KISSsoft AG - +41 55 254 20 50
Uetzikon 4 - +41 55 254 20 51
8634 Hombrechtikon - info@KISSsoft. AG
Switzerland - www. KISSsoft. AG

KISSsoft Tutorial: Sizing a Planetary Gear Set for Precision Mechanics

1 Task

To size a planetary gear set with an input torque of 450 Nmm (0.45 Nm) at 10000 rpm. The nominal transmission ratio is 4.25. The required service life is 20,000 hours, with an application factor of K_A=1.25.
The package size (external diameter of the gear rim) is 35 mm, including 3 mm material between the root diameter and the external diameter. The gears are made of sintered powdered metal. The module must be greater than 0.5 mm (due to manufacturing requirements). The tooth form must be optimized to make full use of the fact that the gears are not manufactured using the generation process. The calculation method used here is the one specified in AGMA: 2101-D04.

2 Starting KISSsoft

2.1 Starting the software

You can call KISSsoft as soon as the software has been installed and released. Usually you start the program by clicking "Start→Program Files→KISSsoft→03-2011KISSsoft". This opens the following KISSsoft user interface:

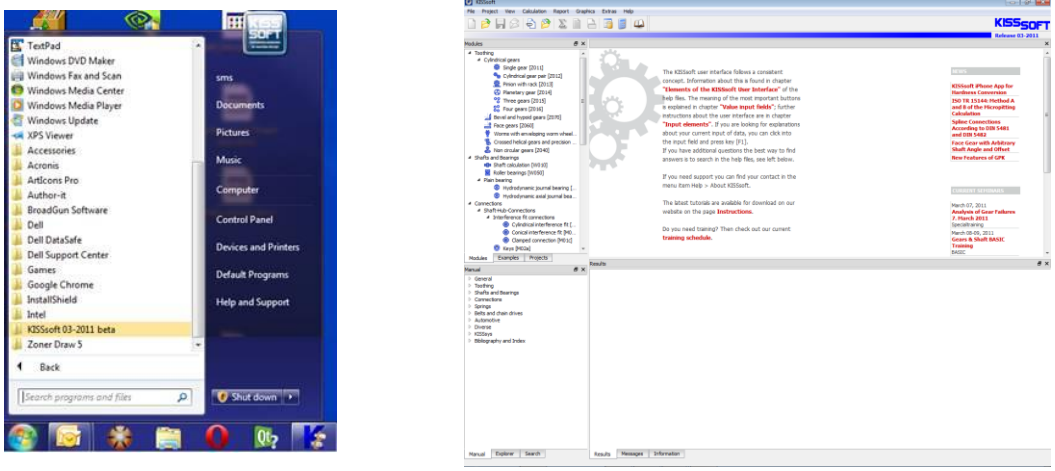


Figure 2.1 KISSsoft main window

2.2 Starting the "Planetary gear" calculation module

In the **"Modules tree"** window, double-click the **"Modules"** tab to call the calculation for a **"Planetary gear"**, see Figure 2.2.

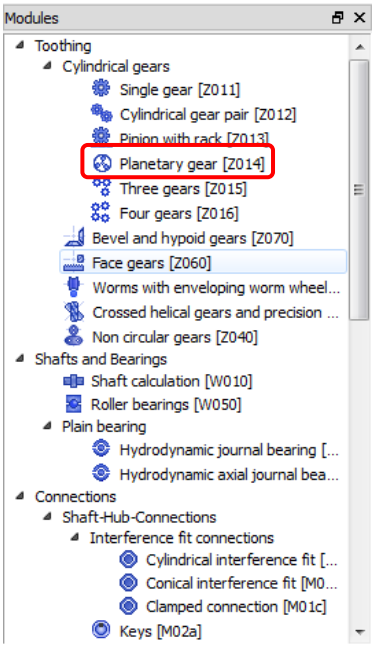
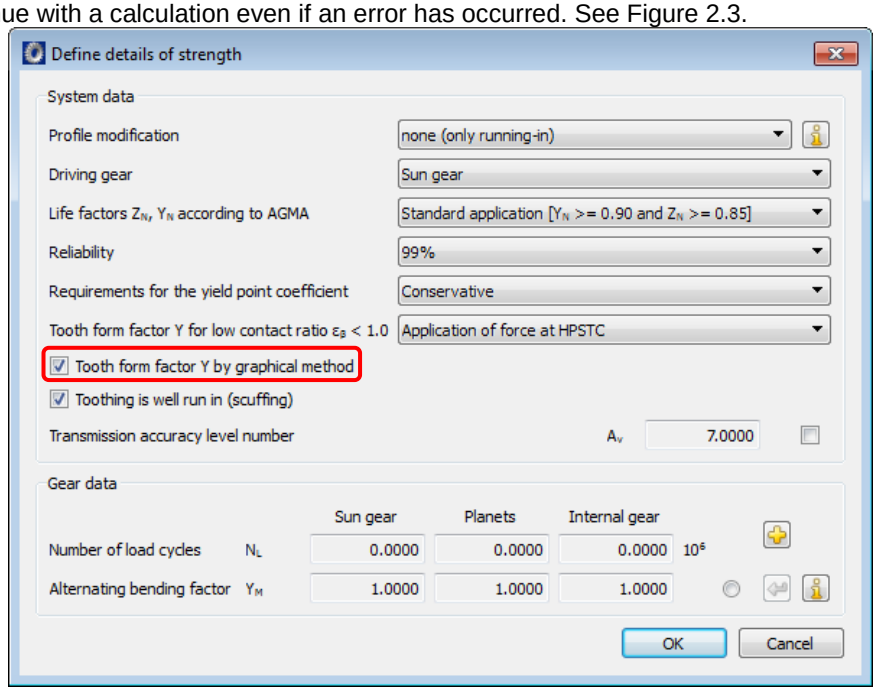


Figure 2.2 Selecting the "Planetary gear" calculation module from the "Modules" window

2.3 Basic settings

If the AGMA 2101-method is used for a planetary gear set, it is a good idea to activate the graphical method for factor Y (as this influences the calculation of root stress). To do this, go to the "Strength" tab, select "Details" and click the "Pair data" group. Activate the graphical method and define where the force is to be applied. As some of the solutions found during the draft design phase will have geometric errors (which cause KISSsoft to cancel the calculation automatically), we recommend you go to the module specific settings and activate "Allow large profile shift" and "Don't abort when geometry errors occur". This allows the KISSsoft software to continue with a calculation even if an error has occurred. See Figure 2.3.



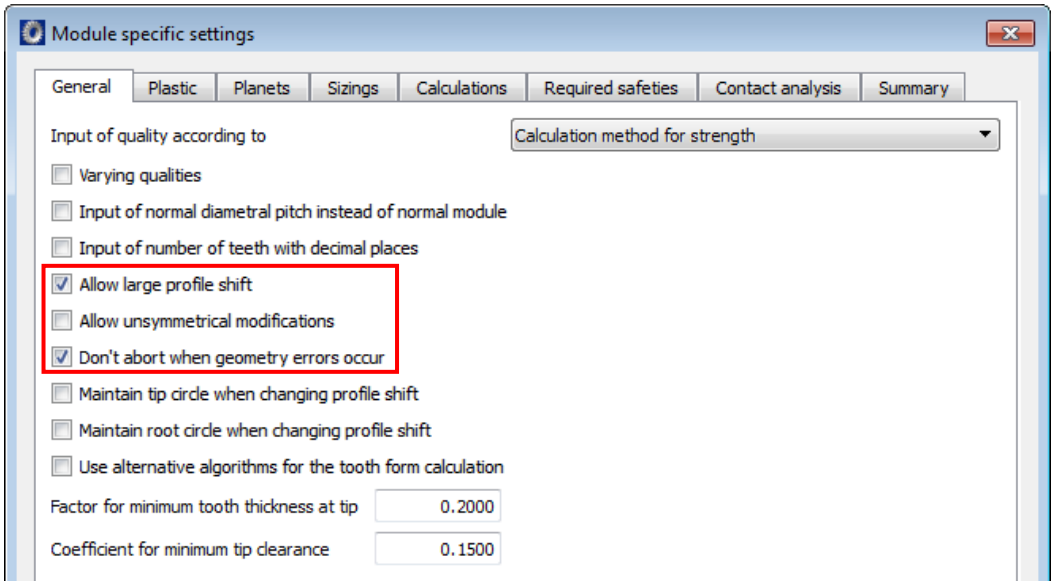


Figure 2.3 "Define details of strength" and "Module specific settings" for this example

2.4 Setting constraints

Go to the "Geometry" tab and input the required number of planets (Figure 2.4). The load distribution coefficient K_γ increases the load placed on an individual planet. In this case, set it to 1.0.

