

KISSsys 03/2014 – Tutorial 2

One Stage Planetary Gearbox

07/11/2014

KISSsoft AG

Rosengartenstrasse 4
8608 Bubikon
Switzerland

Phone: +41 55 254 20 50

Fax: +41 55 254 20 51

info@KISSsoft.AG

www.KISSsoft.AG

Foreword

This Tutorial is divided into three parts, which you, the user, should work through in the sequence in which they are provided.

It assumes that you already have a basic knowledge of the KISSsoft planet gear and coaxial shaft calculation functions.

PART I

- explains how to start the KISSsys system.

PART II

- shows how to handle an existing KISSsys model. This part also describes some basic operating steps, and it is assumed that you will be familiar with them by the time you reach Part III.

PART III

- is the most important part of this tutorial. It illustrates how you can create your own model.

If, whilst working through these tutorials, you have any questions or encounter any problems, simply call the KISSsoft hotline team on 00 41 55 254 20 56, and they will be happy to help.

Table of Contents

1	Starting KISSsys	4
1.1	Calling KISSsys	4
1.2	Selecting the project directory	4
1.3	Opening a KISSsys model	5
2	Buttons and Views	6
2.1	Views in KISSsys	6
2.1.1	User interface and windows	6
2.1.2	Cross references between the diagram, 3D view and tree structure	7
2.1.3	Working in the 3D view	8
2.1.4	Refresh All and Calculate Kinematics	8
2.2	Inputting and outputting data	8
2.3	Calling KISSsoft calculations	8
3	Using the "KISSsys-Tutorial-002" Model	9
3.1	Modifying load data and calculating kinematics	9
3.2	Calculate root and flank safeties	9
3.3	Changing gear data, roller bearing or shaft geometry	10
4	Task	11
5	System Structure	11
5.1	Sketching the system	11
5.2	Starting KISSsys	12
5.3	Loading the template file	12
5.4	Basic data	13
5.4.1	Elements, templates	13
5.4.2	Copy, rename, delete (classic method for creating a model)	14
5.4.3	Insert	14
5.4.4	Naming conventions	15
5.5	Inserting the shaftgroups (main structure)	16
5.6	Adding machine elements	18
5.7	Adding constraints	20
5.8	Notes about multi-level planetary gear units	22
5.9	Defining kinematics	23
5.10	Defining calculations	25
5.10.1	Defining a gear pair	25
5.10.2	Defining a shaft calculation	26
5.11	3D view and positioning	31
5.11.1	Positioning the shafts relative to each other	32
5.11.2	3D View	33
6	Designing the User Interfaces	34
6.1	Overviews of toothings and bearing data	34
6.2	User Interfaces	35
6.2.1	Adding a user interface table	35
6.2.2	Entering text	36
6.2.3	Adding output fields	36
6.2.4	Inserting input fields	38
6.2.5	Calling functions	40
7	Final Tasks	42
7.1	Setting the 3D view	42
7.1.1	Displaying the couplings	43
7.1.2	Cut View	43
7.2	Displaying all the planets	43
7.3	Running a simulation of the planetary gear unit	44

PART I, STARTING KISSSYS

1 Starting KISSsys

1.1 Calling KISSsys

To start KISSsys, in Windows, click on Start → Programs → KISSOFT 03-2014 → KISSsys.

1.2 Selecting the project directory

You have the option of working with projects in the KISSsys system. First, you need to define a project directory. You can then save all the KISSsys models and the corresponding KISSsoft files to the same directory whilst you are working.

To define a project directory, select File → Open project.

If you do not select a project directory, all the results will be stored in the default directory. This is, for example, Users\KISSsoft.

Click the buttons that are marked in the figure below to select the project directory. In this case it is C:\Program Files (x86)\KISSsoft 03-2014\kisssoft\tutorial. KISSsys opens after you confirm your selection.

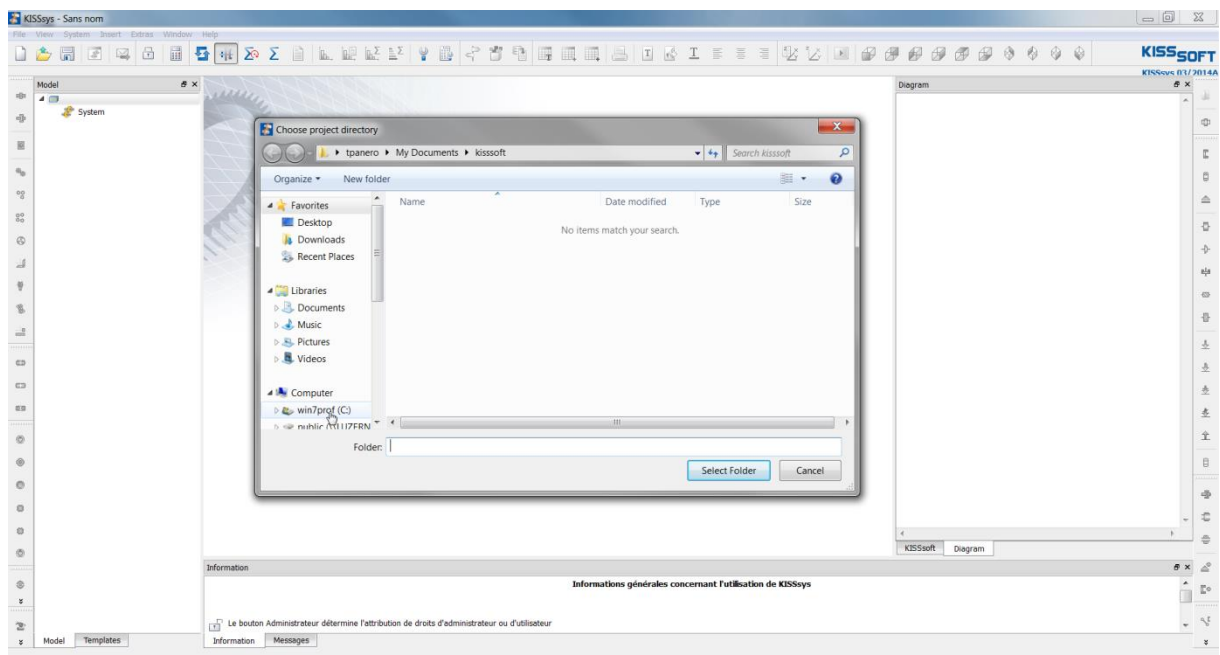


Figure 1. Selecting the project directory

1.3 Opening a KISSsys model

Now, after you have defined the project directory, click File → Open to open the KISSsys models that are already present in it. Then open the KISSsys-Tutorial-002.ks model. The KISSsys screen should now look like this:

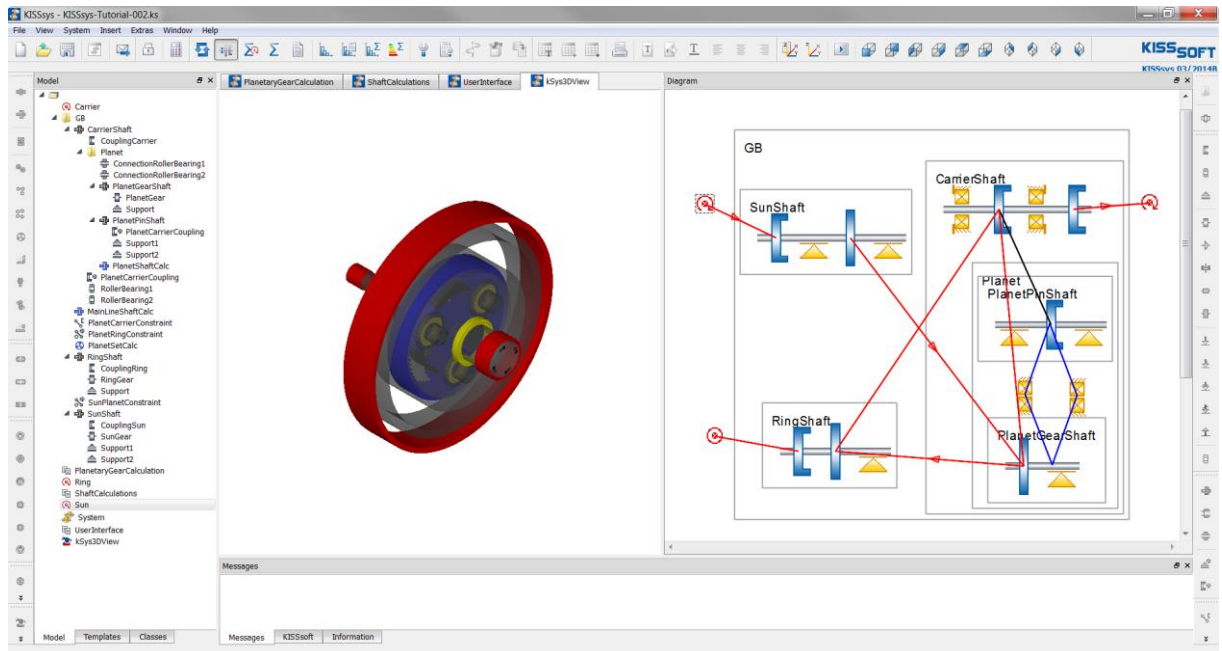


Figure 2. KISSsys view after this model has been opened

Please note: you should only select files from the project directory you have just selected.

Part II, WORKING WITH THE FINISHED MODEL

2 Buttons and Views

2.1 Views in KISSsys

2.1.1 User interface and windows

You can structure the KISSsys user interface like this:

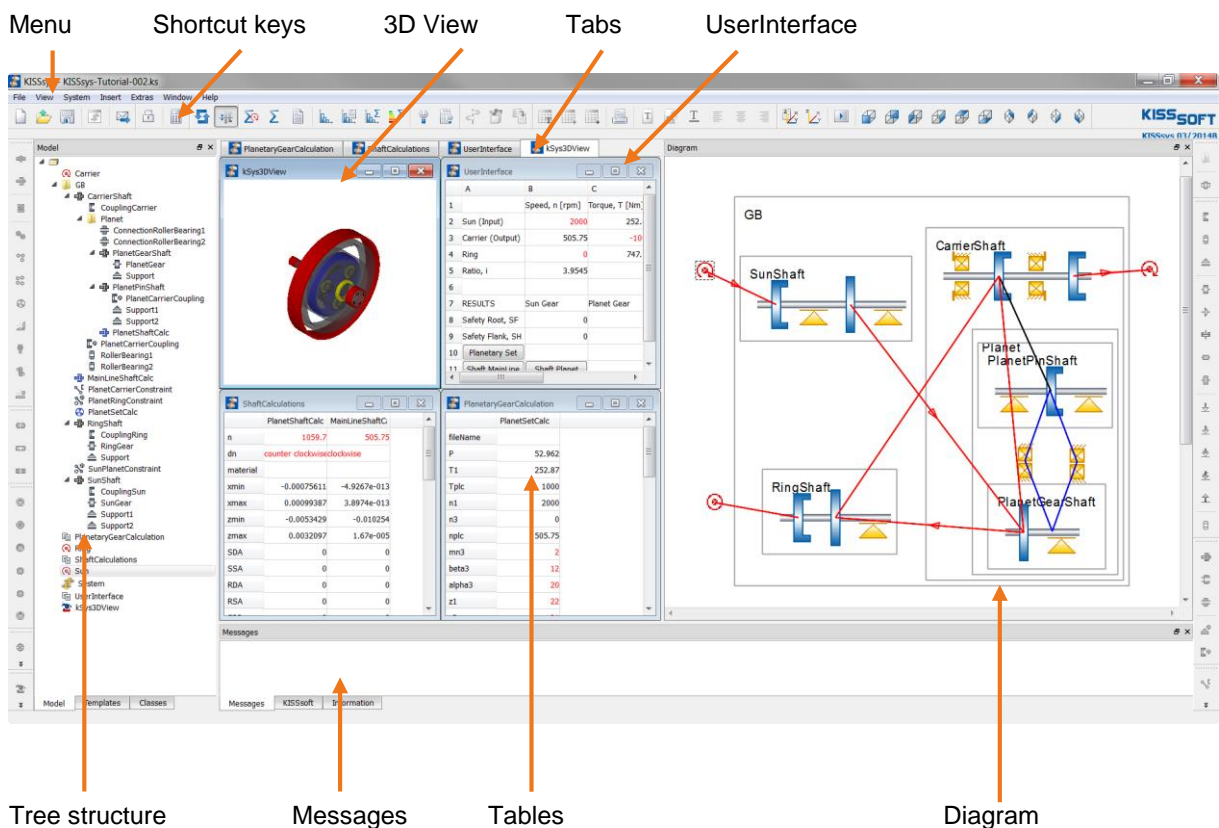


Figure 3. The KISSsys user interface

Click Menu → View to display or hide the classes, the model (tree structure), the templates, the messages, the KISSsoft (messages from KISSsoft calculations) and the diagram as required:

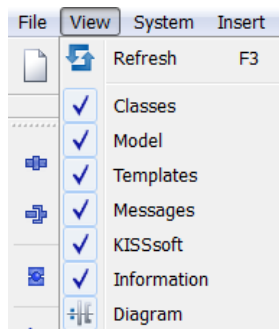


Figure 4. Buttons used for the tree structure, diagram, and message window.

You can also minimize, restore, and close, the UserInterface and 3D View, as necessary. Click either "Window" or the tabs to toggle between individual windows. You can display a closed window by clicking the right-hand mouse button on the relevant element in the tree structure and then clicking "Show". You can also open a window by double-clicking on the appropriate element.

2.1.2 Cross references between the diagram, 3D view and tree structure

When you click on an element in the tree structure, it is displayed in red. At the same time, the relevant element in the 3D view is displayed with a coordinate system to identify it:

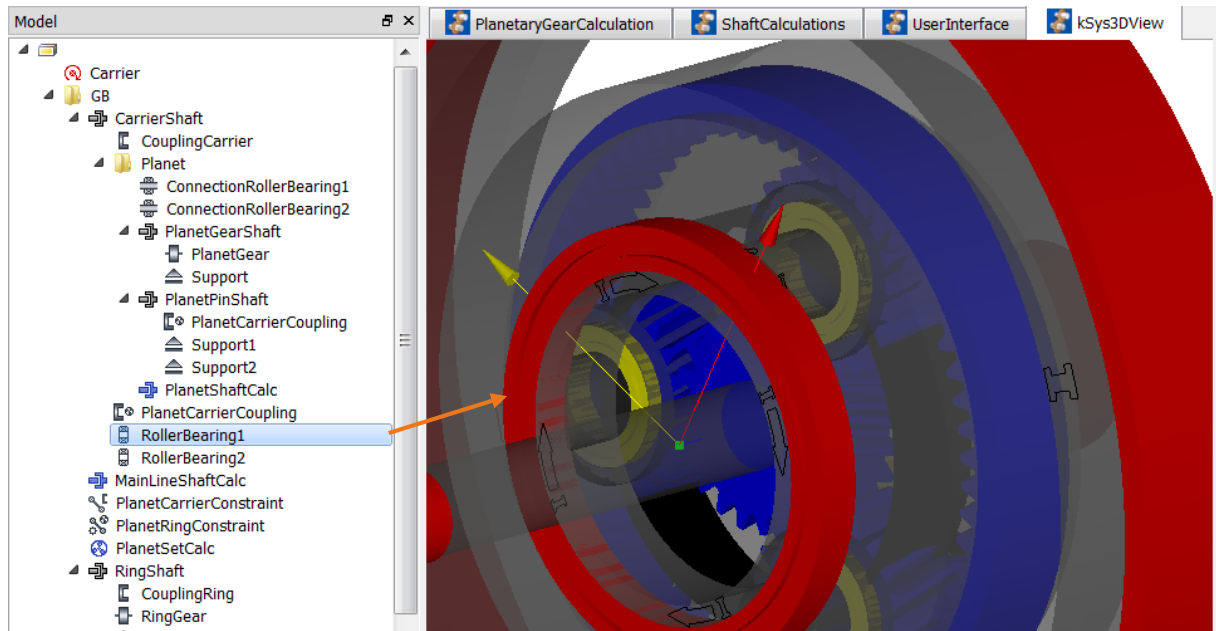


Figure 5. The element selected in the tree structure is highlighted in the 3D view.

