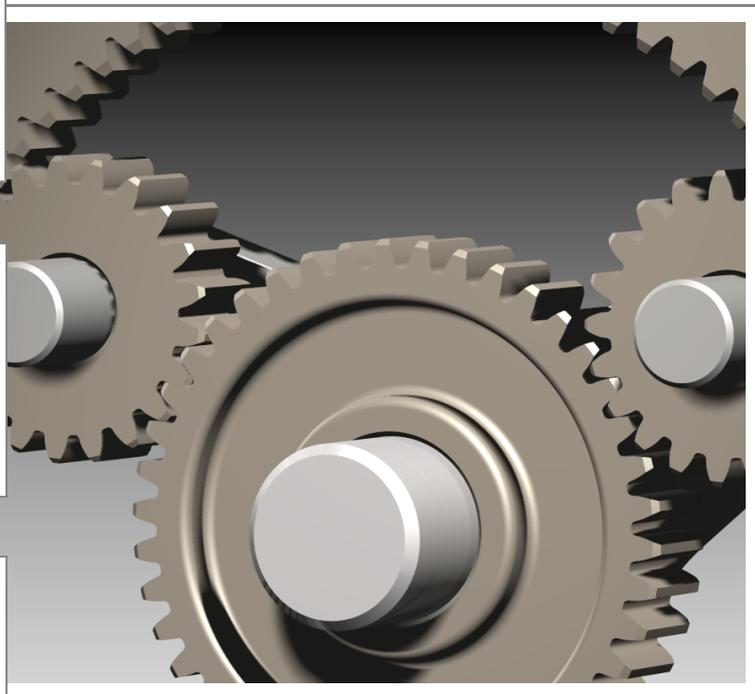
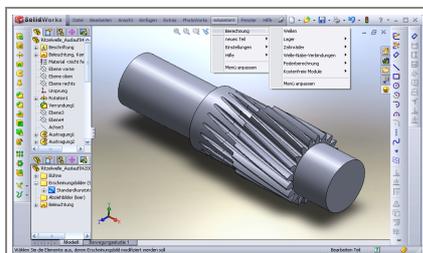
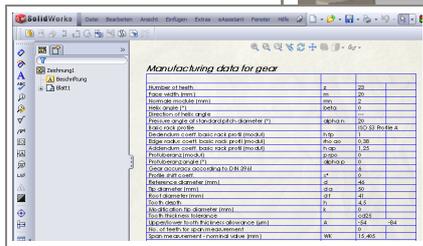
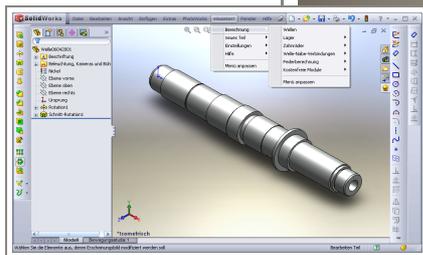
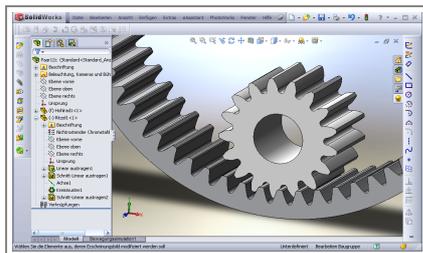


Rollendurchmesser (mm)
 Zwei-Rollenmaß (spielfrei) 10.502
 (obere) 10.407
 (untere) 10.357
 Zahndickenmaß 7.675
 Höhe über der Zahn (mm)

The eAssistant/TBK CAD Plugin for SOLIDWORKS

The Intelligent Connection of Calculation and CAD for
the Calculation Software eAssistant/TBK with SOLIDWORKS



GWJ Technology GmbH
 Celler Strasse 67-69
 38114 Braunschweig, Germany

Phone: +49 (0) 531 129 399-0
 Fax: +49 (0) 531 129 399-29
 Email: info@gwj.de

Rollendurchmesser (mm)
 Zwei-Rollenmaß
 -
 Zahndickensystem
 Höhe über der Welle (mm)

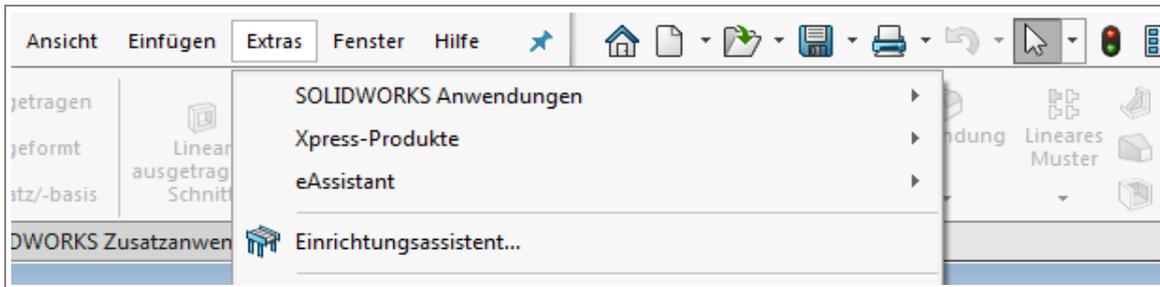
(spielfrei)	10.502
(obere)	10.407
(untere)	10.357
	7.675

Contents

1. Installation.....	3
2. Start.....	3
3. Program Scope.....	3
4. Menu.....	4
Calculation.....	4
New Part.....	5
User Data (applies only for eAssistant).....	6
Options.....	7
Language.....	8
Help.....	8
5. Create a Shaft.....	9
6. Create a Cylindrical Gear.....	9
7. Create a Gearing on an Existing Part.....	10
8. Add Manufacturing Data to Engineering Drawing.....	11
9. Open a Calculation of an Active Part.....	12
10. Add a Tool Runout to the 3D Model.....	13
11. Bidirectional Connection between eAssistant/TBK and SOLIDWORKS.....	13
12. Create a Shaft in SOLIDWORKS and Import it into eAssistant/TBK.....	15

1. Installation

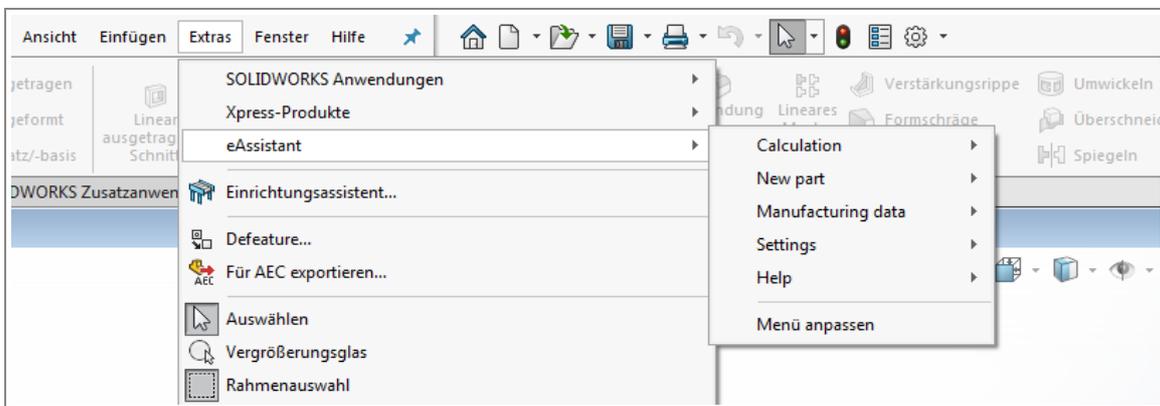
Please start the installation file and follow the instructions. The CAD plugin will be installed on your system. Select "eAssistant" or "TBK" during installation depending on the product. Once the CAD plugin is installed, start SOLIDWORKS and open a new part. The **Tools** drop-down menu in the SOLIDWORKS menu bar provides the entry "eAssistant" or "TBK".



