KISSsys 03/2016 – Instruction 026

Group modelling

15/03/2016

KISSsoft AG

Rosengartenstrasse 4
8608 Bubikon
Switzerland

Tel: +41 55 254 20 50
Fax: +41 55 254 20 51
info@KISSsoft.AG
www.KISSsoft.AG

SHARING KNOWLEDGE

Contents

1. Introduction

This tutorial explains how to model a complete gearbox by using some predefined groups of elements/ parts of a gearbox. For that purpose, the user can insert in the tree structure the different groups and then connect them by adding some gear connections, coupling connections or even merging the shafts. The positions in the 3D view are then set automatically in the background.

2. Selection of the groups

The first thing to do to see properly what is happening is to activate the 3D view like below (double click or "Show")
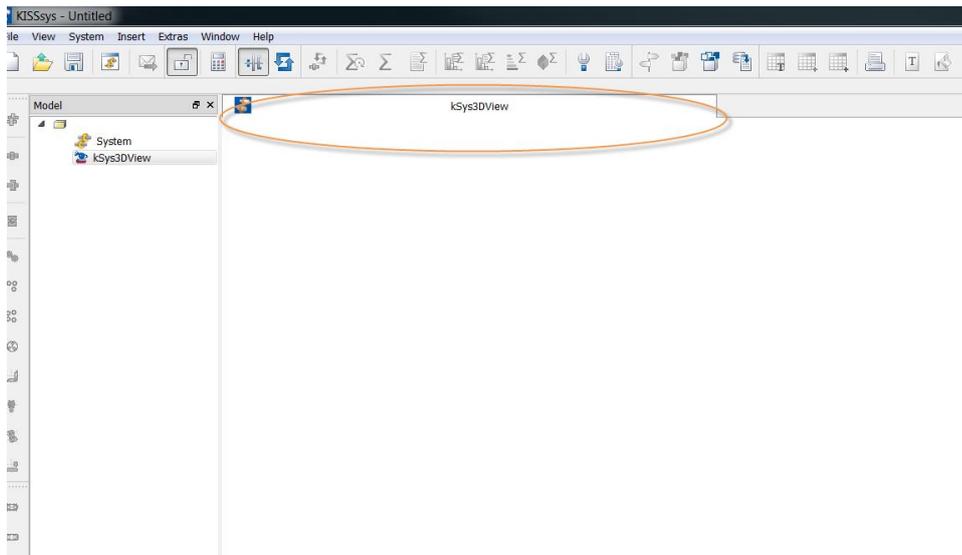

