing Type. There are two options available:

- **Internal**;
- **External**.

Changing the type of gearing is available only during the creation of the gearing. If you edit the parameters of an already created gearing, changing the type of gearing is not available.

Model. There are two options available:

- **Center Distance**;
- **Teeth Number**.



When choosing the **Center Distance** model, the input of the center distance is always available, and the input of the number of teeth and the gear ratio depends on the selected combination of parameter input (for more details, see the General Parameters section). When selecting the **Teeth Number** model, the entry of the number of teeth is always available. Gear ratio and center distance are calculated and not available for input.

Calculation. There are five options available:

- **By Nominal**;
- **By Middle of Tolerance Range (DIN ISO 21771, DIN 21773)**;
- **By Middle of Tolerance Range (DIN ISO 21771)**;
- **By Middle of Tolerance Range (GOST 1643-81)**;
- **By Middle of Tolerance Range (DIN 3962, 3964, 3967)**.

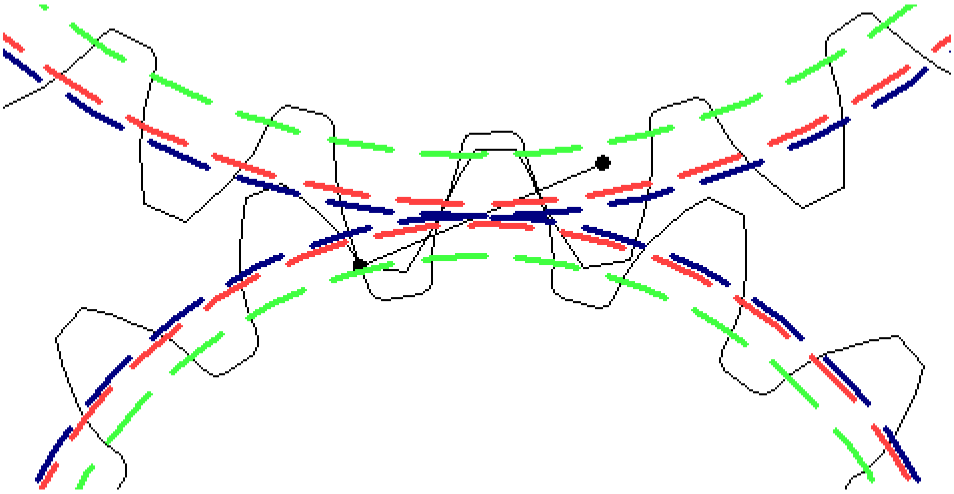
In detail, the choice of calculation options by middle of the tolerance range is described in the Calculation by Middle of Tolerance Range section.

Unit Corrections. There are four options available

- **Equidistributed**;
- **Manually;**
- **By Contact Stress;**
- **By Bending Stress.**

For **Teeth Number** model, the **Equidistributed** option is not available. The methods for calculating shifts are described in detail in the Shift Calculation section.

Create 2D Model. On a separate page in the current document, the contour of the gears in the transverse section, as well as the line of gearing, will be drawn. On the 2D model, the base, pitch and working pitch diameters of each gear are also indicated.



To display a 2D gearing model, open **Page 2** of the document.

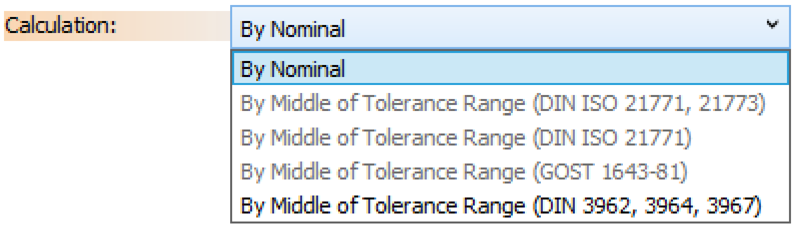
The flag of creating a 2D model can be cleared only during the creation of a gearing. If you edit the parameters of an already created gearing, you cannot clear the flag. The flag can also be set when editing a gearing.

Topics in this section:

- Calculation by Middle of Tolerance Range
- Shfit Calculation

Calculation by Middle of Tolerance Range

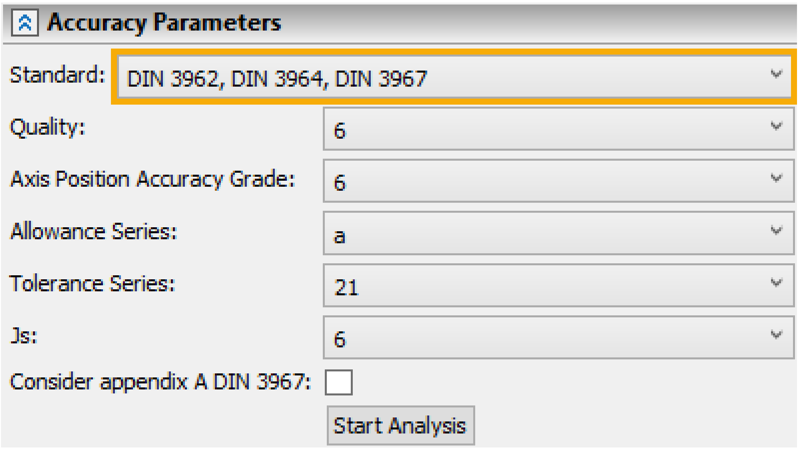
Shifts calculation options are available in the list:

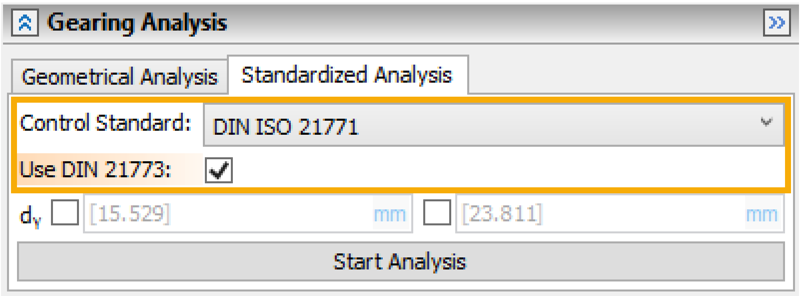


By Nominal. It is the main version of the calculation.

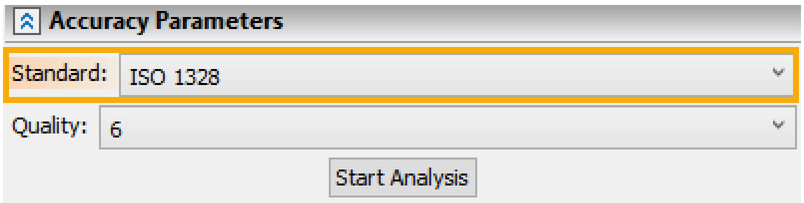
It is recommended that all corrections of the initial calculation data be made in the **By Nominal** mode.

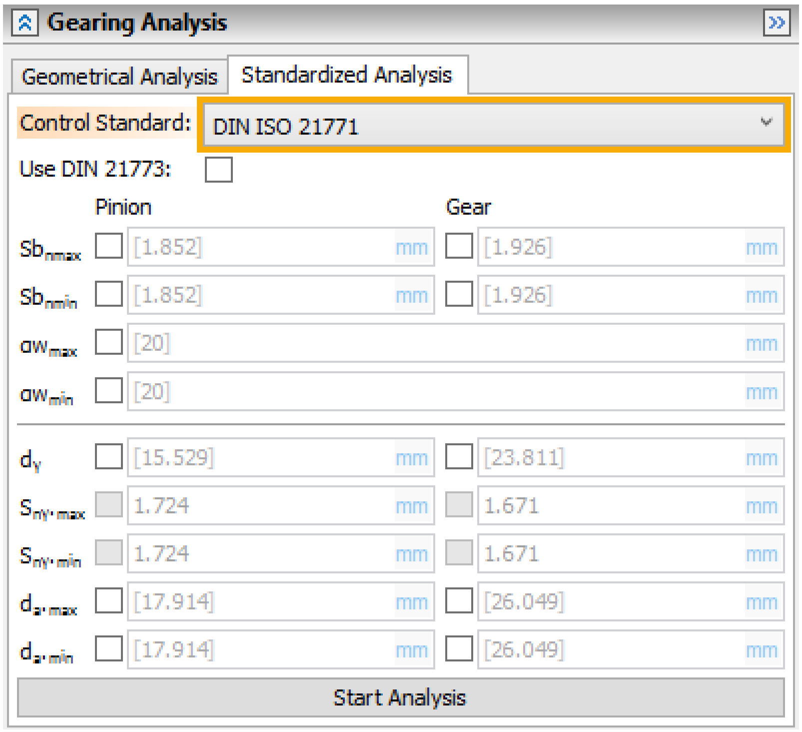
It is necessary to make calculations of accuracy, as well as standardized calculation of geometry analysis in the **By Nominal** mode, because these calculation types are based on nominal geometry values. When you select one of the options for calculating in the middle of the tolerance range, the specified types of calculations will be closed.
By Middle of Tolerabce Range (DIN ISO 21771, 21773). This option allows you to calculate the geometry in the middle of the tolerance range according to DIN 21773. This standard is an addition to DIN ISO 21771, and for calculating deviations it refers to DIN 3967. Therefore, in order for this calculation option in the middle of the tolerance range to be available, an analysis of deviations must be done according to DIN 3962, 3964 and 3967, and indicate the use of DIN 21773 when calculating according to DIN ISO 21771.



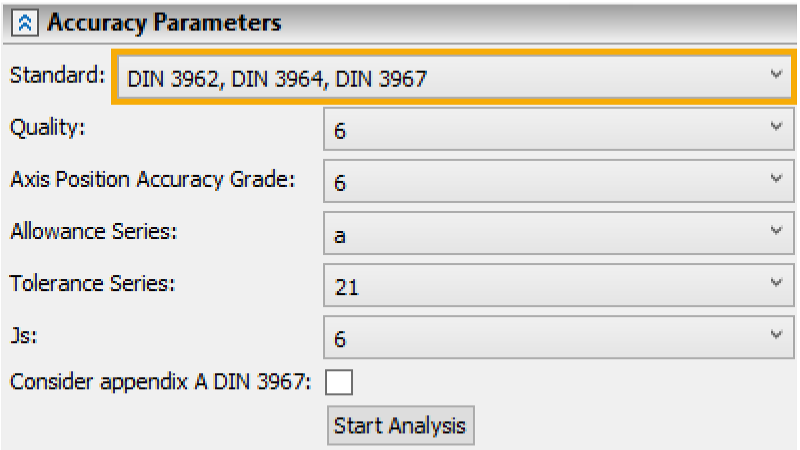


By Middle of Tolerance Range (DIN ISO 21771). This option allows you to calculate the geometry in the middle of the tolerance range according to DIN ISO 21771, which in turn refers to ISO 1328 for calculating deviations. Therefore, in order for this calculation option in the middle of the tolerance range to be available, you need to analyze the deviations according to ISO 1328, and also carry out an analysis according to DIN ISO 21771.

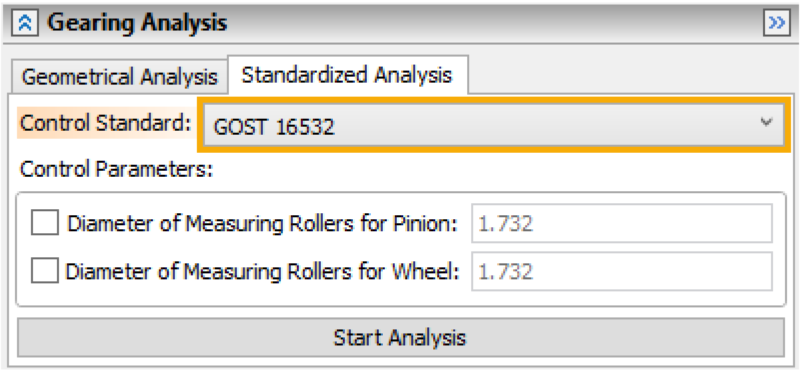


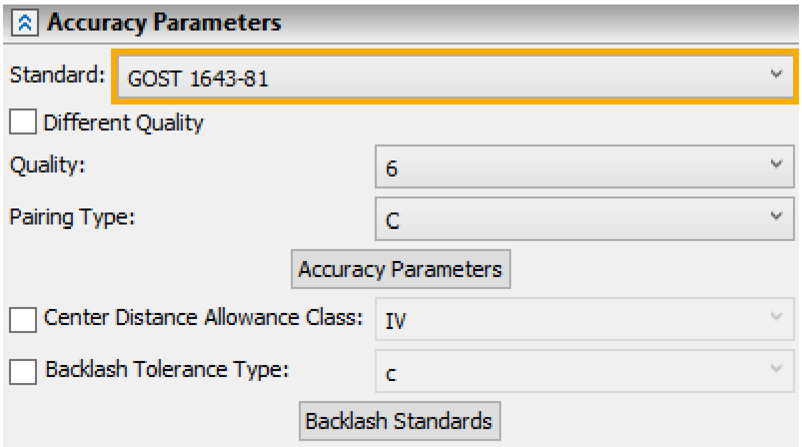


By Middle of Tolerance Range (DIN 3962, 3964, 3967). This option allows you to calculate the geometry in the middle of the tolerance range according to DIN 3967, which in turn is related to DIN 3962 and DIN 3964. In order for this option to calculate in the middle of the tolerance range to be available, you need to analyze the deviations in accordance with DIN 3962, 3964 and 3967.



By Middle of Tolerance Range (GOST 1643-81). This option allows you to calculate the geometry in the middle of the tolerance range according to the GOST 1643-81 standard. The control dimensions, the deviations of which this standard specifies, are calculated, in turn, according to GOST 16532 (or GOST 19274, if internal gearing is selected). Therefore, in order for this calculation option in the middle of the tolerance range to be available, it is necessary to analyze the geometry in accordance with GOST 16532 (or GOST 19274), as well as an analysis of deviations in accordance with GOST 1643-81.





When calculating in the middle of tolerance range by GOST 1643-81, the additional option Select a tolerance field is available. The list contains three options:

- **Shift Tolerance** - calculates the average tooth thickness based on the shift tolerance;
- **Tooth Thickness Tolerance** - calculates the average tooth thickness based on the tooth thickness tolerance;
- **Base Tangent Length Tolerance -** calculates the average tooth thickness based on the tolerance for base tangent length tolerance;

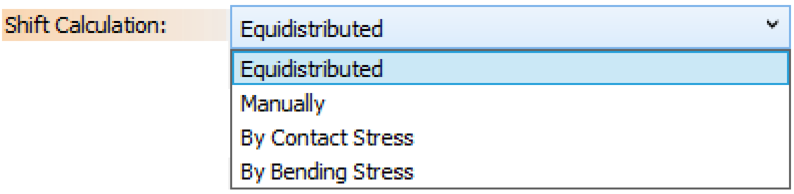
> All custom values that affect center distance, backlash, tooth thickness or tip diameters will be taken into account when calculating to the middle of the tolerance range.

When calculating in the middle of the tolerance range, the input fields display the nominal values relative to which the calculation is carried out. The actual geometric dimensions and backlashes can be seen on the Calculated Geometric Parameters tab.

> Animation, stress plots and the created 3D model will be calculated from the model in the middle of the tolerance range if the appropriate mode is selected.

Shift Calculation

Shift calculation options are available in the list:



Equidistributed. The option is only available for the **Center Distance** model. The total shift required to properly mate the pinion and gear will be in proportions proportional to the number of teeth distributed to the pinion and gear.
Manually. The option allows you to set the shift both on the pinion and on the gear for the **Teeth Number** model. With the **Center Distance** model, you can set the shift only on the pinion or only on the gear; on the second object, the shift will be calculated according to the total shift.
The **By Contact Stress** and **By Bending Stress** options allow you to select shifts at which the contact or bending stresses will be minimal. To start and configure the optimization of stress shifts, click the **Calculate** button.

The button becomes available when you select the calculation of shift stresses. The button calls the modal dialog for optimizing shifts, in which you can enter the parameters limiting the search, configure the solution, start the optimization of shifts, and display the parameters at the current step of the solution. After the optimization is complete, the user will be shown the achieved result in terms of stress values and shifts. If the result is satisfactory, you need to close the optimization window by clicking the **OK** button. In this case, all shift-dependent values will be recalculated and the animation model will be updated. Otherwise, you need to close the optimization window using the **Cancel** button.

Shift Optimization Dialog

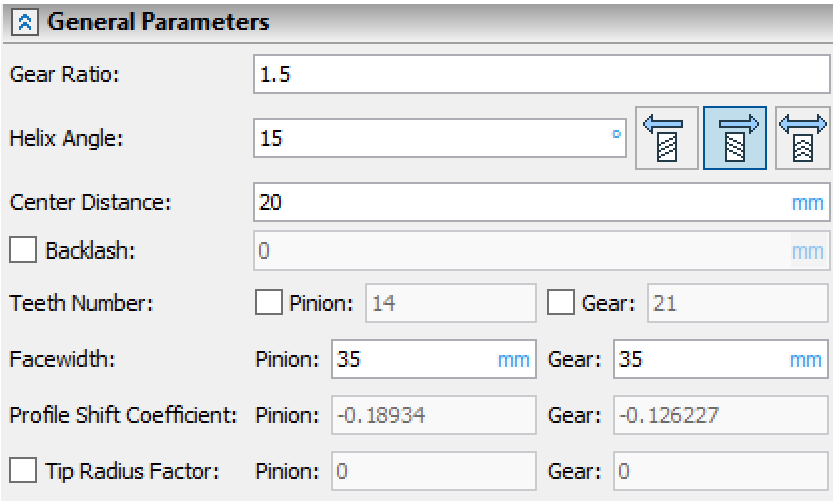
Precision of Solution at Border. There are two options available:

- Normal;
- Increased.

The **Increased** calculation option clarifies the calculation of the stresses for the helical tooth at the moment of entry and exit from the gearing.
Method. So far, only a step by step iteration of the shift values is available.
Pinion (Gear) Sharpening Ratio. The minimum permissible relative (to the module) size of the arc of the tip of the pinion (gear) tooth.
Positive (Negative) Shift Limit. Maximum and minimum shifts on the pinion and gear.
Transverse Contact Ratio. Minimum permissible transverse contact ratio.
Step Size. The step size for enumerating the shift values: from 0.01 to 0.1.
Maximum Step Number. Maximum number of steps to find a solution.
Start Optimization. Optimization start button.
Stop Optimization. Optimization can take a relatively long time, so there is a button for stopping the optimization process: the solution will be completed at the current solution step, after which the system will stop the optimization.
The lower fields of the dialog are only informative and not available for input.
Step Number. The field displays the current solution step.
Pinion (Gear) Shift. The field displays the value at this step of calculating the shift on the pinion (gear).
Maximum Stress. The field displays the maximum stress on the tooth during the entire gearing (for more details on calculating stresses in the Strength Analysis section) obtained at this step of the calculation.
Pinion (Gear) Sharpening Ratio. The field displays the value at this step of calculating the point of a pinion (gear).
Transverse Contact Ratio. The field displays the value at this step for calculating the transverse contact ratio.

General Parameters

The **general parameters** determine the type of gearing, spur, helical or double helical, gear ratio, shift values, number of teeth and center distance.



Gear Ratio. The field allows you to enter the required gear ratio. The actual, calculated gear ratio may differ from the given one, but it will always be as close as possible to the given one. The field is available for input only when the model is selected **Center Distance** and calculated (not specified by the user) values of the number of teeth on the pinion and gear. For the **Teeth Number** model (where the number of teeth is always specified by the user) and when entering the number of teeth on a pinion or gear for the **Center Distance** model, the field is calculated and is closed for input.

Helix Angle. The field allows you to set the helix angle of the teeth at the pitch diameter. The angle can be entered in minutes and seconds through a special command of the context menu of the input field. The direction of helix angle of the teeth on the pinion is determined by the **Left Hand/Right Hand** switch. The direction of helix angle of the teeth on the gear is determined by the type of gearing: for the **External** type, the helix angle on the gear will be reversed, for the **Internal** type - the same as on the pinion. An additional **Double Helical Gear** button allows you to create a chevron gear. In this case, the **Left Hand/Right Hand** switches remain available and they determine the type of double helical gear on the pinion: "Left Hand-Right Hand" if **Left Hand** is selected and "Right Hand-Left Hand" if **Right Hand** is selected.