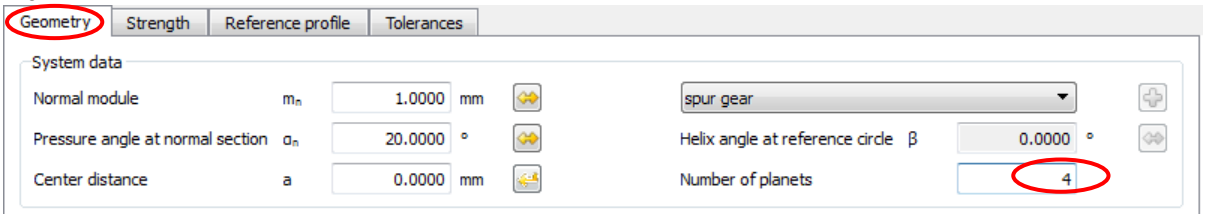


Figure 2.4 Defining the number of planets

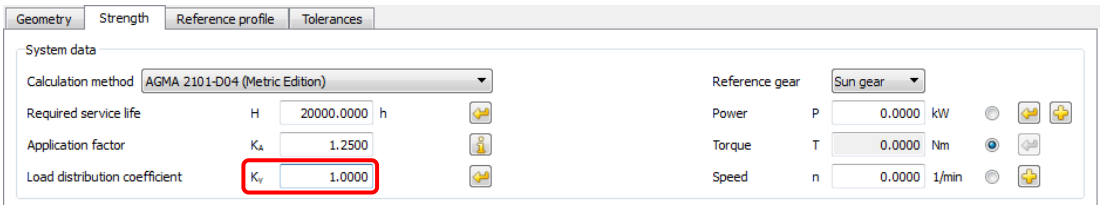


Figure 2.5 Defining the load distribution coefficient

2.5 Rough sizing

Click [OK] to return to the main dialog. Open Rough sizing and specify the required calculation method (1) and the material (2). Then input the application factor (3) and the service life (4). Click the radio button next to the Power field to define the load (5), see Figure 2.7.

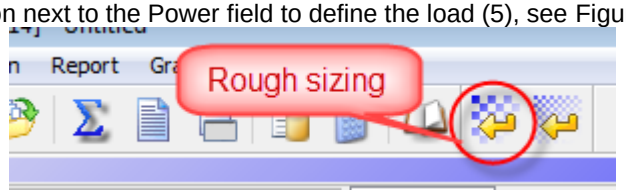


Figure 2.6 Call the Rough sizing function

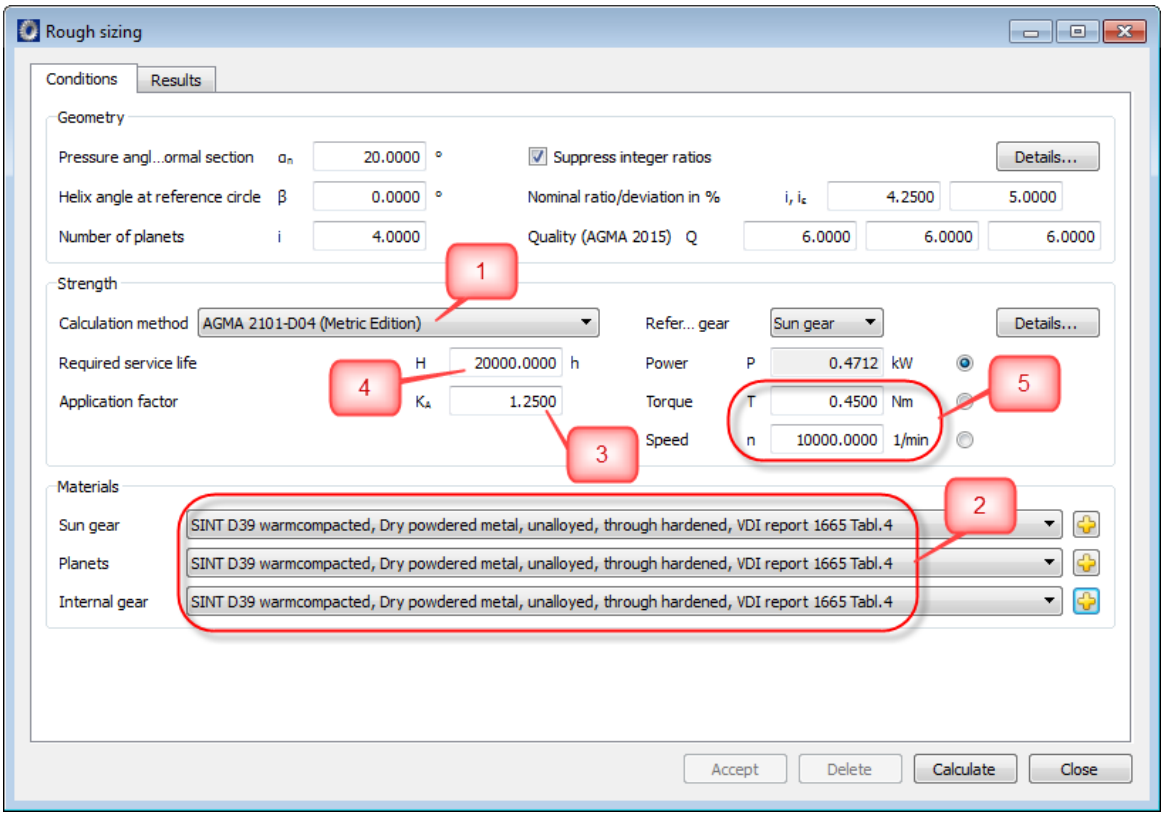


Figure 2.7 Setting the materials, calculation method, application factor and required service life

Specifying the load

To specify the unit used for torque, click the right-hand mouse button on the appropriate field (Figure 2.8).

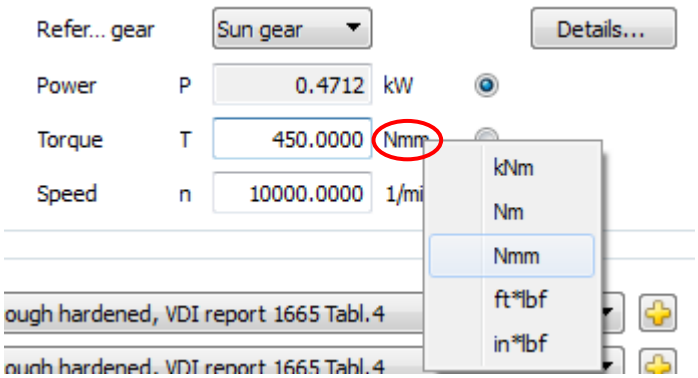


Figure 2.8 Specifying the unit for torque

Define the reference gear (1), the calculated value (2) (if the torque and number of rotations have been defined, the performance will be calculated) and input the data for the number of rotations and torque (3) (see Figure 2.9).

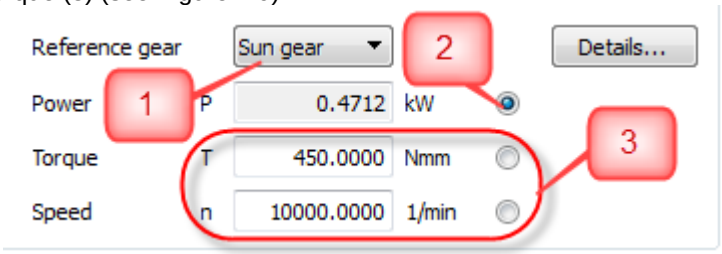


Figure 2.9 Specifying the load

Then enter the nominal transmission ratio (6).