In case the CAD plugin is not visible in the SOLIDWORKS Tools drop-down menu, you must enable the eAssistant/TBK add-in application. Click **Tools => Add-Ins** from the SOLIDWORKS menu bar and activate the eAssistant/TBK application.

2. Start

Click the **Tools** drop-down menu to use all functions of the CAD plugin in SOLIDWORKS.



3. Program Scope

Currently, the following features can be created in SOLIDWORKS based on the calculated eAssistant/TBK data:

- External and internal cylindrical and helical gears
- Rack/pinion gear pairs
- Straight, helical and spiral bevel gears

- Serrated shafts, splined shaft connections (shaft/hub)
- Solid and hollow shafts with an unlimited number of cylindrical and conical shaft segments
- Involute splines (shaft/hub)

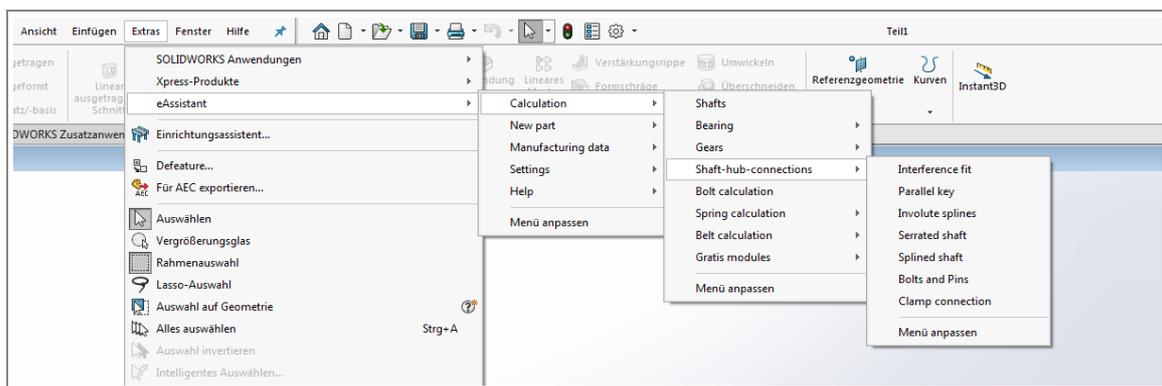
as well:

- Saving of calculation data in the 3D model
- Placing manufacturing data of cylindrical gears, bevel gears and involute splines on the manufacturing drawing
- Cylindrical gears can be created directly on an existing part.
- Bidirectional connection between eAssistant/TBK and SOLIDWORKS (shaft, gear, involute spline profile as single design component) (from SOLIDWORKS 2008)
- Accuracy/point distance for the tooth form can be defined in the options menu.
- Tooth flanks as splines or alternatively as arcs for cylindrical gears and involute tooth profiles possible

4. Menu

Calculation

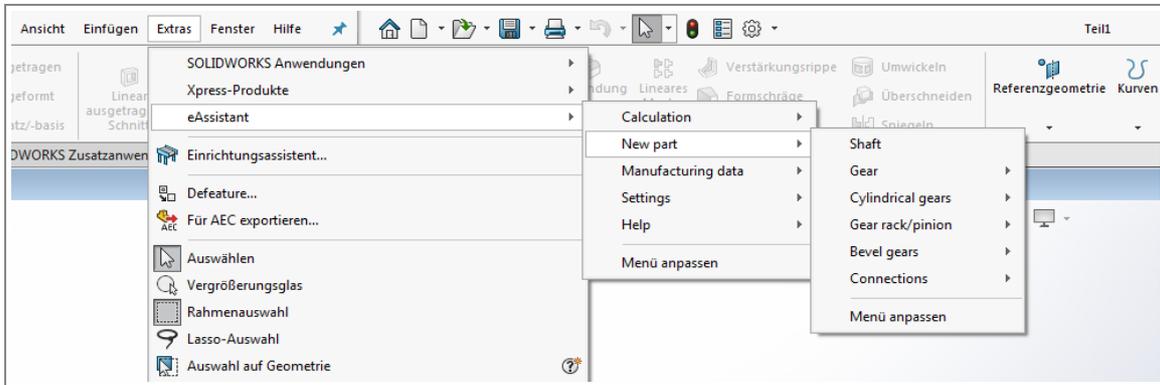
Once the CAD plugin is started, all eAssistant/TBK calculation modules can be opened directly through the **Calculation** menu in SOLIDWORKS.



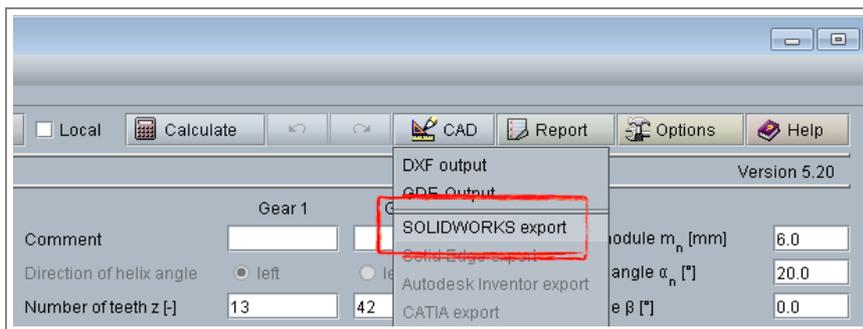
Note on eAssistant: Please note that you need an eAssistant username and password to start the calculation modules through the **Calculation** menu (see **Settings**). Only the free-of-charge modules can be used without the eAssistant access data.

New Part

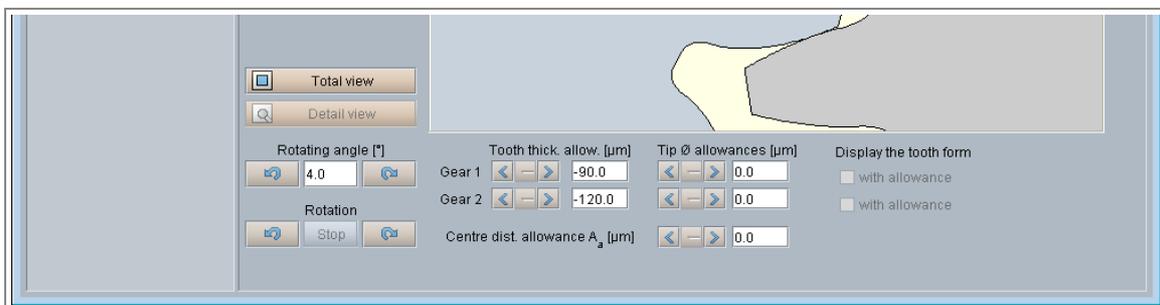
Based on the eAssistant/TBK calculation, the parts can be created directly in SOLIDWORKS by clicking the menu item **New part**.