If you click on an element in the diagram, it is also highlighted in the tree structure and in the 3D view. Hover the mouse arrow over an element in the diagram to display the element's name together with its path:

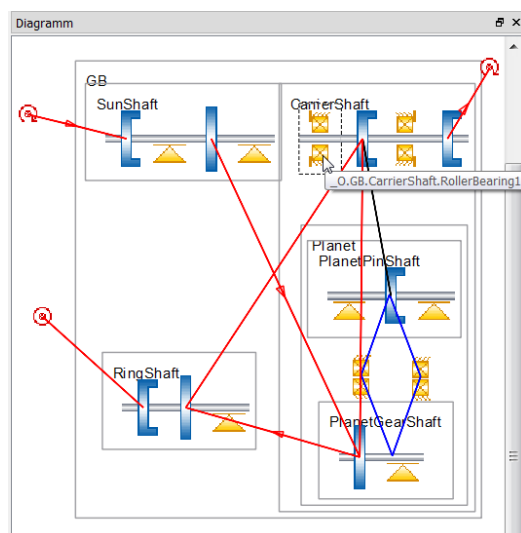




Figure 6. Information about element names in the diagram.

2.1.3 Working in the 3D view

In the 3D view you use the left-hand mouse button to grab and rotate the gear. To zoom in and out, hold down the right-hand mouse button and move the mouse at the same time. To move the model to the correct position, hold down the middle mouse button and move the mouse at the same time.

2.1.4 Refresh All and Calculate Kinematics

Click the "Refresh All"  command (or press F4) to update the data and the displays. Use this command to update the 3D View, for example, if you have modified the tip diameter of a gear. Click on  or press F4 to calculate the gear's kinematics. This function also updates the powerflow in the diagram.

2.2 Inputting and outputting data

Every table has the following text elements

Feature	Type	Use
Black text	Output/text	Results that change according to which calculation is used are shown in black text. Text elements, such as captions, that cannot be changed are also shown in black text.
Red text	Inputs	You can type numerical values directly into fields that contain red text or double-click to select values from a list. The values you input or select are then saved to the appropriate variables.
Gray background	Functions	Double-click on a field to call its functions.

2.3 Calling KISSsoft calculations

From the tree structure, simply double-click on the KISSsoft icons to access the KISSsoft calculation for the corresponding element, for example, a shaft. The KISSsoft icons are:



Figure 7. Extract from KISSsoft calculations

3 Using the "KISSsys-Tutorial-002" Model

3.1 Modifying load data and calculating kinematics

To start calculating the kinematics, double-click on the "Kinematics" function. This function calculates all the speeds, torques and bearing forces. The outputs for the reduction "i tot", the initial speed and the input torque are displayed in the user interface:

A	B	C	D	
1	Speed, n [rpm]	Torque, T [Nm]	Power, P [kW]	
2	Sun (Input)	2000	252.87	52.962
3	Carrier (Output)	505.75	-1000	52.962
4	Ring	0	747.13	0
5	Ratio, i	3.9545		
6				
7	RESULTS	Sun Gear	Planet Gear	Ring Gear
8	Safety Root, SF	2.0518	1.441	2.1326
9	Safety Flank, SH	1.0142	1.012	1.6723
10	Planetary Set			Kinematik
11	Shaft MainLine	Shaft Planet		Strength

Figure 8. Outputs in the user interface after the kinematics calculation has been performed

You can input the speed on the sun shaft and the torque on the planet carrier directly. Note that the torque sign shows whether input or output power is involved. If you modify the input speed data or the output torque data, you must run the kinematics calculation again (double-click on "Kinematics") to ensure the results are kept up to date.

The relationship between the speeds results in the reduction "Ratio, i".


3.2 Calculate root and flank safeties

When you run the "Strength" function (double-click), the kinematics are first calculated again and then the strength calculations are performed for the gears, bearings, and shafts. The resulting flank and root safeties for the gears in the model are displayed directly in the UserInterface table (see section 6.2.3 Outputting results). To update these views, click the "Refresh" function (double-click).

A	B	C	D	
1	Speed, n [rpm]	Torque, T [Nm]	Power, P [kW]	
2	Sun (Input)	2000	252.87	52.962
3	Carrier (Output)	505.75	-1000	52.962
4	Ring	0	747.13	0
5	Ratio, i	3.9545		
6				
7	RESULTS	Sun Gear	Planet Gear	Ring Gear
8	Safety Root, SF	2.0518	1.441	2.1326
9	Safety Flank, SH	1.0142	1.012	1.6723
10	Planetary Set			Kinematik
11	Shaft MainLine	Shaft Planet		Strength

Figure 9. Output containing the calculated safeties for root and flank

This method takes the service life into account, as specified in the KISSsoft calculation file (default setting). If you want to perform the calculation for a different service life, simply change the required service life in KISSsoft. Double click on the "Planetary Set" calculation element to open the KISSsoft screen for calculating planets. Open the "Rating" tab in this screen and change the required service life H from 20,000 h to, for

example, 30,000h. To ensure this value is saved correctly, either click "Calculate"  or press "F5" to close KISSsoft.

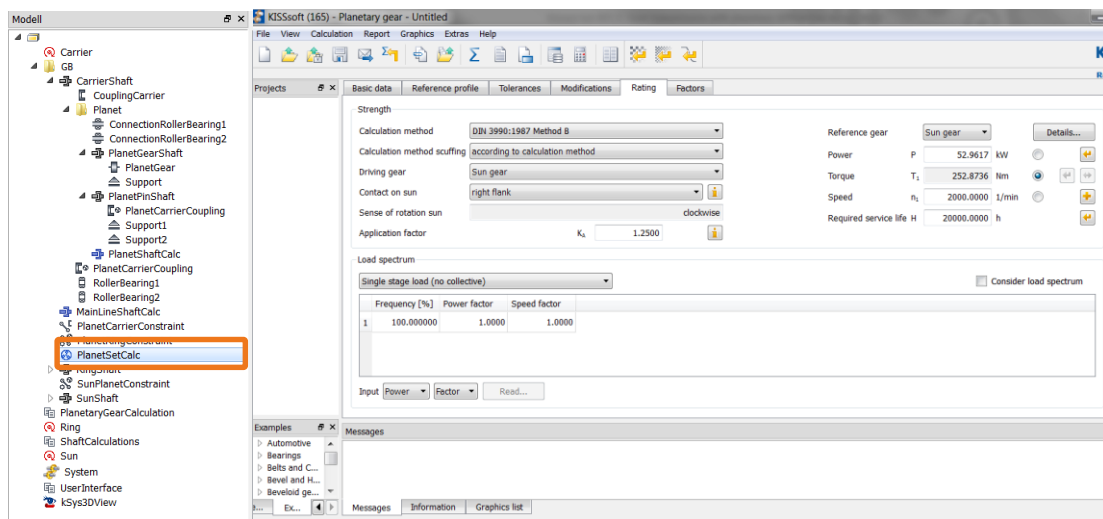



Figure 10. Changing the required service life

After this, call the "KISSsoft" function again to rerun the strength calculation.

3.3 Changing gear data, roller bearing or shaft geometry

You can modify gears, shafts, and bearings, in the KISSsoft system in the usual way. To do this, double-click on the KISSsoft calculations listed in the tree structure to call the KISSsoft system. This is where you can now, for example, perform fine sizing in a gear calculation or define the shafts and specify the bearings in a shaft calculation. To ensure the values you changed in the KISSsoft screen are also stored for the underlying variables, either click on  or press F5 in the calculation. You must then close KISSsoft after making the changes.

If you are using the graphical Shaft editor, please note that none of the elements defined in the tree structure (for example, bearing) can be removed from or added to the editor.

The number and type of elements arranged on the shafts can only be defined in the KISSsys tree structure.

Part III, STRUCTURE OF THE MODEL

4 Task

The task is to create a KISSsys model for calculating a single stage planetary gear unit with an integrated planet gear, coaxial shafts, and bearing calculation. This model can then be used to size or verify these types of systems.

At the end of the tutorial, the model you create should look like the model used in "KISSsys-Tutorial-002".

5 System Structure

The new system is made up of elements such as gears, shafts, etc. as well as the relevant KISSsoft calculations. These elements are taken from a library of "templates."

5.1 Sketching the system

Before you start using the actual model, we recommend you make a rough draft of the system. This can then be used in KISSsys for the descriptions of the individual elements. The sketch in this example is divided into a gear diagram and a planet shaft diagram. The system displays an application which includes general supports for the sun shaft and two bearing-supported gears for each planet gear. Although the internal gear is usually static, you can input an internal gear speed in the model.

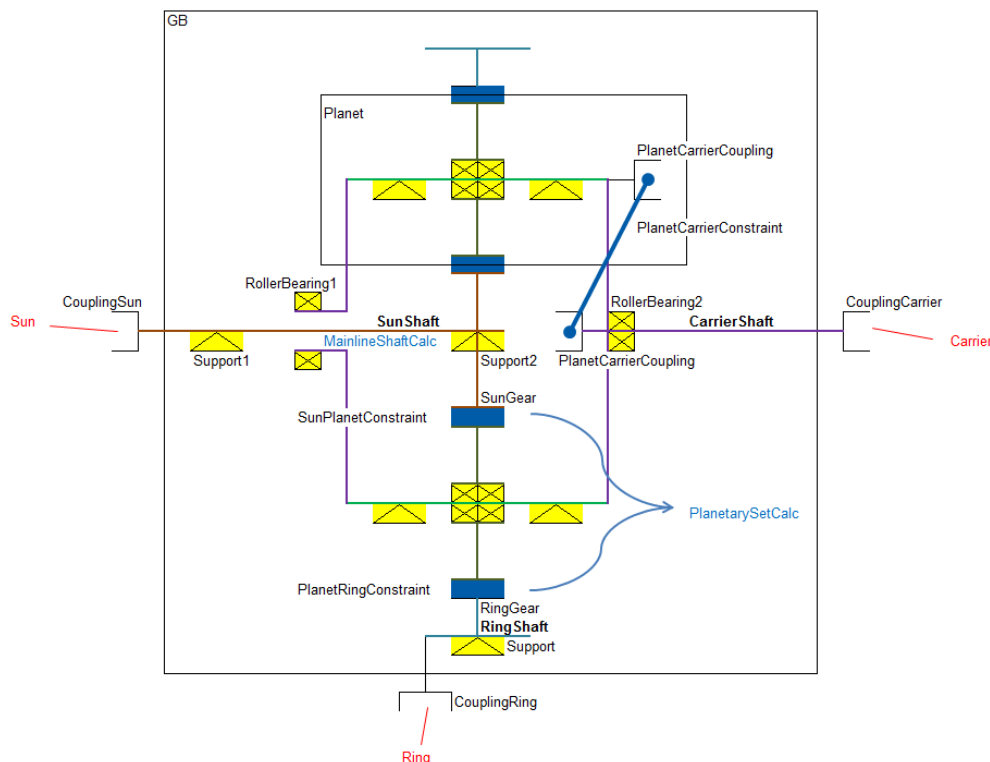


Figure 11. Diagram of a planetary gear unit

The planet shaft diagram looks like this. The planet pin is supported on both sides on general supports. This also means deformations or housing stiffnesses can be taken into account.

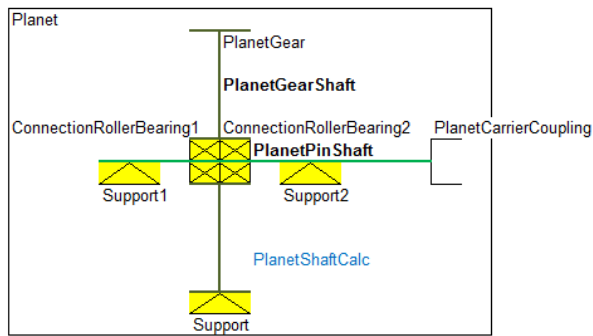


Figure 12. Diagram of a planetary gear unit

5.2 Starting KISSsys

First of all, create a new directory, for example, on the C:\ drive, with the name "MyTutorial". Then open KISSsys and the "MyTutorial" project directory. The KISSsys system opens with an empty model. Now click File/Save as... to save this empty model with a name, for example "KISSsysTutorial".

To create a new system, or modify an existing system, you must run KISSsys in "Administrator" mode. You enable this mode in the "Extras" menu:

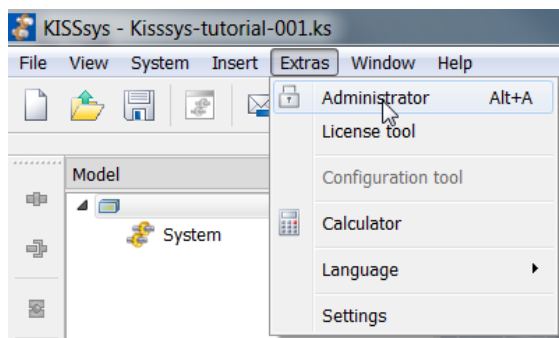


Figure 13. Toggling to "Administrator" mode

If you cannot select the "Administrator" option, this means you do not have the appropriate KISSsys license, and you should contact KISSsoft AG to get one.

5.3 Loading the template file

The "Template" library of standard elements is called automatically when you enable Administrator mode in the "Template" tab. If you want to load a specific library, click File → Open Templates.

The "standardTemplates" folder contains all the templates that can be used in the KISSsys system. You can also import user-defined templates from other folders. If the templates.ks template is not already open, you should open it now.

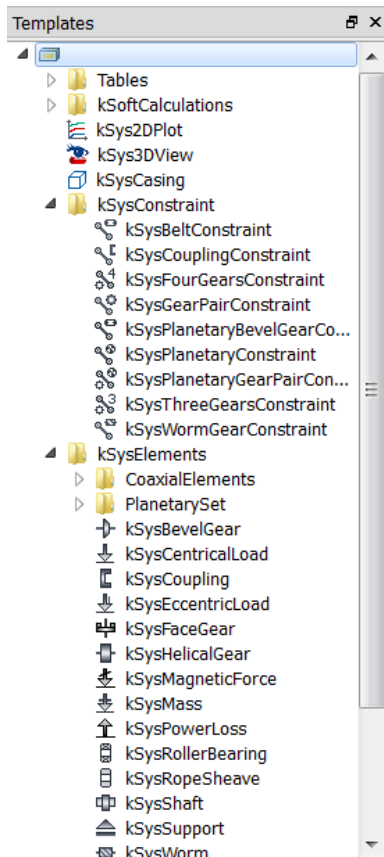


Figure 14. Elements library after loading the templates.ks template file.

After this step, you can now start creating the new system.

5.4 Basic data

5.4.1 Elements, templates

Every KISSsys model is made up of different elements. They are arranged in a tree structure. There are different types of elements:

- Groups (folder icons)
- Machine elements (gray icons)
- Connections (gray icons)
- KISSsoft calculations for the elements (light blue icons)
- System elements (graphics, tables)

These are all listed in the template library. You can also modify these templates to meet specific user requirements (however, this is only recommended for experienced users).

Click the relevant tabs to navigate between the model and the templates:

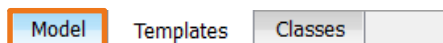


Figure 15. Selecting models and templates

You create your KISSsys model under "Model".

5.4.2 Copy, rename, delete (classic method for creating a model)


You can copy elements from the templates to the tree structure by using the "Ctrl+C"/"Ctrl+V" key combinations, by clicking the right-hand mouse button to "copy"/"insert" them, or by dragging and dropping them. To delete elements you can either press "Delete" or click on them with the right-hand mouse button and then click "Delete" in the context menu. Click the right-hand mouse button and select "Rename" to rename an element.

Attention: If you rename an element it may cause the connections to that element to become invalid.

This is the "classic" method for creating a model.

5.4.3 Insert

This menu provides a range of options you can use to create your own model. There is an assistant which will guide you step by step through the process of creating your model and prompt you by suggesting which elements should be added. The assistant can also be used to create parallel shafts and planetary stages.

There is also an elements box which contains all the available elements. These are categorized and displayed with icons. The default templates are described in greater detail in section 5.3. You can also select which one of the different methods you want to use by clicking on the  icons in the Task bar

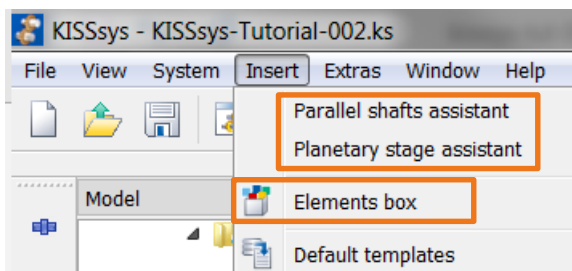


Figure 16. Insert menu

In this tutorial, the "Elements box" is used to create the planetary gear unit.