Center Distance. The field allows you to enter the center distance. For external gearing, the center distance shifts the gear center up along the Z axis by a given amount, for internal gearing shifts down. With the **Teeth Number** model, the field is calculated.

Backlash. The option is available when using the design standard DIN ISO 21771 (the tab Geometric Parameters, the **Calculation**, **Standard** field). Allows you to set the nominal normal backlash.

Teeth Number. Two input fields are available: the number of pinion teeth and the number of gear teeth. Only positive integer values can be entered. This must be taken into account when parameterizing. The number of teeth of the internal gear is also given by a positive integer - this should be remembered by users who are guided by standards and methods, where the number of teeth of the inner gear is given by a negative integer. In the model **Teeth Number** is selected, both fields are always available for input: the number of teeth of the pinion and the number of teeth of the gear. In the **Center Distance** model, the input fields for the number of teeth are closed, but when you check the box next to the field, you can open it for input. The system allows you to enter the number of teeth only on one of the objects: either on a pinion or on a gear, on the second object the number of teeth will be calculated automatically.

Facewidth. Two input fields are available: pinion thickness and gear thickness.

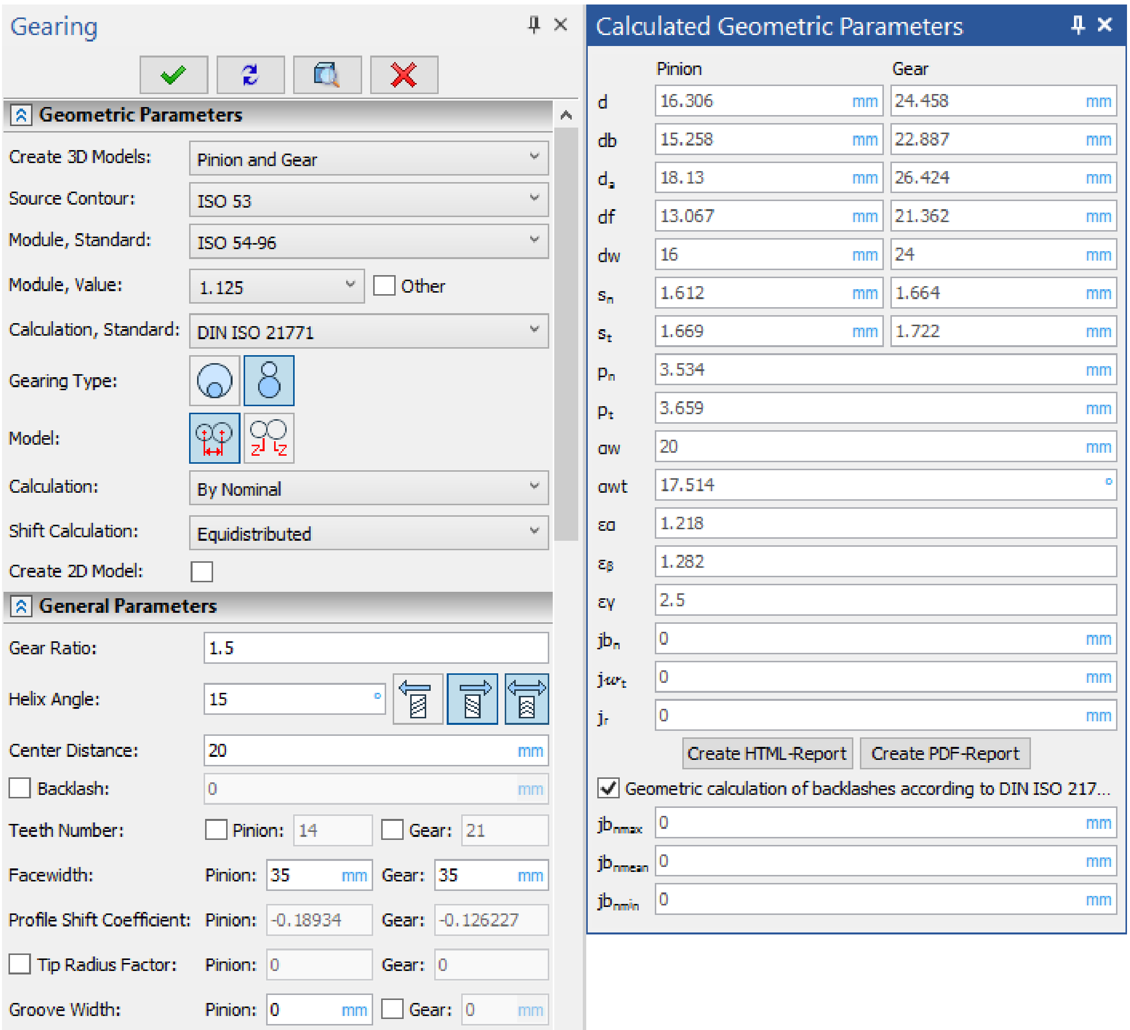
Profile Shift Coefficient. Two input fields are available: pinion shift and gear shift. The shift is specified as a proportional factor to the modulus. Entering shifts is available only for the variant of calculating the shifts of the **Manually** shifts. More details about the options for calculating shifts can be found in the section Shift Calculation.

Tip Radius Factor. The option is available when using the calculation standard DIN ISO 21771.

Groove Width. The field becomes available if the double helical gear creating is selected. Sets the width of the groove on the chevron pinion and gear. For a gear, the default groove width is equal to the width of the groove on the pinion. When a flag is checked next to the field, it will be possible to enter an independent value for the width of the groove on the gear.

Calculated Geometric Parameters

The tab contains only calculated information fields and does not contain fields for entering values. This tab displays the main geometric calculation parameters of gears and gearing, such as pitch diameters and base diameters, tooth thickness, overlap coefficients, backlash size. All values are calculated "on the fly". the fields instantly display the calculated values according to the parameters entered by the user. The tab has a special button that opens the tab in a separate window. Due to this, the window with the main design parameters can be placed next to the input area of the gearing parameters and the influence of the input parameters can be monitored.



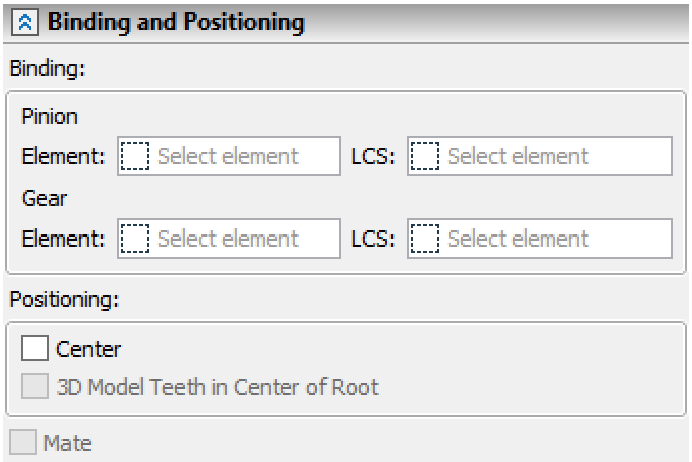
All tab fields have hints with the full name of the parameter.

For geometric analysis according to DIN ISO 21771, it is convenient to use the option **Geometric calculation of backlashes according to DIN ISO 21771** to evaluate the influence of the introduced deviations on the backlashes.

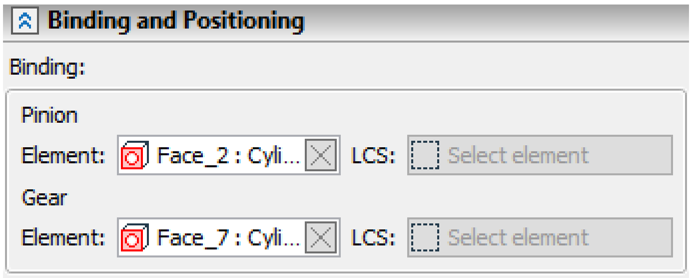
At the bottom of the tab there are buttons for creating reports in *.pdf and *.html formats.

Binding and Positioning

By default, the center of the pinion will be at the origin of the global coordinate system, and the center of the gear will be shift along the Z axis of the global coordinate system by the center distance. To change the orientation of the transfer or its binding to the models already existing in the document, use the **Binding and Positioning** tab.

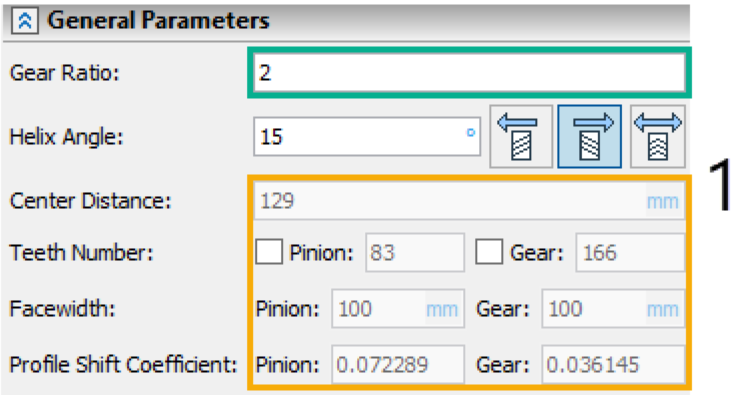


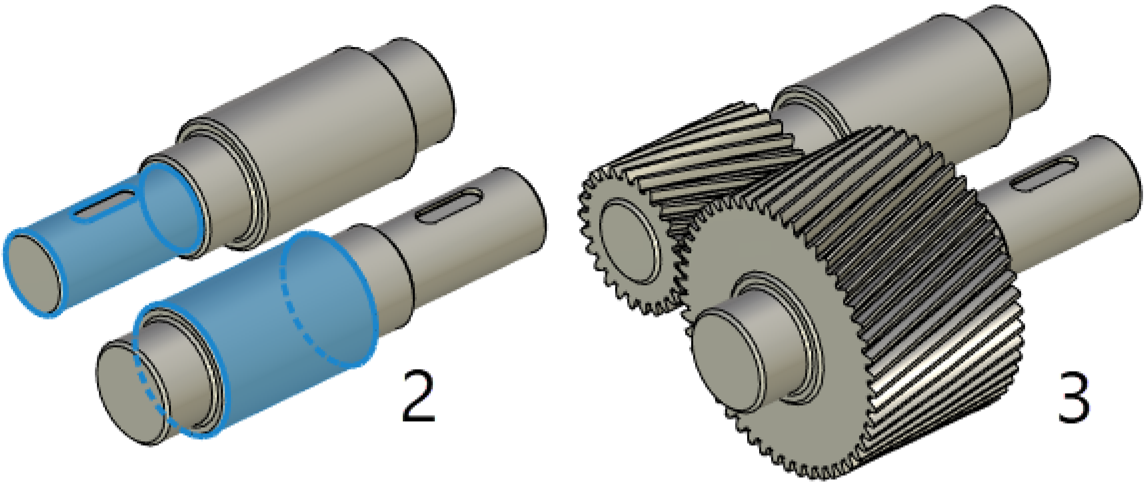
The sequence of selection of the pinion and gear binding fields (by elements or by LCS) is automatically filtered, suggesting one of the possible binding options.



The following binding options are available:

- Binding on two shafts: on cylindrical faces, on transverse faces or mixed;
- Binding by pinion: cylindrical face, end face or LCS;
- Binding by gear: cylindrical face, end face or LCS;

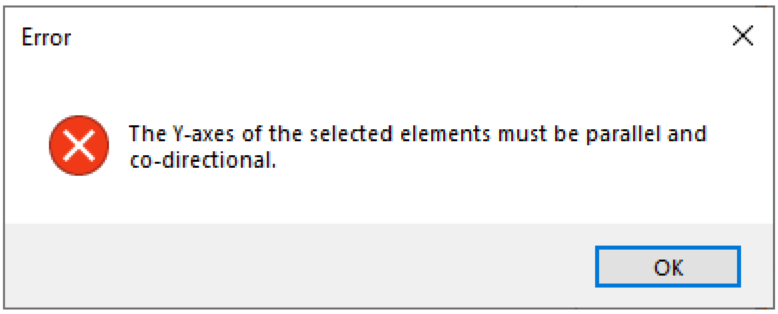




When the binding is carried out, the alignment is checked, the parameters depending on the given location of the pinion and gear are automatically calculated, and the gear itself is always geared. So, for example, by selecting two cylindrical faces (2), you can set the center distance and the number of teeth on the gears, which will be calculated from the values of the modulus and gear ratio (1).
When selecting elements for binding, it is necessary that their axes be parallel, and also in the case of selecting the LCS for binding, observe the parallelism condition for the Y axes of the LCS.

> When binding along two cylindrical faces, it is necessary that the bodies or fragments to which they belong are defined by the LCS with co-directional axes along the axis of the cylindrical faces.

Otherwise, the system will diagnose these cylindrical faces as not co-directional, and a warning will be issued about this.

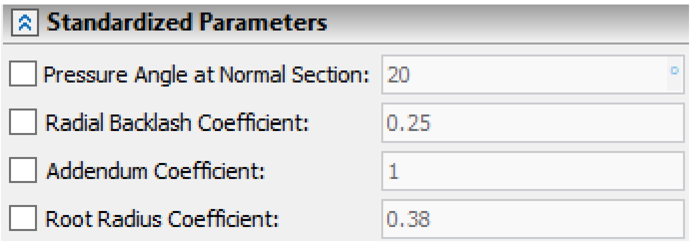


When working on the **Teeth Number** model, only one of the gearing elements will be available to bind.
The **Center** option allows you to align the gears in the center relative to each other if they have different facewidths. The option is not available for double helical gears, because they are always

centered. The option flag is automatically set when creating a gearing if the thickness of the pinion and gear are different.
The option **3D Model Teeth in Center of Root** becomes available and is automatically activated if the backlash is greater than zero.
The **Mate** option allows you to automatically mate **Gear - Gear** between the gears in gearing. The option is closed until the pinion and gear are attached to one of the objects available for binding. For mates to work correctly in the **Move** mode, you need to fix the bodies to which the LCSs of the gears are attached. The **Mate** option will not be available when working on the **Teeth Number** model.

Standardized Parameters

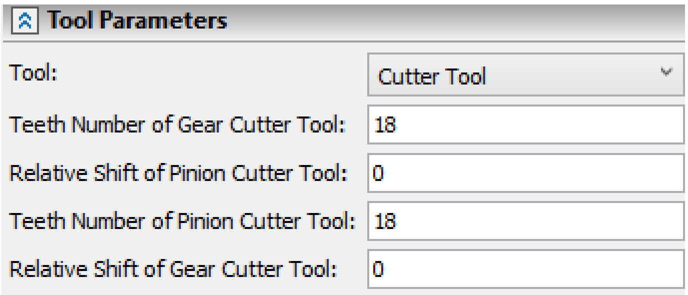
This tab displays the standardized parameters of the source contour.



The list, input method and availability of input depend on the selected standard in the **Source Contour** field (Geometric Parameters section). Setting the flag next to the parameter input field makes it possible to enter a value different from the standardized one. All coefficients determine the geometric parameters of the source contour in proportion to the modulus.

Tool Parameters

The tool type determines the shape of the transition curve and the calculation of the root diameter.



The tool parameters depend on the selected tool type. The following tool types are available:

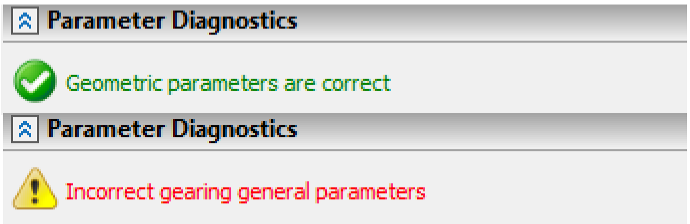
- **Milling Cutter Tool**;
- **Rack Tool**;
- **Cutter Tool**.

The choice of the tool type is carried out from the list. For internal gearing, only **Cutter Tool** is available, which is explained by the impossibility of obtaining an internal wheel with tools such as **Milling Cutter Tool** or **Rack Tool**.
Milling Cutter Tool. The tool simulates cutting with a sharp-edged rail, which corresponds to the operation of a hob cutter or knurling a rail without rounding. The geometry of the tool is entirely determined by the source contour, no additional tool parameters need to be entered.
Rack Tool. The tool simulates knurling with a rounded edge. The option is the **Rounding Radius Factor**. The factor sets the rounding on the rail in proportion to the modulus.
Cutter Tool. The tool simulates cutting with a cutter. The input fields for the number of teeth of the cutter and the shift on the cutter are available. The shift is set as a factor of proportionality to the modulus and determines the shift of the conditional rack making the cutter. The source contour of the

gear and cutter tool teeth is the same. The specified cutter tool parameters can be entered separately for the pinion and gear.

Parameter Diagnostics

On this tab, the system informs about possible errors when entering parameters. If the entered parameters are correct, the message **Geometric parameters are correct** is displayed, but there are no messages in the **Diagnostics** window. If the entered parameters are invalid or acceptable with a note, a corresponding warning will be issued, duplicated in the **Diagnostics** window.



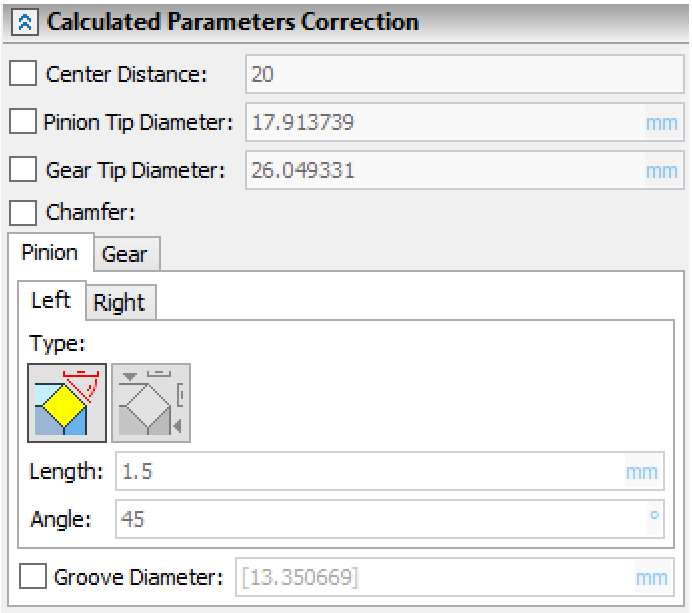
The following problems are automatically diagnosed:

- Tip diameter of the internal gear is less than the base diameter, which makes it necessary to adjust: the message - **Tip diameter is smaller than the base diameter, the value has been corrected** (gearing is available).
- Incorrect relationship of the center distance, shifts and the number of teeth, leading to the impossibility of calculating the geometry of the gears: the message - **Incorrect gearing general parameters** (gearing is not available).
- Intersection of involutes of opposite sides of the tooth: the message - **Impermissible tooth sharpening** (gearing is not available).
- Intersection of transition surfaces of opposite sides of the root: the message - **Impermissible tool shift** (gearing is not available).
- Checking if the selected tool can be used: the message - **Tool or tool parameters are impermissible** (gearing is not available).
- Checking for the correctness of the custom value of the tip diameter of the pinion and gear, if the custom value completely "cuts" the involute, a message will be displayed: **Impermissible pinion tip diameter** (gearing is not available).
- Checking if the chamfer size is entered correctly. The chamfer should not completely cut off the crown, but for the double helical gears it should cut into the groove: the message - **Impermissible chamfer size** (gearing is not available).