Complete a calculation using the eAssistant/TBK. Click on the button **CAD => SOLIDWORKS output** in the top menu bar of the eAssistant/TBK. Confirm the message "Data for CAD was written successfully." with the button "OK".



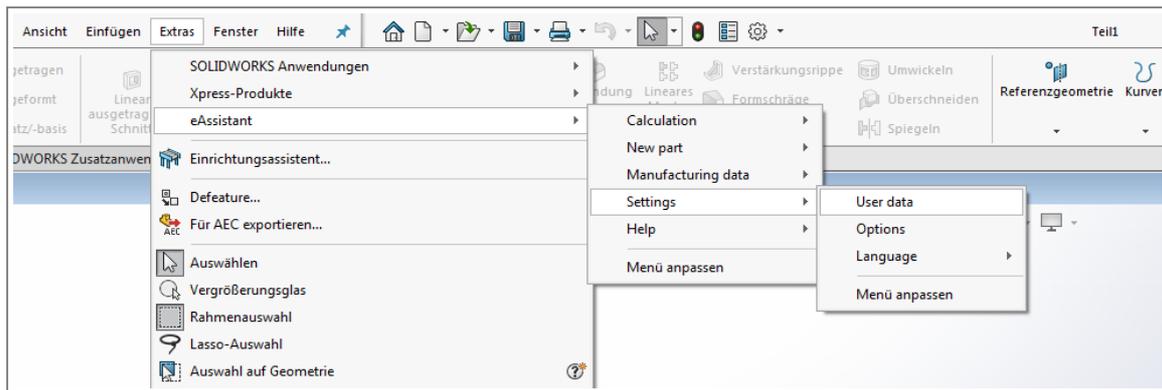
During the generation of involute gearings (cylindrical gears, single cylindrical gears, gear racks, involute splines), the mean tooth thickness allowances and the mean tip diameter allowances **should be selected**. The CAD model is then created with these allowances. You can modify the allowances by opening the mask **Tooth form** in the respective gear modules.



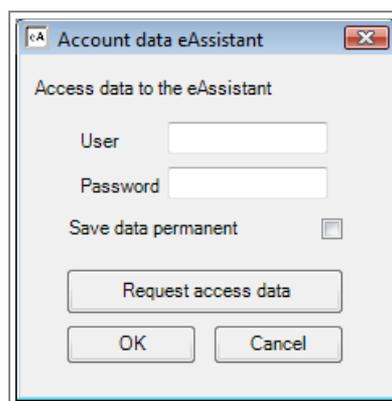
Then click the menu item **New part** in the CAD plugin and select the corresponding element. Now the 3D model is created.

User Data *(applies only for eAssistant)*

Click on the menu item **Tools => eAssistant => Settings => User data** to enter your eAssistant username and your password. You need an account to use all eAssistant calculation modules.



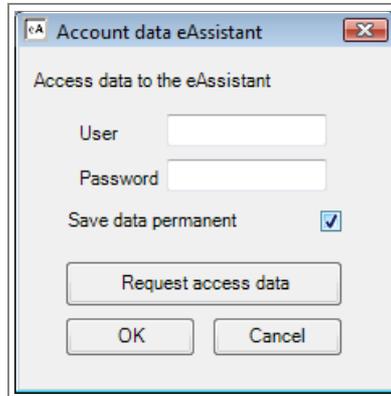
In case you do not have an eAssistant account, please visit our web site www.eAssistant.eu to register for a free test account. Click on the button **Request access data** to get to the eAssistant web site as well. The account provides a time credit of five hours to test the eAssistant without any charges or obligation. Are the five hours expired, the test account is deactivated automatically. If you want continue to use the eAssistant, you need to purchase an account. Decide between hour packages or flat rates.



As soon as you entered your username and password into the window, you can start all eAssistant calculation modules directly through the menu item **Tools => eAssistant => Calculation** without logging in again. The eAssistant and SOLIDWORKS will be connected automatically.

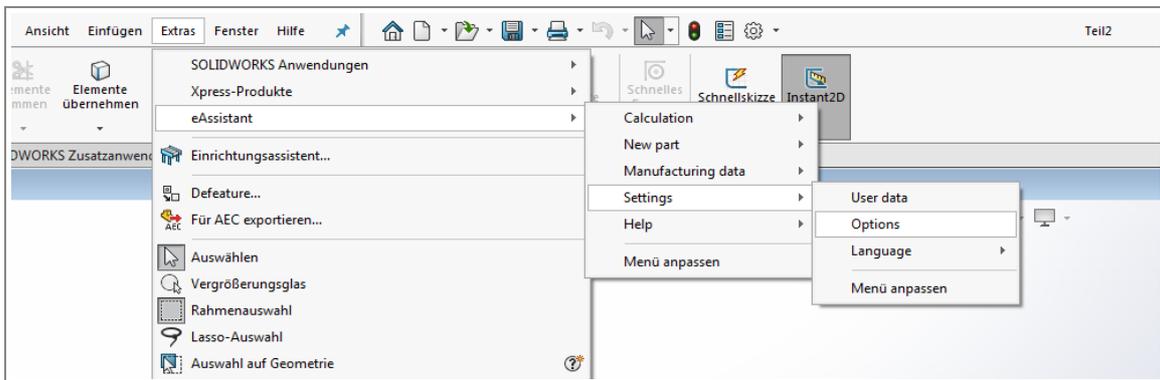


Activate the option "Save data permanent" to save your username and password and to open all calculation modules immediately without entering username and password again.

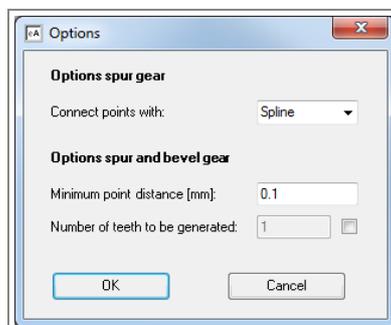


Options

Click the menu item **Tools => eAssistant (TBK) => Settings => Options** to adjust different settings for cylindrical and bevel gears. Select spline or arcs (tangential) to connect the points.

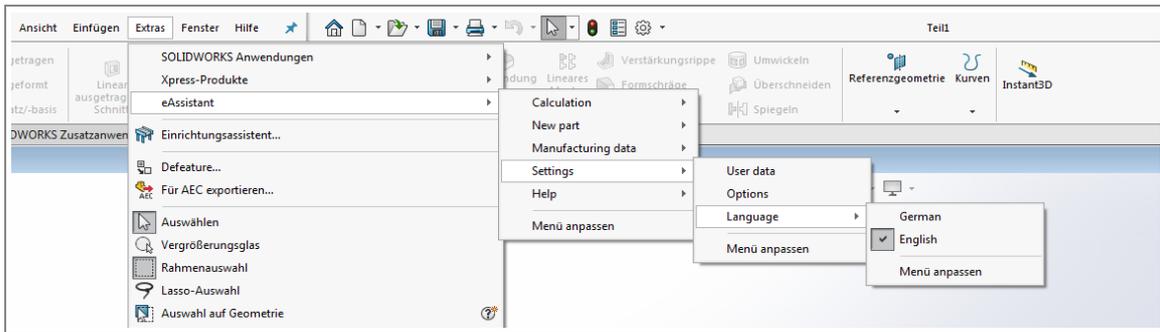


You can, for example, define the distance between the points (accuracy) or the number of teeth to be generated. Just activate the checkbox and enter the appropriate number of teeth.