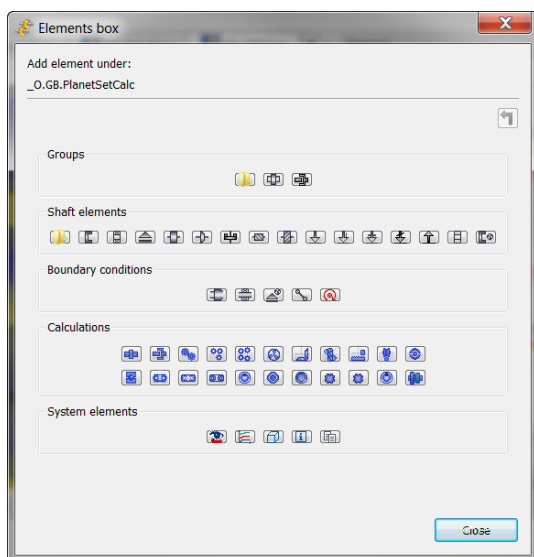


Figure 17. Elements box: categorized elements

In this method, click on the element you require (group or shaft) from the preselected elements to insert it into the model structure tree.

5.4.4 Naming conventions

Click on Extras → Setting → Tab elements to make global settings for adding elements. You can either have the elements named automatically, or the system will prompt you to enter a new label for the element every time.

Basically, you can either use long names to make them easier to read and understand, or short names to stop the variable and path names from becoming too long.

Element	Name in KISSsys (class)	Name in the KISSsys Tutorial (detailed name)	Example of short names
Coaxial shaft	kSysCoaxialShaft	SunShaft, RingShaft, etc.	ss, rs, etc.
Cylindrical gear	kSysHelicalGear	SunGear, RingGear, etc.	g
Coupling	kSysCoupling	Coupling	c
Roller bearing	kSysRollerBearing	RollerBearing	b1
Connecting roller bearing	kSysConnectionRollerBearing	ConnectionRollerBearing	b11
...			

In this tutorial, you should use names that match the formats shown in this table. KISSsys will therefore need to ask for the names. Set the name assignment fields for all the classes and calculations to "Ask".

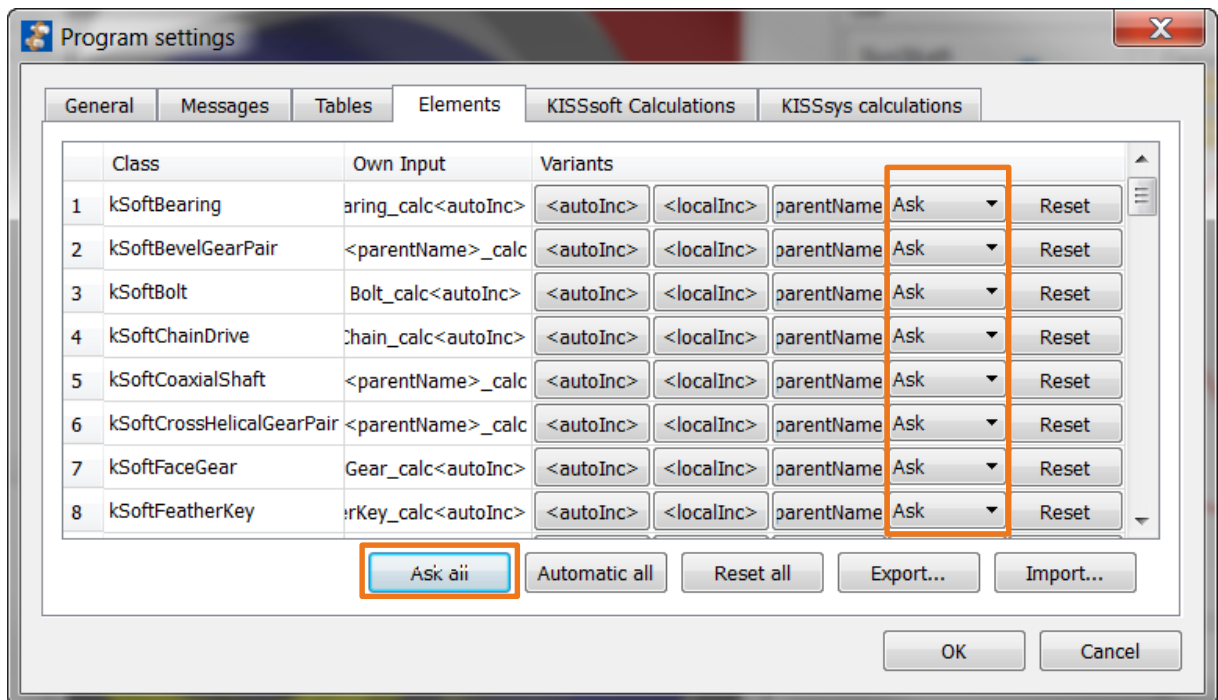


Figure 18. Settings for assigning names to the elements automatically

5.5 Inserting the shaftgroups (main structure)

The first step in this tutorial is to add the shafts and groups to provide an overview of the structure. After this, the elements (bearings and cylindrical gears) are added to the relevant shafts.

Planetary gear units with coaxial shafts require more subgroups than cylindrical gear units (see Tutorial 1). This is because the KISSsoft calculation with coaxial shafts includes all the shafts present in the particular group. These subgroups are needed to keep the coaxial shafts separate from each other. In the case of planetary stages, these are an initial coaxial shaft calculation with 3 shafts (sun shaft, internal gear shaft and planet carrier shaft - all the shafts are in the "GB" group) and a second coaxial shaft calculation with 2 shafts (planet pin and planet gear - all the shafts are in the "Planet" sub-group).

To help illustrate the structure more clearly, this tutorial uses the procedure described below to explain the basic principles behind creating a model. Otherwise we recommend you use the procedure in which you add the elements (bearings and cylindrical gears) to the shaft at the same time as you add the shafts.

1. Add the "GB" main group
2. Add coaxial shafts for the main shaft ("SunShaft", "RingShaft" and "CarrierShaft").
3. Add the "Planet" group to the "CarrierShaft" shaft
4. Coaxial shafts for planet pin ("PlanetPinShaft" and "PlanetBodyShaft")

First, add the main group:

1. Select the main directory in the model tree structure
2. Add the main group:
 - 1 x group (kSysGroup, with the name "GB")

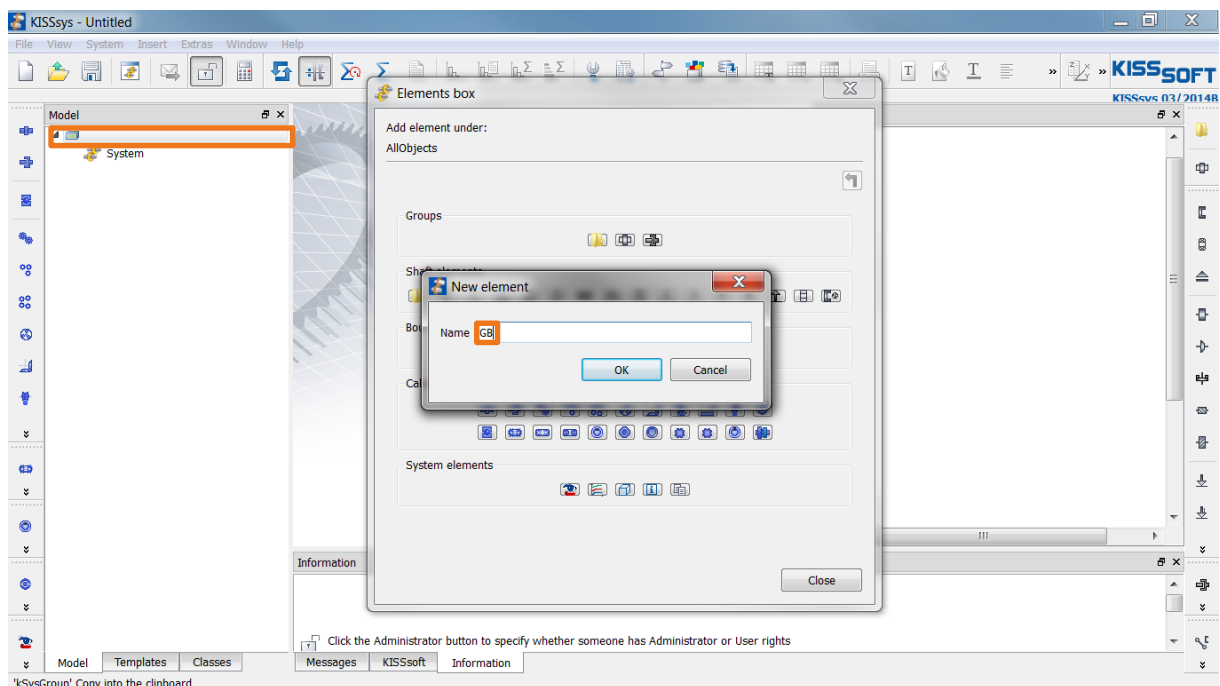


Figure 19. Add the "GB" group

Now add three coaxial shafts from the Elements box to this:

1. Select the "GB" group in the model tree structure
2. Add coaxial shafts for the main shaft (select the "GB" group again each time):
 - 1 x coaxial shaft (kSysCoaxialShaft, with the name "SunShaft")
 - 1 x coaxial shaft (kSysCoaxialShaft, with the name "RingShaft")
 - 1 x coaxial shaft (kSysCoaxialShaft, with the name "CarrierShaft")

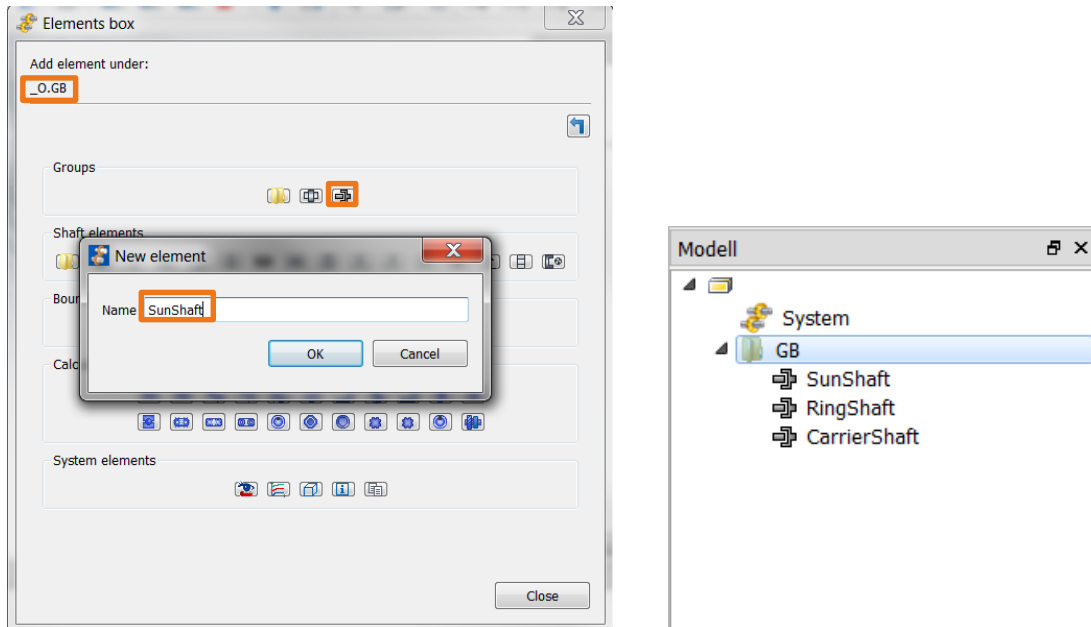


Figure 20. Adding coaxial shafts to the main shaft

The coaxial shaft of the planet bolt is regarded as a shaft system, which - as in real planetary gear units - is located within the planet carrier ("CarrierShaft"). To do this, first create the "Planet" group within the "CarrierShaft" shaft.

To create the "Planet" group:

1. Select the "CarrierShaft" shaft in the model tree structure
2. Add a group:
 - 1 x group (kSysGroup, with the name "Planet")

Then add the shafts for the planet pin and the planet gear body:

1. Select the "Planet" group in the model tree structure
2. Add the coaxial shafts:
 - 1 x coaxial shaft (kSysCoaxialShaft, with the name "PlanetPinShaft")
 - 1 x coaxial shaft (kSysCoaxialShaft, with the name "PlanetGearShaft")

The structure is displayed in the "Diagram" window. Use the mouse to size the frame to make the diagram easier to see. The model now has this structure:

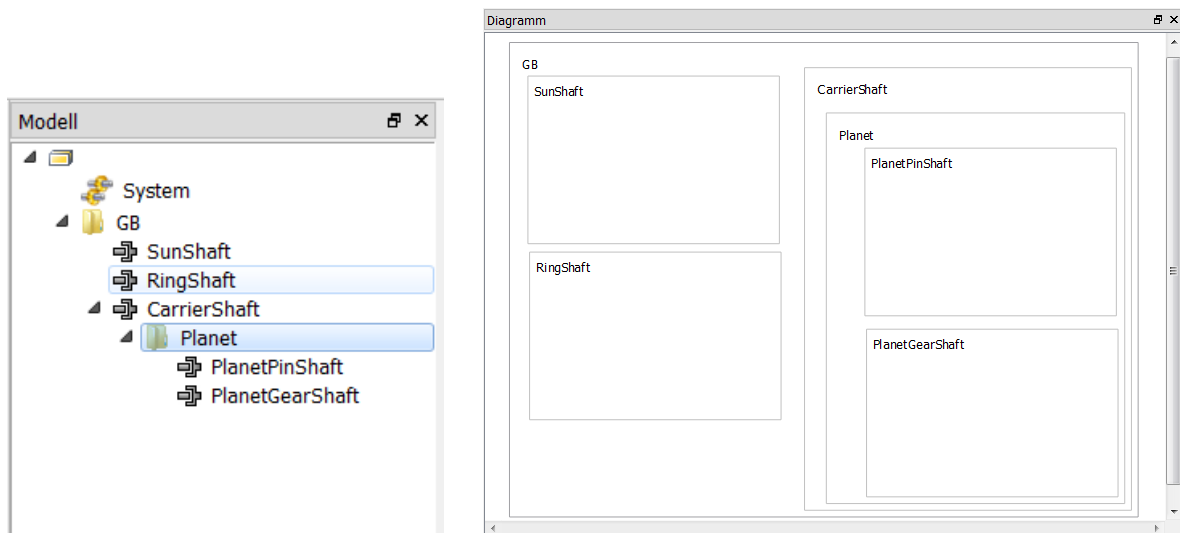


Figure 21. Model structure with main group, sub-group and shafts

The model's structure is now complete.

5.6 Adding machine elements

You can now add machine elements to the shaft. Take care to ensure that each element is added in the correct place.

1. Select the "SunShaft" shaft in the model tree structure
2. Add the machine elements which include the "SunShaft" shaft:
 - 1 x coupling (kSysCoupling, with the name "CouplingSun")
 - 2 x general support (kSysSupport, with the names "Support1, Support2")
 - 1 x gear (kSysHelicalGear, with the name "SunGear")

The elements appear in the shaft calculation in the same sequence as they were added. You can change this arrangement later on in the Shaft editor.

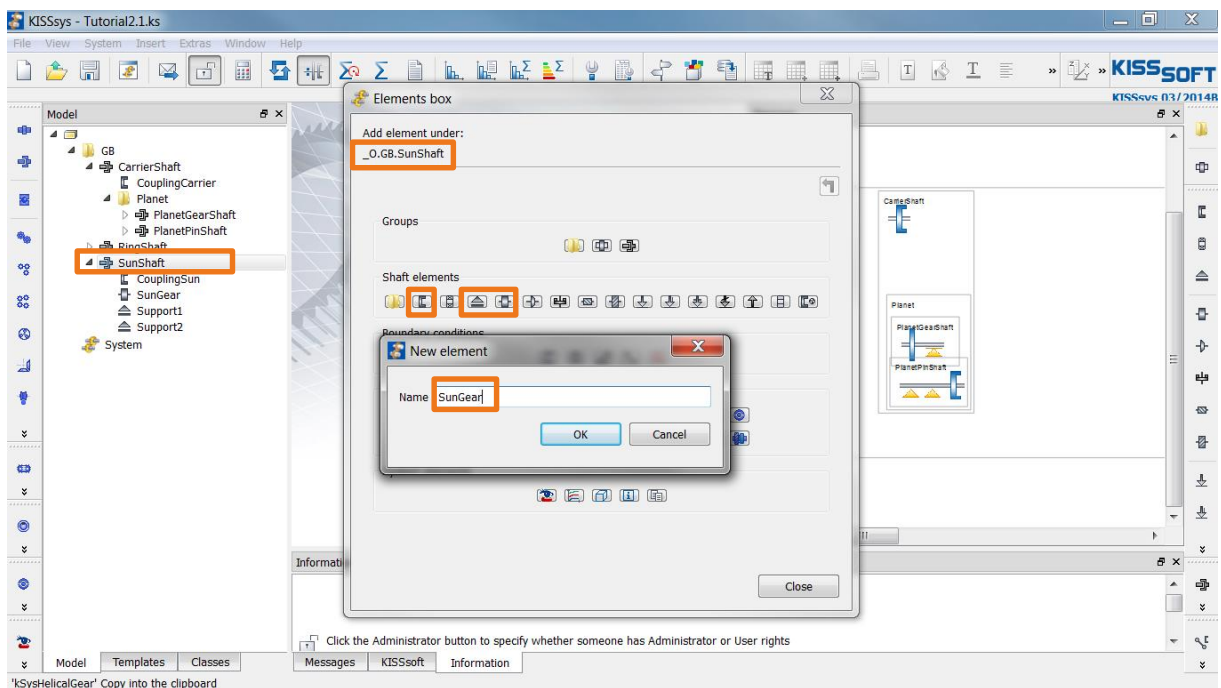


Figure 22. Adding machine elements to a SunShaft

Use the same method to add all the other machine element to the other shafts.