You can then go in the "Group Box" button and select which group you want to add in the three structure. Be careful, the group will be added on the level below the current selection. It is then recommended to add them on the root of the tree structure.
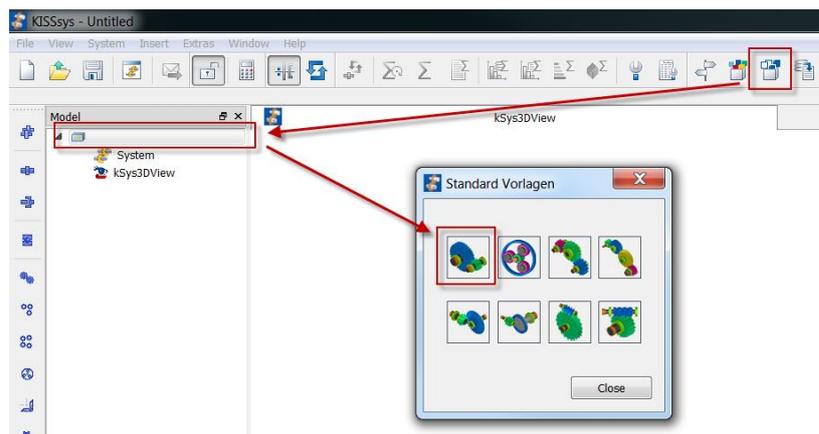
The group appears then directly on the screen as the 3D view is active.
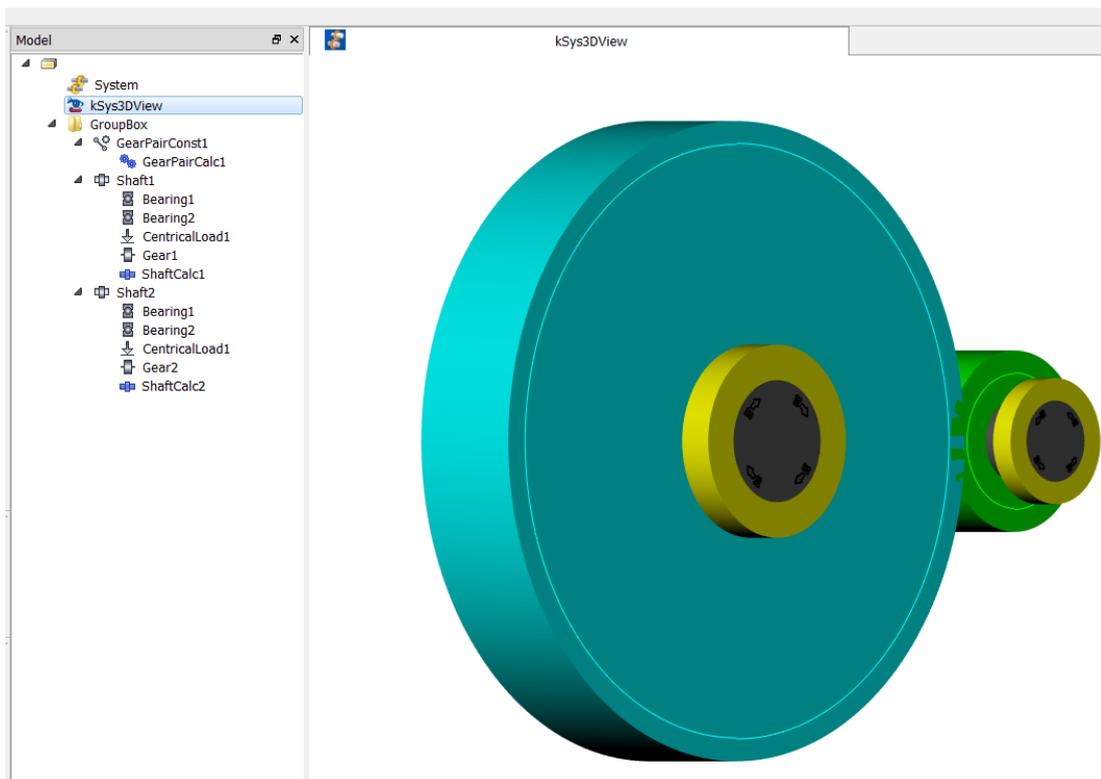
You can then add a second group to combine with first one. In the example below the planetary.
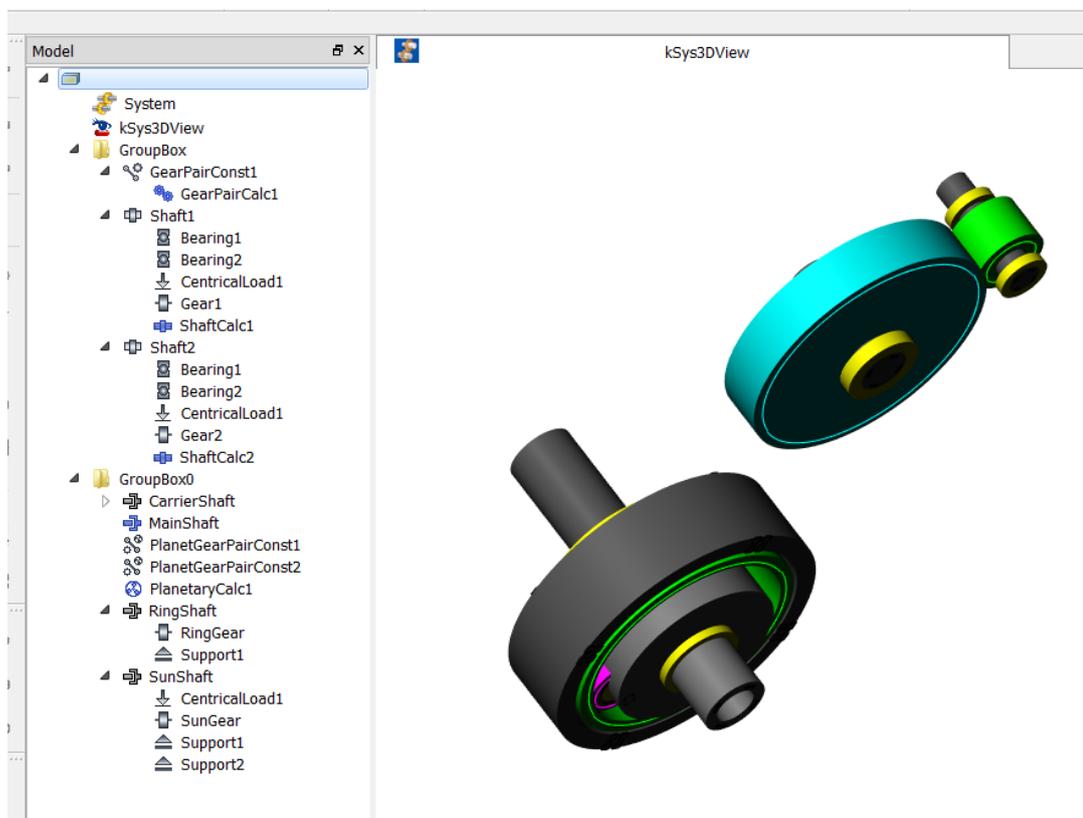
The different possibilities to connect the two groups are then explained after.
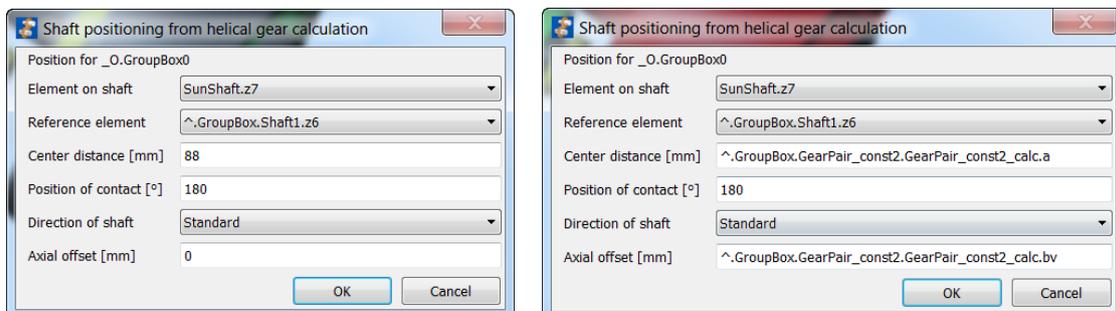
3. Connection and positioning by gears