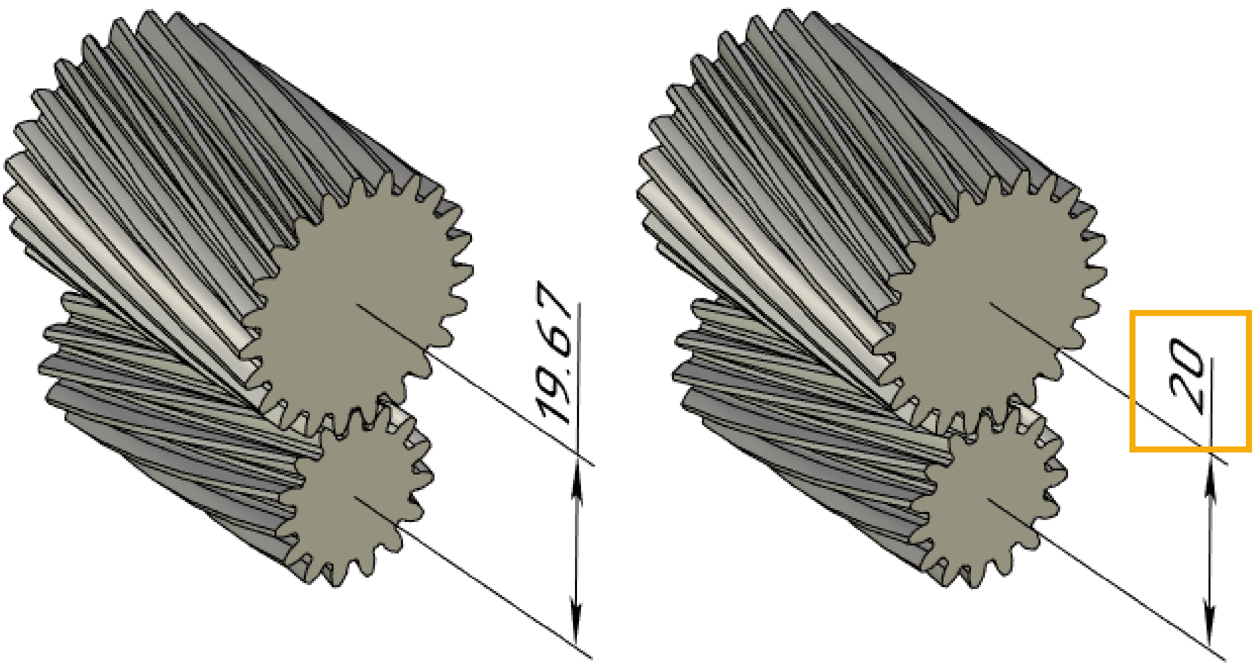
In addition, this tab displays information about the correction of the diameter of the measuring rollers (balls) used in the standardized analysis of geometry: the message - **Diameter of pinion measuring rollers (balls) is invalid and has been corrected.**

Calculated Parameters Correction

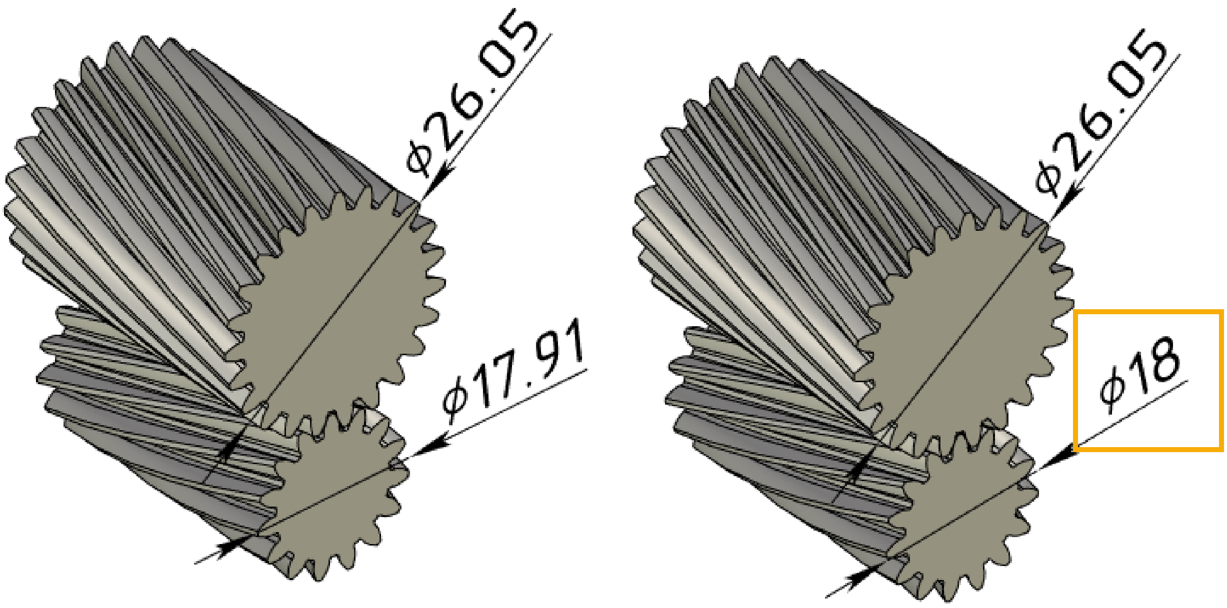
Correction of the calculated parameters is available, which, due to the manufacturing technology and assembly accuracy, cannot be performed with the calculated accuracy.



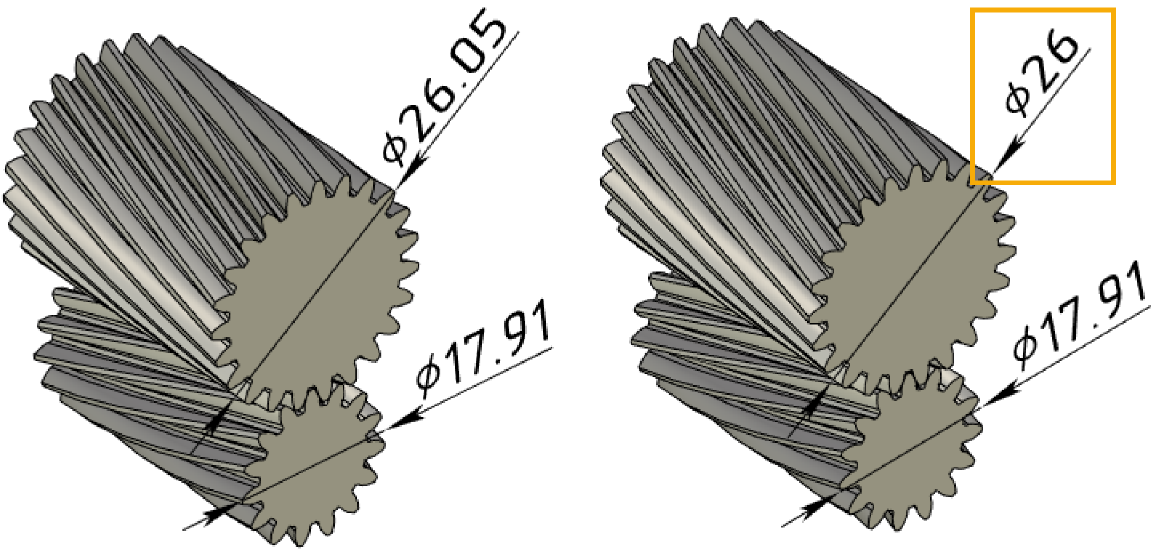
Center Distance. When you set the flag, you can correct the center distance. For external gearing, the center distance can be increased relative to the calculated distance, and for internal gearing, it can be reduced.



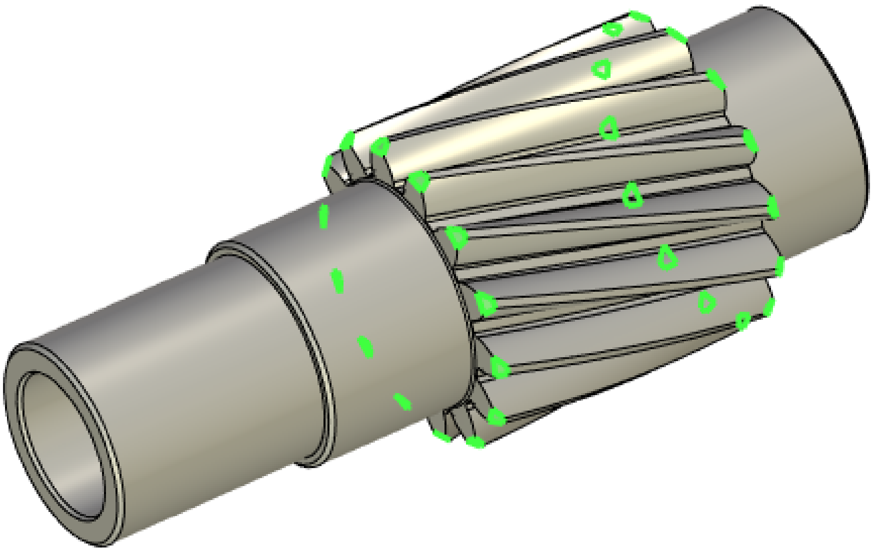
Pinion Tip Diameter. When you set the flag, you can correct the value of the pinion tip diameter.



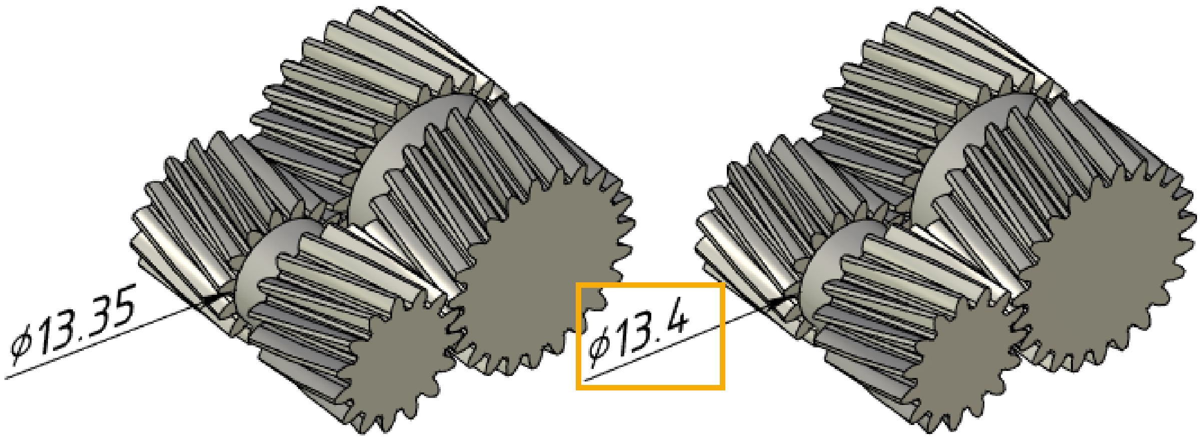
Gear Tip Diameter. When you set the flag, you can correct the value of the gear tip diameter.



Chamfer. When the flag is set, chamfers will be added to the ends of the pinion and gear models. Entering the chamfer parameters of the pinion and gear are divided into the corresponding tabs. There are two ways to enter the chamfer: **Length - Angle** and **Offsets**. For the gear, the **Same as Pinion** option is enabled by default. If the option flag is unchecked for the gear, you can enter chamfer parameters other than the pinion chamfer parameters.



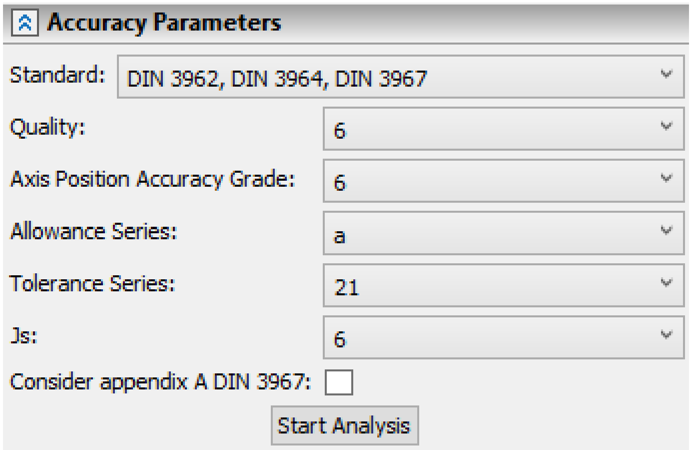
Groove Diameter. By default, the groove diameter is one-tenth of the module smaller (larger if the internal gear is) than the diameter of the roots. When you set the flag next to the field, you will be able to correct the groove diameter.



Accuracy Parameters

On this tab, you can calculate the tolerances and deviations according to the following standards:

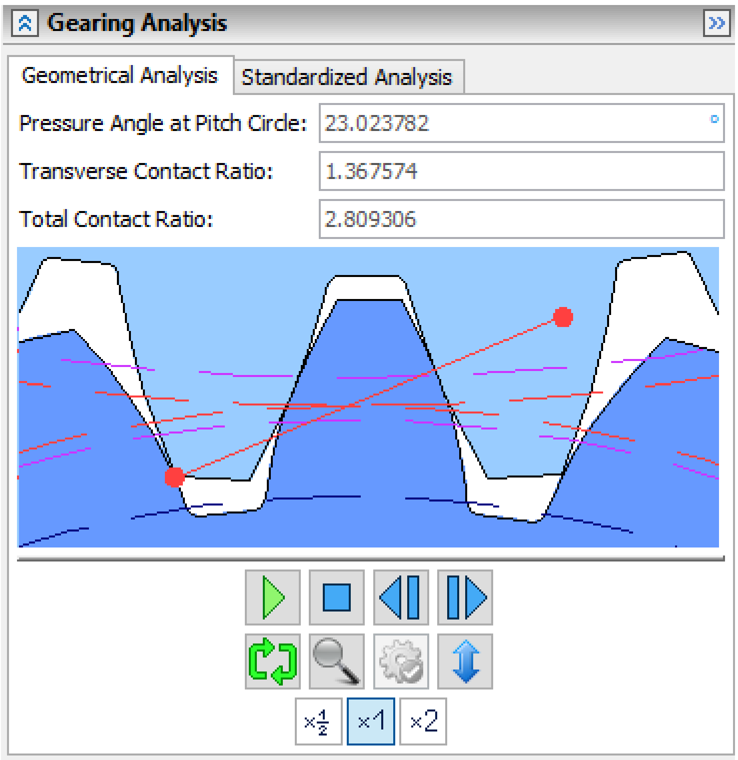
- ISO 1328;
- DIN 3962, DIN 3964, DIN 3967;
- GOST 1643-81;



For each standard, it is necessary to specify the quality of accuracy and other standards for the manufacture and mating of gears, according to the initial data provided by the corresponding standard.
At the bottom is the **Start Analysis** button, which opens a modal window that shows all the calculated parameters of the tolerances and deviations provided by the standard. For each parameter, there is a symbol and a hint indicating the full name of the parameter according to the standard. At the bottom of the modal window, there are buttons for creating reports in *.pdf and *.html formats.
Manual input is available for all calculated values. The manual input of the deviations affecting the strength calculation is synchronized with the manual input of the same parameters in the strength calculation dialog according to the corresponding standard: ISO 1328-ISO 6336, DIN 3962-DIN 3990, GOST 1643-81-GOST 21354,

Gearing Analysis

Gearing analysis involves a geometric analysis of both the resulting gear pair as a whole, and a separate evaluation of the geometry of the pinion and gear. Gearing analysis is divided into visual analysis, using animation, located on the **Geometrical Analysis** tab, and analysis according to the selected standard, which is located on the **Standardized Analysis** tab.



The tab has a special button that opens the tab in a separate window. This allows you to place the gearing analysis window next to the gearing parameter input area and monitor the impact of the input parameters. In addition, working with this tab in a separate window makes it convenient to view the gearing animation. The animation window can be stretched or compressed to the required size without changing the size of the other tabs in the dialog.

Topics in this section:

- Geometrical Analysis
- Standardized Analysis

Geometrical Analysis

You can move a tab to a separate window by using the icon in the upper-most corner of the tab. Working in a separate window is convenient when working with animation. The animation window can be scaled with the mouse wheel.
The general characteristics view fields are available on the geometrical analysis tab:

- **Pressure Angle at Pitch Circle;**
- **Transverse Contact Ratio;**
- **Total Contact Ratio.**

An animation of the gearing is also available. Animation control is carried out by special pictograms.

Start Animation. The rotation of the gearing animation. The gear rotates clockwise. The rotation speed can be changed using the icons described below.

Stop Animation. The animation stops, the gearing returns to the specified position. The default is the specified position at the start of the gearing of the extreme section. You can control the specified position on the Stress Analysis Plots tab using the **Selection Position in Gearing** slider, or using the **Step Backward** and **Step Forward** buttons.

Step Backward. Toggles the **Selection of Transverse Section** slider (see Stress Analysis Plots section) one position to the left.

Step Forward. Toggles the **Selection of Transverse Section** slider one position to the right.

The animation steps, like the slider, are limited to the gearing segment in the first section.

Refresh gearing animation and graphic window. To update the animation according to the changed parameters or according to the selected transverse section defined by the **Selection of Transverse Section** slider (see the Stress Analysis Plots section), click this button. The button has three states. - the animation window is up to date. - the animation window is not up-to-date and can be updated. - the entered parameters are not correct, the animation window cannot be updated until the error in entering the parameters is corrected.

Zoom. By clicking this icon, you can zoom the animation to the scale of the gearing line or show the gears in gearing completely. You can also change the scale of the animation display with the mouse wheel.

No Intersections. If there is this icon under the animation window, there are no intersections between the gears.

Intersections. If this icon is under the animation window, intersections appear between the gears when turning (in other words, the gearing will jam). Clicking on this icon moves the animation to the position of the beginning of the intersection, indicating the intersection point. In the event of multiple intersections, pressing the icon again will sequentially transfer the animation from one beginning of one intersection to the next.

 **Expand View**. By clicking this icon, you can increase or decrease the size of the animation window.

 **Slow Down Animation 2x**. To slow down the rotation of the gear twice, you need to click this icon and click **Start Animation** (you can change the animation speed during the animation).

 **Normal Speed.** Normal speed is calculated according to RPM on the Operating Characteristics tab.

 **Speed Up Animation 2x**. To speed up the rotation of the gear twice, you need to click this icon.

The animation window is closely interconnected with the work on the Stress Analysis Plots tab, allowing you to use a two-dimensional model to understand the stages of helical or spur gearing. Animation, stress analysis graphs and sliders **Selection of Transverse Section** and **Selection of Position in Gearing** show the history of loading of the gear tooth and the gear tooth in the gearing (taking into account the gearing of other pairs of teeth). The contact of the considered pair of gearing in the animation is indicated by a red circle on the segment of the gearing. In extreme positions, the point of contact of the teeth coincides with the beginning and end of the gearing segment.

The animation shows the working, pitch and base diameters of the pinion and gear.

Standardized Analysis

The following geometric analysis options are available:

- **DIN ISO 21771 (optionally taking into account DIN 21773)**;
- **GOST 16532**;
- **Numerical Calculation**.

DIN ISO 21771. When this standard is selected, the diameter of the rollers (balls) is determined according to the calculation specified in DIN ISO 21771. The calculated diameter of the rollers (balls) will be indicated in the standardized geometric analysis window. When choosing this standard, before starting the calculation, you can indicate the need to take into account the specifics of DIN 21773 by setting the flag.

According to Appendix A of DIN ISO 21771, the designer himself determines the deviations in the thickness of the tooth, the center distance and the tip diameters. It also determines the diameter by which the thickness of the tooth is controlled. All specified fields are available for input when the flag is set. The default values for the deviation fields are set based on the theoretical zero backlash. When the DIN 21773 counting flag is set, the clearance calculation according to Appendix A of DIN ISO 21771 is not carried out. In this case, only the diameter input field is available, by which the tooth thickness is controlled.

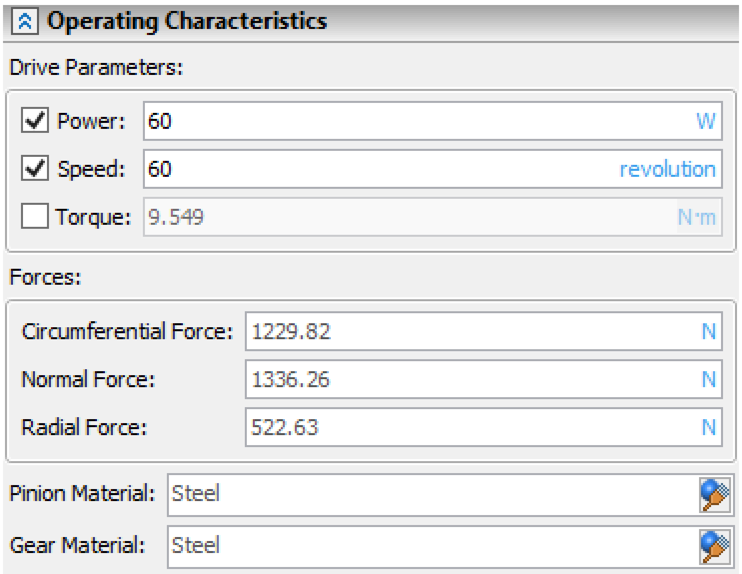
GOST 16532. When choosing this standard, before starting the calculation, you can enter the diameters of the measuring rollers (balls) for the pinion and gear. By default, the diameter of the rollers (balls) will be selected according to the assortment in accordance with GOST 2475-88. The matched diameter is shown in the corresponding fields. To enter a custom value, you must put a flag next to the field name.