"RingShaft" coaxial shaft

1. Select the "RingShaft" shaft in the model tree structure
2. Add the machine elements which include the "RingShaft" shaft:
 - 1 x coupling (kSysCoupling, with the name "CouplingRing")
 - 1 x gear (kSysHelicalGear, with the name "RingGear")
 - 1 x general support (kSysSupport, with the name "Support")

The planet carrier coupling connects the planet pin to the planet carrier with the same speed (no slipping). First add the "planet carrier coupling" elements, then connect the couplings (see section 5.7 Adding connections).

"CarrierShaft" coaxial shaft

1. Select the "CarrierShaft" shaft in the model tree structure
2. Add the machine elements which include the "CarrierShaft" shaft:
 - 1 x coupling (kSysCoupling, with the name "CouplingCarrier")
 - 1 x planet carrier coupling (kSysPlanetCarrierCoupling, with the name "PlanetCarrierCoupling")
 - 2 x roller bearing (kSysHelicalGear, with the names "RollerBearing1, RollerBearing2")

"PlanetPinShaft" coaxial shaft

1. Select the "PlanetPinShaft" shaft in the model tree structure
2. Add the machine elements which include the "PlanetPinShaft" shaft:
 - 2 x general support (kSysSupport, with the names "Support1, Support2")
 - 1 x planet carrier coupling (kSysPlanetCarrierCoupling, with the name "PlanetCarrierCoupling")

"PlanetGearShaft" coaxial shaft

1. Select the "PlanetGearShaft" shaft in the model tree structure
2. Add the machine elements which include the "PlanetGearShaft" shaft:
 - 1 x gear (kSysHelicalGear, with the name "PlanetGear")
 - 1 x general support (kSysSupport, with the name "Support")

The diagram now looks like this:

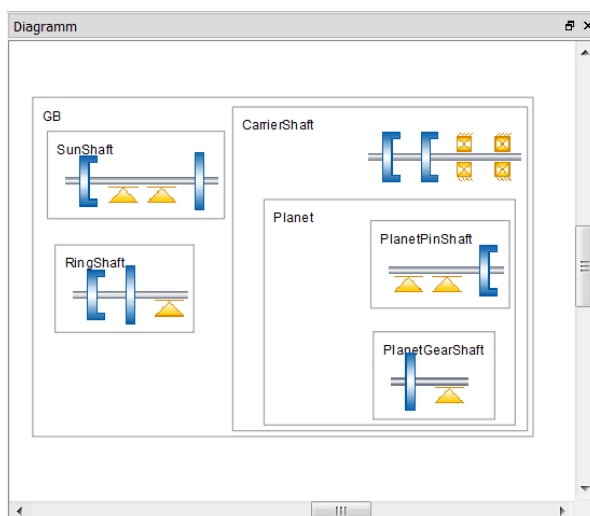


Figure 23. Diagram after adding the machine elements

5.7 Adding constraints

The next step involves adding different constraints. You will find these in the Elements box in the Boundary conditions section.

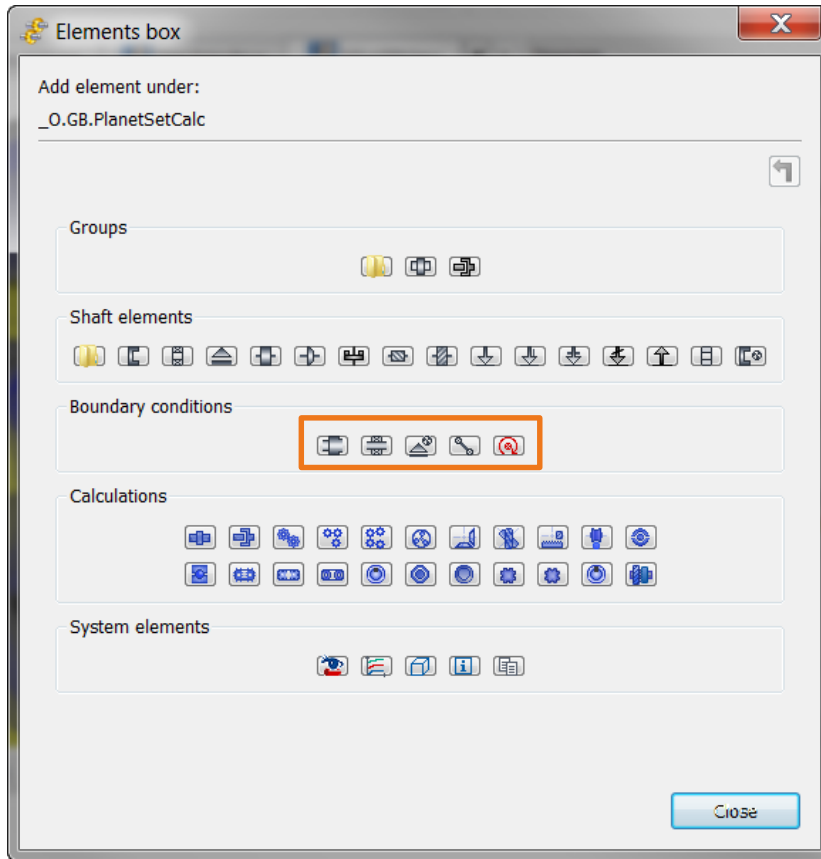


Figure 24. Connections

Now add the following constraints:

- Connecting roller bearing between planet pin and planet gear
- Coupling constraint for the planet carrier couplings on the planet pin shaft "PlanetPinShaft" and the planet carrier shaft "CarrierShaft"
- Toothing constraints for connecting the sun-planet and planet-internal gear (input the number of planet gears here)

In the KISSsys system, the roller bearings that connect the planet pin to the planet gear are added to the same level as the shafts themselves, in the same way as in the KISSsoft shaft calculation with coaxial shafts. In this case, they are added to the "Planet" group.

- Select the "Planet" group in the model tree structure
- Add the connecting bearings which connect the "PlanetPinShaft" and "PlanetGearShaft" shafts:
 - 2 x connecting roller bearing (kSysConnectionRollerBearing, with the name "ConnectionRollerBearing1, ConnectionRollerBearing2")
- Defining connection shafts:
 - Inner race: PlanetPinShaft
 - Outer race: PlanetGearShaft

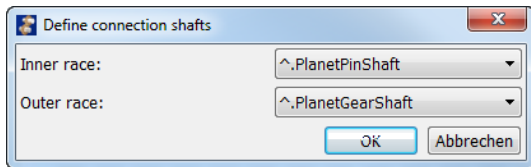


Figure 25. Add connecting roller bearings

The planet carrier coupling connects the planet pin to the planet carrier with the same speed (no slipping). You can now connect the couplings.

1. Select the "GB" group in the model tree structure
2. Add the constraint under Boundary condition in the Elements box (kSysConstraint). The menu in which you select the constraint types now appears.



3. Add the coupling constraint (kSysCouplingConstraint) and name it "PlanetCarrierConstraint"
4. Input the coupling constraint details:
 - Element 1: ^.CarrierShaft.PlanetCarrierCoupling
 - Element 2: ^.CarrierShaft.Planet.PlanetPinShaft.CarrierCoupling
 - All other points in accordance with the default setting

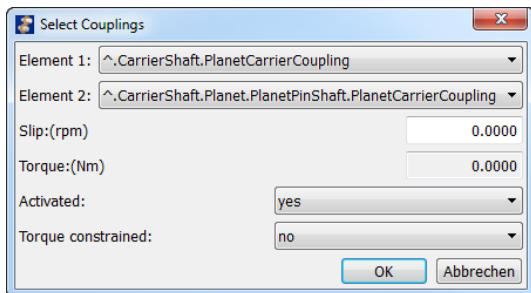


Figure 26. Adding a constraint to planet carrier couplings

The "SunShaft", "CarrierShaft" and "RingShaft" shafts are connected via the sun-planet and planet-internal gear toothings, just like a real gear unit.

Adding constraints to toothings:

1. Select the "GB" group in the model tree structure
2. Add a constraint for the sun-planet toothings (kSysConstraint). The menu in which you select the constraint types now appears.
3. Add a planetary constraint (kSysPlanetaryGearPairConstraint) and name it "SunPlanetConstraint"
4. Input the planetary constraint details:
 - Select configuration: sun/planet
 - Select sun gear: "^_SunShaft.SunGear"
 - Select planet gear: "^_CarrierShaft.Planet.PlanetGearShaft.PlanetGear"
 - Select planet carrier coupling: "^_CarrierShaft.PlanetCarrierCoupling"
 - Define the number of planets: 3

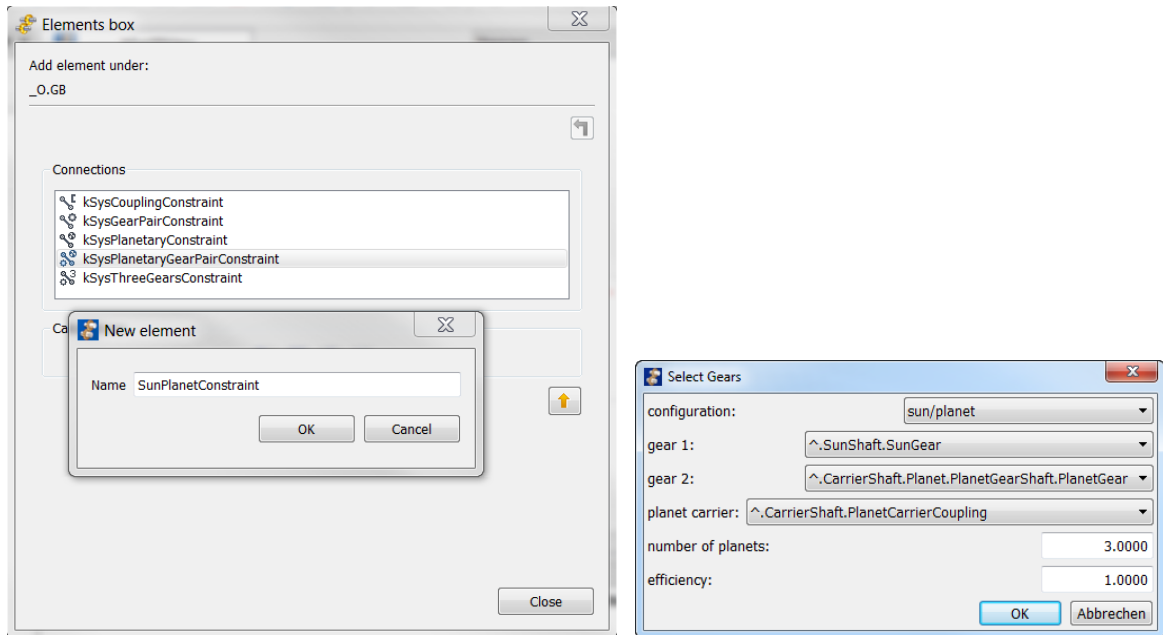


Figure 27. Sun gear-planet gear constraints

Use the same method to define the constraint for the planet-internal gear tothing.

The diagram now looks like this. You have now added all the elements, only the kinematic conditions and the KISSsoft calculations are missing.

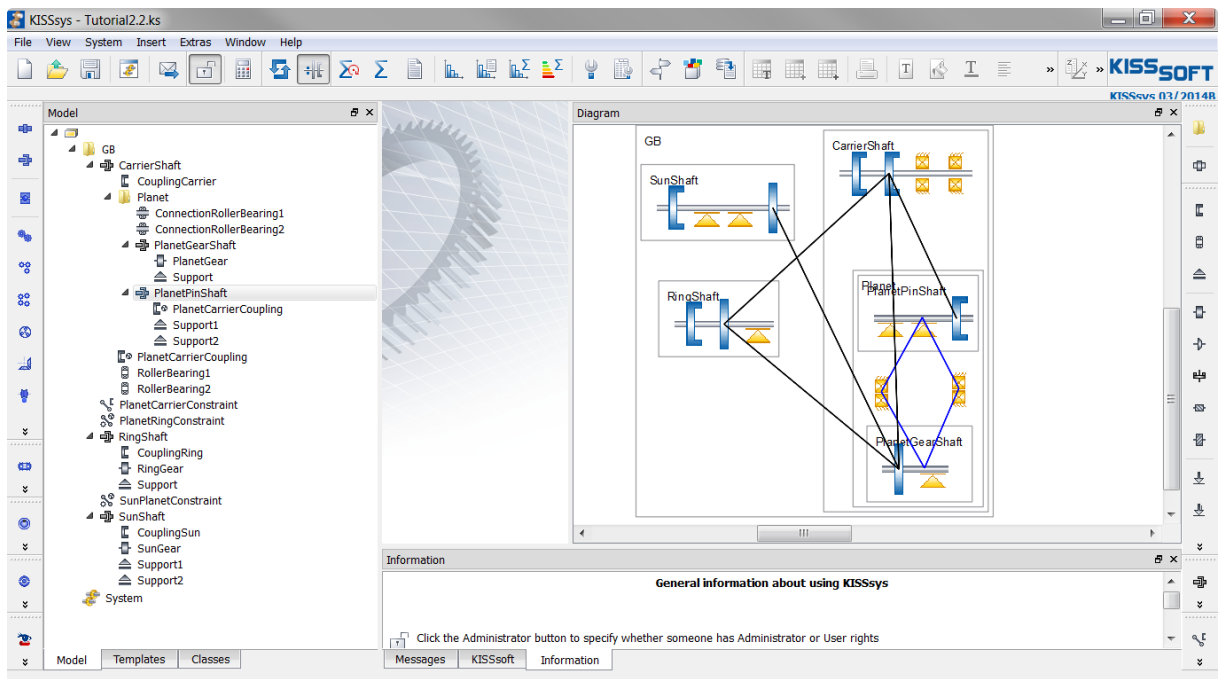


Figure 28. KISSsys model with elements

5.8 Notes about multi-level planetary gear units

You can use the variants described below to add another planetary gear, or other tothing stages:

- Variant I: Separate group for each planetary stage
Right from the start, the current planetary stage has been structured as an additional sub-group, for example "Stage1". This group has then been copied as a second group, for example "Stage2". The

benefit of this is that you can add more stages very quickly. The disadvantage is that information about the deflection of planet carrier Stage 1 on sun gear Stage 2, and how the new stages are positioned in KISSsys, is missing.

- Variant II: the second planetary stage is added to the same group, "GB". The KISSsoft shaft calculation now has the two main shafts (Stage 1 and Stage 2) in the same KISSsoft file. This has the benefit that the deflections are transferred from the Stage 1 planet carrier to the Stage 2 sun shaft, and positioning can take place in the KISSsoft system.


If you have any further questions, please contact the Support team.

5.9 Defining kinematics

The input (or output) elements (kSysSpeedOrForce) are used to define the kinematics. They are placed outside the "GB" group and are used to represent motors (input element) and processing machines (output element) which drive, or are driven by, the gear unit.

The sun shaft speed and the torque on the planet carrier are predefined in this tutorial. The internal gear is static, in other words, its speed = 0 rpm. This provides a generally applicable type of modeling which can also be used if the internal gear rotates or displays a deviation in performance.

Add drive element for a sun shaft

1. Select the "AllObjects" main directory in the model tree structure
2. Add the drive element  (click on Elements box – Boundary conditions)
 - 1 x drive element (kSysSpeedOrForce, with the name "Sun")
3. Input the drive element details
 - Link the drive element with the coupling from the "SunShaft" sun gear shaft, "^\.GB.SunShaft.CouplingSun"
 - Speed constraint → Yes, Speed:(rpm) = 2000 ¹/_{min}
 - Torque constrained → No, because the torque on the planet carrier shaft is predefined.