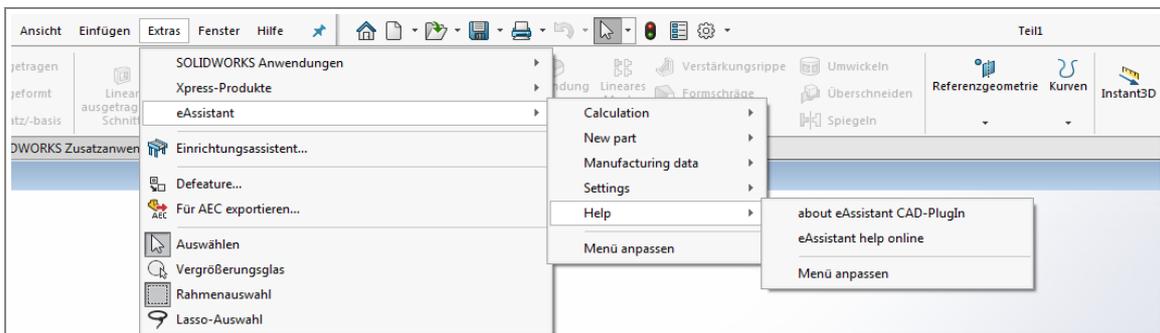
Language

Click the menu item **Tools => eAssistant (TBK) => Settings => Language** in order to change the language. German and English are available. Restart SOLIDWORKS or the CAD plugin (by clicking **Tools Add-Ins**). The menu interface of the CAD plugin adapts itself automatically to the selected language. The eAssistant/TBK calculation modules are started in the selected language as well.



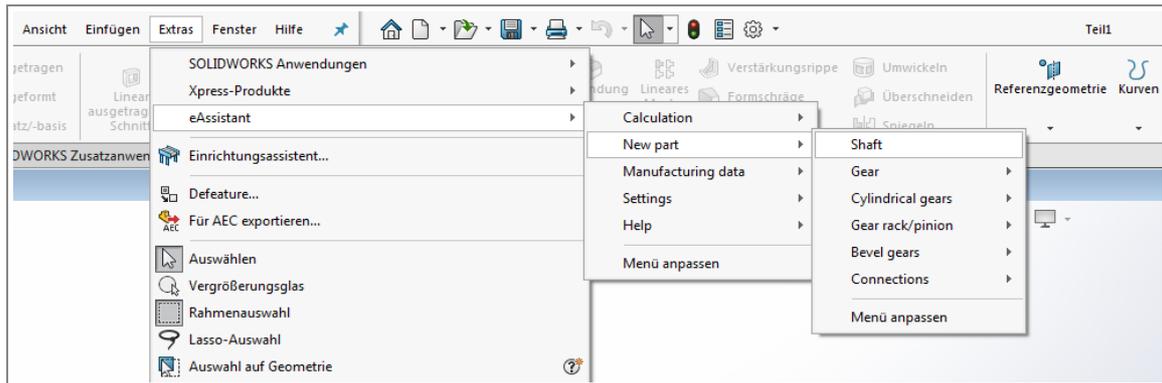
Help

Click on the menu item **Tools => eAssistant (TBK) => Help => eAssistant (TBK) help online** to open the eAssistant/TBK manual where you can find additional information on the eAssistant/TBK.



5. Create a Shaft

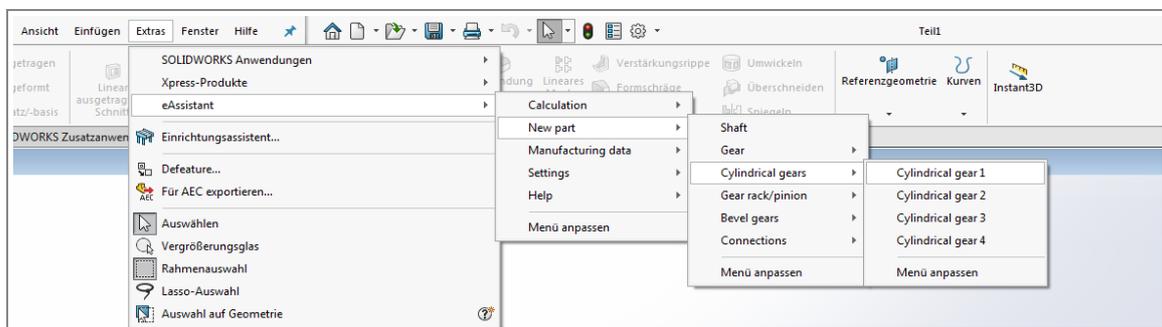
Based on the eAssistant/TBK calculation, shafts with an unlimited number of cylindrical and conical segments can be created as a 3D part. Click on the menu item **Tools => eAssistant (TBK) => Calculation => Shaft** to start the eAssistant/TBK module. Then make a shaft calculation. Click on the button **CAD => SOLIDWORKS export** in the eAssistant/TBK module. Create the shaft by clicking on the menu item **Tools => eAssistant/TBK => New part => Shaft** to create the shaft in SOLIDWORKS.



You can also create a 2D sketch of a shaft in SOLIDWORKS and then import it to eAssistant/TBK. Please see [Section 12](#) for more information.

6. Create a Cylindrical Gear

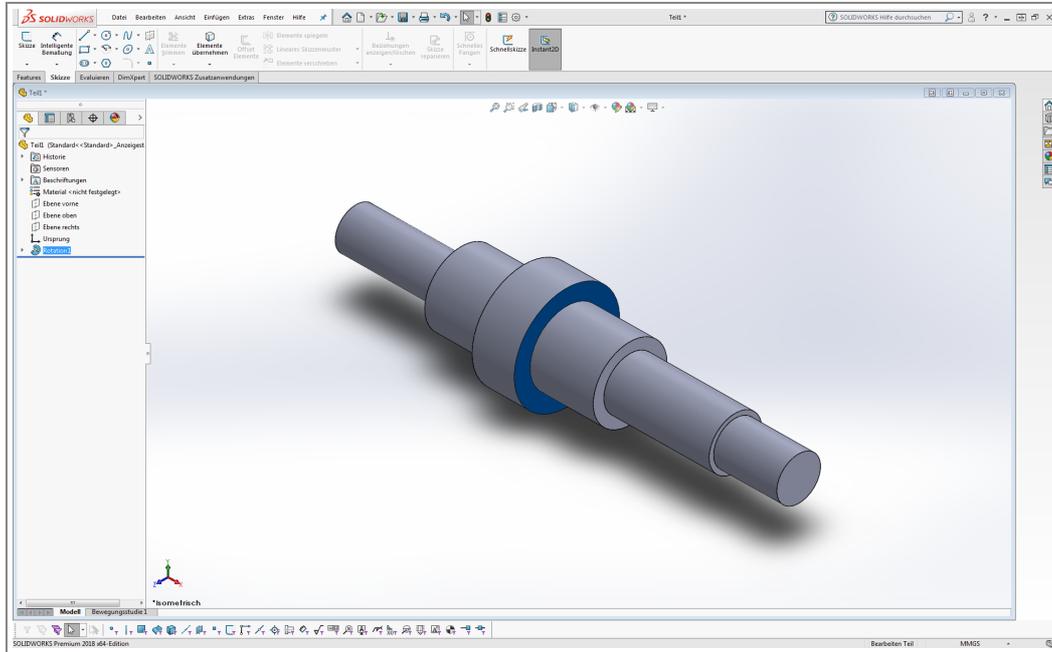
Click on the menu item **Tools => eAssistant (TBK) => Calculation => Gears => Cylindrical gear pair** in order to start the eAssistant/TBK calculation module. Then make a calculation. Click on the button **CAD => SOLIDWORKS export** in the eAssistant/TBK module. Click on the menu item **Tools => eAssistant (TBK) => New part => Cylindrical gears => Cylindrical gear 1** or cylindrical gear 2, 3, 4 to create the cylindrical gear in SOLIDWORKS.