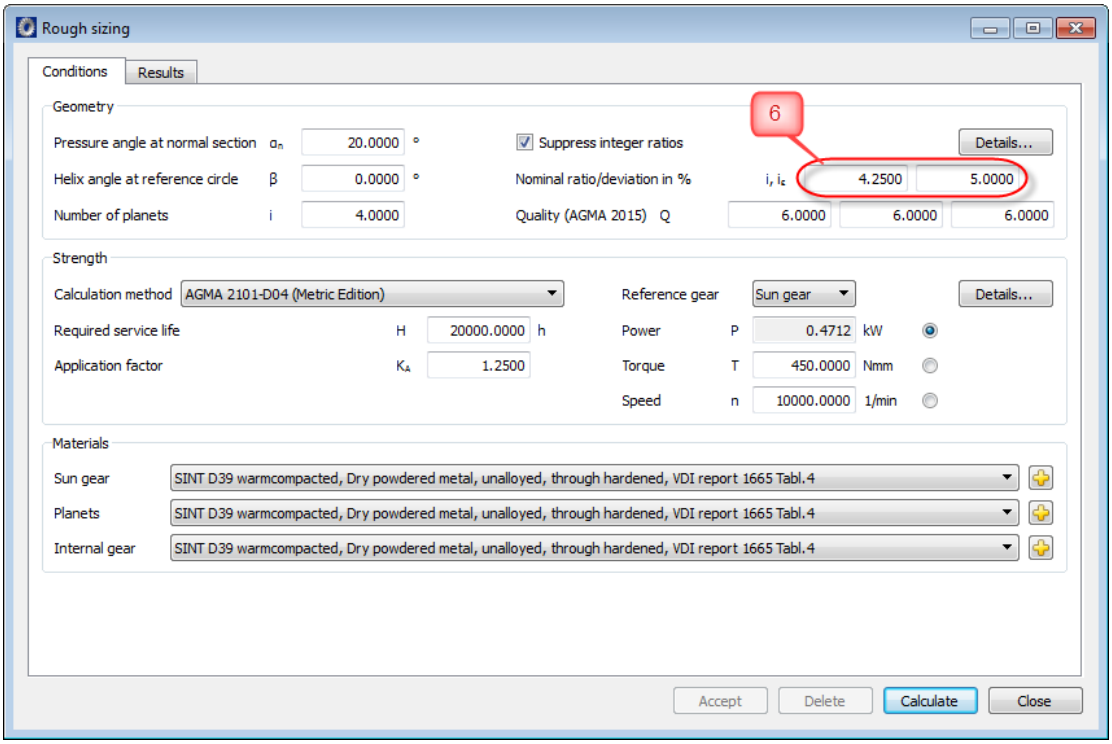


Figure 2.10 Rough sizing settings

If KISSsoft were to calculate a design with the basic settings, it would result in an extremely small module. For this reason, you should lower the value range for the number of teeth from 9 to 12 (7), to force KISSsoft to select a larger module. It is not usually necessary to change the default value for the number of teeth. Then click **"OK"** to close the input window.

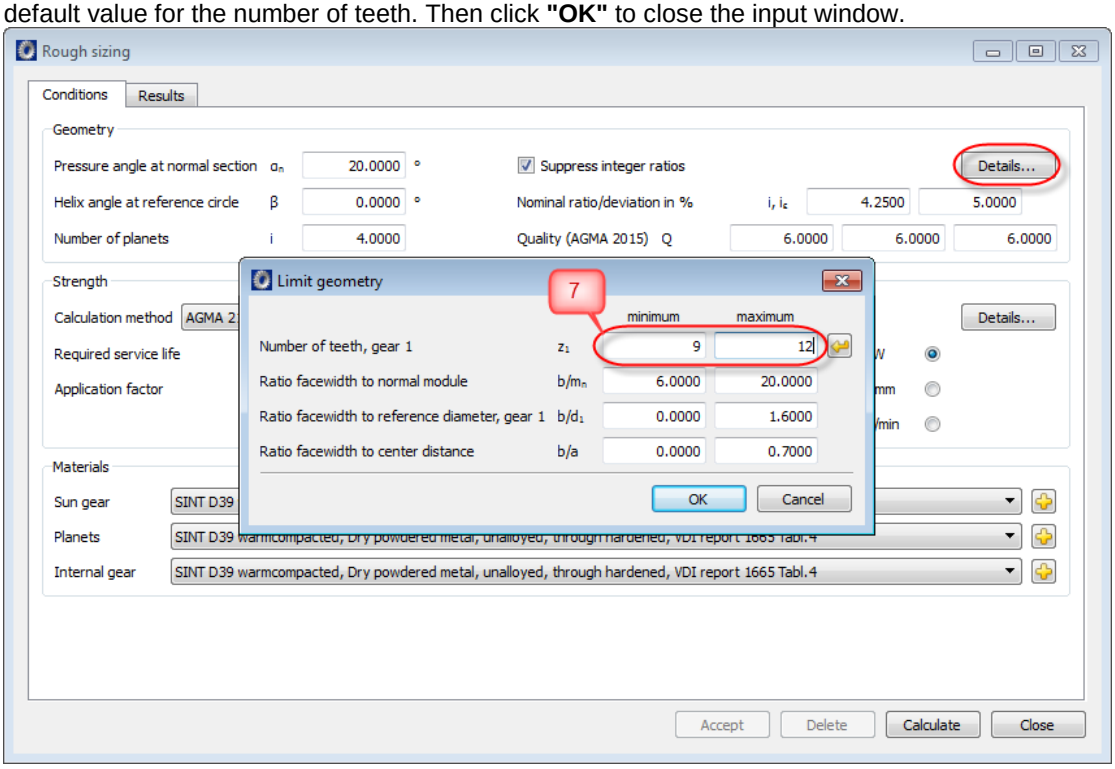


Figure 2.11 Rough sizing default settings "Limit geometry"

Click [Calculate] (8). Look at the results.

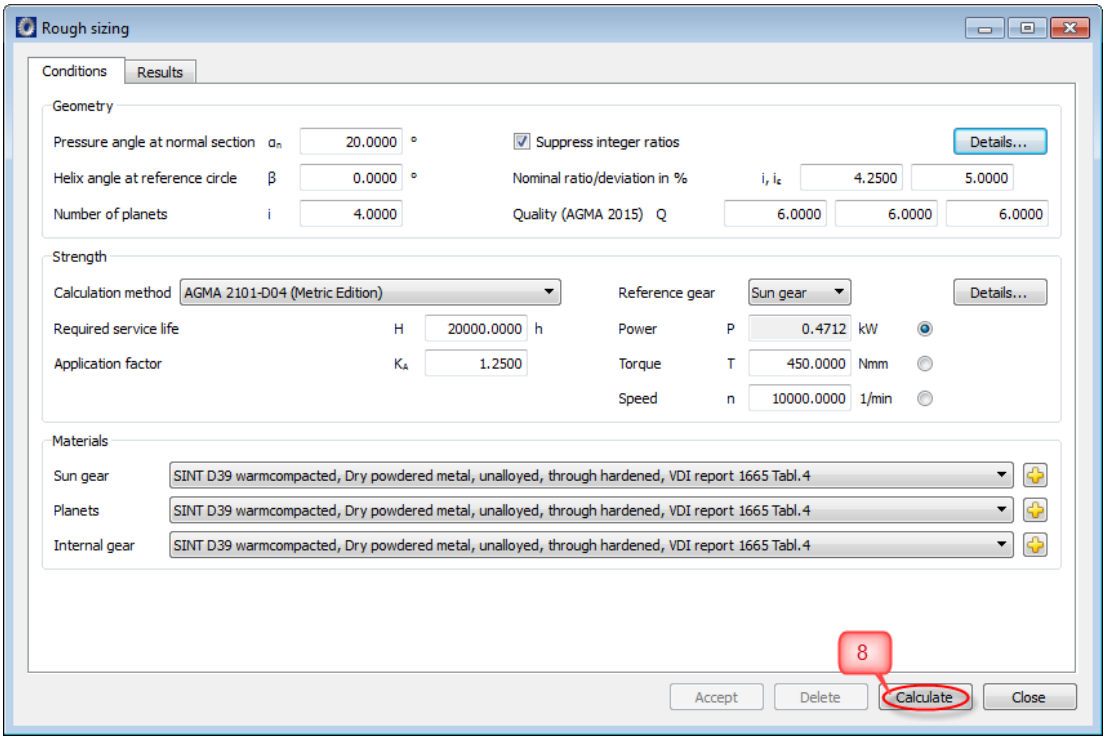


Figure 2.12 Calculating rough sizing

If you are happy with this, return to the dialog by closing the editor and clicking on [Accept] (9), see Figure 2.13.