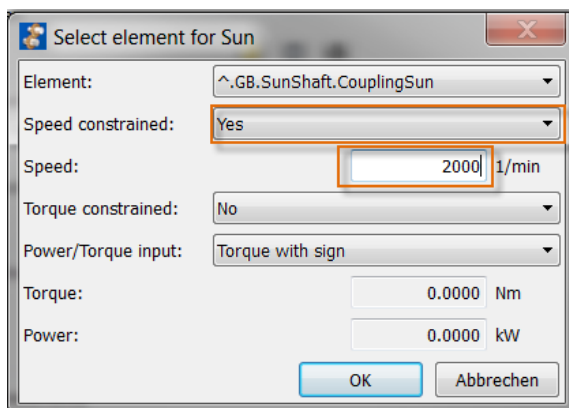


Figure 29. Kinematic definition for sun shaft

Add a drive element for the planet carrier shaft

1. Select the "AllObjects" main directory in the model tree structure
2. Add the drive element (click on Elements box – Boundary conditions)
 - 1 x drive element (kSysSpeedOrForce, with the name "Carrier")
3. Input the drive element details
 - Link the drive element with the coupling from the "CarrierShaft" planet carrier shaft, (^\.GB.CarrierShaft.CouplingCarrier)
 - Speed constraint → No
 - Torque constrained → Yes

- Power/Torque input → Torque with sign. As you want to make this model as universally applicable as possible, a torque with sign is defined here instead of a fixed driven value with Torque driven / Output. This setting means you can control the power flow directly using the prefix.
- Torque = -1000 Nm

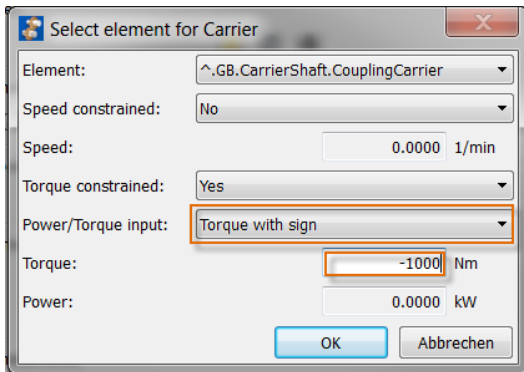


Figure 30. Defining kinematics for planet carrier shaft

Add a drive element for the internal gear shaft

1. Select the highest level, "AllObjects", in the model tree structure
2. Add the drive element (click on Elements box – Boundary conditions)
 - 1 x drive element (kSysSpeedOrForce, with the name "Ring")
3. Input the drive element details
 - Link the drive element with the internal gear shaft coupling "RingShaft" (^..GB.RingShaft.couplingRing)
 - Speed constraint → Yes, Speed = 0 1/min
 - Torque constrained → No

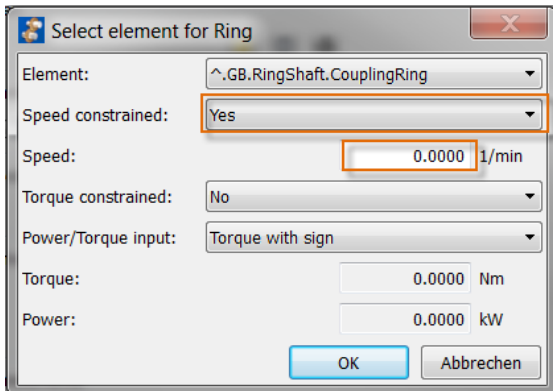


Figure 31. Defining kinematics for an internal gear shaft

To make the powerflow visible, you must calculate the kinematics. In this step, you can also tell whether the kinematics are running smoothly (red lines).

When the individual shafts (groups) are created, they are positioned on top of each other. They are now repositioned. Up to this point, the finished model now looks like this:

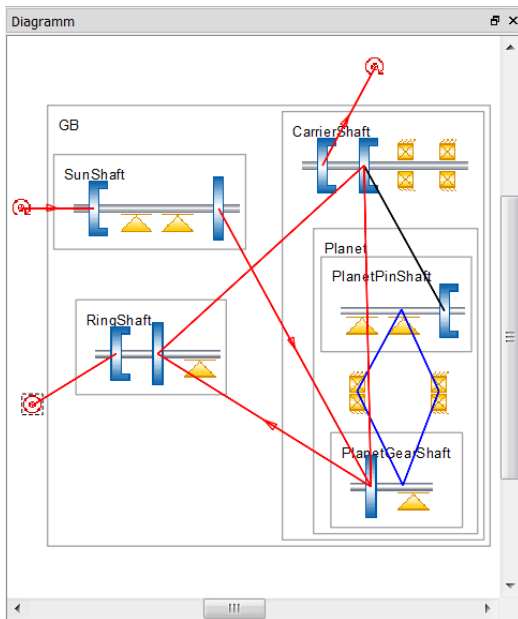



Figure 32. kinematics of a one-level planetary gear unit

5.10 Defining calculations

The calculations always start with the gear calculation because the geometry data from this calculation is later used for the force elements in the shaft calculation. As already mentioned in this tutorial, this model is used for sizing a new gear or for verifying an existing gear. Simplified versions of gear and shaft data are used in this tutorial.

5.10.1 Defining a gear pair

Use the Elements box to add the planet calculation element.

1. Select the "GB" group in the model tree structure
2. Add the KISSsoft calculation (click on Elements box - Calculations)
 - 1 x planet calculation  (kSoftPlanetGearSet, with the name "PlanetarySetCalc")
3. Constraint type: Select "Planetary gear pair constraint"
4. Input the planet calculation details:
 - Input the connections
 - Saving mode: Save File in KISSsys

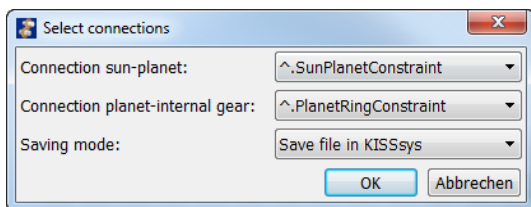


Figure 33. Add KISSsoft planet calculation

The gear data has been defined using the KISSsys rough sizing functions. The gear pair calculation also has an additional function, "GearSizing". In KISSsys this function is used to size the planetary stage and dimension the toothing to achieve the required safeties. The user enters the transmission ratio, meshing angle, and helix angle, in this function.

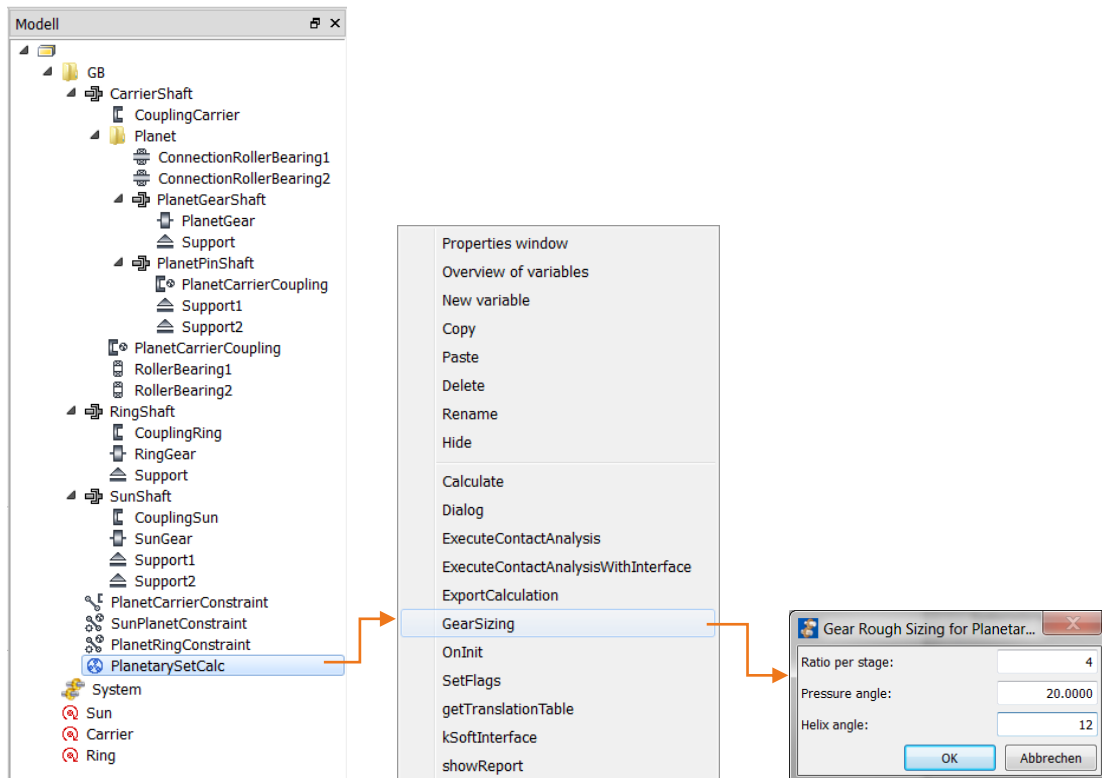


Figure 34. Rough sizing the planet calculation

This data is required for the first stage:


- $i = 4$
- $\alpha = 20^\circ$
- $\beta = 12^\circ$

You can modify the tothing data again later on in the KISSsoft system if necessary.

5.10.2 Defining a shaft calculation

Use the Elements box to add the coaxial shaft calculations. Add the shaft calculation for the main shafts (sun shaft, internal gear shaft and planet carrier shaft "MainLineShaftCalc") to the "GB" group. This covers all the shafts in this group. The shaft calculation for the planet pin and the planet gear is added to the "Planet" group.

Adding the shaft calculation for the main shaft:

1. Select the "GB" group in the model tree structure
2. Add the KISSsoft calculation (click on Elements box - Calculations)
 - 1 x coaxial shaft calculation  (kSoftCoaxialShaft, with the name "MainLineShaftCalc")

Double click on the shaft calculation to start the KISSsoft shaft calculation function. You can now model the shaft in the graphical Shaft editor. When you open the graphical Shaft editor, you can see that the machine elements are already present, as defined in the structure in KISSsys. Enter these values to generate the shaft file.

Input values for the "SunShaft" sun gear shaft (3rd shaft from the top):

Element	Position y [mm]	Length l [mm]	Diameter d [mm]	Degree of freedom	Comment
SunShaft	0				
Outer contour of cylinder	0	150	20		
"Coupling" CouplingSun(Sun)	10	20	20.5		
"Cylindrical gear" SunGear (SunPlanetConstraint)	135				
"Cylindrical gear" SunGear (SunPlanetConstraint)2	135				
"Cylindrical gear" SunGear (SunPlanetConstraint)3	135				
Support1	10			1-1-1-0-0-0	1=fixed
Support2	135			1-0-1-0-0-0	

The torque is applied to the sun shaft at the shaft entry via the "coupling/motor" force element (drive), it is then divided into thirds via the "cylindrical gear" force element due to the meshing of the three planets (one third torque for each planet) (driven).

The sun gear shaft has no bearings. In this example it is positioned on the hub of an electrical motor. The shaft can tilt freely (with these settings). However, if you want to display a restricted tilting action, you can select "fixed with stiffness" for the x and z rotations as an alternative. The displacements are fixed to the left side in every direction.

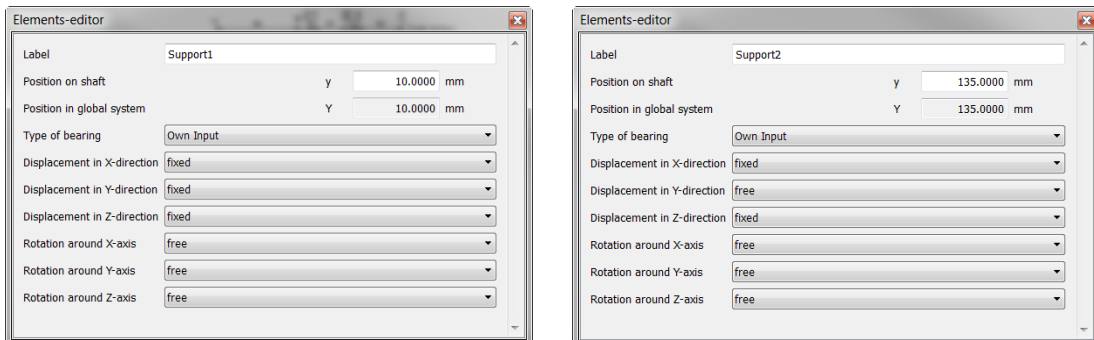


Figure 35. Properties of left support Properties of right support

Input values for the "CarrierShaft" planet carrier shaft:

Element	Position y [mm]	Length l [mm]	Diameter d [mm]	Degree of freedom	Comment
CarrierShaft	96				
Outer contour of cylinder	0	10	80		
Outer contour of cylinder	10	60	160		
Outer contour of cylinder	70	50	40		
Inner contour, cylindrical bore	0	20	70		
Inner contour, cylindrical bore	20	40	140		
CouplingCarrier(Carrier)	110	20	45		
PlanetCarrierCoupling (PlanetcarrierConstraint)	38	0			

PlanetCarrierCoupling (PlanetRingConstraint)	38	0			
PlanetCarrierCoupling (SunPlanetConstraint)	38	0			
RollerBearing1	5			Fixed bearing, left	SKF 61816
RollerBearing2	74			Fixed bearing, right	SKF 61808

The torque is applied to the planet carrier at the tothing position via the "coupling/motor" force element of the sun-planet and planet-internal gear connection (drive, torque dependent on the current operating pitch diameter). The entire torque is then distributed by the "coupling/motor" force element to the planet carrier shaft (driven).

The roller bearings support the dead weight of the shaft.

Input values for the "RingShaft" internal gear shaft:

Element	Position y [mm]	Length l [mm]	Diameter d [mm]	Degree of freedom	Comment
RingShaft	115				
Outer contour of cylinder	0	40	230		
Inner contour, cylindrical bore	0	40	210		
"Coupling" CouplingRing(Ring)	20	40	250		
"Cylindrical gear" RingGear (PlanetRingConstraint)	20				
"Cylindrical gear" RingGear (PlanetRingConstraint)2	20				
"Cylindrical gear" RingGear (PlanetRingConstraint)3	20				
Support	20			1-1-1-1-0-1	1=fixed

The torque is applied via the "cylindrical gear" force element with the meshing of the three planets. Each planet takes one third of the total torque (drive), which is supported at the internal gear shaft on the shaft entry via a "coupling/motor" force element (driven).

The internal gear is supported by a general support and absorbs the axial forces from the helical gear teeth. The degree of freedom, "Rotations y", must be non-locating, otherwise the kinematics would be over-determined.

After you have made additional graphical settings in the Shaft editor (position shafts one after the other, assign colors to the shaft) the "MainLine" shaft looks like this:

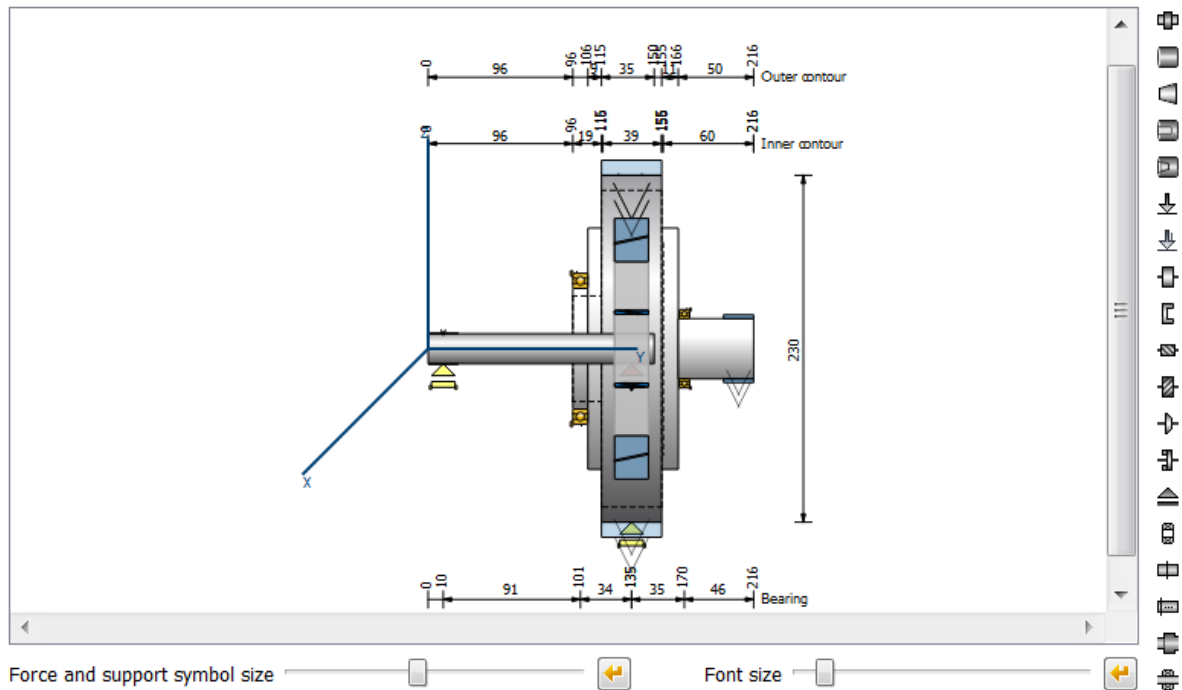


Figure 36. MainLine shaft geometry

Adding a planet shaft calculation:

1. Select the "Planet" group in the model tree structure
2. Add the KISSsoft calculation (click on Elements box - Calculations)
 - 1 x coaxial shaft calculation (kSoftCoaxialShaft, with the name "PlanetShaftCalc")

Double click on the shaft calculation to start the KISSsoft shaft calculation function. You can now model the shaft in the graphical Shaft editor. When you open the graphical Shaft editor, you can see that the machine elements are already present, as defined in the structure in KISSsys.