Numerical Calculation. In contrast to the calculations according to the standard, where all the control parameters are determined by the expressions given in the specified standards and according to the entered gearing parameters, in the numerical calculation the control parameters are determined by the mathematical model of the gears without using the formulas of the standards. By analogy with GOST 16532, before starting the calculation, you can enter the diameters of the measuring rollers (balls) for the pinion and gear.

To start the analysis according to the selected option, click the **Start Analysis** button. The calculation takes some time, during which a modal window for waiting for the result appears with the option of stopping the calculation.
The calculation results are displayed in the **Geometrical Analysis** modal window. All control parameters are labeled according to the standard and provided with prompts with the name according to the standard. All parameters for which the standard specifies the limit are compared with it, and in case of exceeding the range they will be marked in red.
For calculations according to DIN ISO 21771 (and also taking into account DIN 21773) the window is divided into two tabs - **Geometrical Analysis** and **Accuracy Parameters**. This is due to the fact that these standards regulate deviations for control dimensions, shifts and backlashes. If calculated in accordance with DIN 21773, backlash deviations shall be calculated in accordance with DIN 3967.
For numerical calculation and calculation according to DIN ISO 21771, a choice of the number of teeth for measuring the base tangent length is available. When choosing a new number of teeth in the measurement of the base tangent length, it is necessary to recalculate the results using the corresponding button at the bottom of the window. In the numerical calculation, the limits on the number of teeth in the measurement are determined by the possibility of taking measurements according to the length and curvature of the involute and taking into account its cut with the selected tool. In the numerical calculation, the diameter limits of the measuring roller (ball) are calculated based on the ability to find points of contact on the surface of the teeth. The thickness limits are determined according to the possibility of measuring along the base tangent length for a given number of teeth.
At the bottom of the **Geometrical Analysis** window, there are buttons **Create HTML-Report** and **Create PDF-Report**, by pressing which the data on geometric analysis will be summarized in the report tables. For numerical calculation and calculation according to DIN ISO 21771, a button to recalculate the result is also available.

Operating Characteristics

On this tab, the operating characteristics of the gearing and the material of the gears are set.

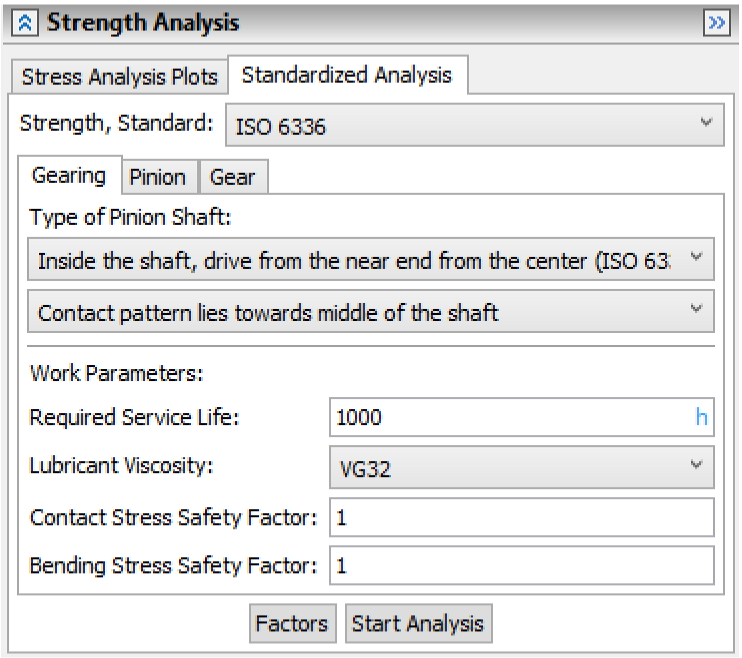


The fields for entering the values of power, speed and torque are interconnected: you can enter any two of the three parameters, the third will be calculated automatically. The fields available for input are determined by the user by setting a flag next to the name of the corresponding field.
The **Power**, **Speed** and **Torque** fields define the parameters of the driving link.

The **Circumferential Force**, **Normal Force**, and **Radial Force** fields are read-only. These are calculated fields that "on the fly" change the values according to the parameters entered by the user. The fields are necessary to understand the load on the gearing.
Material can be set separately for pinion and gear. The choice of material is made from the standard window for viewing materials, which is invoked by clicking the icon on the right side of the material input field. The material determines the values of Young's modulus and Poisson's ratio required for strength analysis calculations.

Strength Analysis

Strength analysis can be carried out as a qualitative assessment of the effect of the specified parameters on the stresses arising on the pinion and gear teeth, implemented on the **Stress Analysis Plots** tab, and a quantitative assessment of the strength characteristics calculated according to the selected standard, implemented on the **Standardized Analysis** tab.



The tab has a special button that opens the tab in a separate window. Thanks to this, the strength analysis window can be stretched or shrunk to the required size without resizing the rest of the dialog tabs. This makes working with stress analysis graphs much more convenient.

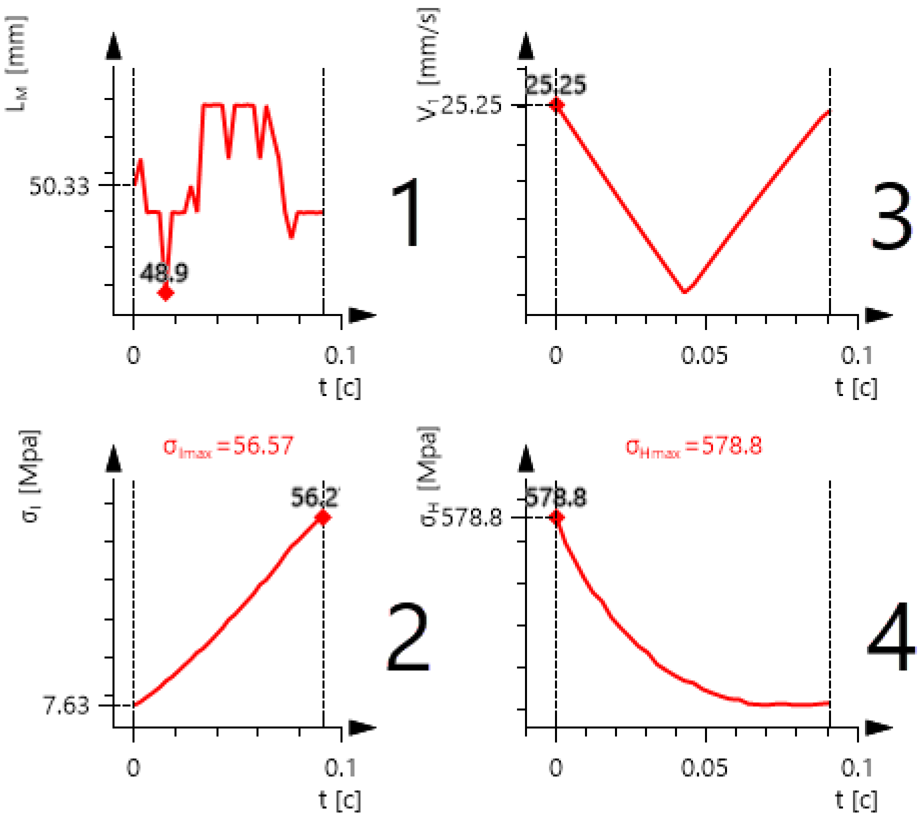
Topics in this section:

- Stress Analysis Plots
- Standardized Analysis

Stress Analysis Plots

On this tab, four graphs are calculated, allowing you to evaluate the influence of all entered parameters, from the geometry of the source contour and the selected tool, to the material of the pinion and gear, on the nature of the load of the gearing:

- Plot of length of the gearing line (marked with number 1 in the figure);
- Plot of slip rate, or slip ratio (marked with number 2 in the figure);
- Bending stress plot (marked with number 3 in the figure);
- Contact stress plot (marked with number 4 in the figure).



All plots are plotted against time, calculated according to the gearing of one pair of teeth, essentially arbitrary, in a typical transverse section, which can also be arbitrary, but for clarity, this is the end face of the gear. Zero corresponds to the beginning of the gearing of the pair under consideration, the final value of time - corresponds to the exit from the gearing of this pair of teeth. The current position of the considered pair of teeth is indicated on the plots with a cross and corresponds to the position of the teeth in the animation window. Plots can be built for any design transverse section: to do this, you need to use the **Selection of Transverse Section** slider. To create the effect of viewing the stress state along the length of the oblique tooth, all plots of all calculated sections will be built in the time coordinates of the first transverse section of the gear.

To update the plots according to the entered parameters, or change the plot display settings, press the button (the button duplicates the animation update button in the animation window).

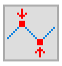
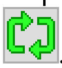
Pressing on the plot line will show the coordinates of the point closest to the cursor.

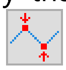
All plots show extremes: for the plot of the length of the gearing line, the minimum, for the others, the maximum. For stress plots, in addition to the extremum of the current section, the maximum value for all calculated transverse sections is also indicated.

Plot of length of the gearing line. Shows how the total length of the line of gearing changes across all contacting pairs of teeth, depending on the current position in gearing of the pair of teeth in question in a typical transverse section.

By default, we assume that all possible pairs of teeth are geared along the entire possible length (according to the ideal geometric model). If it is necessary to take into account that the fit of the teeth in contact is not accurate, then you can reduce the length of the gearing line as a percentage (from 10 to 100). The option of calculating one pair of teeth is also available. To do this, in the **Contact Calculation Type** list, change the option **All Teeth** to the option **One Tooth**. This option is particularly relevant when calculating a low-precision spur gear, for which it is customary to calculate based on the fact that only one pair of teeth is in contact. In this case, the calculation is carried out only on the contact line of the pair of teeth considered in the animation.

Plot of slip rate, or slip ratio. Shows how the slip speed changes (in mm/s) depending on the position of the teeth in the gearing. You can also estimate the slippage by its relative value (slip coefficient). To do this, in the **Plot View** list, change the option of the **Slip Speed** to the option of the **Slip Coefficient**. The graph is located above the contact stress graph, because the combination of the slip speed and the amount of contact stress affects the surface wear of the teeth.

Bending stress plot. The calculation of bending stresses is made for the smallest longitudinal section of the tooth in case there is a cutting. If there is no undercut, and the transition surface does not have a section with the smallest longitudinal section of the tooth, then the calculation is made by the transverse section at the junction of the involute to the base of the tooth. The bending moment on the tooth is calculated according to the current position of the gearing point in each calculated transverse section and according to the specified performance characteristics. The number of calculated transverse sections is set by the user in the **Number of Section Calculation Steps** located next to the **Selection of Transverse Section** slider. In each case, it is determined at which position of the gearing and in which of the calculated transverse sections the greatest bending moment will occur. To display the maximum, there is a special button  located under the plot. When it is pressed, the **Selection of Transverse Section** and **Selection of Position in Gearing** sliders will be set to the position where the greatest bending stresses occur on the pinion or gear. Plots will be plotted for the set transverse section, and the crosshair will show the current position in the gearing. The animation window will also display a picture corresponding to the plots both by the transverse section and the position in the gearing. The bending stress plot for the pinion and gear is different. To change the displayed plot for the pinion to the plot for the gear or vice versa, use the **Pinion** and **Gear** buttons and then click the update button .

Contact stress plot. Contact stresses are calculated using the model of contact between two cylinders, where the radii of the cylinders are determined by the current radii of the involutes of the contacting teeth and according to the position of the gearing point in each calculated transverse section. The model takes into account Young's modulus and Poisson's ratio specified in the material parameters. The normal contact force is calculated according to the current position of the point of gearing in each calculated transverse section and according to the specified performance characteristics. As well as for the plot of bending stresses, it is possible to display the position of gearing at which the maximum contact stresses occurs, for this you need to use the  button below the plot.

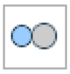
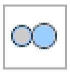
Above the plots there are the following controls and display settings, some of which have already been briefly described earlier.

Contact Quality. A slider that changes the percentage of teeth adherence along the contact lines from the maximum possible 100%, when the contact line for all teeth is considered completely to a minimum of 10% of the maximum length of the contact line.

Contact Calculation Type. The list contains two options **All Teeth** and **Single Tooth**. The first option is universal, the second should be used to calculate the spur gearing with low manufacturing accuracy.

Number of Section Calculation Steps. The field indicates the number of calculated transverse sections. The first and the last will be the transverse surfaces of the gears. The rest will be equally spaced along the length. An increase in the number of calculated sections significantly slows down the work of the module. Reduction - makes the calculation rough, you cannot set the number of sections less than 20.

Selection of Transverse Section. Slider that changes the value from 0 to a user-specified value for the number of steps for sections. Affects the display of all plots, except for the plot of the length of the contact line, since viewing the selected section will not change the length of the contact line. Affects the display of animation (if the gearing is not spur), because for a helical gear in each subsequent section, the contact point will be shifted to the right or left (depending on the direction of the twist of

the teeth). For all sections except the initial one, there will be some area of no contact at the considered interval of rotation for the considered pair of teeth. On the plots of stresses and slippage, the time during which there is no contact between the considered teeth in a given section will be displayed as zero values. With some combinations of the angle of twist and the thickness of the gear during the considered period of time, at sufficiently distant transverse sections, there will be no contact at all between the teeth under consideration. This means that the twist of the teeth has reached an angle greater than or equal to the angle of rotation at which there is gearing.
To the right of the section selection slider there is a field showing the number of the selected section.
Number of Section Calculation Steps. The field specifies the number of points of calculation along the line of contact, respectively, and the number of points on each plot. The first and last points correspond to the gearing and shift of the first section. The rest with equal steps will be distributed over the gearing time. Increasing the number of calculated points slows down the module. Decrease - makes the calculation rough, you cannot set the number of steps less than 20.
Selection of Position in Gearing. Slider that changes the value from 0 to a user-specified value for the number of gearing calculation steps. Affects the rotation of gears in gearing in the animation window. On plots, affects the position of the crosshair indicating the current time in the gearing.
To the right of the gearing position selection slider there is a field showing the number of the calculated pitch.
The **Pinion**  button and the **Gear**  button. Determine the displayed bending stress plot.
Button **Refresh gearing animation window and plots window**. Refreshes plots and animation according to the specified settings and parameters.
Precision of Solution at Border. The option is created to eliminate inaccuracies when finding stresses and the contact line near the extreme sections. Allows you to eliminate incorrect extrema, but slows down the module. If it is necessary to clarify the calculation near the extreme sections, you need to change the accuracy option from **Normal** to **Increased**.
Plot View. Slippage can be estimated on the plot by speed in mm/s, for this you need to select the **Slip Speed** option, or by the relative value of the slip coefficient, in this case, you need to select the **Slip Coefficient** option.

Standardized Analysis

The standardized analysis allows the calculation of strength and durability according to the selected standard.
The calculation can be performed according to the following standards:

- ISO 6336;
- DIN 3990.
- GOST 21354;

The dialogs for entering parameters for the calculation according to ISO 6336 and DIN 3990 are practically identical due to the fact that the calculation methods described in these standards are similar. The dialog for entering parameters for calculation in accordance with GOST differs quite strongly from the dialog for entering parameters for ISO and DIN.
In the dialogs of all standards, some of the parameters related to the pinion and gear must be entered on the corresponding tabs **Pinion** and **Gear**. On the Gearing tab, you must enter the general parameters for the gearing. When entering the hardness, the unit can be selected: HB, HRC or HV. The choice is made from the list. Hints are provided for parameters indicated by symbols. On the **Gear** tab, the option **Same as Pinion** is available: when the flag is set, all parameters entered for the pinion will be duplicated for the gear. On the **Gearing** tab, in the **Maximal Torque** field (when calculating according to GOST), the moment calculated or specified by the user on the Operating Characteristics

tab is set by default. By setting the checkbox next to the field name, you can enter a different value for the maximum moment, known to the user from the load plot of the projected gearing.

Factors. The button opens the modal window of the same name, where the calculated values of all factors and some other parameters of the calculation methods of the standard are displayed. As in the case with the main dialog of the standardized strength analysis, the number of coefficients and additional parameters for calculating GOST differ from similar DIN and ISO.

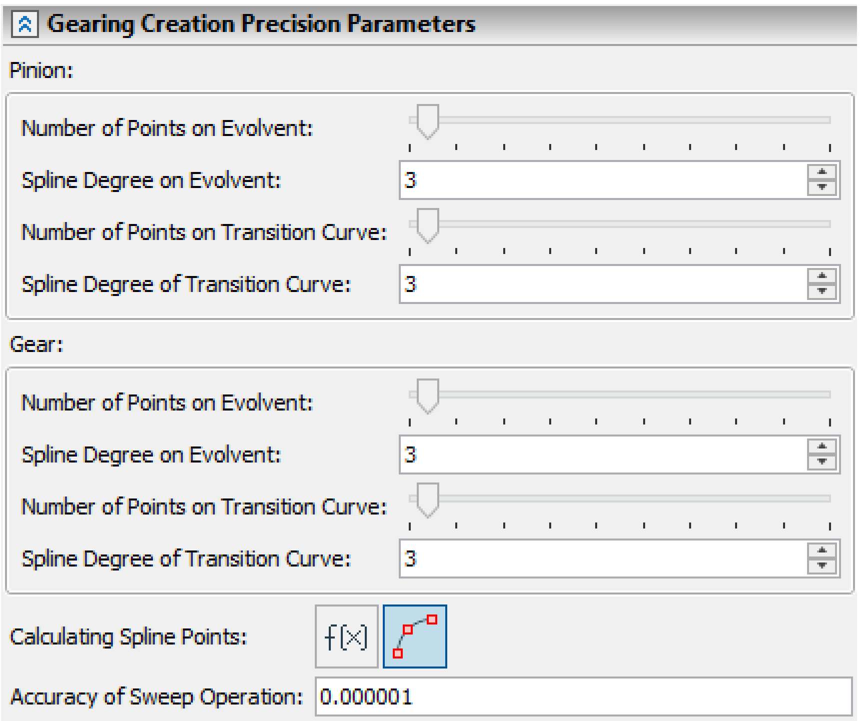
There are two tabs in the window that divide the factors into those that are used in the calculation of contact stresses and those that are used in the calculation of bending stresses. For all factors that, according to the logic of the calculation methods of the standard, can be changed - a flag setting is available that opens manual entry. If the parameter being changed affects other calculated values shown in this window, they will change their values "on the fly". For all calculated parameters, which are indicated by symbols in the window, there are hints containing the name of the parameter according to the standard.

Start Analysis. The button starts the calculation, the results of which are displayed in the modal window. According to the listed standards, verification is carried out according to the list of control parameters separately for contact and bending stresses. Accordingly, the window is divided into two tabs. The controlled parameter and its limit are indicated on each tab. The limit in some cases is a calculated value according to the methods of the standard, and in some it is set by the user, for example, the required resource. If the monitored parameter is outside the permissible limit, it will be highlighted in red. For all monitored parameters, indicated by symbols in the window, there are hints containing the name of the parameter according to the standard.

At the bottom of the results window, the buttons **Create HTML-report** and **Create PDF-report** are available, when you click them, the strength analysis data and the specified calculation parameters will be summarized in the report tables.

Gearing Creation Precision Parameters

The involute in the tooth transverse section and the transition curve are approximated by a spline. To improve the approximation accuracy for the user, approximation settings are available on the **Gearing Creation Precision Parameters** tab. Parameters can be set separately for pinion and gear.



Number of Points on Evolvent. Slider defining the number of points of the spline approximating the evolvent. You can set a value from 5 to 50.

Spline Degree on Evolvent. The field in which you can set the degree of the spline approximating the evolvent. Valid values are from 1 to 5. The unit corresponds to a spline with linear interpolation.