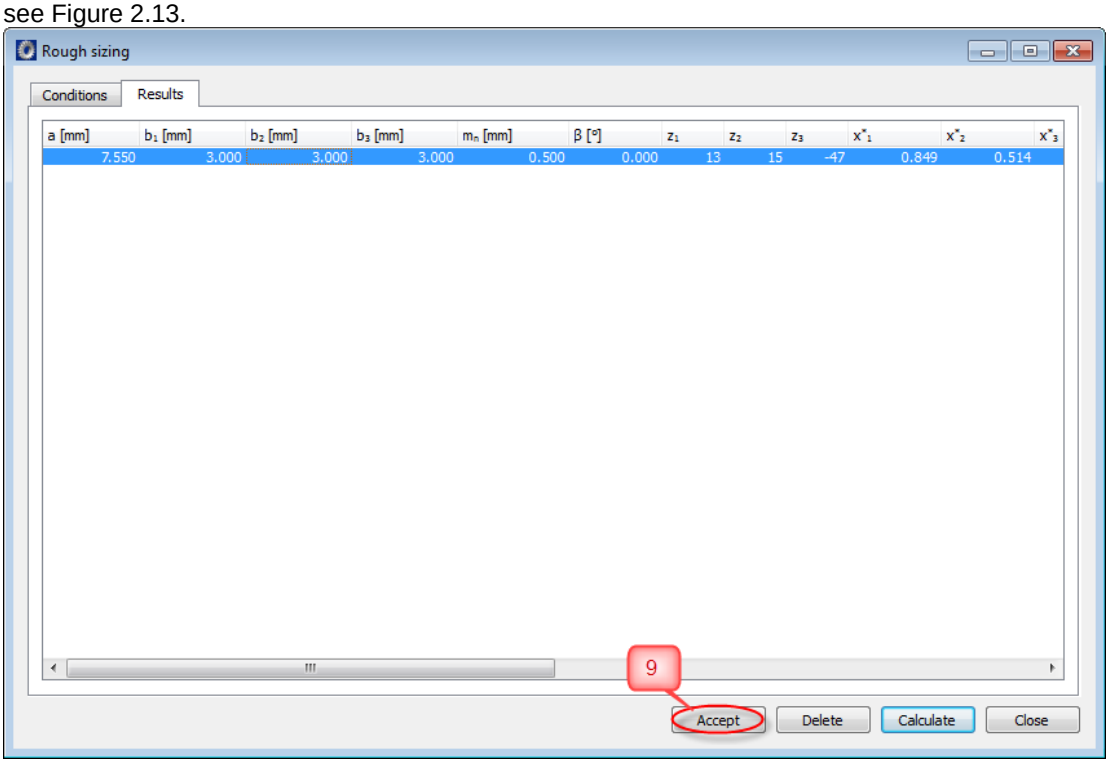


Figure 2.13 Rough sizing result

The main dialog is now filled with data from the solution generated by the Rough sizing function. As the module is smaller than 1.0, we recommend you use a different standard for the tolerances. To do this, go to the "Tolerances" tab. In the Tolerances dialog, select the tooth thickness deviation "DIN 58405 10e" for each gear, Figure 2.14. Here, 10 indicates the quality (interval width), where 10 means "lower quality". The letter "e" represents the definition of the interval limit and therefore also the backlash.

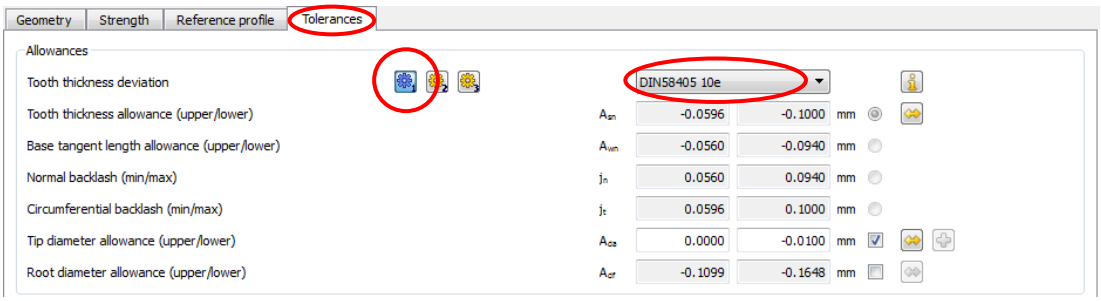


Figure 2.14 Setting the tolerance

After you have defined the tolerances, you may want to input a better value for the facewidth (in this example we are leaving the values unchanged (1), see Figure 2.15. Click [Calculate F5] (2). You now see the first results of the roughly sized planetary gear set in the "Results overview" (2a) (Figure 2.15).

2.6 Fine Sizing

This completes the presizing step. This is used to provide the Fine sizing function with values that are approximately the correct size for the subsequent processing steps. Now, to generate an optimized solution, click the Fine sizing (3) button Figure 2.15.