Enter these values to generate the shaft file.

Input values for the "PlanetPinShaft" planet pin shaft (2nd shaft from the top):

Element	Position y [mm]	Length l [mm]	Diameter d [mm]	Degree of freedom	Comment
PlanetPinShaft	0				
Outer contour of cylinder	0	60	20		
PlanetCarrierCoupling (PlanetCarrierConstraint)	50				
Support1	10			1-1-1-0-0-0	1=fixed
Support2	50			1-0-1-0-0-0	

The planet pin shaft is only subject to bending, not to torque.

The calculated deformation (bending) depends on the position of the general supports.

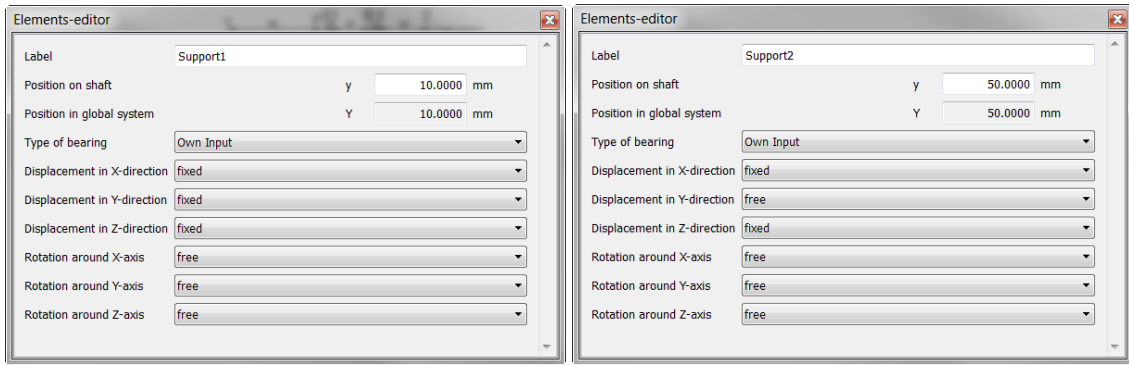


Figure 37. Properties of left support

Properties of right support

Input values for the "PlanetGearShaft" planet gear shaft:

Element	Position y [mm]	Length l [mm]	Diameter d [mm]	Degree of freedom	Comment
PlanetGearShaft	15				
Outer contour, cylinder	0	30	40		
Inner contour, cylindrical bore	0	30	32		
"Cylindrical gear" PlanetGear (PlanetRingConstraint)	15	30			
"Cylindrical gear" PlanetGear (SunPlanetConstraint)	15	30			
Support1	15			0-1-0-0-0-0	1=fixed

The torque is introduced via the "cylindrical gear" force element with the sun-planet meshing (drive) and transferred via the "cylindrical gear" force element with the planet-internal gear meshing (driven).

The planet gear body is supported on general supports to absorb the resulting axial forces. The resulting axial forces are due to the uneven operating pitch diameter and the helix angle on the operating pitch diameter.

Element	Position y [mm]	Length l [mm]	Diameter d [mm]	Degree of freedom	Comment
ConnectionRollerBearing1	20			Non-locating bearing	SKF 61804
ConnectionRollerBearing2	40			Non-locating bearing	SKF 61804

After you have made additional graphical settings in the Shaft editor (position shafts one after the other, assign colors to the shaft) the "Planet" shaft looks like this:

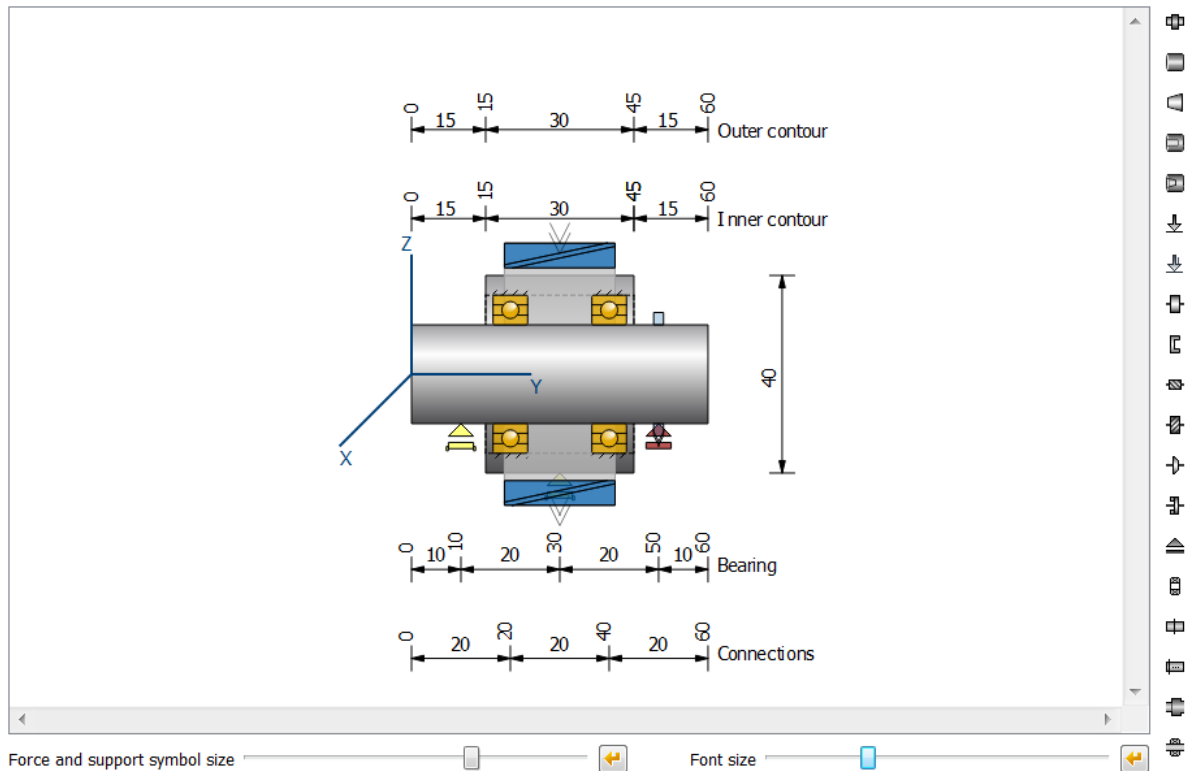



Figure 38. Shaft geometry of a planet

5.11 3D view and positioning

Add the 3D View from the Elements box to display gear units in the KISSsys system.

Adding the 3D View:

1. Select the "AllObjects" main directory in the model tree structure
2. Add 3D View  (click on Elements box - System elements)
 - kSys3DView, accept suggested name

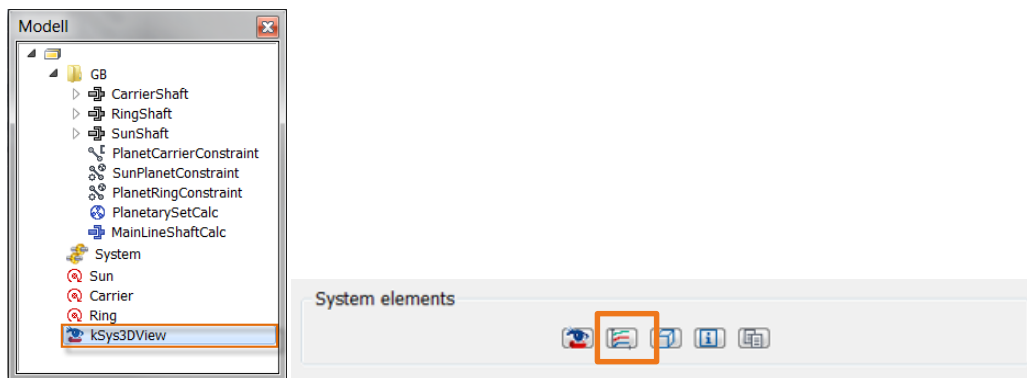


Figure 39. System elements elements box

Either double-click or click the right-hand mouse button on the kSys3Dview element under Show. KISSsys opens the 3D view. Initially, all the shaft elements overlap each other at their coordinate origins.

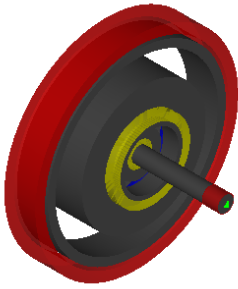


Figure 40. 3D view not positioned

5.11.1 Positioning the shafts relative to each other

Then position the shafts relative to each other. This example uses the "MainLine" coaxial shaft as a reference. The "Planet" shaft is referenced to this shaft. The "Planet" coaxial shaft should be parallel to the sun gear shaft. The distance between the two shafts matches the center distance of the gears, taken from the overall planet calculation.

The positioning of coaxial shafts is defined by groups, not by shafts, as is the case for simple shafts. To do this, right-hand mouse click on the groups and select "Dialog".

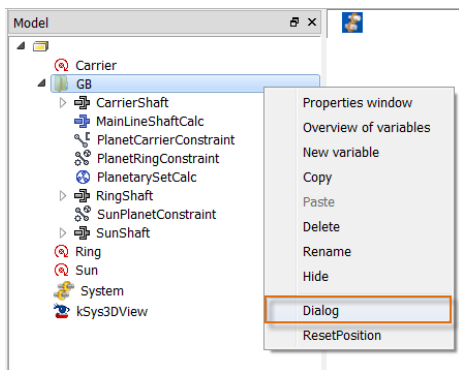


Figure 41. Calling the positioning dialog

Select "Parallel to Shaft/Group" for the "GB" group. Select "SunShaft" as the element. Leave the parameter at the origin (0/0/0).

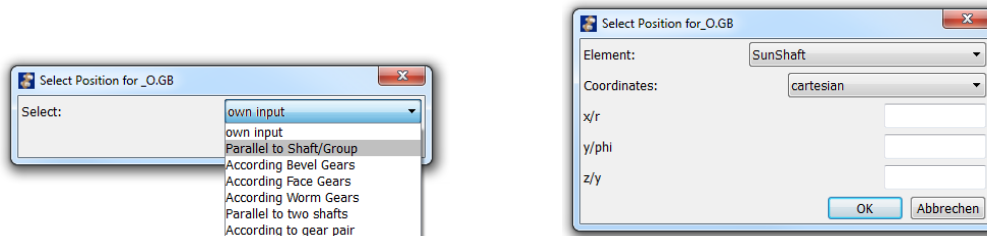


Figure 42. Positioning the "GB" group

Now position the "Planet" group relative to the "GB" group. Then, select "According to gear pair" under "Dialog". Then, select the planet calculation and select the definition according to "Formulas":

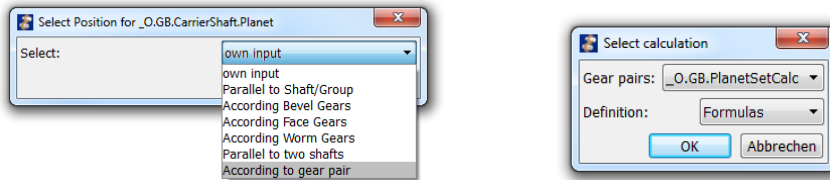


Figure 43. According to gear pair

The sun shaft is suggested as the reference element in the positioning details. Select this element. The parameters for center distance etc. are transferred automatically from the calculation file.

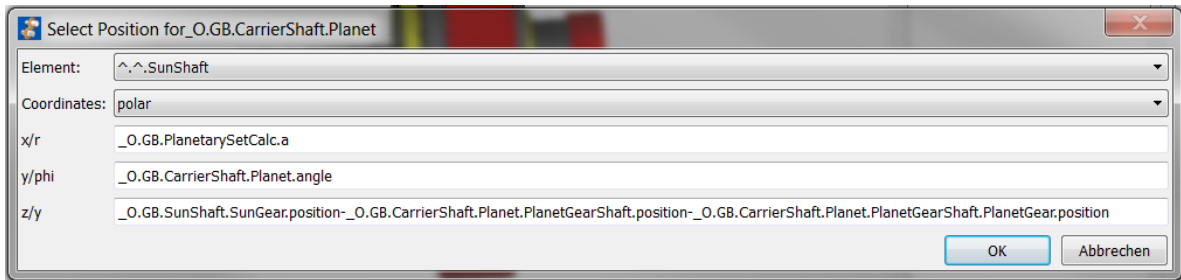


Figure 44. Formulas in the positioning dialog

You have now defined the position of the shafts.

5.11.2 3D View

Now that the elements are positioned relative to each other, the 3D View can be generated again. To do this, click on the "Refresh" button. You now see this display, in which you can clearly see the axis of the planetary bolt.

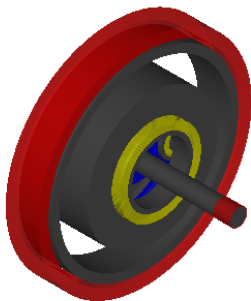


Figure 45. 3D view of the system after positioning

You can set a number of different color properties to improve the display. Click the right-hand mouse button on "CarrierShaft" and input Transparency = 0.5 for SetColor. The shaft now appears to be slightly transparent.

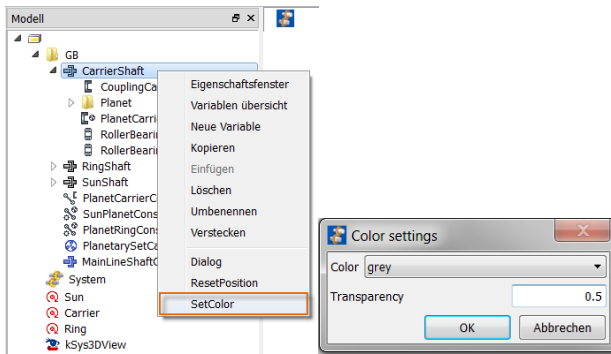


Figure 46. Transparency settings for the planet carrier shaft

Toothings should be displayed with their teeth. To do this, set "Representation in" in the 3D settings to "Solid Elements".

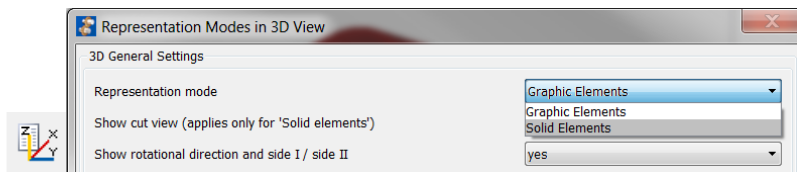


Figure 47. Displaying the teeth

The gear unit is now displayed like this. You will find more information about graphical displays in Chapter 7 "Final Tasks".

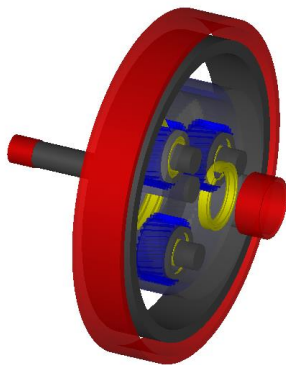


Figure 48. Final display

6 Designing the User Interfaces

6.1 Overviews of toothings and bearing data

A number of pre-configured tables are provided. These already contain extensive overviews of results which you can select and then load directly into the model. This means you can test your results very quickly.