Number of Points on Transition Curve. Slider that determines the number of points of the spline that approximates the transition curve. You can set a value from 5 to 50.

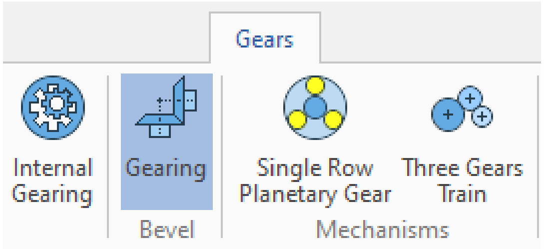
Spline Degree of Transition Curve. A field in which you can set the degree of the spline that approximates the transition curve. Valid values are from 1 to 5. The unit corresponds to a linearly interpolated spline.

Calculation Spline Points. DIN ISO 21771 and GOST 16532 standards have expressions for determining the thickness of a tooth in an arbitrary section. Calculation of the evolvent by expressions from the standards will correspond to the choice of the **By Standard** option. If the source contour was selected according to ISO or DIN, then the formulas of the DIN ISO 21771 standard will be used. If the source contour was selected according to GOST, then the evolvent calculation will be using the GOST 16532 standard. The evolvent can also be calculated using the formulas known from mathematical theory. This method of calculation will correspond to the choice of the **Analytically** option.

Accuracy of Sweep Operation. Pinion and gear bodies are obtained by applying the **Sweep** operation. The accuracy of this operation is configurable. Decreasing this parameter will increase the accuracy of the **Sweep** operation.

Bevel Gearing

To create a bevel gear, use the **Gearing** command in the **Bevel** group.



Construction and calculation of bevel gearing is possible according to the following standards at the user's choice: ISO 23509, DIN 3971, GOST 19624-74 and GOST 19326-73. These standards regulate the calculation of the geometry and the source contour. The list of parameters of the source contour and their default values correspond to the above standards, or the standards referred to in the standards. In other words, the choice of the geometry calculation standard automatically determines the corresponding standard of the source contour. This distinguishes the calculation of bevel gears from cylindrical ones, where the source contour does not depend on the selected calculation standard. This is due to the fact that the ISO 23509 standard describes the source contour in the general case, including for hypoid gearing, while in GOST and DIN the description of the source contour is less universal. In the future, if necessary, the separation of the calculation standards and the source contour can also be added.

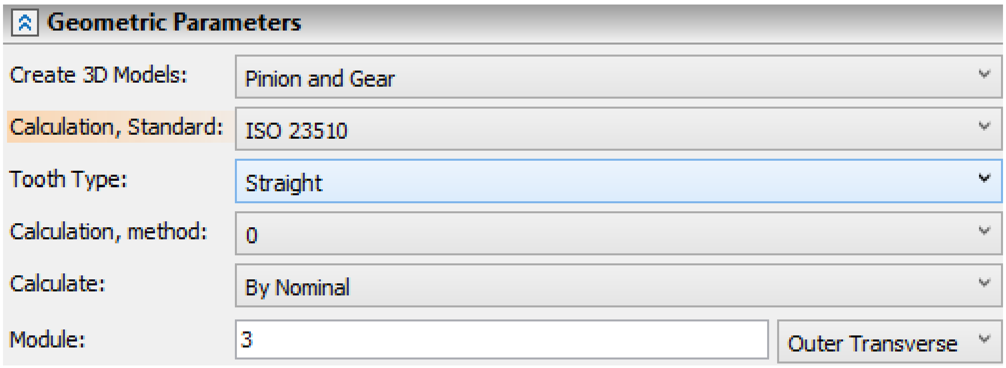
The choice of the calculation standard also determines the available types of teeth (straight, spiral, etc.), options for their shape and tool parameters. Some similar parameters, such as profile shift coefficient, thickness modification coefficient and some others in different standards, are located on different tabs. This is due to the fact that the principle of presentation of the calculation methodology and the grouping of parameters by chapters and tables in the standards under consideration is different. In the future, if necessary, the dialog can be slightly changed towards the unification of fields in different standards.

Topics in this section:

- Geometric Parameters
- General Parameters
- Calculated Geometric Parameters
- Binding and Positioning
- Standardized Parameters
- Calculated Parameters Correction
- Tool Parameters
- Accuracy Parameters
- Gearing Analysis
- Operating Characteristics
- Strength Analysis
- Gearing Creation Precision Parameters

Geometric Parameters

On this tab, you can select methods for calculating geometry.



Create 3D Models. The option allows you to select from the list which 3D objects to add to the scene:

- **Pinion and Gear;**
- **Pinion;**
- **Gear.**

This option is necessary when creating mechanisms based on multiple connected conical gearings, where one of the objects is common to two or more gearings and should not be duplicated.

Calculation, Standard. The option allows you to select the calculation of the geometry of gears according to the standards:

- **ISO 23509;**
- **DIN 3971;**
- **GOST 19624-74 and GOST 19326-73.**

The calculation standard defines the formulas for calculating the geometric parameters, the parameters of the source contour, the relationship and the availability of entering the gearing parameters.

Tooth Type. The option allows you to select the type of teeth. The list is different depending on the selected standard of calculation.

For ISO:

- **Straight**;
- **Helical**: on the sweep of the cone, the tooth lines are straight lines tangent to one circle;
- **Spiral (Constant):** on the dividing conical surface, the tangent to the tooth line has a constant helix angle;
- **Spiral:** the shape of the tooth lines is determined by the parameters of the tool;
- **Hypoid** (not available in the current implementation of the command).

For DIN:

- **Straight**;
- **Helical**;
- **Spiral** (analogue of **Circular** in GOST).

For GOST:

- **Straight**;
- **Circular** (analogue of **Spiral** in DIN): the shape of the tooth lines is determined by the tool parameters.

Calculation, Method. Available only for calculation according to ISO 23509. Defines the calculation method according to ISO 23509 terminology. Method "0" - calculation of bevel gearing. Methods "1" to "3" - options for calculating the hypoid gearing. Methods "1" through "3" are not available in the current implementation of the command.

Calculate. The option allows you to switch between the options for constructing geometry by nominal values, and taking into account the conversion to the middle of the tolerance field according to the selected standard. In the current implementation of the command, only the **By Nominal** option is available.

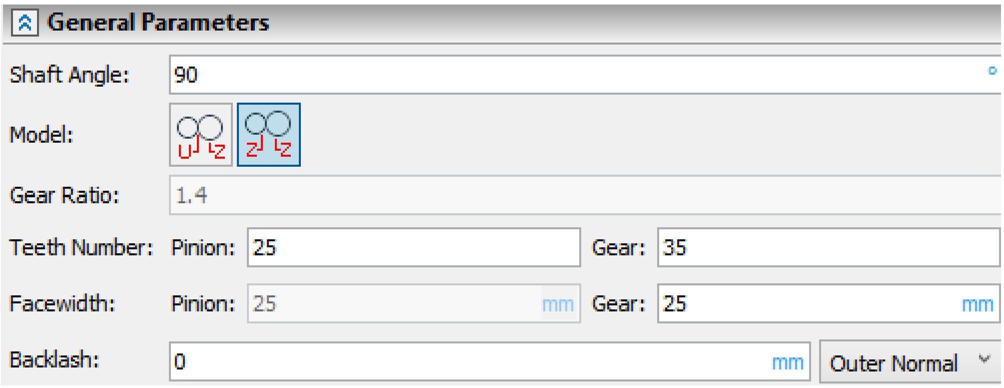
Module. Module value input field. The module can be specified as:

- **Mean Normal** (the main input method for calculation according to GOST 19326-73);
- **Mean Circumferential** (ISO only);
- **Outer Circumferential** (the main input method for calculation according to GOST 19624-74).

For GOST with the **Mean Normal** option, if the **Circular** type is selected, a list of module values is available according to GOST 19326-73. When the **Other** flag is set, input not from the list is available.

General Parameters

The general gearing parameters determine the gear ratio, the number of teeth, the thickness of the working surface of the cones and other parameters depending on the selected design standard and the type of teeth.



Mean gear spiral angle. The parameter is available if the tooth type is different from straight. Determines the mean helix angle of the tooth (helix angle in the calculated section) pinion and gear, if the gear is not hypoid, and the mean angle of the gear, if the gear is hypoid.

Shaft Angle. The angle between the axes of the bevel gears.

Model. The option allows you to select the type of data entry:

- **Gear Ratio:** the input of the gear ratio and the number of teeth on the pinion or gear is available (by setting the flag);
- **Teeth Number:** input of the number of teeth on the pinion and gear is available.

Gear Ratio and **Teeth Number** fields can be calculated or open for input depending on the selected input model.

Facewidth. Specifies the facewidth of the bevel gear ring. Only the gear facewidth field is open for input. Pinion facewidth is calculated. For non-hypoid gears, the facewidth of the pinion is equal to the facewidth of the gear.

Other fields on this tab depend on the selected calculated standard.

For ISO 23509

Backlash. Allows you to set the value of the backlash in the selected section:

- **Mean Normal**;
- **Mean Circumferential;**
- **Outer Normal;**
- **Outer Circumferential.**

For GOST 19624-74 and GOST 19326-73

Mean Section Shift Coefficient. When the value is equal to 1, the calculated section is in the middle of the gear rim. With values from 0.8 to 1.2, it shifts in proportion to the specified coefficient: with a value less than 1 inward, with a value greater than 1 - outward from the middle of the crown.

Profile Shift Coefficient. Only the pinion field is open for input. Gear shift coefficient always equal to the pinion shift coefficient with the opposite sign.

Parameters, Input Method. Determines the method of entering the thickness modification coefficient and the spread of the cutters of the gear head, options are available:

- **Entering Values**;
- **By Table:** available if the tooth type is **Circular** and the module is set as **Mean Normal**, with a value from the suggested list.

Thickness Modification Coefficient. Input is available only in the **Pinion** field. The gear thickness modification coefficient is always equal to the pinion thickness modification coefficient with the opposite sign. Thickness modification coefficient can be selected from the table, according to GOST 19326-73, if the input method is set as **By Table**.

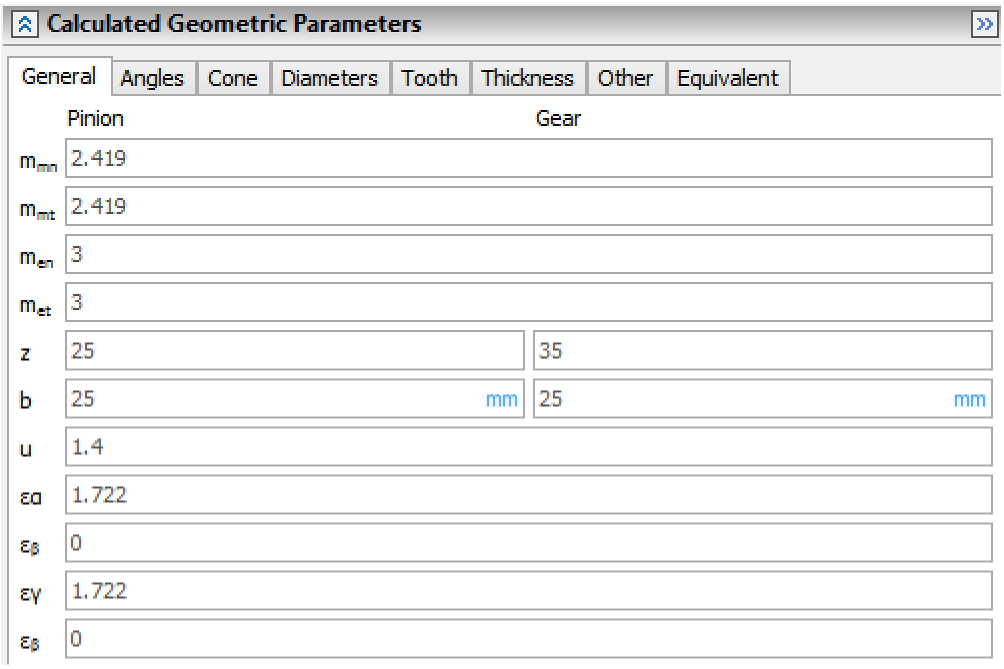
For DIN 3971

Profile Shift Coefficient. Input is available only in the **Pinion** field. Gear profile shift coefficient is always equal to the pinion profile shift coefficient with the opposite sign.

Addendum Angle. Because In DIN 3971 there are no guidelines for determining the shape of the teeth along the generatrix of the cone, but the relationship between addendum and dedendum angles is indicated, then when calculating according to DIN 3971 it is possible to set user-defined values for the addendum angles. It should be borne in mind that pinion addendum angle is equal to the gear dedendum angle, and the gear addendum angle is equal to the pinion dedendum angle. By default, the gear and pinion addendum angles are calculated in such a way that the shape of the teeth is similar to the option **Tooth in Depth**: Standard (according to ISO 23509) and **Depth Taper: Tooth Depth Taper I** (according to GOST 19326-73). When a flag is checked next to the field, the head angle value becomes available for input.

Calculated Geometric Parameters

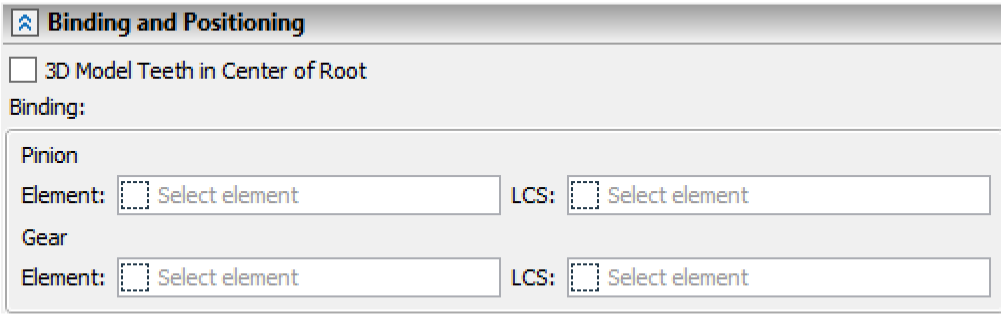
The tab contains only calculated information fields and does not contain fields for entering values.



This tab displays the general geometric calculated parameters of bevel gears and gearing. All values are calculated immediately. the fields instantly display the calculated values according to the parameters entered by the user. Calculated values are grouped thematically on tabs. The tab has a special button that opens the tab in a separate window. Due to this, the window with the general calculated parameters can be placed next to the input area of the gearing parameters and the influence of the input parameters can be monitored. All tab fields have hints with the full name of the parameter.

Binding and Positioning

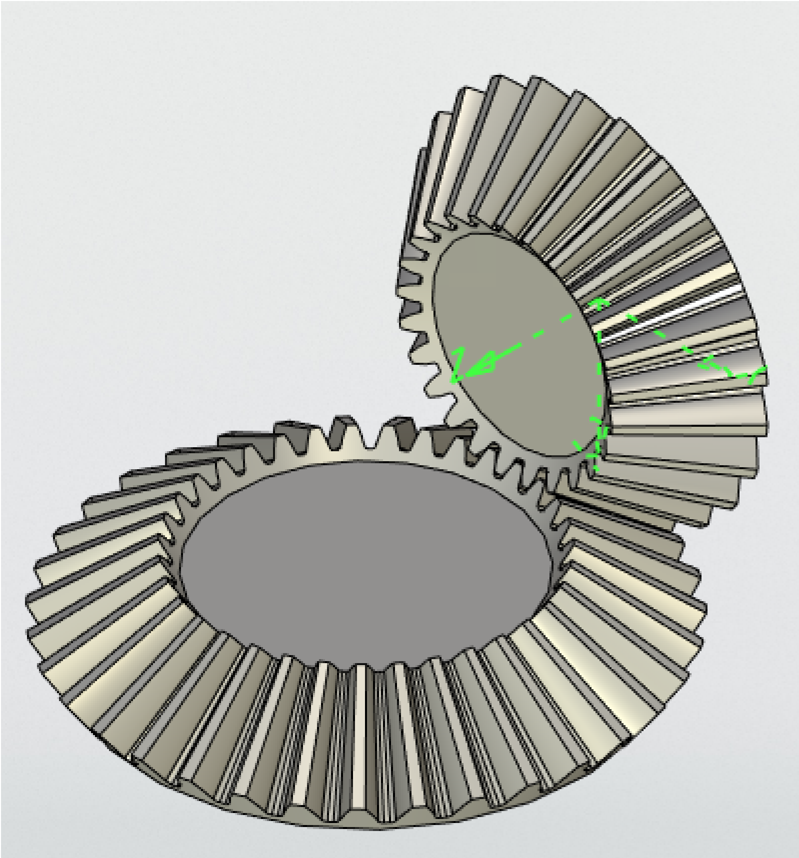
By default, the point of intersection of the axes of the pitch cones of the gear and the pinion (the point of the vertices of the pitch cones) will be at the origin of the global coordinate system, and the axis of the cone of the gear will coincide with the Z-axis of the global coordinate system. The center angle, which determines the position of the pinion, is plotted to the right of the Z axis in the XZ plane. To change the orientation of the gearing or its binding to the models already existing in the document, use the **Binding and Positioning** tab.

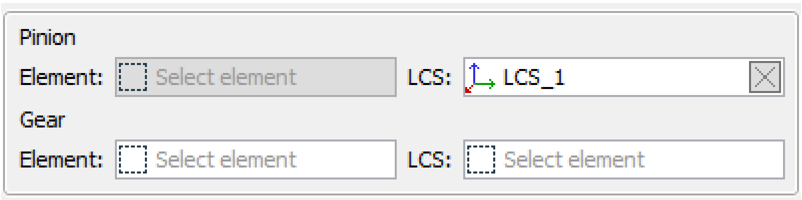


You can bind one object - a pinion or a gear, or two objects - both a pinion and a gear. Either a circular face (shaft end) or LCS (the Z-axis of the LCS coincides with the axis of the cone) can be used as a bind object. To select a binding method, you need to specify the appropriate field and select an object in the 3D scene.

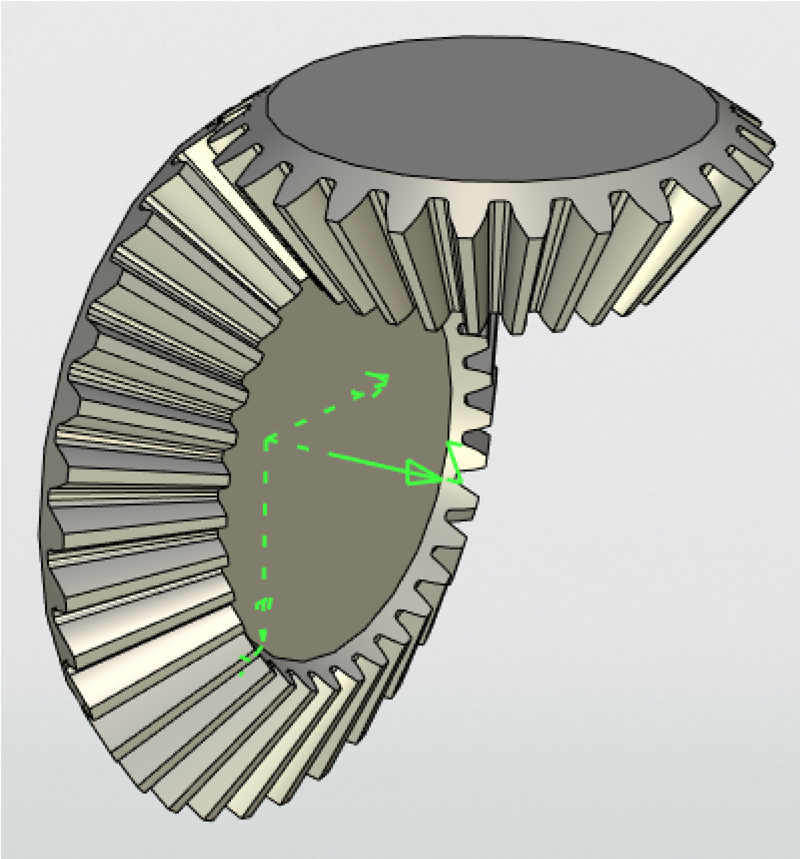
Binding one object at a time

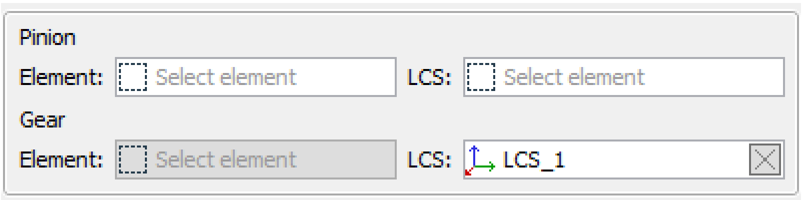
- Binding pinion by LCS. The center of the base of the pinion dividing cone coincides with the center of the LCS. The axis of the cone of the pinion coincides with the Z-axis of the LCS. The shaft angle, which determines the position of the gear, is plotted to the left of the Z-axis in the XZ plane of the selected LCS.