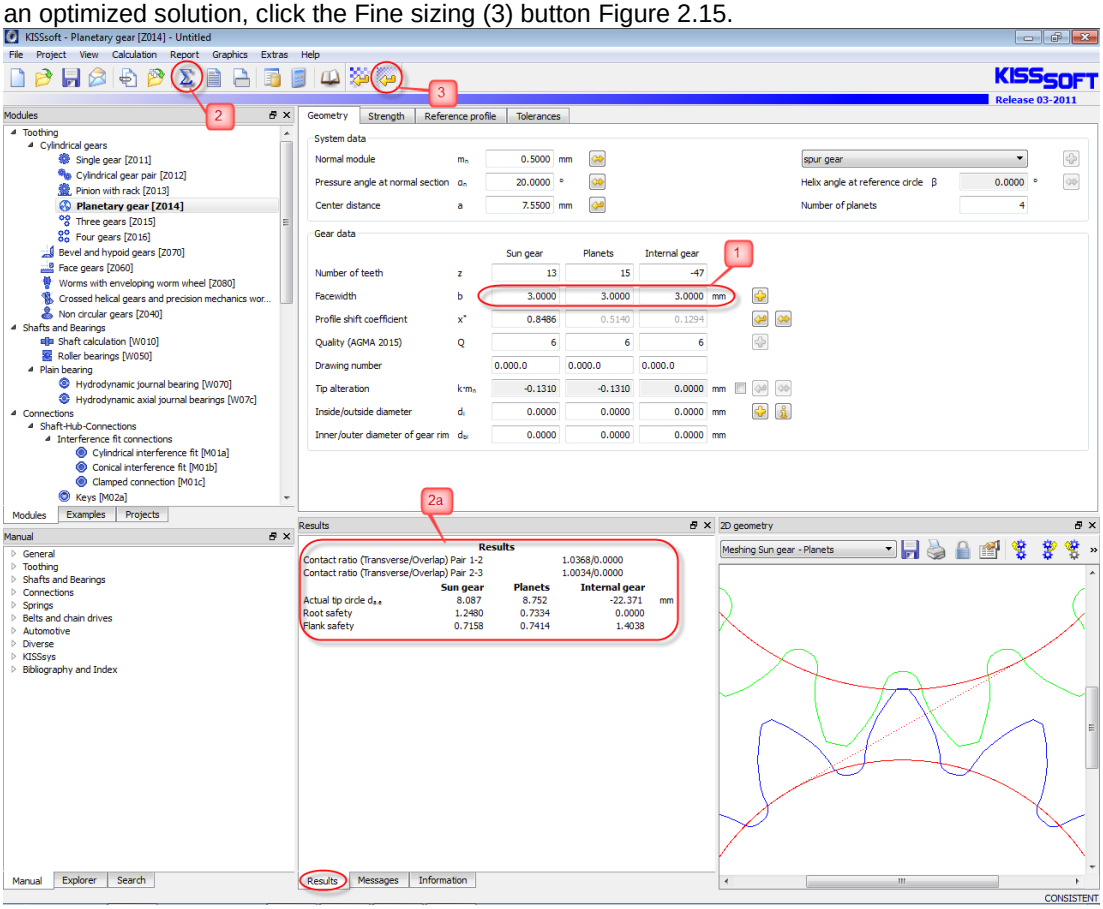


Figure 2.15 Calculate, check the results and call the Fine sizing function

Firstly, check the value for the nominal conversion (this may have changed slightly during Rough sizing) (1). Then input the required values range and the increments for the module (KISSsoft will automatically select very small values) (2). Define the target value for the gear rim reference circle (3). To define the correct diameter, deduct 3 mm of the material underneath the root diameter twice from the gear's external diameter (35 mm). This gives a measurement of 29 mm. Then reduce this again by a further 2*1 mm for the dedendum (this value does not need to be correct because the permissible deviation is set to 10%).

Click the "Sizing button" for the center distance (4) to display the possible value range for this value. In order to force the gear's root diameter to be small enough to ensure sufficient material

remains below the tooth root, you must input a suitable value in the relevant field (here: 35-2*3 mm = 29 mm) (5), see Figure 2.16.

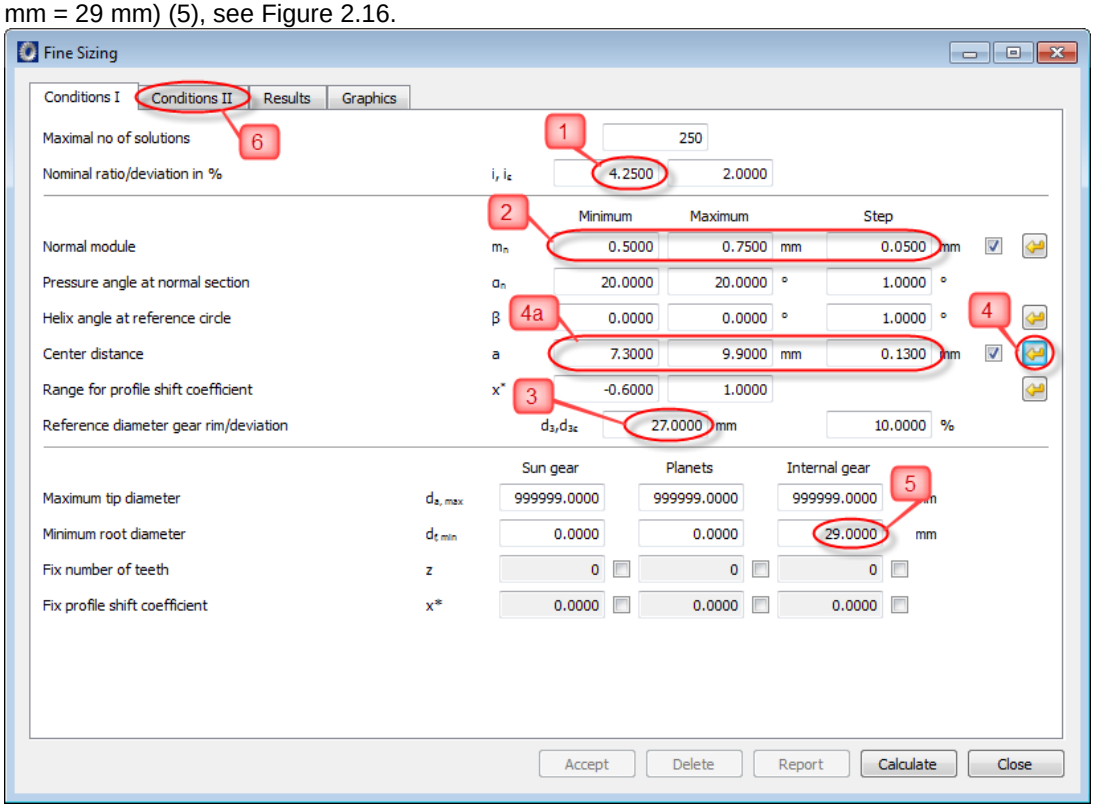


Figure 2.16 Settings for Fine sizing

As you are using the graphical method to determine the root stress, you must calculate the tooth form for every solution that is found. To do this, click [Conditions II] (6).

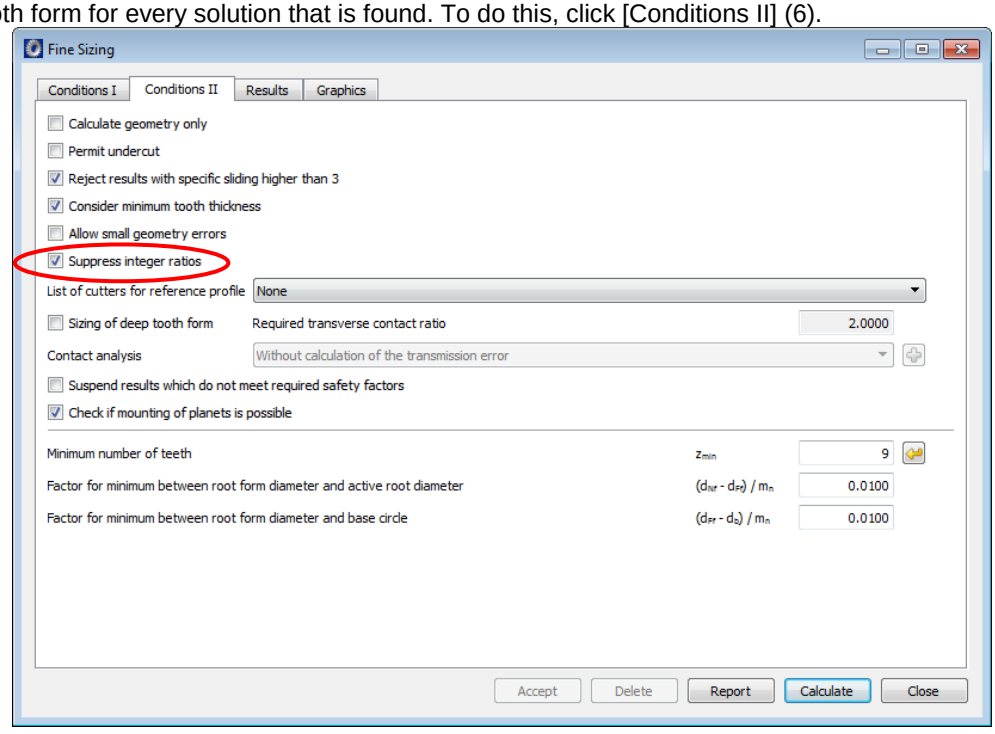


Figure 2.17 Calculating mesh stiffness

Click [Calculate] to start Fine sizing. If a solution is found (if not, a message appears to tell you no solutions could be found), click on the "Graphics" tab.

If you now only want to include the safety factors, Solution 18 looks very promising: the safety factor for the root is large enough and the flank safety is more than adequate. As you can usually improve tooth root safety by modifying the root geometry, flank safety is more important in this case.

Normally, you would also check the other criteria (such as transverse contact ratio, specific sliding, etc.). As this is closely connected with the particular problem you are dealing with, this issue is not discussed here, and the tutorial moves on to solution 18. Please refer to the tutorials list at the end of this document for more information about the Fine Sizing function.