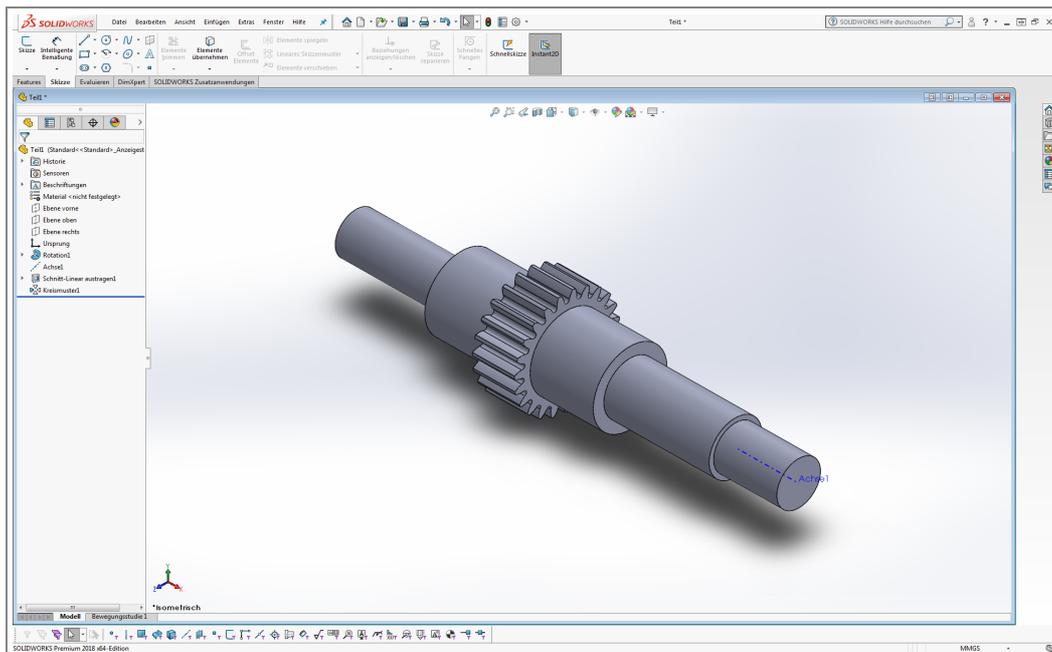
Rollendliniener (mm) (spielfrei) 10.502
 Zwei-Rollenanlag (obere) 10.407
 Zahnstärkenstärk (untere) 10.357
 Zahnstärkenstärk 7.675
 Höhe über der Zahn (mm)

7. Create a Gearing on an Existing Part

The eAssistant/TBK plugin provides a simple way to create a gearing directly on an existing part (pinion shaft).



Make a gear calculation in the eAssistant/TBK. Click on the button **CAD => SOLIDWORKS export**. Select a front face where you want to place the gearing. The face will then be highlighted. Select the menu item **Tools => eAssistant/TBK => New part => Cylindrical gears => Cylindrical gear 1** to add the gearing to the existing part.



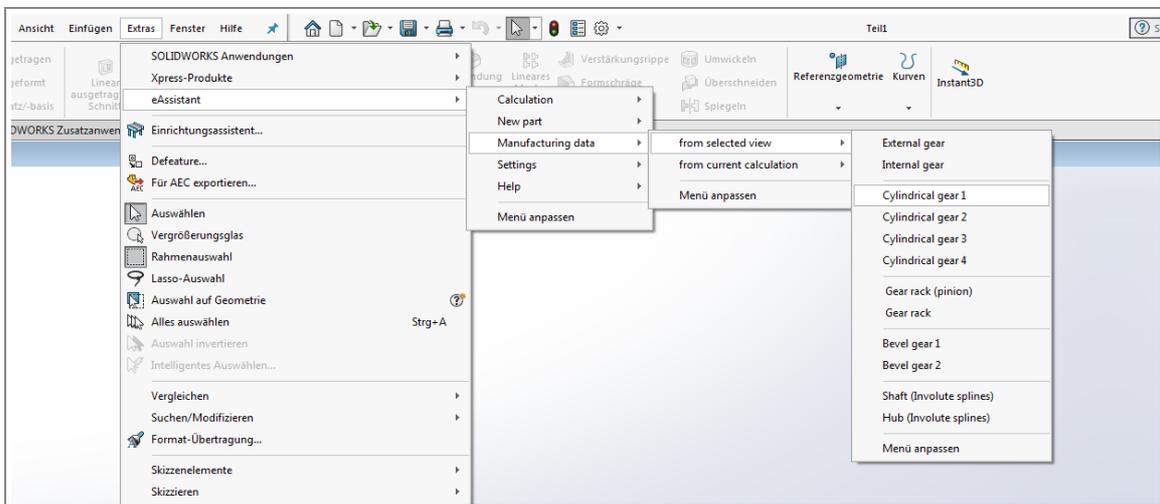
When the dialog window for the "Runout configuration" appears, confirm with the "OK" button.



Please note that the tip circle diameter of the cylindrical gear gear and the diameter of the selected shaft segment must have the same size.

8. Add Manufacturing Data to Engineering Drawing

With just one click, the design table with all manufacturing details can be placed on the manufacturing drawing. Open a drawing in SOLIDWORKS. Select the menu item **Tools => Assistant (TBK) => Manufacturing data => from selected view or from current calculation => Cylindrical gear 1** to create the table.

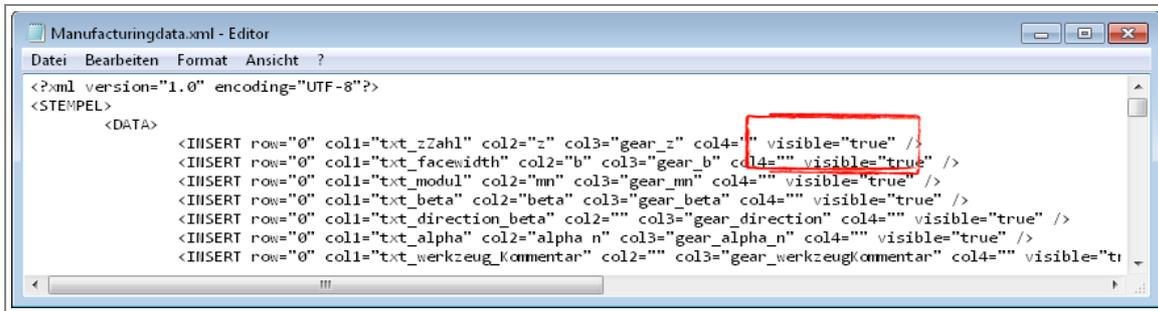


After installation of the CAD plugin, the file "Manufacturingdata.xml" appears in the program directory. When calling the manufacturing data via the CAD plugin, a new folder is created in the directory **C:\User\[Username]\AppData\Roaming** on Windows Vista or Windows 7. Various templates (for example "ManufacturingdataBG.xml") for the design tables are saved in this folder and only these xml files are used to configure the table. Based on these files, the design table can be modified to your preferences.