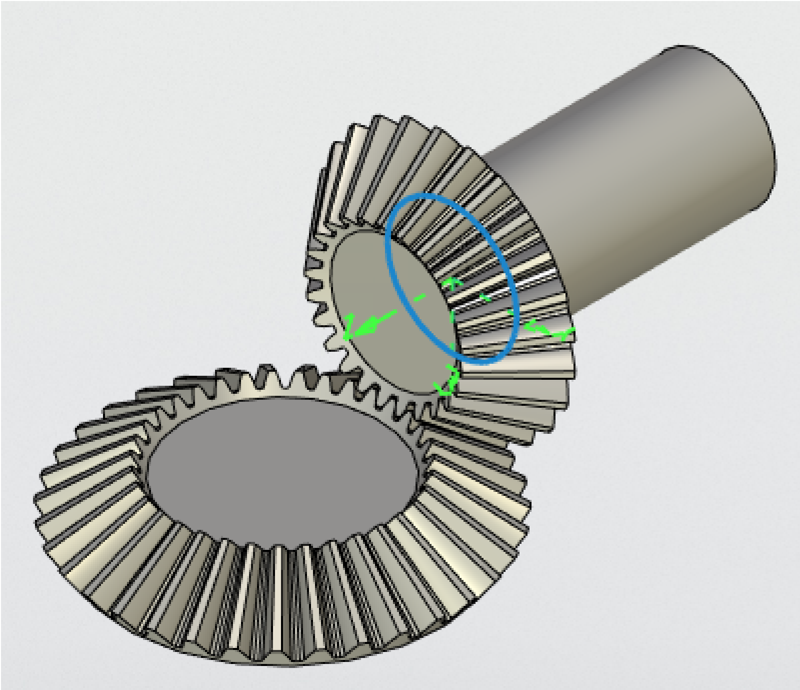


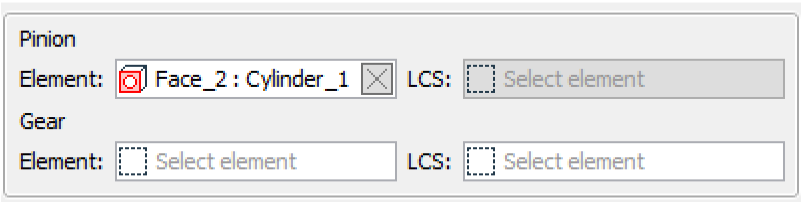
- Binding gear by LCS. The center of the base of the gear dividing cone coincides with the center of the LCS. The axis of the cone of the gear coincides with the Z-axis of the LCS. The shaft angle, which determines the position of the gear is plotted to the right of the Z axis in the XZ plane of the selected LCS.





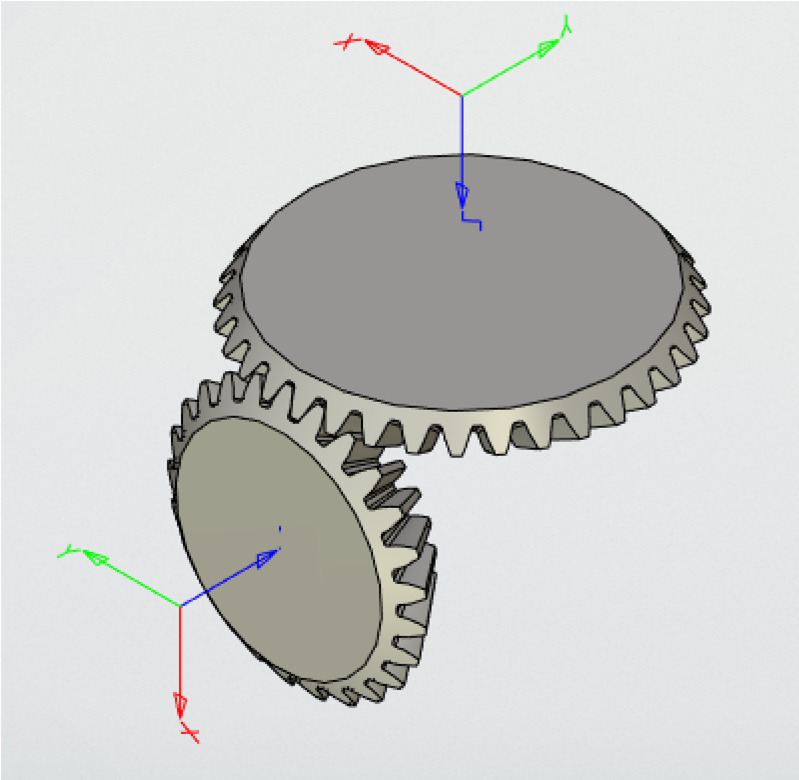
- Binding pinion or gear to a circular face. An LCS is created on the circular face, the center of which will coincide with the center of the circular face. The Z-axis of the LCS coincides with the normal of the circular face. Further positioning is similar to snapping by LCS.

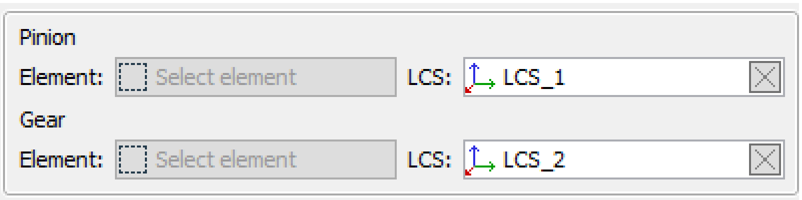




Binding two object at a time

As in the case of binding one object at a time, when you select a circular face, an LCS is created on it, the center of which coincides with the center of the circular face, and the Z-axis of the LCS coincides with the normal of the circular face. The Z-axes of the selected LCS, or LCSs obtained by selecting a circular face, must lie in the same plane. The point of intersection of the Z axis of the LCS will be the point of intersection of the axes of the dividing cones (the point of the vertices of the dividing cones). The center angle will be set automatically according to the angle between the Z-axes of the LCS.



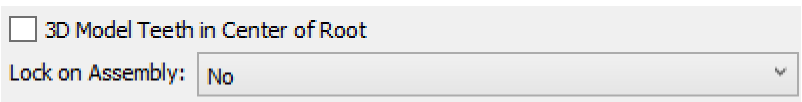


Additional options

3D Model Teeth in Center of Root if there is a backlash (the backlash is set or the geometry is calculated in the middle of the tolerance field), the option allows you to set the teeth in the center of the root.

Lock on Assembly. When pinions and gears are geared, they automatically receive rotation transformations around the cone axis. If it is necessary to create an gearing between a gear and several gears, or vice versa, between a pinion and several gears, then the common object must be fixed from transformations. To do this, you need to use this option:

- **No**: the default state when objects are not fixed from the rotation transformation around the axis;
- **Pinion**: the pinion does not rotate around the axis - only the gears rotates to engage in gearing;
- **Gear**: the gear does not rotate around the axis - only the pinions rotates to engage in gearing.

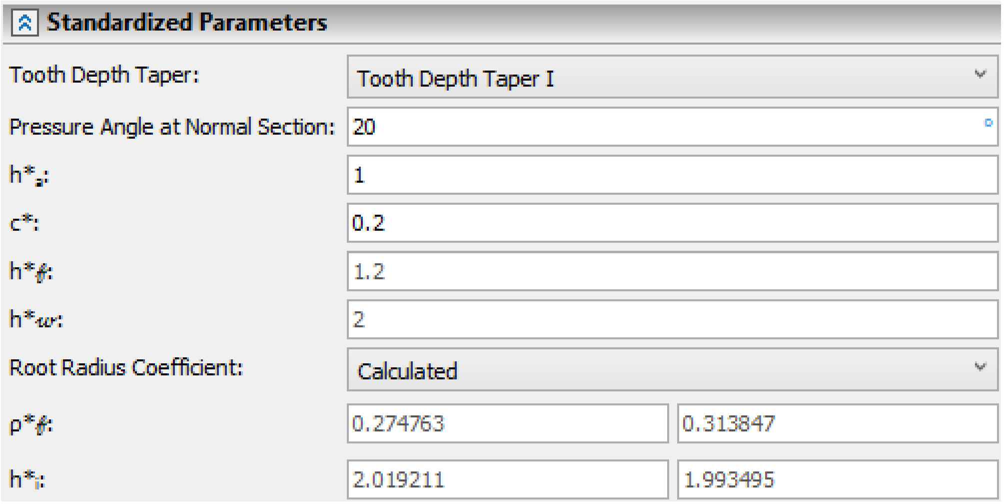


Creation of multiple gearing around one object assumes that the position of the axes is already known, therefore the option is available only when binding by two objects.

In order not to create a common object every time, you need to use the Create 3D Models option.

Standardized Parameters

This tab displays the standardized parameters of the source contour. And also the parameters, according to the description in the calculated standards, related to the source contour. The composition of the parameters of the source contour entirely depends on the selected calculated standard.



For ISO 23509

Data Input Type. The option according to ISO 23509 allows you to choose two options for entering the parameters of the source contour:

- **I**: input option close to other European standards;
- **II**: input option close to AGMA (not available in the current implementation of the command).

Pressure Angle at Normal Section. The normal angle of the profile in the calculated section.
Profile Shift Coefficient. Sets the pinion profile shift coefficient. With a non-hypoid gear, the gear profile shift coefficient is always equal to the coefficient of the pinion with the opposite sign.
Addendum Coefficient and **Basic Crown Gear Dedendum Factor** - typical coefficients of proportionality of the size of the tooth relative to the module.
Thickness Modification Coefficient. Sets the coefficient of modification in the thickness of the gear tooth. The influence of this coefficient on the thickness of the tooth of the gear is determined by the calculation methods ISO 23509.
Tooth in Depth. Defines the shape of the tooth along the generatrix of the cone according to the options specified in ISO 23509:

- **Standard**;
- **Uniform Depth** (not available in the current implementation of the command);
- **Constant Slot Width** (not available in the current implementation of the command);
- **Modified Slot Width** (not available in the current implementation of the command).

For DIN 3971

Pressure Angle at Normal Section. The normal angle of the profile in the calculated section.
Addendum Coefficient и **Dedendum Coefficient** - typical coefficients of proportionality of the size of the tooth relative to the module.
Thickness Modification Coefficient. The pinion field is always open for input, whereas the gear field is only open when a flag is set. If the flag is cleared, gear thickness modification coefficient is equal to the pinion thickness modification coefficient with the opposite sign.

For GOST 19624-74 and GOST 19326-73

Tooth Depth Taper. Determines the shape of the tooth along the generatrix of the cone according to the options specified in GOST 19326-73:

- **Tooth Depth Taper I;**
- **Tooth Depth Taper II;**
- **Tooth Depth Taper III.**

In the standard for spur gearing (GOST 19624-74), axial tooth shapes are not indicated, and the calculation is made according to formulas similar to the axial shape I option from GOST 19326-73. However, for spur gearing, you can specify other tooth shapes and get the result. The use of axial shapes II and III for spur gearing is at the discretion of the user.

Pressure Angle at Normal Section. The normal angle of the profile in the calculated section.

Addendum Coefficient and **Tip Clearance** - typical coefficients of proportionality of the size of the tooth relative to the module, available for input.

Dedendum Coefficient, Common Depth Coefficient и **Boundary Depth Coefficient -** typical coefficients of proportionality of the size of the tooth relative to the module, displayed as calculated.

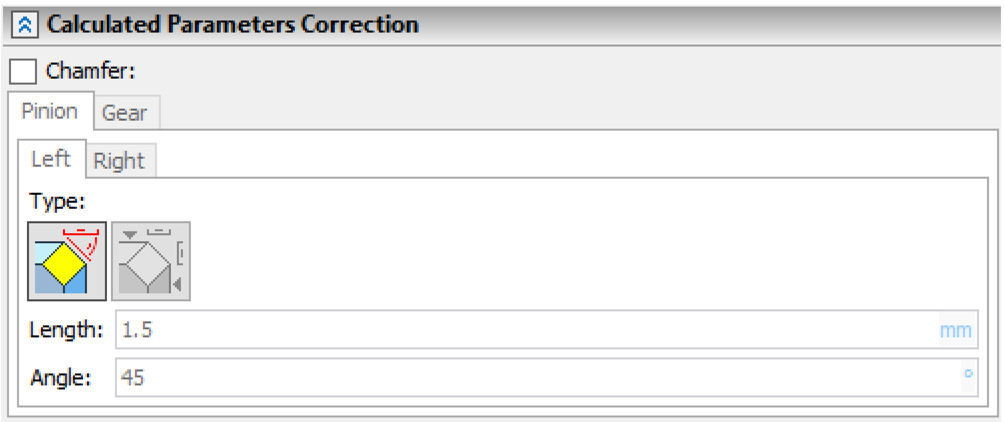
Root Radius Coefficient. Defines the option for calculating the coefficient of the radius of curvature of the transition curve:

- **Calculated:** automatically selects the coefficient of curvature at which the involute part of the tooth is minimally overlapped, but which is sufficient to build a curve;
- **By GOST:** sets the curvature coefficient according to GOST 13754-81 or GOST 19624-74 (depending on the type of tooth), while a situation may arise that at a given coefficient it is impossible to create a transition curve;
- **Custom:** user input of curvature coefficients.

Root Radius Coefficient Value. The fields are available for input if the **Custom** option is selected in the list.

Calculated Parameters Correction

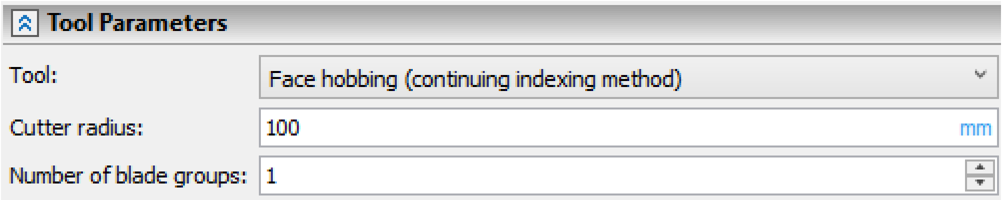
As in the command for creating a cylindrical gearing, a special tab is provided for adjusting the calculated parameters, which, due to the manufacturing technology, assembly accuracy, design features, cannot be performed without correcting the calculated values. In the current implementation of the command, adding a chamfer is available.



Chamfer. Chamfering will be added to the gear rim of the pinion and gear when the flag is set. The input of the parameters of the gear and pinion chamfer is divided into corresponding tabs. There are two ways to enter a chamfer: **Length - Angle** and **Offsets**. For a gear, the **Same as Pinion** option is enabled by default. If the flag for the gear is cleared, it will be possible to enter the chamfer parameters different from the gear chamfer parameters.

Tool Parameters

The tool parameters depend on the selected calculation standard.



For ISO 23509

The tool options are only available if the Spiral or Hypoid option is selected.
Tool. According to ISO 23509, there are two main typed ways to create spiral teeth:

- Face hobbing (continuing indexing method);
- Face milling (single indexing method).

Cutter radius and **Number of blade groups** - tool parameters according to ISO 23509.

For DIN 3971

The tool options are only available if the Spiral option is selected.
Nominal Cutter Radius. Defines the diameter (radius) of the arc tooth.

For GOST 19326-73

Instrument options are available only if Circular is selected.
Nominal Cutter Radius. Defines the diameter (radius) of the circular tooth.
Cutting Processes Type:

- **One-sided processing method;**
- **Two-sided processing method.**

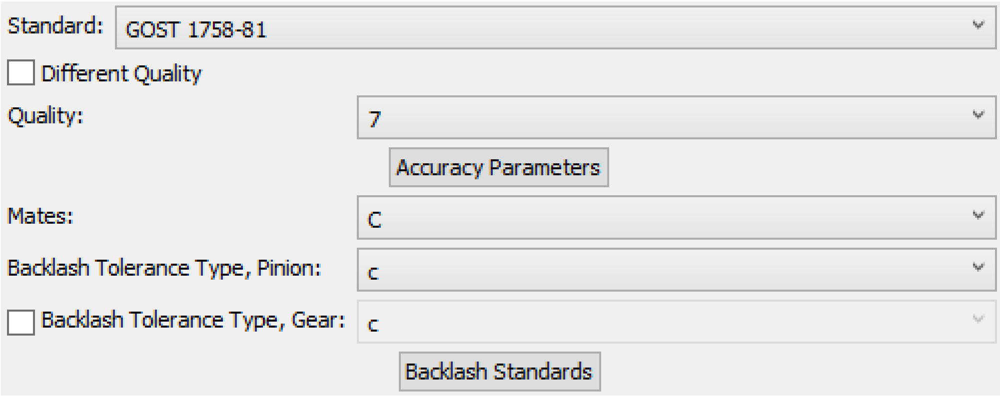
Cutter point width for two-sided finishing of gear. The parameter is available if the two-sided processing method is selected. The parameter will be selected from the table if the input method By Table is specified in **Parameters, Input Method**. If the option **Parameters, Input Method: Entering Values** is selected, a list will be available:

- **Calculate**: the value of the set of cutters will be calculated;
- **Round off according to GOST 11902-66**: the value of the divorce of the cutters will be calculated and rounded to the nearest according to GOST 11902-66;
- **Entering Values**: user input.

Accuracy Parameters

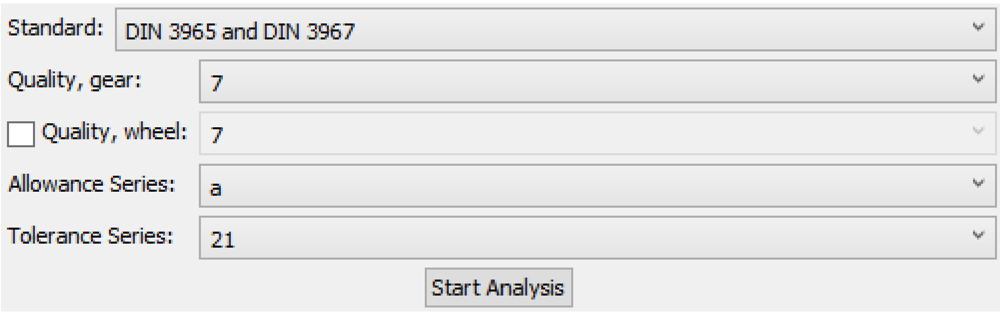
Current tab contains accuracy grade settings these are necessary for tolerance limit and deviation calculation of bevel gearing. Calculation is optionally possible according to **GOST1758-81**, **DIN 3965 and DIN 3967**, **ISO 17485**. To choose one use combo box. Result of this calculation is used in Inspection Table on Drawing and also as Input Data in Strength Analysis.
Each Standard has own Data input template. So Input data type and their fields to fill can be different.