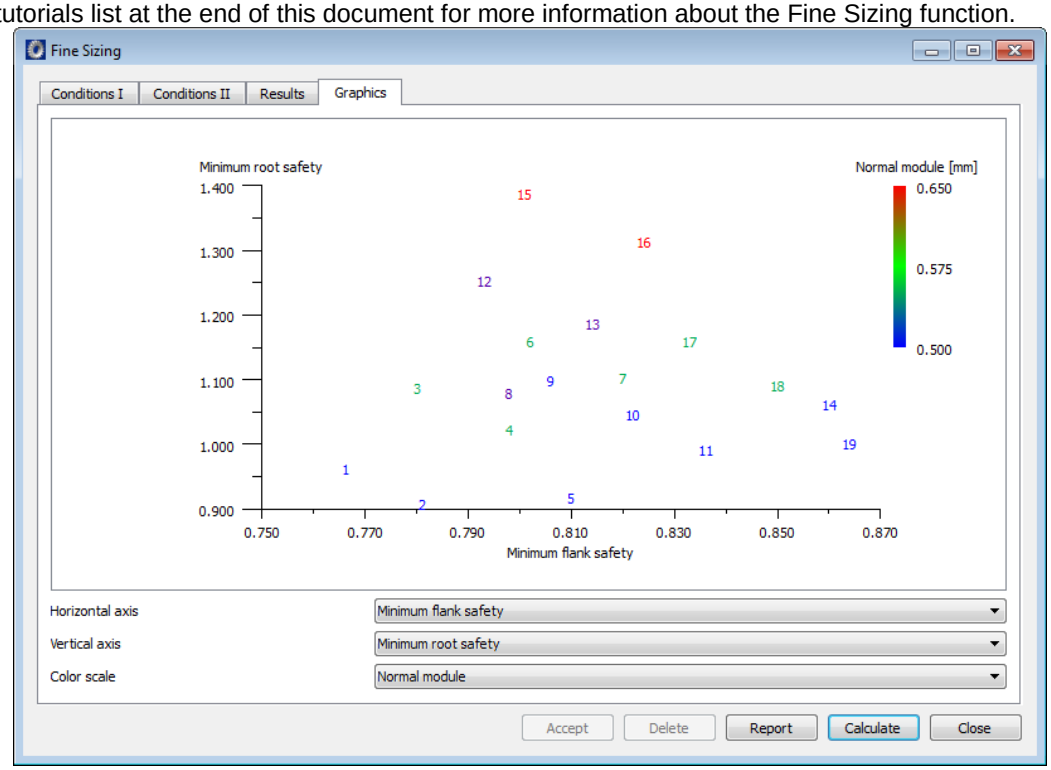
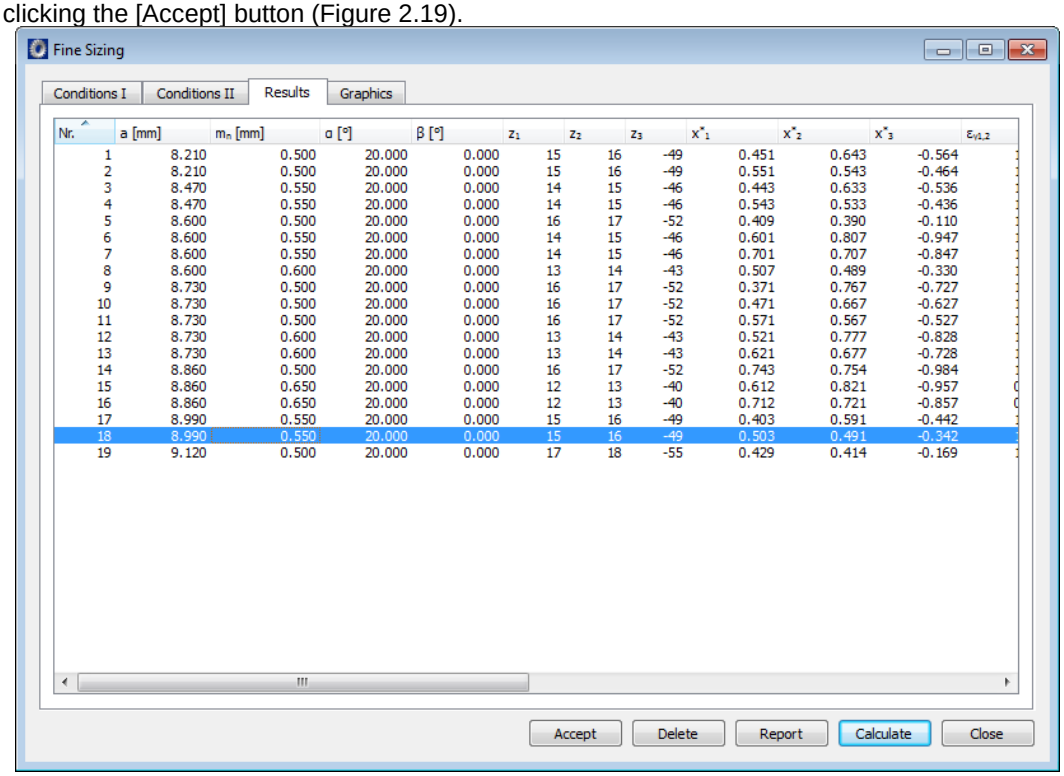


Figure 2.18 Graphical display of the results

Go to the Fine Sizing "Results" tab and select a variant by double-clicking on variant 18 or clicking the [Accept] button (Figure 2.19).



Nr.	a [mm]	m_n [mm]	α [°]	β [°]	z_1	z_2	z_3	x^*_1	x^*_2	x^*_3	$\varepsilon_{\gamma 1,2}$
1	8.210	0.500	20.000	0.000	15	16	-49	0.451	0.643	-0.564	1
2	8.210	0.500	20.000	0.000	15	16	-49	0.551	0.543	-0.464	1
3	8.470	0.550	20.000	0.000	14	15	-46	0.443	0.633	-0.536	1
4	8.470	0.550	20.000	0.000	14	15	-46	0.543	0.533	-0.436	1
5	8.600	0.500	20.000	0.000	16	17	-52	0.409	0.390	-0.110	1
6	8.600	0.550	20.000	0.000	14	15	-46	0.601	0.807	-0.947	1
7	8.600	0.550	20.000	0.000	14	15	-46	0.701	0.707	-0.847	1
8	8.600	0.600	20.000	0.000	13	14	-43	0.507	0.489	-0.330	1
9	8.730	0.500	20.000	0.000	16	17	-52	0.371	0.767	-0.727	1
10	8.730	0.500	20.000	0.000	16	17	-52	0.471	0.667	-0.627	1
11	8.730	0.500	20.000	0.000	16	17	-52	0.571	0.567	-0.527	1
12	8.730	0.600	20.000	0.000	13	14	-43	0.521	0.777	-0.828	1
13	8.730	0.600	20.000	0.000	13	14	-43	0.621	0.677	-0.728	1
14	8.860	0.500	20.000	0.000	16	17	-52	0.743	0.754	-0.984	1
15	8.860	0.650	20.000	0.000	12	13	-40	0.612	0.821	-0.957	0
16	8.860	0.650	20.000	0.000	12	13	-40	0.712	0.721	-0.857	0
17	8.990	0.550	20.000	0.000	15	16	-49	0.403	0.591	-0.442	1
18	8.990	0.550	20.000	0.000	15	16	-49	0.503	0.491	-0.342	1
19	9.120	0.500	20.000	0.000	17	18	-55	0.429	0.414	-0.169	1

Figure 2.19 Selecting a solution

When you return to the main dialog (Figure 2.20) the "Results overview" window gives a brief overview, whereas the entire data record for the selected solution is displayed in the report. Click "F6" to display the report. At this point, you have finished sizing the planetary gear.