Cylindrical gears, bevel gears and involute splines get a template as soon as you call the manufacturing data. For bevel gears, a distinction is made between straight/helical bevel gears and spiral bevel gears. The appearance and size of that table are both individually configurable. Open the xml file in a text editor and change the corresponding row, for example from visible="true" into visible="false" to show or hide specific table rows. Save the xml file and place it to the manufacturing drawing. In case you need more extensive customizations (font size, column width, etc.) of the table, please ask for a more detailed description.



If you delete the folder "eAssistant" in the user directory, the templates (xml files) are deleted, too. To avoid this, please create a backup of the folder.



In case the path `C:\User\[Username]\AppData\Roaming` is not displayed, the folder options must be adjusted. Open the folder options by clicking `Start => Control panel => Folder options`. Click the View tab. Select "Show hidden files, folders, and drives" under "Hidden files and folders". Click "OK" to save the new setting.

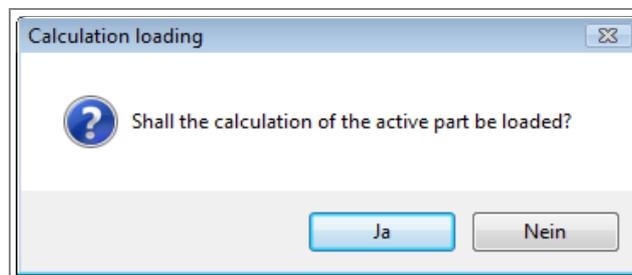


The modifications of the xml files will be lost after an update of the CAD plugin. You have to subsequently maintain the modifications manually.

9. Open a Calculation of an Active Part

The calculation information is saved in the 3D model. The calculation, which belongs to the 3D model, can be opened at any time throughout the entire design phase. If a component contains several different calculation elements, it is possible to open the corresponding calculations.

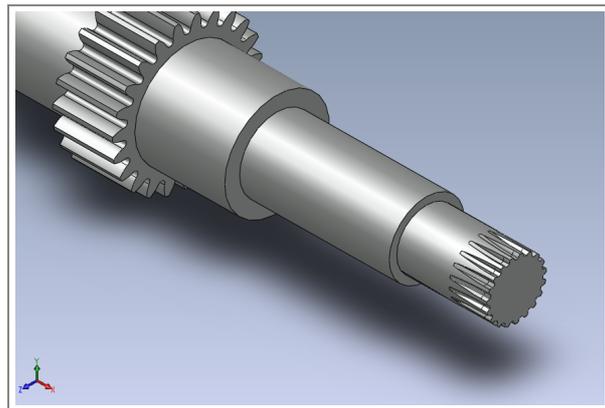
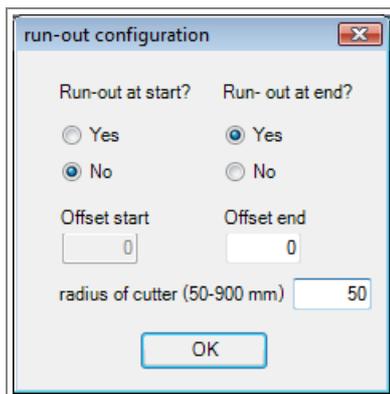
To start the calculation, which belongs to the opened 3D model, click on the menu item `Tools => eAssistant (TBK) => Calculation` and select the corresponding calculation module. The following dialog window appears. Click the "Yes" button and the eAssistant/TBK opens the calculation module with all calculation values.



Even if there are several different gear calculations in one 3D model, you can open every calculation file again. For the gear, for example, click on the feature tree on the left side in SOLIDWORKS to select the corresponding cutout. Click on the menu item `Tools => eAssistant (TBK) => Calculation` and then click on the calculation module.

10. Add a Tool Runout to the 3D Model

For pinion shafts you can add a tool runout to the 3D model. For this purpose, you have to specify a cutter radius. Select the menu item **Tools => eAssistant (TBK) => Calculation => Gears => Cylindrical gear pair** and run a cylindrical gear calculation. Click the button **CAD => SOLIDWORKS export**. Choose a plane of the shaft in SOLIDWORKS and click the menu item **Tools => eAssistant (TBK) => New part => Cylindrical gears => Cylindrical gear 1**. The dialog window is opened. Activate the option "Yes" for "Run-out at start" or "Run-out at end". Add a cutter radius and confirm with "OK".

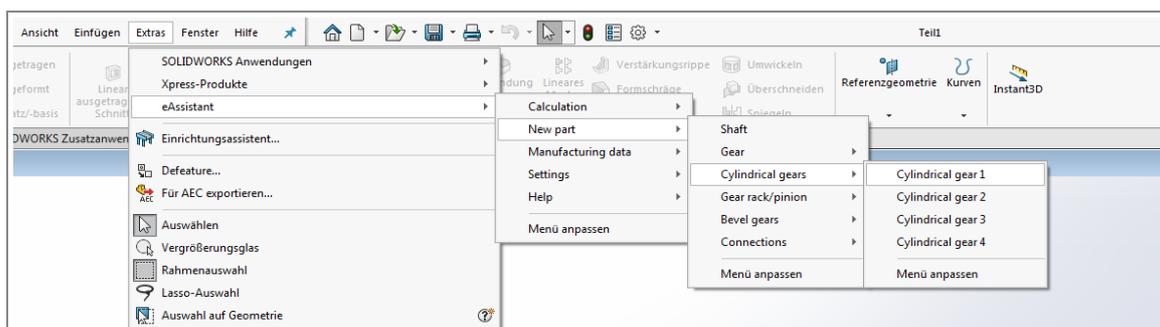


Now you can check if the tool runout collides with the following shaft segment, so that you can correct it. If necessary, you can add a straight element by using the option "Offset".

11. Bidirectional Connection between eAssistant/TBK and SOLIDWORKS

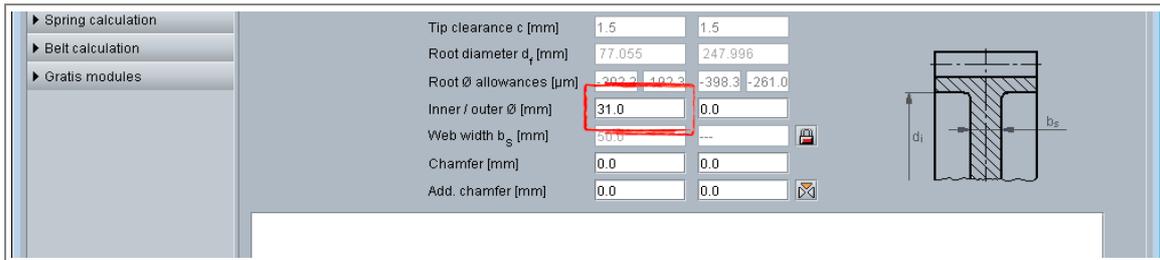
The attractiveness of the CAD plugin is additionally enhanced by the bidirectional connection between eAssistant/TBK and SOLIDWORKS. As changes are made to a calculation, the 3D model can be updated (shafts, cylindrical gear, or involute spline profile/hub profile as single part).