GOST 1758-81:



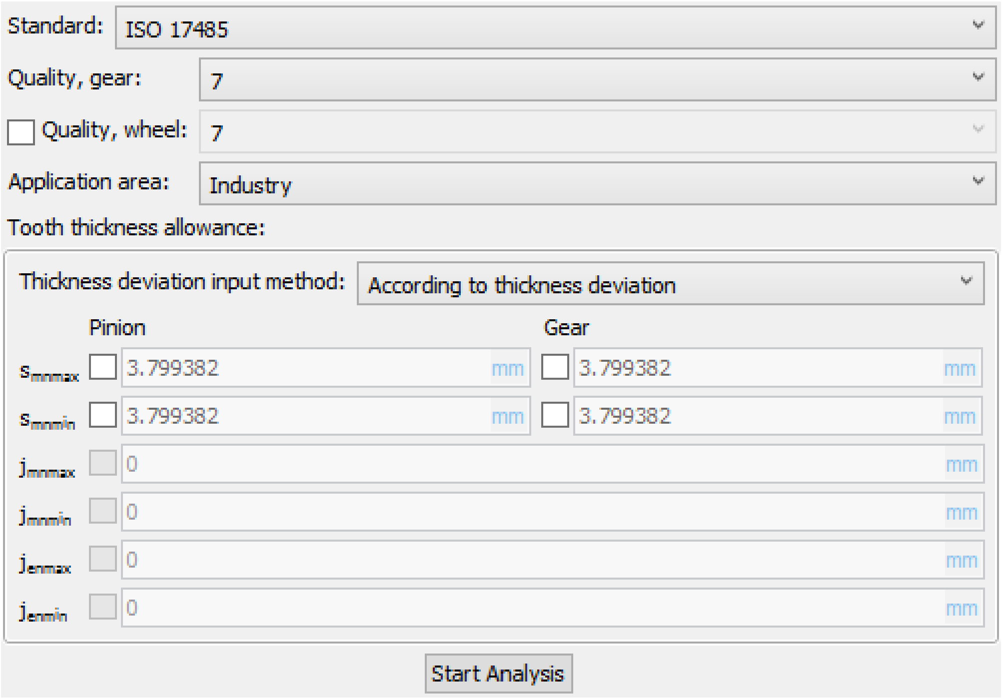
- The field **Different Quality** with tick mark on left allows to enter different quality settings for pinion and gear. Default settings (there is no tick mark) mean that pinion and gear quality is equal;
- Combo box **Quality** gives to choose grades from 4 to 12;
- **Accuracy Parameter** button opens the window with calculated tolerances and deviations for pinion and gear. The calculated deviations divided in three groups - three tabbed pages: **Smoothness Standards**, **Kinematic Standards**, **Tooth Contact Standards**;
- The next combo box **Mates** has to choose values from **A** to **H**;
- Two lines below **Backlash Tolerance Type** has values from **a** to **h** according to standard. First line is for pinion parameters setting. The second one is to set parameter for gear, but inactive till tick mark is On. In the other case pinion and gear by default have equal Backlash Tolerance Type settings.
- The button **Backlash Standards** bellow opens window with calculated values.

DIN 3965 and DIN 3967:



- Two combo boxes **Quality** has grades from 4 to 12 to choose. First one is for pinion parameter. Second one is for gear parameter, but inactive till tick mark is On. In the other case pinion and gear by default have equal **Quality** settings;
- **Allowance Series** with values (a-h, ab, bc, cd) to choose;
- **Tolerance Series** (21-30).

ISO 17485:



Here **Accuracy parameter** tab is divided on two parts. The upper one contains:

- Two combo boxes **Quality** has grades from 4 to 12 to choose. First one is for pinion parameter. Second one is for gear parameter, but inactive till tick mark is On.
- Combo box **Application area: Auto, Truck transport, Industry, Aviation.**

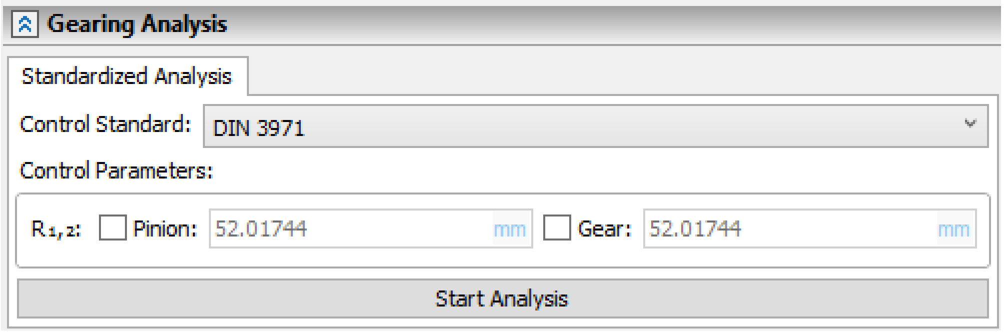
Lower one:

Tooth thickness allowance:

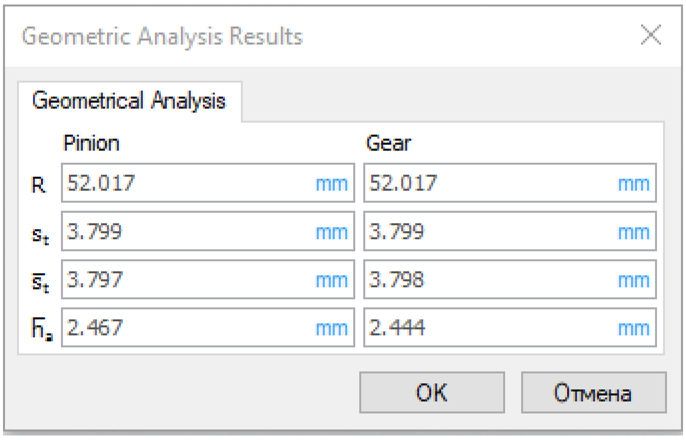
- Combo box **Thickness deviation input method**. There are three pairs of values to unlock for editing. It is possible by choosing one of the variant in combo box.
- So there are six fields: **Mean normal circular tooth thickness** (max/min), **Mean normal backlash** (max/min), **Outer normal module** (max/min). To edit these values it is necessary to set flag on left from them. Availability for editing hangs on value in field **Thickness deviation input method.**
- The button **Start Analysis** located on below opens window with calculated values.

Gearing Analysis

This tab carries out **Standardized Analysis** of gearing according to DIN3971, GOST19624-74 and GOST19326-73. ISO23509. Input data here is Distance from outer end to measured section. The distance is set by default. It also can be entered manually after flag set against relevant field (pinion or gear).

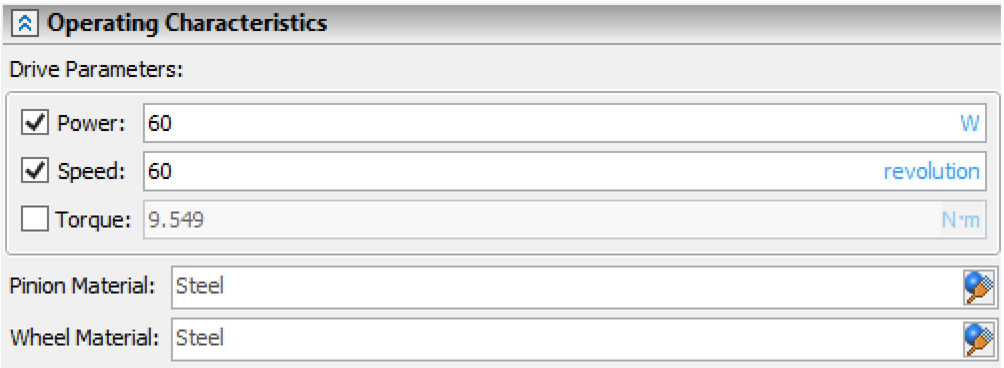


The button **Start Analysis** opens window with results.



Operating Characteristics

Here service conditions and material properties to be entered.



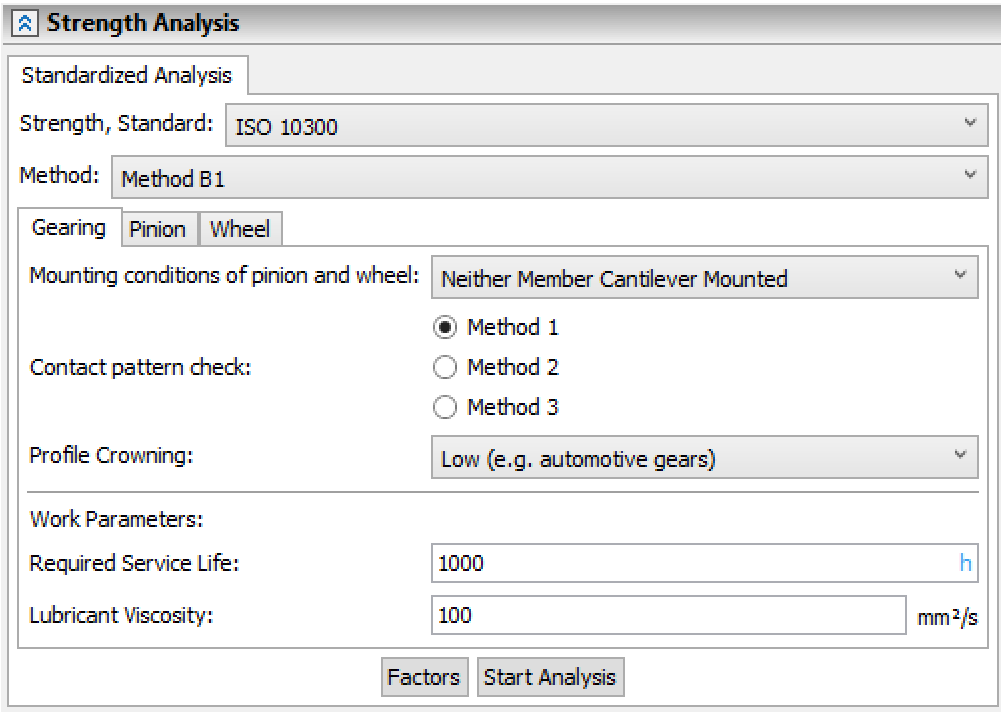
The fields Power, Speed and Torque linked. It means that after two values were entered the third one will be calculated automatically. Field is available for editing if a flag on left side of field is set. Fields Power, Speed and Torque define parameters of driver.

Values in calculated fields change immediately after inputted data. They are needed to estimate load in gearing.

The pinion and gear material can be set independently. To choose material open a window by clicking on pictogram on right side of the material field. Chosen material gives Young's modulus and Poison's ratio for calculation in Strength Analysis.

Strength Analysis

In this tab strength analysis setting is performed. Here calculation according ISO10300 available.



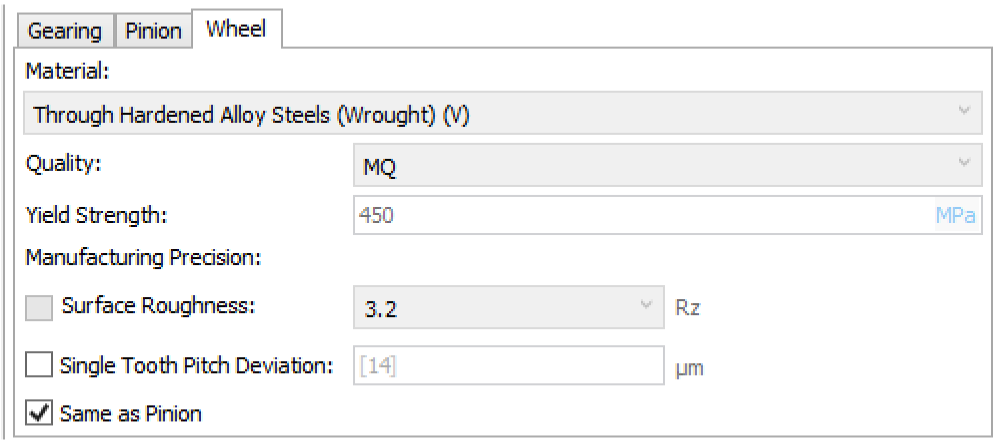
First step of calculation is to choose **Method**: **B1** or **B2**.
Under the line of **Method** are three tabbed pages located **Gearing**, **Pinion** and **Gear**.

Gearing:

- Combo box **Mounting conditions of pinion and wheel**. On this step we choose number of cantilever mounted parts: two, one or none;
- Next element of the interface is field for choosing **Method** of **Contact pattern checking.** Description of each method is available in hints over the Method legend;
- **Profile crowning**: Low or High;
- **Required Service Life and Lubricant Viscosity** to be entered in subdivision **Work Parameters.**

Pinion or Wheel:

In this two tabbed pages material/surface properties of gearing parts to be defined.



- Combo box **Material** contains list of different types of material with various heat treatment;
- **Quality** allows to choose grades of requirement.

Manufacturing precision:

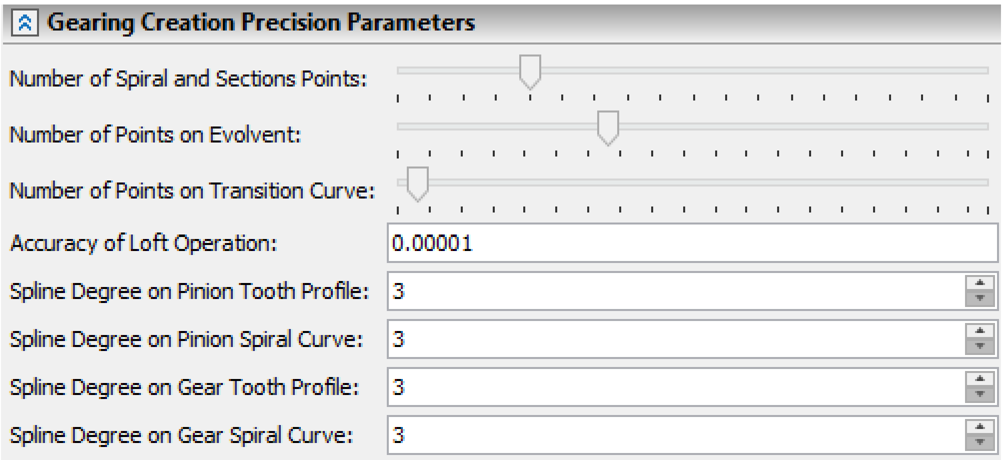
- Surface Roughness;
- Single Tooth Pitch Deviation;
- On the page Wheel there is a check mark **Same as Pinion**, it makes the Wheel settings equal with settings on page **Pinion**.

Calculated **Factors** available by clicking on button bellow. Factors are divided in two tabs **Contact Stress** and **Bending Stress**.

Click on button Start Analysis to see result of calculation. It is also printed in Report.

Gearing Greation Precision Parameters

The evolvent and the transition curve in the circumferential section of the teeth, as well as the curve defining the shape of the tooth, are approximated by a spline. Approximation settings are available on the **Gearing Creation Precision Parameter** tab to improve the precision of the approximation for the user.



Number of Spiral and Sections Points. Slider that determines the number of points of the spline that approximates the tooth shape line. You can specify a value from 10 to 100. A circular section will be drawn at each point of the spline. When constructing a spur gearing, the tooth shape line is a straight line and is constructed from two points. Accordingly, when constructing a spur gearing, this parameter does not affect the construction accuracy.

Number of Points on Evolvent. Slider that determines the number of points of the spline approximating the evolvent. You can set a value from 7 to 100.

Number of Points on Transition Curve. Slider that determines the number of points of the spline that approximates the transition curve. You can set a value from 7 to 100.

Accuracy of Loft Operation. Pinion and gear bodies are obtained by applying the **Loft** operation. The accuracy of this operation is configurable. Decreasing the value of this parameter will increase the accuracy of the **Loft** operation.

Single Degree on Pinion Tooth Profile. The field in which you can set the degree of the spline approximating the evolvent and the transition curve in the circumferential section of the pinion. The value must be an integer greater than or equal to 3.

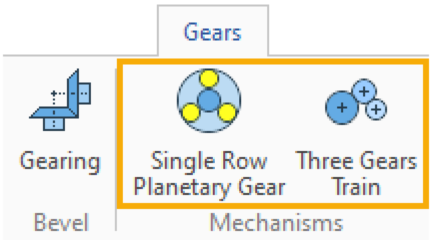
Single Degree on Pinion Spiral Curve. The field in which you can set the degree of the spline that approximates the line of the shape of the tooth of the pinion. The value must be an integer greater than or equal to 3.

Single Degree on Gear Tooth Profile. The field in which you can set the degree of the spline approximating the evolvent and the transition curve in the circumferential section of the gear. The value must be an integer greater than or equal to 3.

Single Degree on Gear Spiral Curve. The field in which you can set the degree of the spline that approximates the line of the shape of the tooth of the gear. The value must be an integer greater than or equal to 3.

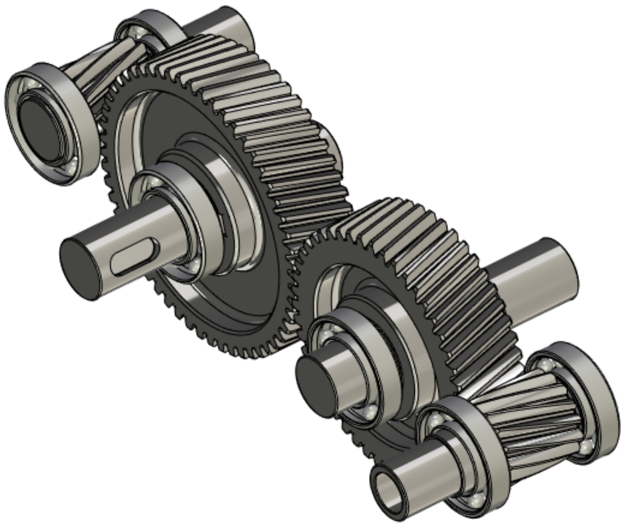
Mechanisms

Mechanisms are a set of commands that make it easy for the user to create connected gears. The user has the opportunity to analyze the possibility of creating a mechanism according to the specified parameters, choose the best option, correct the resulting mechanism after generating a 3D model. Mechanism creation commands are available in the ribbon.



You can create the following mechanisms:

- Single Row Planetary Gear;
- Three Gears Train.



The created mechanisms can be supplemented and combined. In the model tree, all the gears of the mechanism will be combined into a special editable object.

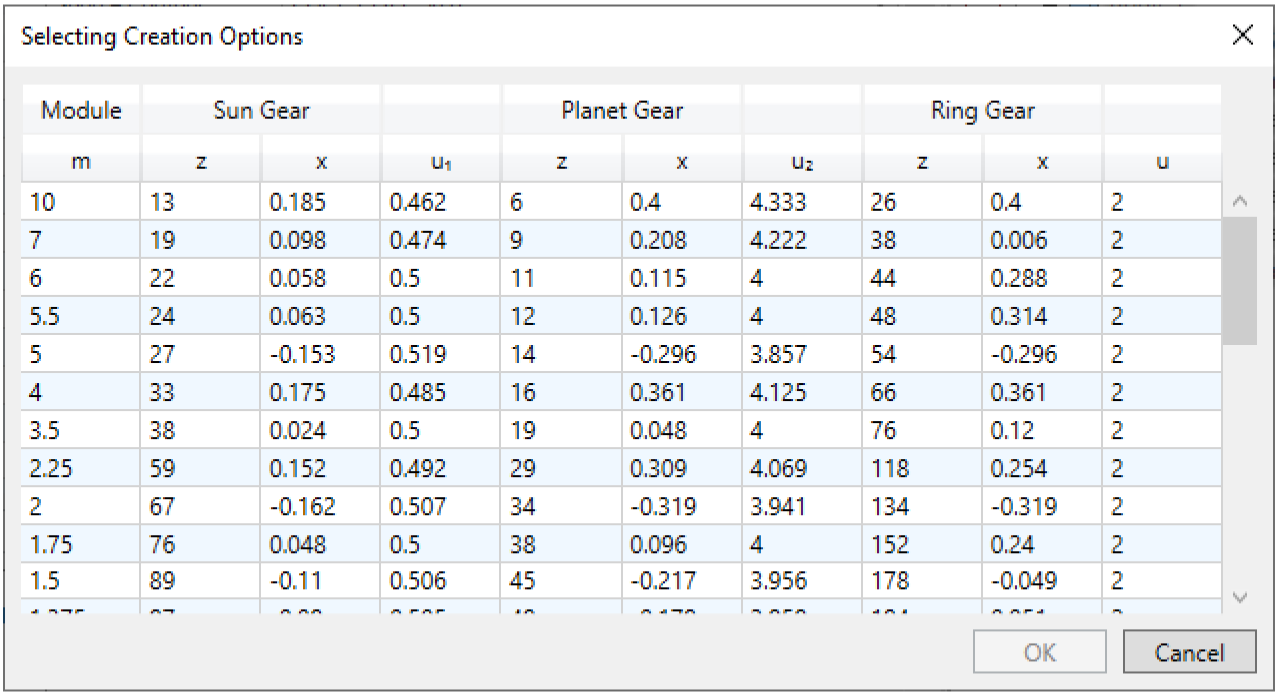
Each gearing is also available for editing, modification, and analysis.