The following two tables from the Elements box are inserted into the main directory ("AllObjects") under System elements: "PlanetaryGearCalculation" and "ShaftCalculations":

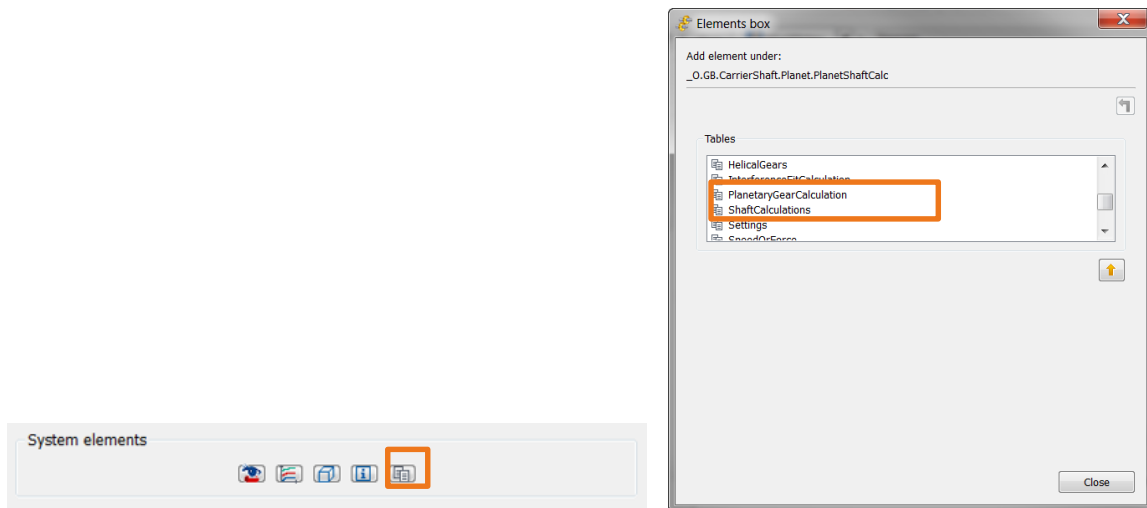


Figure 49. Pre-defined tables

You can assign any name you like to these tables because the names do not affect which values are displayed.

To display the two tables, either click the right-hand mouse button and select "Show" or double-click on them. Click on Window→Parts to display the different windows at the same time:

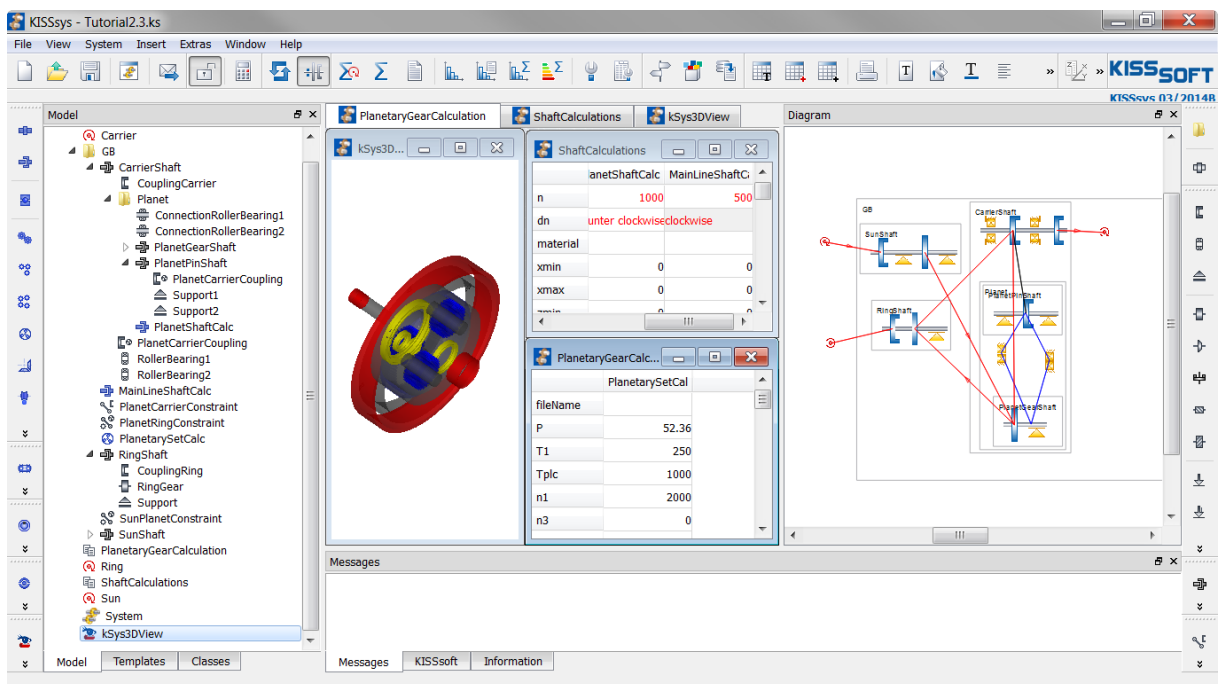


Figure 50. Inserted tables with results from the shaft and planet gear calculation

Click on Window – Parts to display all the windows simultaneously. These tables provide an overview of the parameters that are being used.

6.2 User Interfaces

6.2.1 Adding a user interface table

You now need to add a freely configurable table as an overview of the main entries and the main results. To do this, go to the Elements box, select the "UserInterface" and add it to the main directory.

Click on "Show" or double-click to open the table again. Click on "Dialog" to change the number of rows and columns. Select 11 rows and 4 columns.

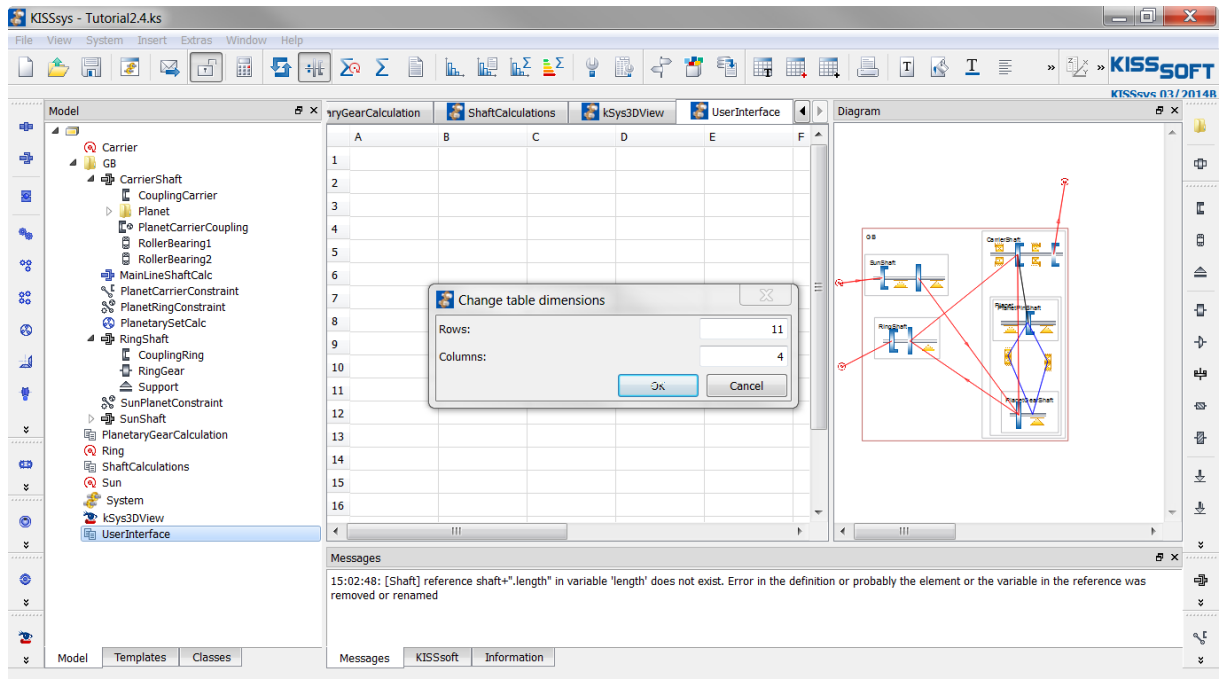


Figure 51. Inserting and defining the UserInterface table.

6.2.2 Entering text

You input the row and column headers directly into each particular field. Enter the texts as shown below.

	A	B	C	D
1		Speed, n [rpm]	Torque, T [Nm]	Power, P [kW]
2	Sun (Input)			
3	Carrier (Output)			
4	Ring			
5	Ratio, i			
6				
7	RESULTS	Sun Gear	Planet Gear	Ring Gear
8	Safety Root, SF			
9	Safety Flank, SH			
10				
11				

Figure 52. Inputting row and column headers as text

6.2.3 Adding output fields

Next, add the result fields. In KISSsys, these are declared as expressions. As an example, the input torque is to be displayed here as a result. As shown in section 5.9 "Defining kinematics", the drive torque (on the planet carrier shaft) is defined in this model, and therefore the input torque (on the sun shaft) is a result.

Before copying the variables to the table, you must determine which insertion type the drag and drop method should be used for. You must specify this globally. Define this under Extras→Settings in the Tables tab. As we initially inserted the variables as an expression, this setting will be transferred here.

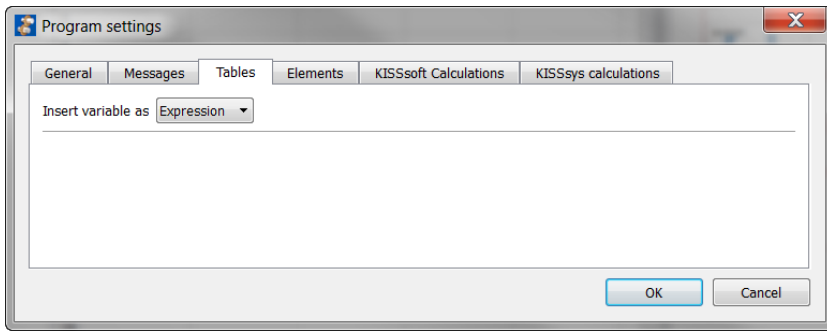


Figure 53. Global setting for the drag and drop method

The "Sun" drive element is inserted into the UserInterface table so that the calculated input torque can be displayed. To display the properties, click on "Sun" → Properties. The properties are displayed in a new window, in the same place as the kinematics diagram.

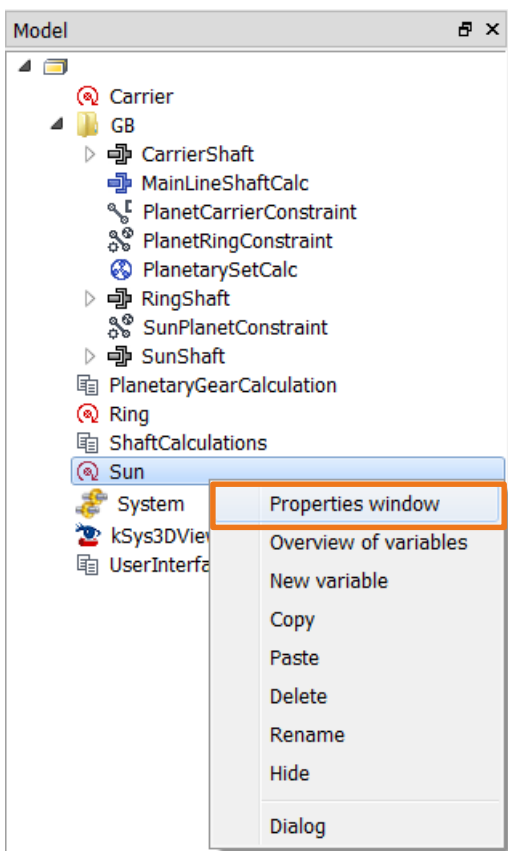


Figure 54. Displaying a drive element property

Click (and hold down) the right-hand mouse button on the "torque" variable, drag it to the line you want to move it to, and then release the mouse button.

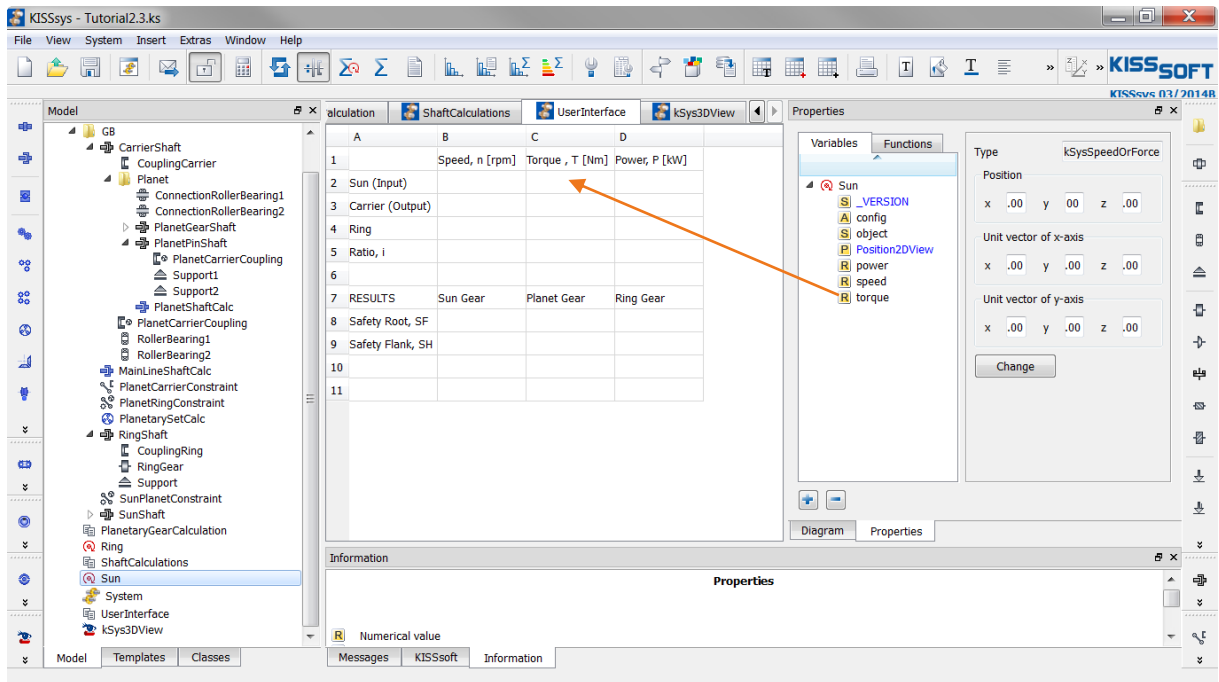


Figure 55. Inserting a results field as an "expression"




Use this method to display the remaining results. The safety values for the tooth root and tooth flank calculation are given as variables SF1, SF2, SF3 and SH1, SH2, SH3 in the properties of the "PlanetSetCalc" planet calculation. The UserInterface now looks like this:


	A	B	C	D
1		Speed, n [rpm]	Torque , T [Nm]	Power, P [kW]
2	Sun (Input)		250	52.36
3	Carrier (Output)	500		52.36
4	Ring		750	0
5	Ratio, i			
6				
7	RESULTS	Sun Gear	Planet Gear	Ring Gear
8	Safety Root, SF	2.0511	1.403	2.2895
9	Safety Flank, SH	1.0054	1.0398	1.784
10				
11				

Figure 56. UserInterface with all the results fields (expression type)

6.2.4 Inserting input fields

Next, add the input fields. In KISSsys, these fields are called references, because these values refer to a variable which takes on the value of the entry. These values are shown in red in the cells.

You can use the method described above to insert reference values (change the global setting to Reference, use the right-hand mouse button to drag the variable into the field). Alternatively, you can use this second method to change the global setting to Expression. First select the variable and the relevant cell. Click on the  icon in the task bar to insert the variable as a reference. This method enables you to insert the variable either as text  or as an expression .

To insert the input window for the sun shaft's speed, click on the "Sun" drive element in the model tree structure. The variables for this element now appear in the properties window. Now select the "speed" variable and the corresponding cell. Then click on  to insert the variable as a reference.