Select the menu item **Tools => eAssistant (TBK) => Calculation => Gears => Cylindrical gear pair** and make a cylindrical gear calculation. Click the button **CAD => SOLIDWORKS export** to transfer the calculation values to SOLIDWORKS. Click the SOLIDWORKS menu item **Tools => eAssistant (TBK) => New part => Cylindrical gears => Cylindrical gear 1** to create the cylindrical gear.

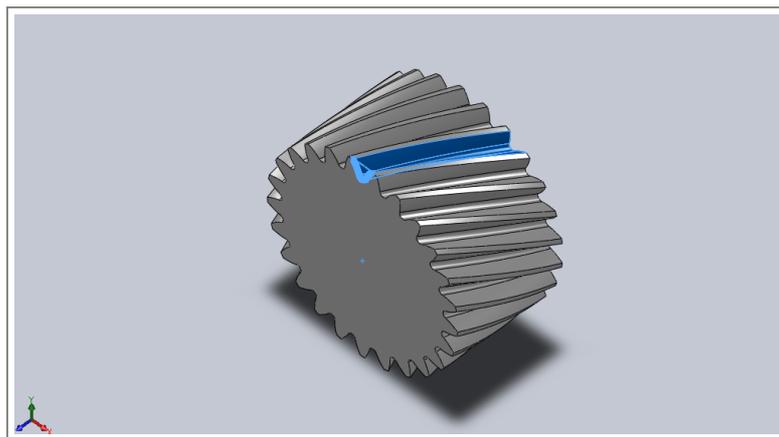


To make necessary changes on the CAD model, re-open the eAssistant/TBK module and modify the calculation. Because the calculation information is saved to the CAD model, you can start the eAssistant/TBK calculation by clicking **Tools => eAssistant => Calculation => Gears => Cylindrical gear pair**. Confirm the following message "Shall the calculation of the active part be loaded?" with the button "Yes" and the related calculation module starts.

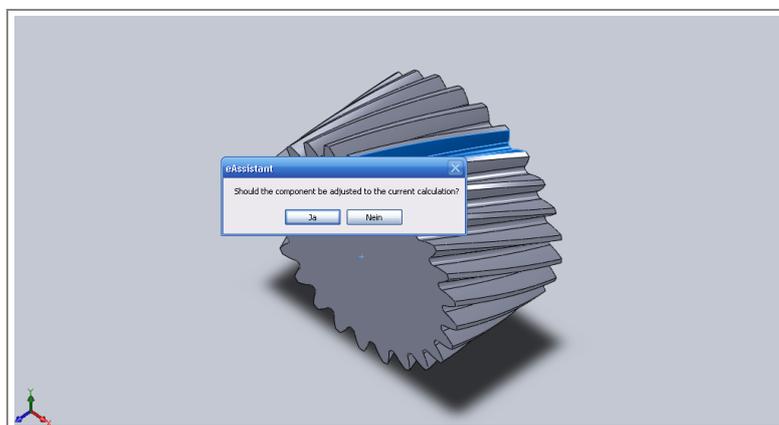
For example, you can add a hole to gear 1. Click on the button **CAD => SOLIDWORKS export**.



Select the item "Extrusion2" from the integrated SOLIDWORKS feature tree. The area between the teeth will then be highlighted.

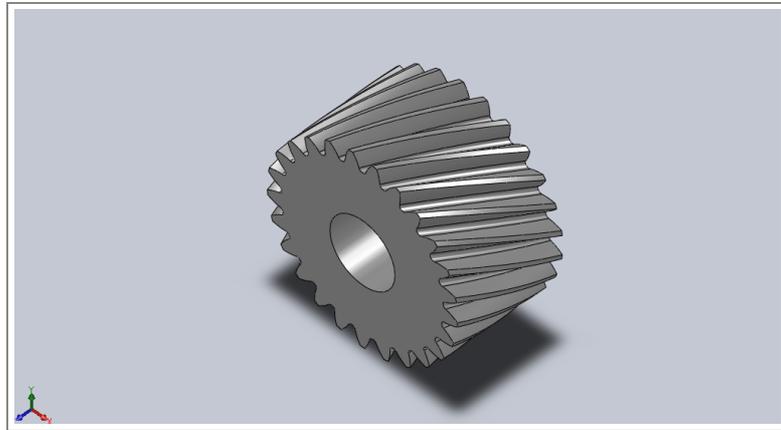


Select **Tools => eAssistant (TBK) => New part => Cylindrical gears => Cylindrical gear 1** and the message "Should the component be adjusted to the current calculation?" appears.



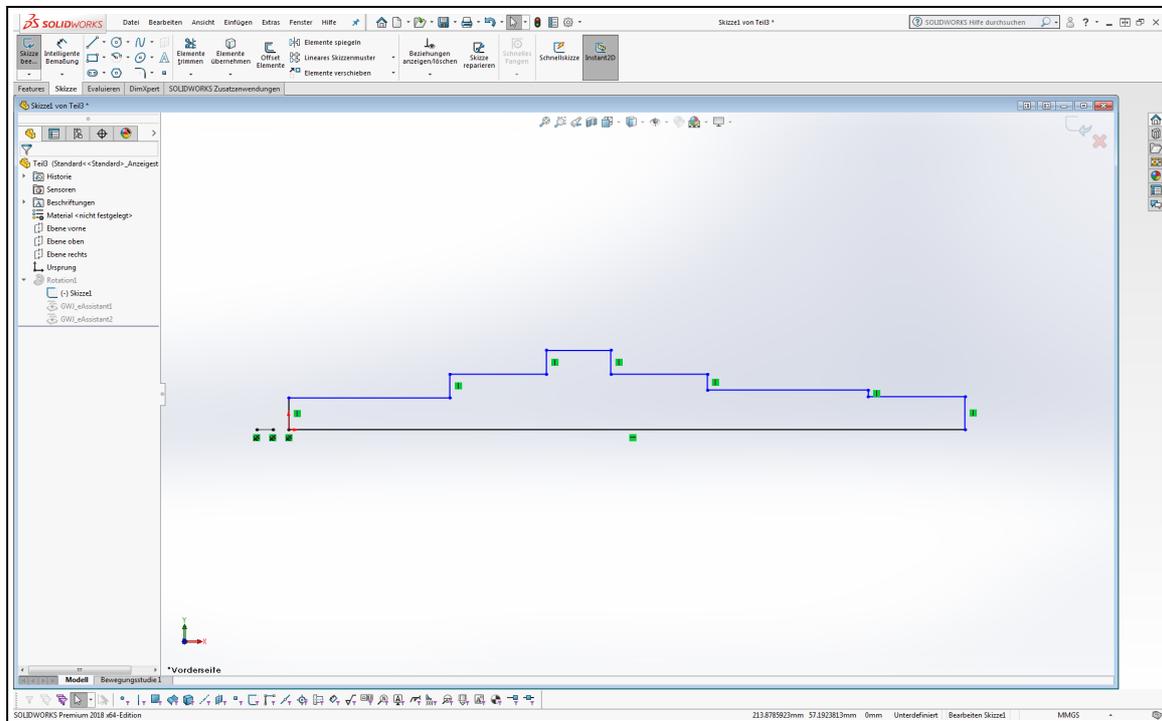
Kollendurchmesser (mm)		
Zwei-Rollenmaß	(spielfrei)	10.502
	(obere)	10.407
	(untere)	10.357
Zahndickenstärke		7.675
Hohe über der Zahn (mm)		

Click the "Yes" button and the CAD model is updated immediately and quickly adjusted to the current calculation.