Topics in this section:

- Single Row Planetary Gear
- Three Gears Train

Single Row Planetary Gear

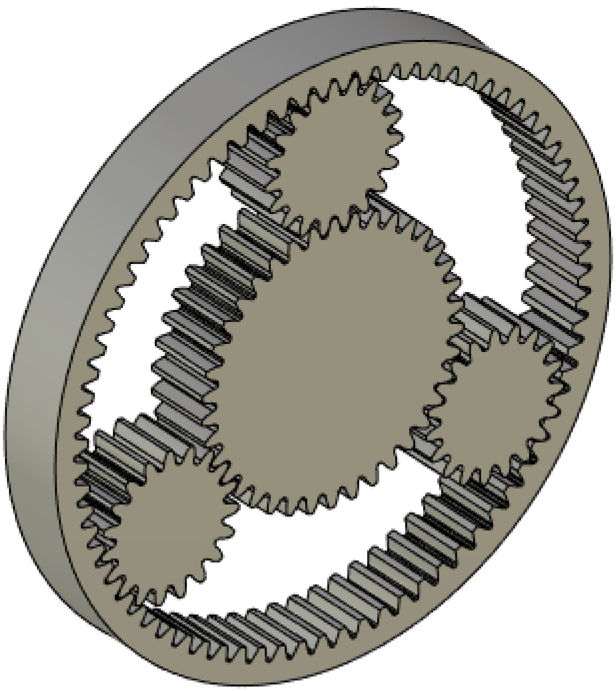
For the **Single Row Planetary Gear** command, the dialog contains the main parameters of the gearing, in which you can define the standard by which the mechanism will be calculated, as well as its mechanical and geometric characteristics. According to the selected parameters, a list of possible solutions with different modulus values is offered, from which the user chooses the optimal one.


Selecting Creation Options

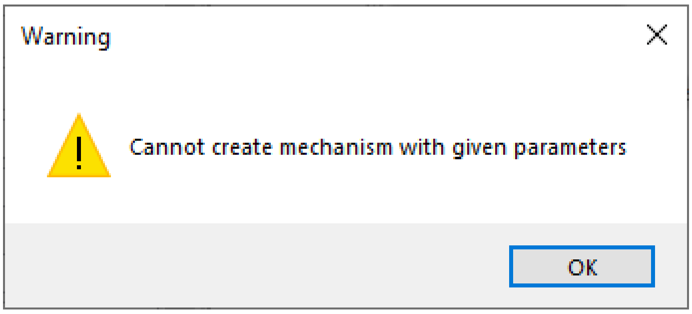
Module	Sun Gear			Planet Gear			Ring Gear		
m	z	x	u_1	z	x	u_2	z	x	u
10	13	0.185	0.462	6	0.4	4.333	26	0.4	2
7	19	0.098	0.474	9	0.208	4.222	38	0.006	2
6	22	0.058	0.5	11	0.115	4	44	0.288	2
5.5	24	0.063	0.5	12	0.126	4	48	0.314	2
5	27	-0.153	0.519	14	-0.296	3.857	54	-0.296	2
4	33	0.175	0.485	16	0.361	4.125	66	0.361	2
3.5	38	0.024	0.5	19	0.048	4	76	0.12	2
2.25	59	0.152	0.492	29	0.309	4.069	118	0.254	2
2	67	-0.162	0.507	34	-0.319	3.941	134	-0.319	2
1.75	76	0.048	0.5	38	0.096	4	152	0.24	2
1.5	89	-0.11	0.506	45	-0.217	3.956	178	-0.049	2

OK Cancel

After selecting a design option and pressing the **OK** button, a 3D model of the mechanism will be created.



If the **Cancel** button is pressed, it will be possible to return to the command dialog.
If it is impossible to find a solution for the given parameters, a corresponding message will be displayed.

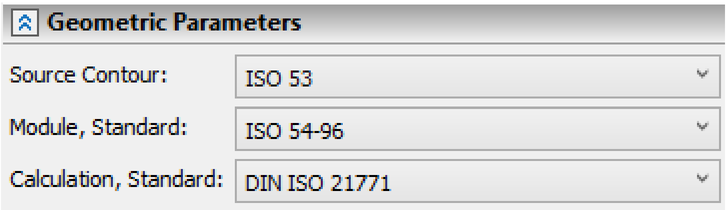


Topics in this section:

- [Geometric Parameters of Single Row Planetary Gear]
- [General Parameters of Single Row Planetary Gear]
- [Additional Options for Creating Single Row Planetary Gear]

Geometric Parameters of Single Row Planetary Gear

This tab provides a choice of standards that determine the construction of the geometry of the gears of the mechanism.



Source Contour. The option allows you to select the following standards from the list:

- ISO 53;
- DIN 867;
- GOST 13755-2015;
- Manually.

Module, Standard. Defines a list of module values allowed by the specified standard:

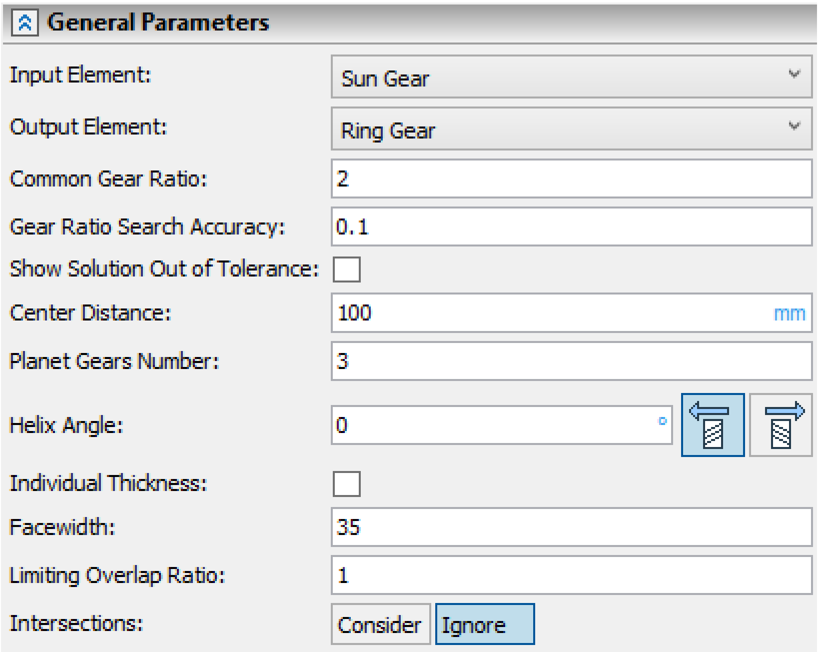
- ISO 54-96;
- DIN 780.
- GOST 9563-60;

Calculation, Standard. The option allows you to select the calculation of the geometry of gears according to the standards:

- DIN ISO 21771;
- GOST 16532.

General Parameters of Single Row Planetary Gear

The tab contains parameters that determine the execution of the mechanism and its mechanical characteristics.



Input Element. Determines which element of the planetary gear will be on the drive shaft:

- Sun Gear;
- Carrier;
- Ring Gear.

Output Element. Determines which element of the planetary gear will be on the output shaft:

- Sun Gear;
- Carrier;
- Ring Gear.

The option when the same element is input and output is automatically excluded.

Common Gear Ratio. Determines the gear ratio between the input and output elements.

Gear Ratio Search Accuracy. The parameter is a filter by the permissible error when displaying a table of possible variants of the mechanism.

Show Solution Out of Tolerance. The option cancels the filtering of solutions that are not included in the tolerance by the calculated gear ratio.

Center Distance. Determines the center distance between the sun gear and the planet gears (the center of the ring and sun gears are the same).

Planet Gears Number. An integer specifying the number of planet gears (satellites). When setting this parameter, remember that a large number of satellites can lead to self-intersections and impossibility of assembly.

Helix Angle. The field allows you to set the helix angle of the teeth at the pitch diameter.

Facewidth. Sets the thickness of the mechanism elements.

Individual Thickness. This option allows you to enter the thickness for each element separately.

Limiting Overlap Ratio. The parameter is a filter when searching for solutions. If the transverse-overlap ratio is less than the specified one, the solution will be eliminated.

Intersections:

- Consider;
- Ignore.

The parameter is a filter. If there are intersections, the solution will be eliminated or not, depending on the option selected. The option allows you not to exclude ring gears, the intersection with which can be corrected by adjusting the tip diameter.

All detailed corrections are made in the specific gearing edit mode.

Additional Options for Creating Single Row Planetary Gear

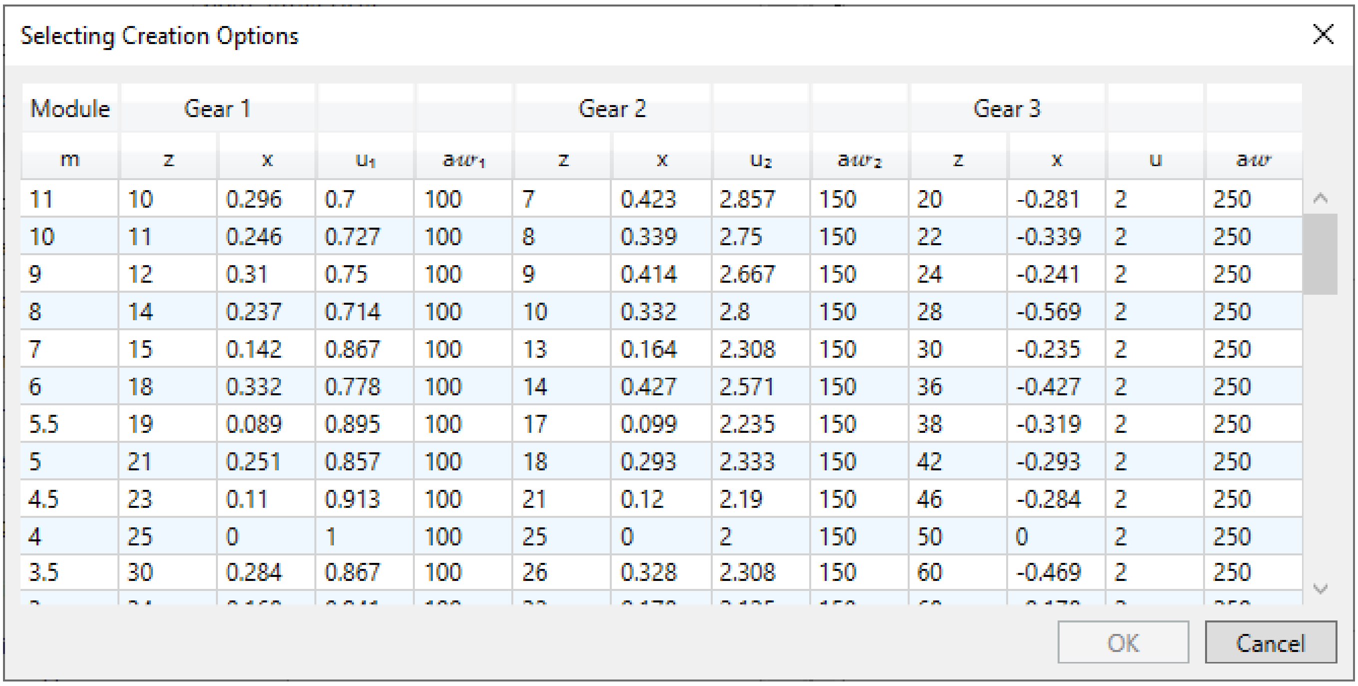
Standardized Parameters tab. The tab is similar to the Gearing command tab. Allows you to correct the parameters of the source contour.

Binding tab. The planetary gear is tied to the center of the sun gear. Binding can be made to an element of a 3D model or LCS. The **Center** option allows you to center the elements of the mechanism if their thickness is different.

Chamfer tab. When the flag is set, chamfers will be added to the ends of the mechanism elements. There are two ways to enter a chamfer: **Length - Angle** and **Offsets**.

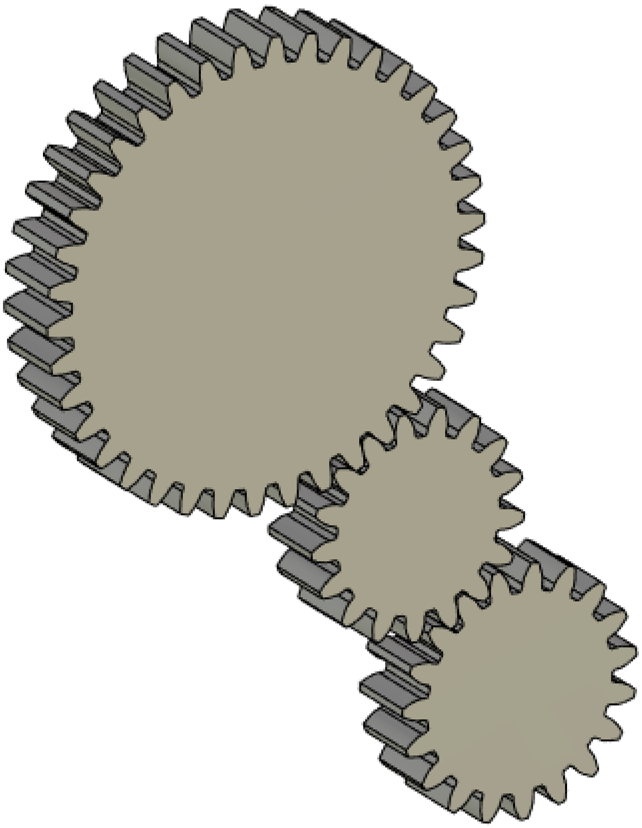
Three Gears Train

In the dialog of the **Three Gears Train** command, you can define the standard by which a chain of three gears will be calculated, as well as set the mechanical and geometric characteristics. According to the selected parameters, a list of possible solutions with different modulus values is offered, from which the user chooses the optimal one.

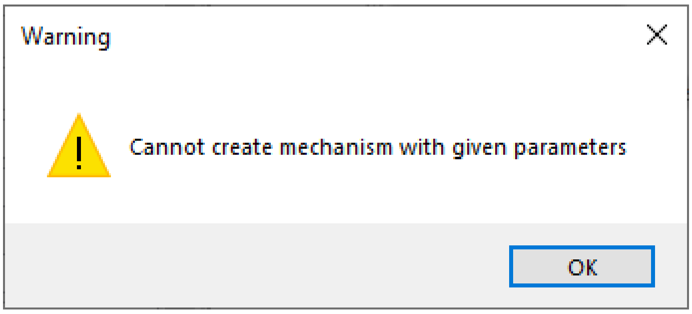


Module	Gear 1				Gear 2				Gear 3			
m	z	x	u_1	a_{w1}	z	x	u_2	a_{w2}	z	x	u	a_w
11	10	0.296	0.7	100	7	0.423	2.857	150	20	-0.281	2	250
10	11	0.246	0.727	100	8	0.339	2.75	150	22	-0.339	2	250
9	12	0.31	0.75	100	9	0.414	2.667	150	24	-0.241	2	250
8	14	0.237	0.714	100	10	0.332	2.8	150	28	-0.569	2	250
7	15	0.142	0.867	100	13	0.164	2.308	150	30	-0.235	2	250
6	18	0.332	0.778	100	14	0.427	2.571	150	36	-0.427	2	250
5.5	19	0.089	0.895	100	17	0.099	2.235	150	38	-0.319	2	250
5	21	0.251	0.857	100	18	0.293	2.333	150	42	-0.293	2	250
4.5	23	0.11	0.913	100	21	0.12	2.19	150	46	-0.284	2	250
4	25	0	1	100	25	0	2	150	50	0	2	250
3.5	30	0.284	0.867	100	26	0.328	2.308	150	60	-0.469	2	250

After selecting a design option and pressing the **OK** button, a 3D model of the mechanism will be created.



If the **Cancel** button is pressed, it will be possible to return to the command dialog.
If it is impossible to find a solution for the given parameters, a corresponding message will be displayed.



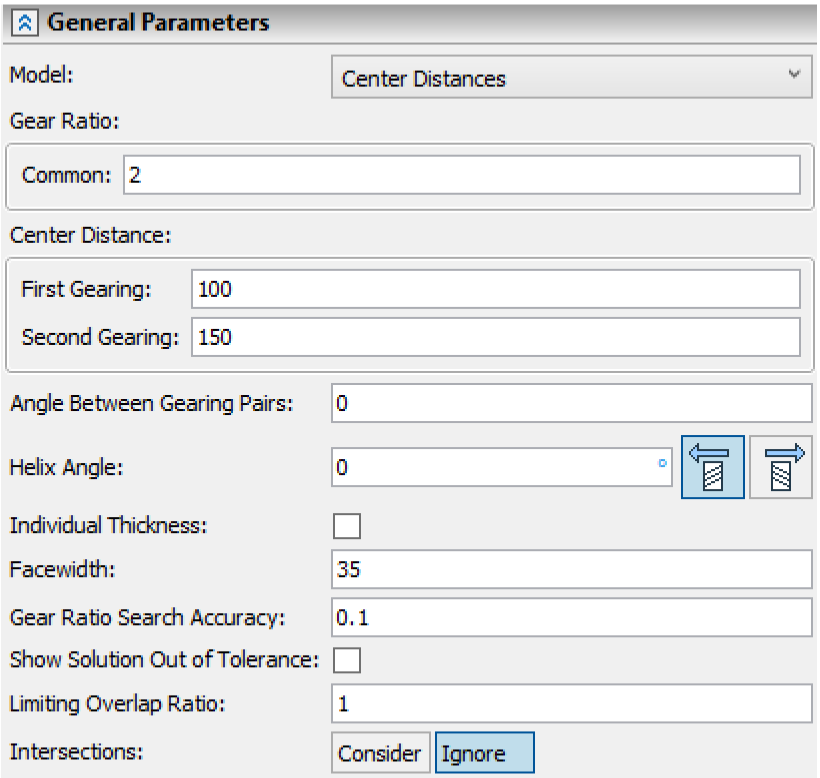
The Geometric Parameters, Standardized Parameters, Chamfers tabs are similar to the command for creating a planetary mechanism.

Topics in this section:

- General Parameters of Three Gears Train
- Binding of Three Gears Train

General Parameters of Three Gears Train

The tab contains parameters that determine the execution of the mechanism, as well as its mechanical characteristics.



Model. Defines the method for calculating the gearing in the mechanism:

- **Center Distances;**
- **Center Distance - Teeth Number**;
- **Teeth Number - Center Distance**;
- **Teeth Number – Teeth Number**.

The selected model determines the type and number of parameters to be entered. For the **Center Distances** model, you need to enter the common gear ratio and two center distances. For the **Teeth Number – Teeth Number** model, enter the number of teeth for all three gears and the shift on all gears. In the two remaining models, you need to enter the number of teeth and shift for two gears, and for one gearing, the center distance and gear ratio.

Angle Between Gearing Pairs. Determines the angle between the segments connecting the centers of the gears. In other words, the angle between the planes in which the gear shafts lie.

Helix Angle. The field allows you to set the helix angle of the teeth at the pitch diameter.

Facewidth. Sets the thickness of the mechanism elements.

Individual Thickness. This option allows you to enter the thickness for each element separately.

Gear Ratio Search Accuracy. The parameter is a filter by the permissible error when displaying a table of possible variants of the mechanism.

Show Solution Out of Tolerance. The option cancels the filtering of solutions that are not included in the tolerance by the calculated gear ratio.

Limiting Overlap Ratio. The parameter is a filter when searching for solutions. If the transverse-overlap ratio is less than the specified one, the solution will be eliminated.

Intersections:

- Consider;
- Ignore.

The parameter is a filter. If there are intersections, the solution will be eliminated or not, depending on the option selected.

Binding of Three Gears Train

The chain of gears binds to the center of the first gear. Binding can be made to an element of a 3D model or LCS.
The **Center** option allows you to center the chain elements if their thickness is different.

Contact us to request information about T-FLEX software, our Academic Program, or if you have ideas on cooperating with Top Systems

www.tflex.com/mail

Contact Us

TOP SYSTEMS

www.tflex.com

(7-499) 973-20-34
(7-499) 973-20-35

You Tube