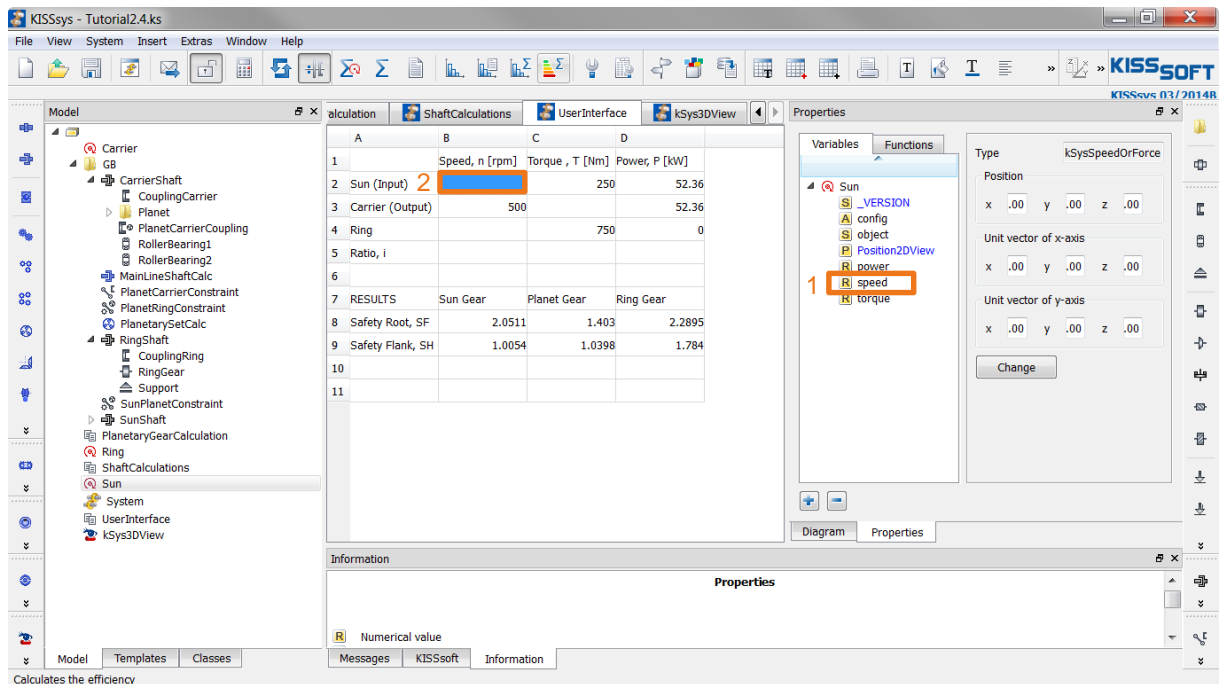


Figure 57. Inserting a variable as a reference

Other input fields are the torque for driven gear (planet carrier shaft-carrier) and the speed for an internal gear (ring). Use the same method to input these values. The UserInterface table now looks like this.

A	B	C	D
1	Speed, n [rpm]	Torque , T [Nm]	Power, P [kW]
2	Sun (Input)	2000	250 52.36
3	Carrier (Output)	500 -1000	52.36
4	Ring	0	750 0
5	Ratio, i		
6			
7	RESULTS	Sun Gear	Planet Gear Ring Gear
8	Safety Root, SF	2.0511	1.403 2.2895
9	Safety Flank, SH	1.0054	1.0398 1.784
10			
11			

Figure 58. UserInterface with all its input fields (References type)

NOTE: the icons do **not** change the type of a variable that is currently present in a field. Instead the variable from the path is written to the field, overwriting any variables that may already be there.

You should then output the total reduction. This is calculated from the input speed and the output speed. This should appear to the right of the "i tot" field. To do this, click the right-hand mouse button on "Insert real" to the right of the "Ratio i" field. Then input the following formula as the expression:

```
#
IF _O.Carrier.speed = 0 THEN
    RETURN 0;
ELSE
    RETURN abs(_O.Sun.speed/_O.Carrier.speed);
ENDIF
```

Figure 59. Calculating the total reduction from the speeds at input and output.

TIPS: Before you input the code, copy the "_O Carrier.speed" variable from the path window to the clipboard (Ctrl+C). This entry is case-sensitive. After you have entered the code, copy the entire text to the Clipboard (with the right-hand mouse button) before closing the window. This is because the text will be lost if any compilation errors occur. This means the text is still available to you and can be corrected if necessary.

6.2.5 Calling functions

You can also run functions from the user interface. Five functions are implemented in this example:

- "Kinematics" function calculates the model's kinematics
- "Strength" function performs the strength analysis
- "Planetary Set" function calls the KISSsoft calculation for the planetary stage
- "Shaft MainLine" function calls the KISSsoft calculation for the main shaft
- "Shaft Planet" function calls the KISSsoft calculation for the planet shaft

You insert these functions into a field in the same way, by clicking the right-hand mouse button on the required cell and then clicking on "Insert function". You must define a name for the function and the commands that are to be performed.

A	B	C	D
1	Speed, n [rpm]	Torque, T [Nm]	Power, P [kW]
2	Sun (Input)	2000	250
3	Carrier (Output)	500	-1000
4	Ring	0	750
5	Ratio, i	4	
6			
7	RESULTS	Sun Gear	Planet Gear
8	Safety Root, SF	2.0511	1.403
9	Safety Flank, SH	1.0054	1.0398
10			
11			

Variable

- ▲ 🔍 Su
- S
- A
- S
- P
- R
- R
- R

- ✎ Edit
- 📄 Copy
- 📄 Paste
- Insert
- Delete
- Insert real
- Insert string
- Insert list
- Insert data base list
- Insert function
- Format
- Adjust content automatically

Information

🔍 Numerical value

Messages | KISSsoft | Information

Figure 60. Inserting a function into a cell

For the "Kinematics" function in cell D10:

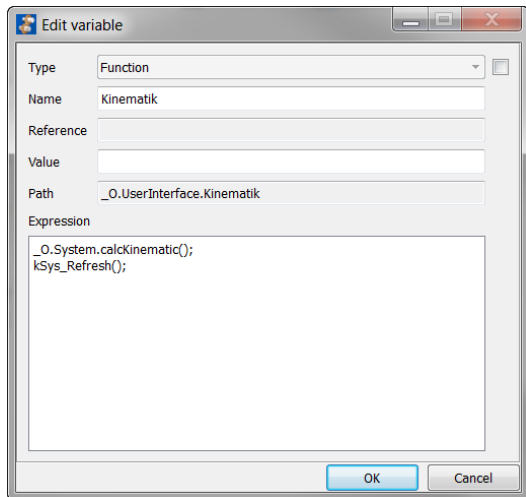



Figure 61. Function for calculating the kinematics

Although the "Kinematics" function name appears in the user interface it has no other meaning.

You need to input this text:
`_O.System.calcKinematic();`
`kSys_Refresh();`

`_O.System.calcKinematic();` means that the `calcKinematic` function was called under "System" (corresponds to Σ), the brackets are empty because no arguments were transferred. The ";" character marks the end of the command.

`kSys_Refresh` corresponds to "Refresh All" . This ensures that the calculated values (in this case the output speed and the input torque) are displayed in the UserInterface.

For the "Strength" function in cell D11:

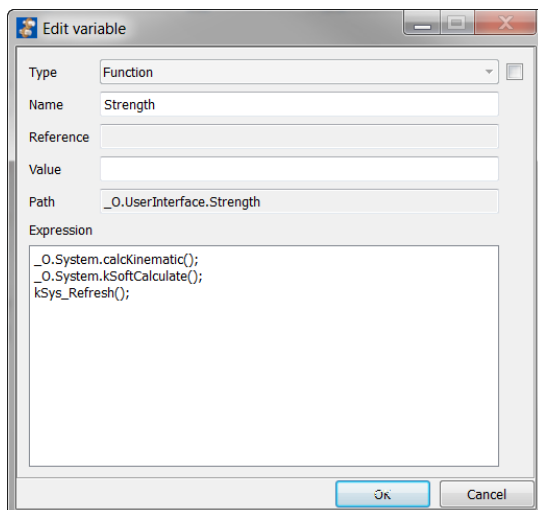


Figure 62. Function for calculating strengths

The first step here is to recalculate the kinematics.

You need to input this text:
`_O.System.calcKinematic();`
`_O.System.kSoftCalculate();`
`kSys_Refresh();`

`_O.System.kSoftCalculate();` means the `kSoftCalculate` function Σ was called under "System" (the brackets are empty because no arguments were transferred. The ";" character marks the end of the command). This calls all the KISSsoft strength calculation operations.

For the "Planetary Set" function in cell A10:

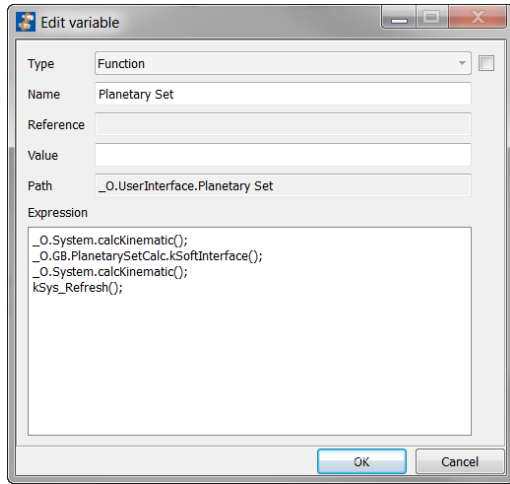


Figure 63. Function for calling KISSsoft for the planetary stage

The "kSoftInterface" command opens the appropriate KISSsoft calculation.

However, you must input the entire path so that it is clear which KISSsoft calculation is being called.

You need to input this text:

```
_O.System.calcKinematic();
_O.GB.PlanetSetCalc.kSoftInterface();
_O.System.calcKinematic();
kSys_Refresh();
```

The functions for KISSsoft shaft analysis (Shaft MainLine in cell A11 and Shaft Planet in cell B11) are implemented in the same way as the KISSsoft planetary stage calculation. The UserInterface now has five functions, with gray backgrounds, that can be called with a double-click:

	A	B	C	D
1		Speed, n [rpm]	Torque, T [Nm]	Power, P [kW]
2	Sun (Input)	2000	250	52.36
3	Carrier (Output)	500	-1000	52.36
4	Ring	0	750	0
5	Ratio, i	4		
6				
7	RESULTS	Sun Gear	Planet Gear	Ring Gear
8	Safety Root, SF	2.0511	1.403	2.2895
9	Safety Flank, SH	1.0054	1.0398	1.784
10	Planetary Set			Kinematik
11	Shaft MainLine	Shaft Planet		Strength

Figure 64. Functions inserted into the UserInterface

7 Final Tasks

7.1 Setting the 3D view

Use the system-specific settings for your system to change the color and appearance of the elements in the 3D view.



Figure 65. 3D-specific settings

7.1.1 Displaying the couplings

In a KISSsoft file, the way in which the sun shaft, internal gear shaft and planet carrier shaft couplings are displayed is controlled by their effective diameter. If this effective diameter is input manually, enter a value that is 10 mm larger than the particular shaft diameter. In the KISSsys 3D View, couplings are usually shown in red. You can change this under "Couplings" in the 3D settings. You can also specify the transparency level of the elements by entering a number between 0 and 1.

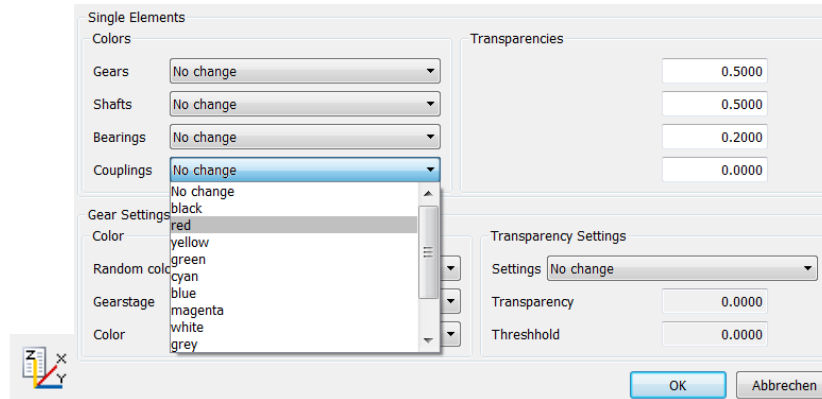


Figure 66. Displaying the couplings

7.1.2 Cut View

It is a good idea to display planetary gear units in Cut View to make the structure of the gears easier to see. You can also use the Cut View in a simulated run (see section 7.3 Simulation). To open the Cut View, go to "Show cut view" and change it to "Yes". Click on "No" to close this view and refresh the view.

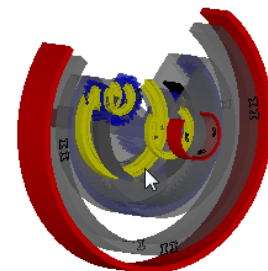
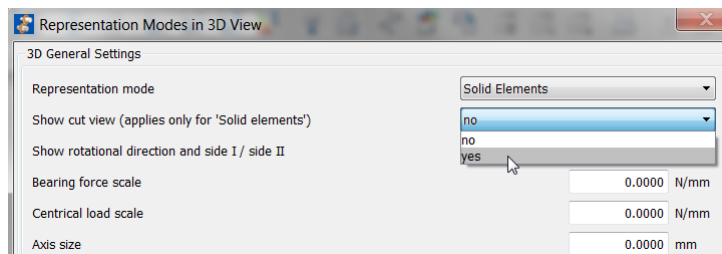


Figure 67. Activating Cut View

7.2 Displaying all the planets

In the default setting, only one planet gear is displayed in the 3D View. To display all the planets, click on the "ShowPlanets" option under "kSys3DView". The planets view returns to its default setting either if the Cut View is deactivated again or if you toggle to a simplified display of the gears.

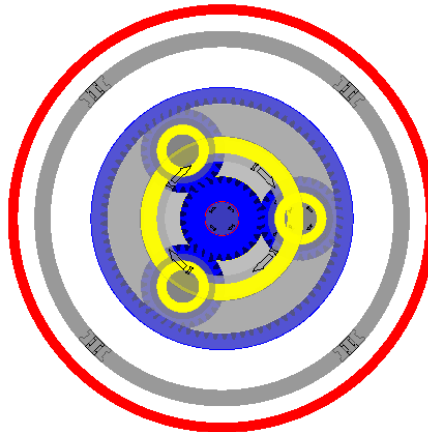
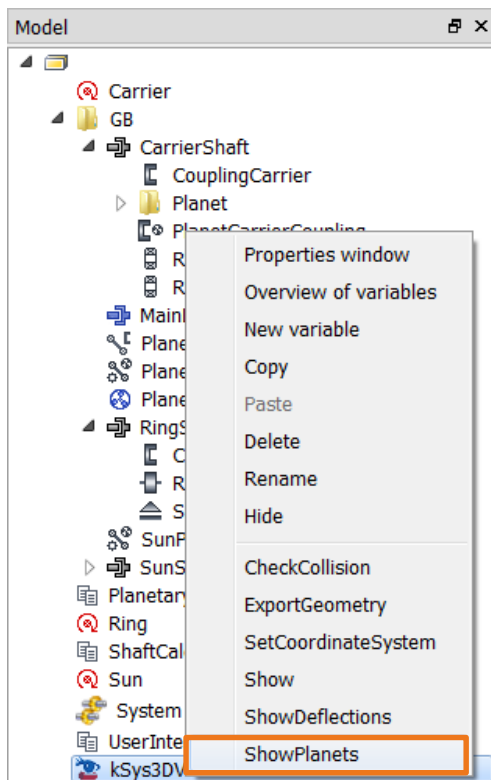



Figure 68. Displaying all the planets

7.3 Running a simulation of the planetary gear unit

The simulation run enables you to visually examine the speeds and directions of rotation of each individual gear. This is very useful if you are working with very complex gear systems (or drive trains). To enable a simulation run, click on the  icon in the task bar or click the right-hand mouse button on System → Activate simulation. Press "Esc" to interrupt the run. Define the speed of the simulation in the next window.

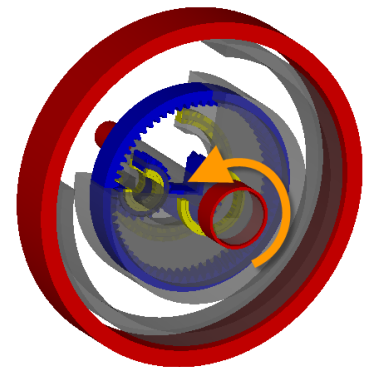
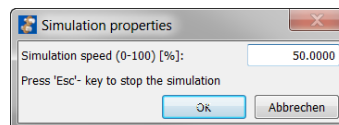
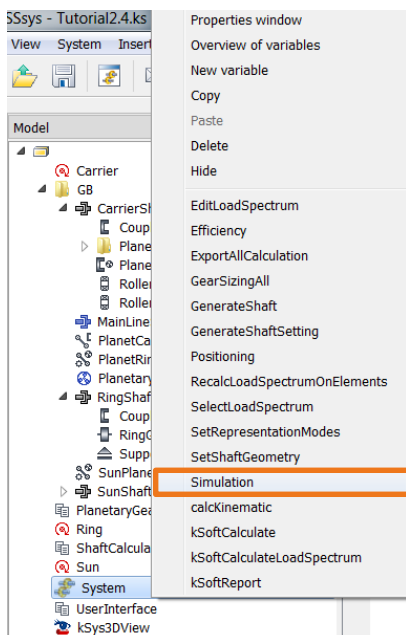


Figure 69. Activating a simulation run