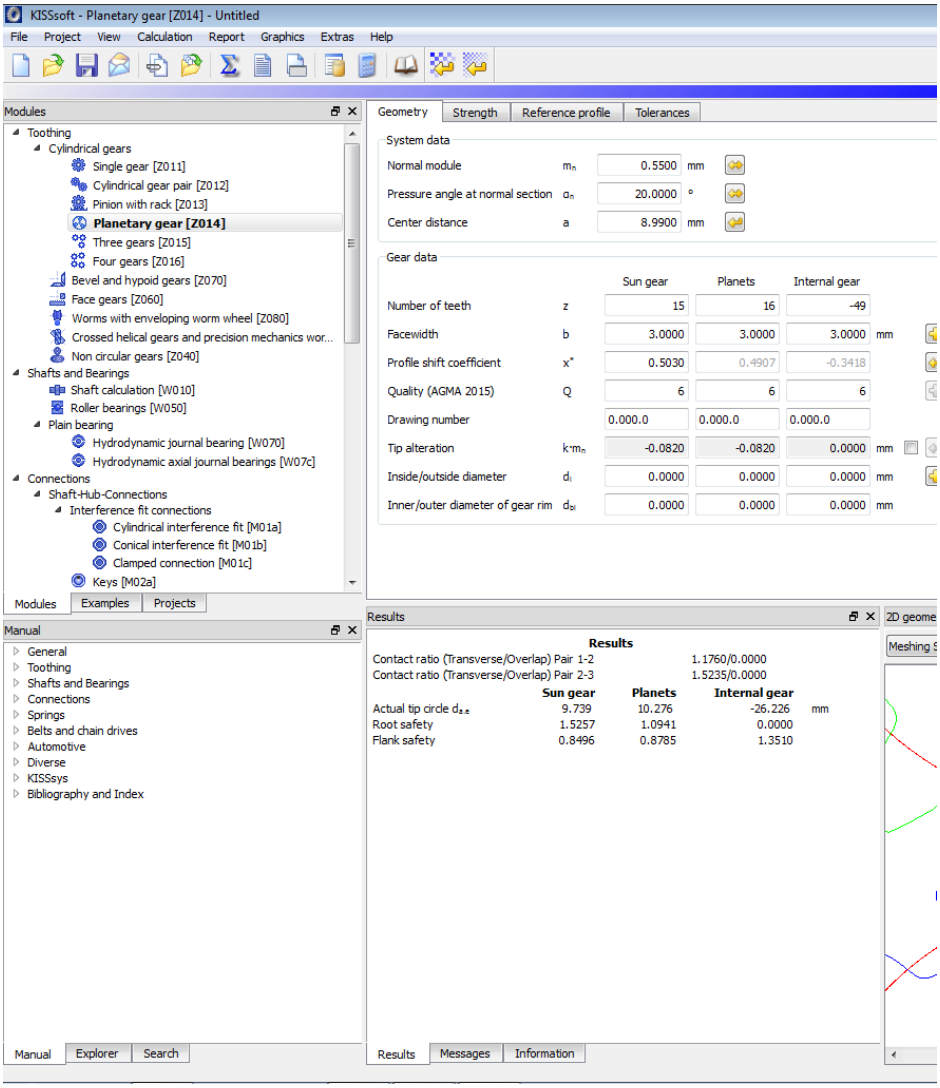


Figure 2.20 Overview of calculation results

2.7 Optimizing the tooth form

After the gear layout process is finished, the next step is to optimize the tooth form. As the gears are not manufactured by the generation process (in this case sintered), you can make modifications at no additional cost. This section describes the most commonly used modifications to the tooth form of sintered gears. For more details on this topic, please refer to the tutorials list at the end of this document.

To input the tooth form modification, click the **"Calculation" -> "Modifications"** menu option. See Figure 2.21.

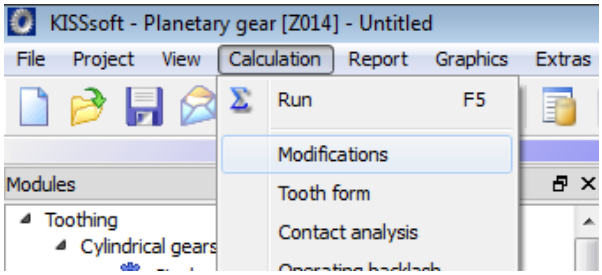


Figure 2.21 Activating the "Modifications" tab page

To improve initial tooth contact, and to take into account the shrinkage associated with the sintering manufacturing process, you must define tip rounding. To do this, select **"Rounding"** from the **drop-down list** for **"Type of tip modification"**, and input a value (here: 0.08 mm, which is already fairly large) in the input field for Gears 1 through 3 (see Figure 2.22).

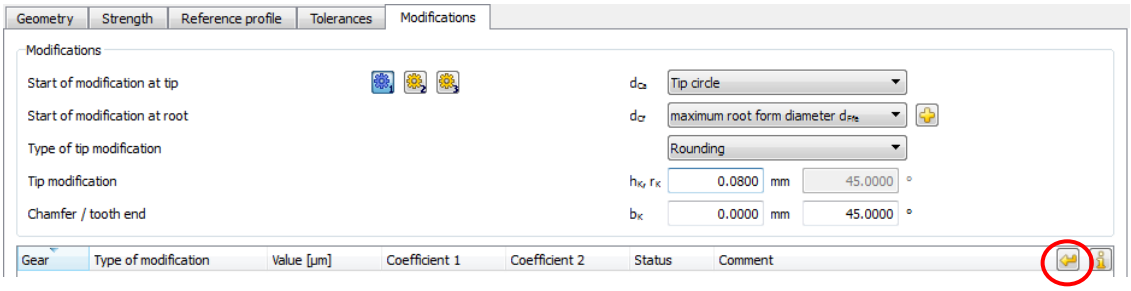


Figure 2.22 Tip modification "rounding" the tooth form

You can now input the tooth form modification in the active **"Modifications"** tab. Click the **"Sizing button"** to open the **"Sizing modifications"** window (Figure 2.23).

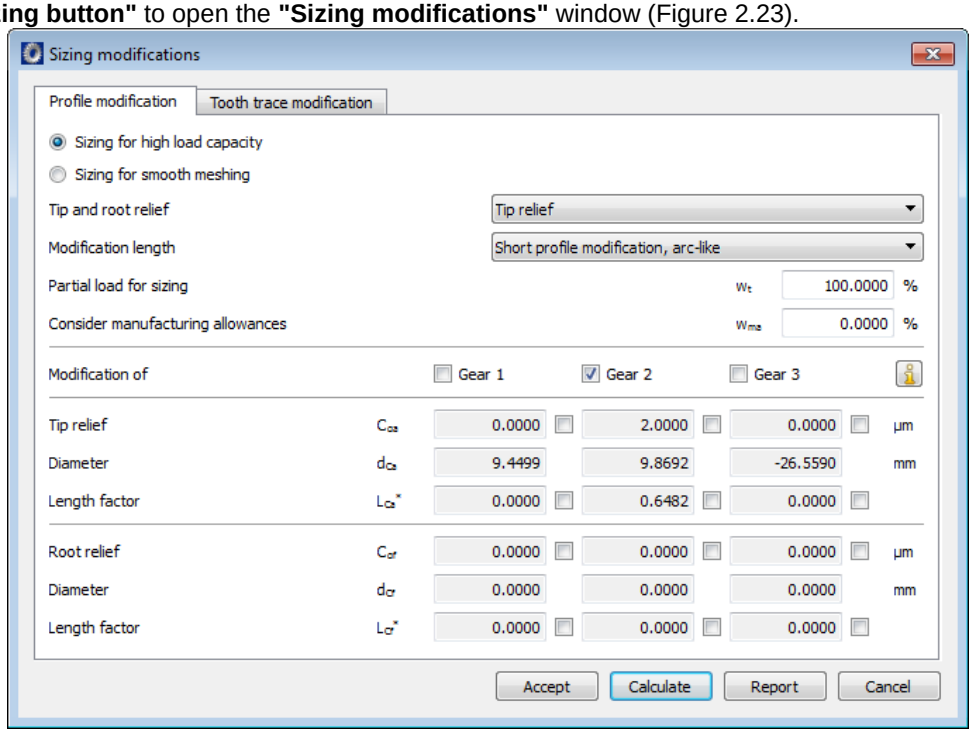


Figure 2.23 Defining profile correction details

The **"Short profile correction, curved"** is selected for the tip relief to keep the contact impact as small as possible. This should then be carried out on Gear 2. Select **"Short profile correction, curved"** and click the **"Calculate"** button. KISSsoft calculates a draft design for the tip relief, which (automatically) starts halfway between the single tooth contact point, with a value based on the calculated tooth bending value.