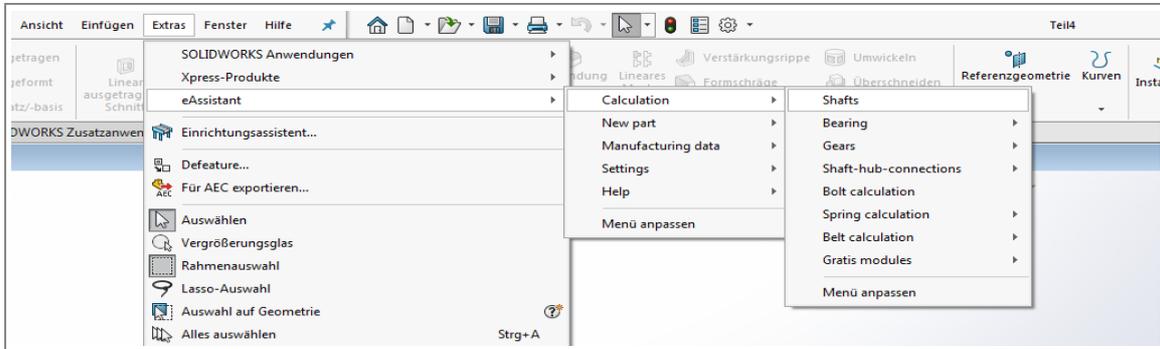


12. Create a Shaft in SOLIDWORKS and Import it into eAssistant/TBK

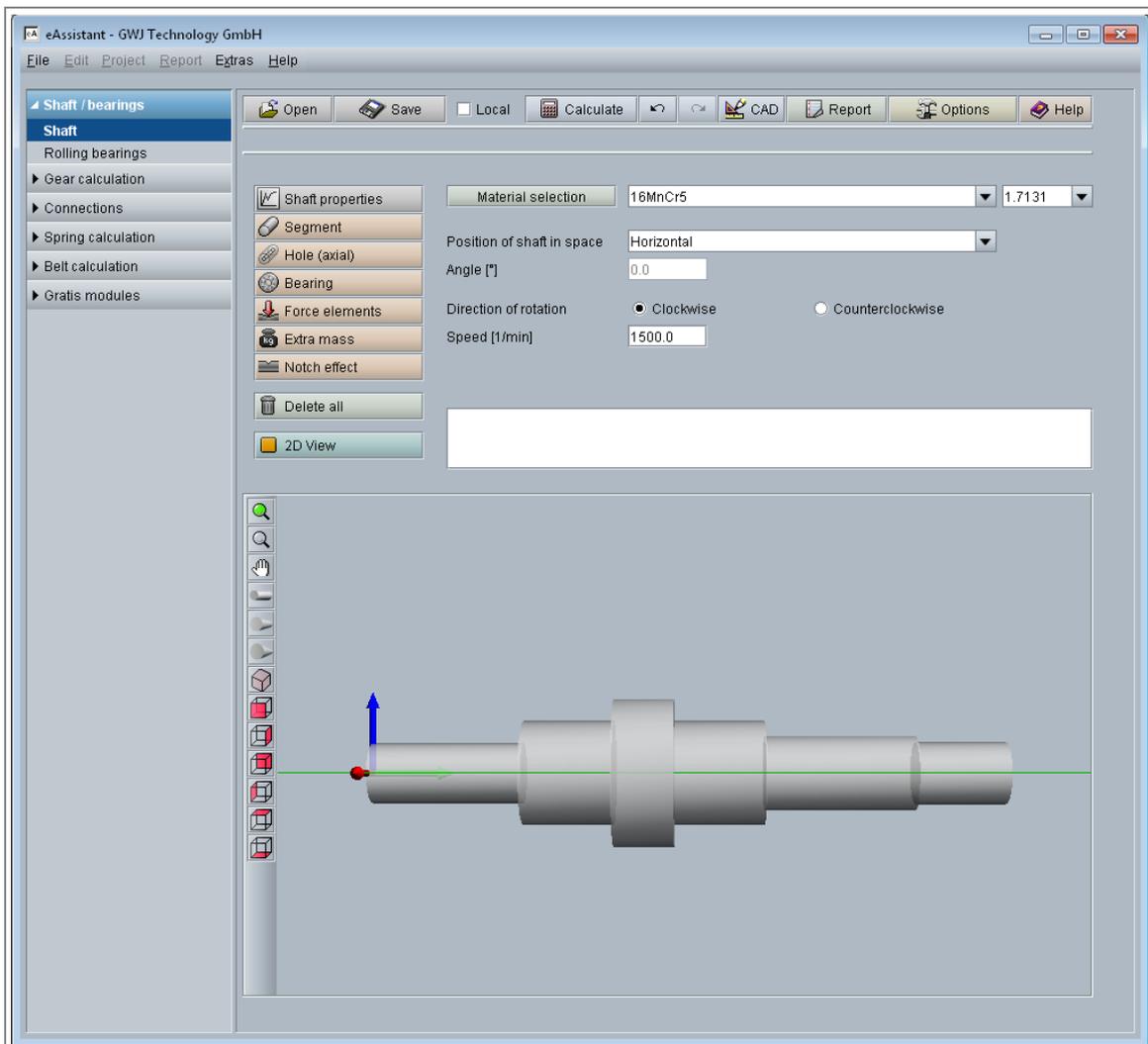
You can create a shaft directly in SOLIDWORKS and then import it into eAssistant/TBK. Solid and hollow shafts with cylindrical and conical shaft segments are possible. To create a shaft as a revolved part, use the CAD's revolve features, the inner shaft geometry can be generated using the revolved cut feature.



To create a shaft as a revolved part, use the SOLIDWORKS **Revolve Feature**. Select the corresponding feature(s) from the FeatureManager design tree and import the shaft into eAssistant/TBK by clicking the menu item **Tools => eAssistant (TBK) => Calculation => Shafts**.



The eAssistant/TBK shaft module opens and the shaft is loaded into the calculation module.



Rollendurchmesser (mm)
Zwei-Rollenmaß
-
Zahndickenmaß
-
Höhe über der Zahn (mm)

(spielfrei)	10.502
(obere)	10.407
(untere)	10.357
	7.675

We are looking for ideas, suggestions or criticism, so we would
like to know what you think.
You will always find a sympathetic ear, no matter what the problem is.

We appreciate your feedback.

GWJ Technology GmbH
Celler Strasse 67-69
38114 Braunschweig, Germany
Phone: +49 (0) 531 129 399-0
Fax: +49 (0) 531 129 399-29
Email: info@gwj.de

Visit our web site www.eAssistant.eu and www.tbksoft.com to find detailed
information or videos on the eAssistant/TBK software and the 3D CAD plugin.

Product information:

eAssistant and TBK are products of GWJ Technology GmbH.