The best solution for the root is usually an ellipse with a larger radius at the end of the involute and a smaller radius in the middle of the root area between two teeth. To calculate this, click the "Tooth form" tab and then click "Insert elliptical root modification". Next click the "Sizing button" to display a suggested value for the elliptical form.

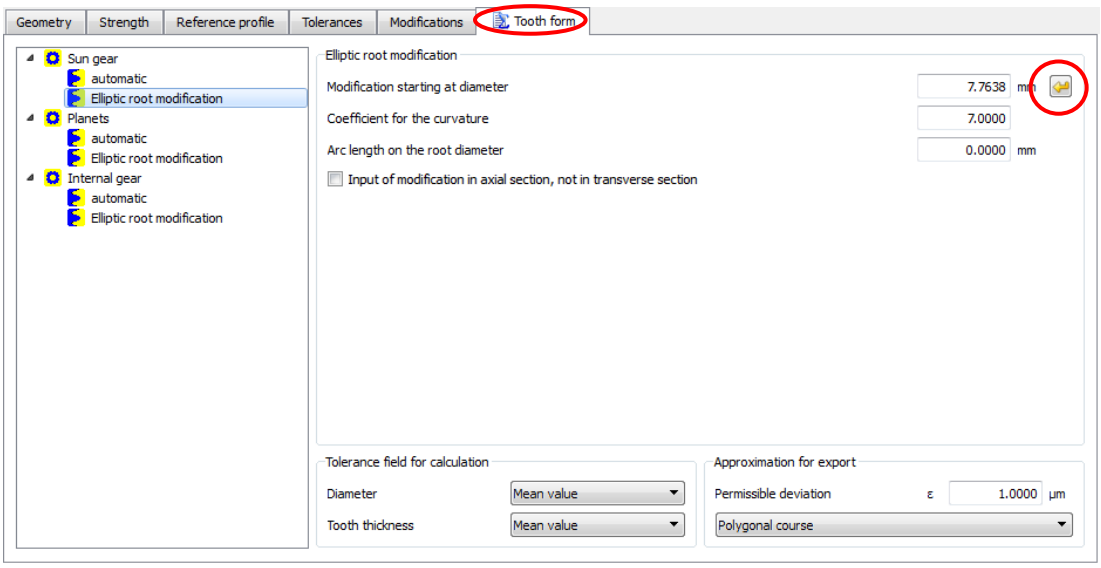


Figure 2.24 "Elliptical root modification" adds the alternative

After this definition has been performed for all the gears, either click the **"Σ"** icon or press "F5".

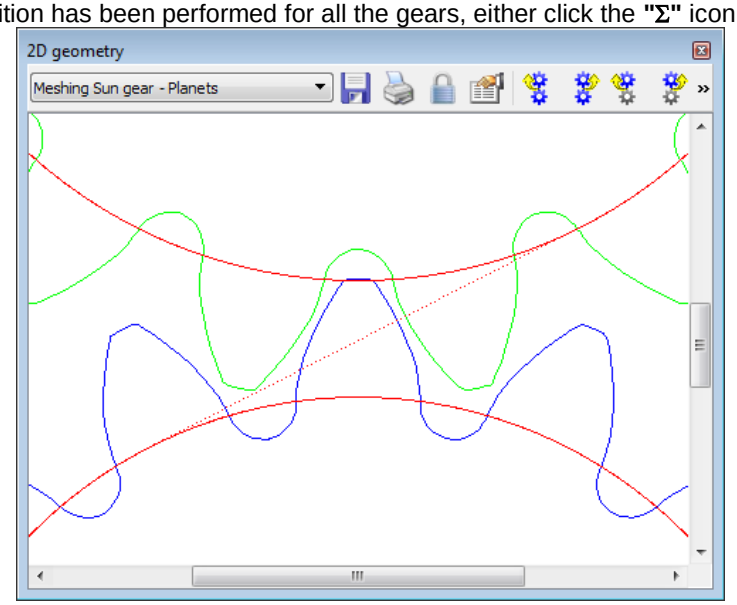


Figure 2.25 Calculating and displaying the tooth form

In the next dialog, activate the collision check.

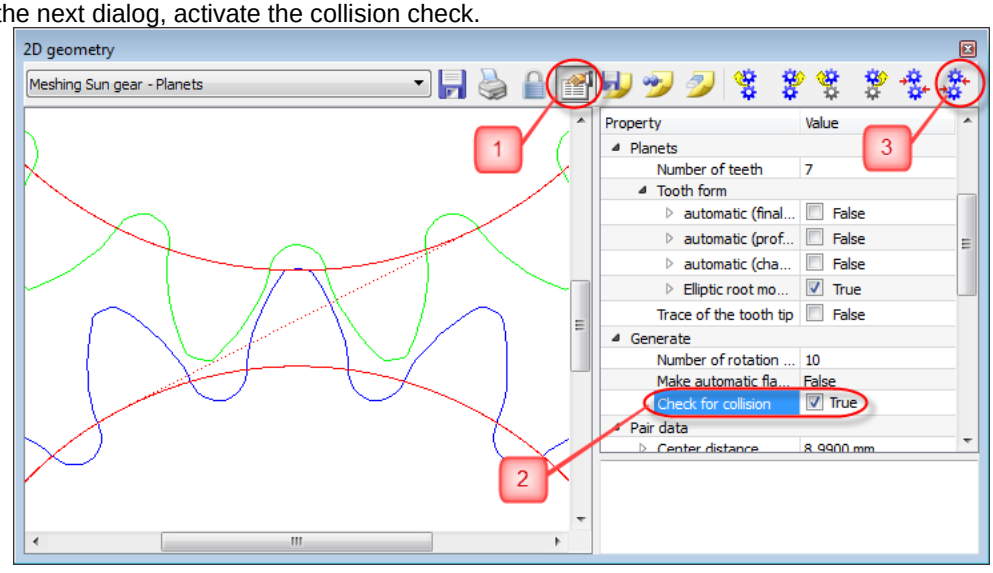


Figure 2.26 Activating collision check and displaying pair 1

Click the  button (2) and select "Apply flank (right)". Now you should see small black dots at the points where the two flanks contact each other. A black dot means "meshing or almost meshing", and a red dot means "collision".

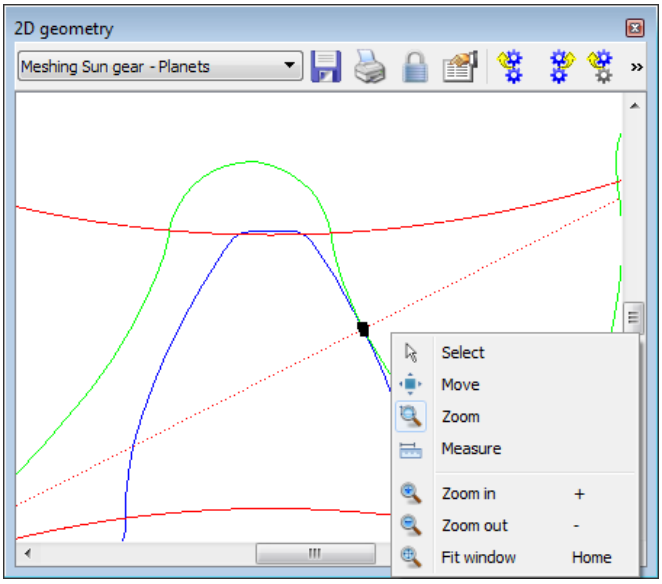


Figure 2.27 Automatic connection of flanks, zoom and collision check

Use the "+"/"-" button to zoom in or out, click the right-hand mouse button to open the menu and, finally, click the  buttons to animate the graphic. Check whether any of the modifications cause a problem. See Figure 2.27.

When you are satisfied with the result, go back to the main dialog. Click [Calculate F5] and use the root safety factor to check whether or not the modifications have improved the root strength.

2.8 Current tutorials

The tutorials listed below include additional details about particular topics that are mentioned in this document:

- Tutorial 009, "Fine Sizing of Cylindrical Gears"
- Tutorial 011: "Tooth Form Optimizations, Tooth Form Modifications"