One possible connection would then be to add gears on two shafts and then use a gear pair connection. See below with the gears "z6" and "z7" in blue.
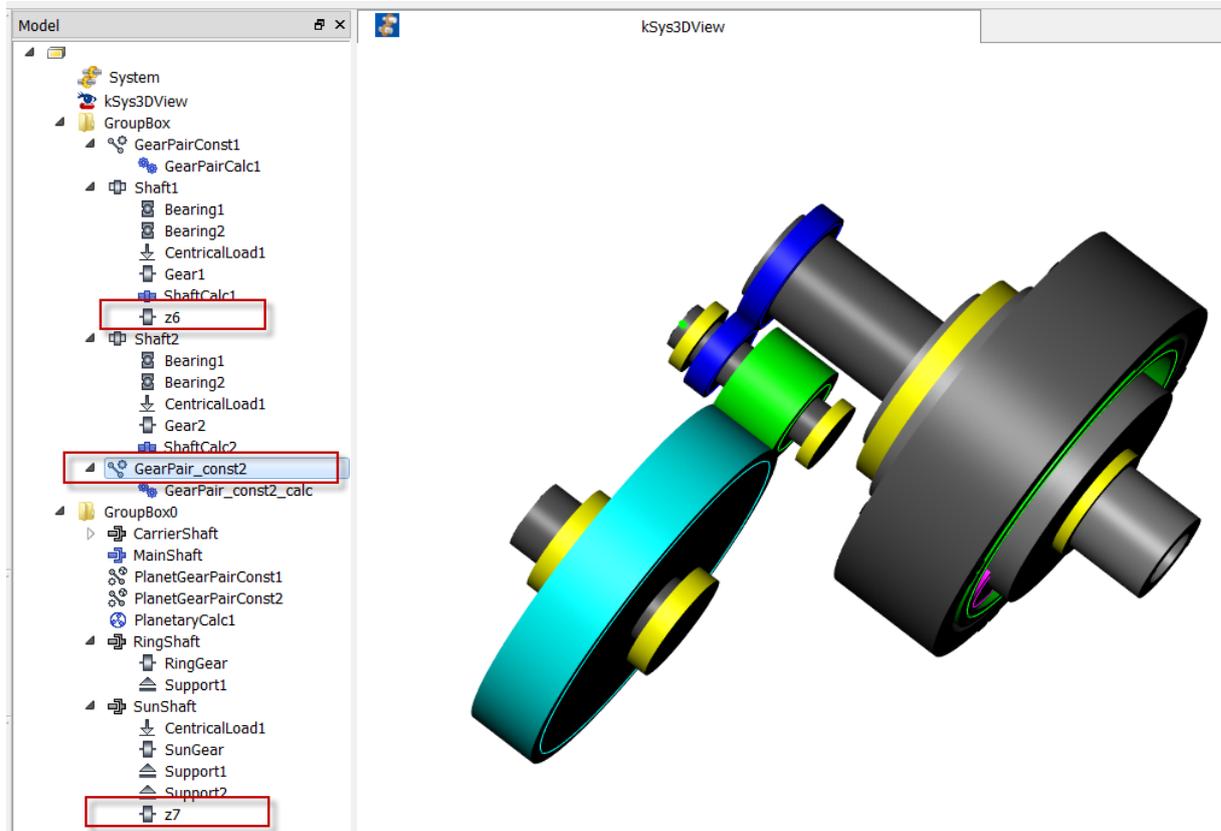
At this point, a default geometry is given to the gears, so the positioning of the shafts is done automatically, even without a gear pair calculation element. The user can then add one to change the geometry of the gears. The positioning of the shafts is then automatically updated according to the new center distance.

Important: If needed, you can change the position of the gear on each shaft through the shaft calculation, and you can also change the position of contact in the dialog function of the group or the shaft containing one of the two gears connected. You can finally also change the direction of the shaft or other parameters…

In this example, for "GroupBox0", before and after adding the gear pair calculation element:


After adding couplings and boundary conditions, the user can also size the gears automatically.

4. Connection and positioning by couplings

Another possibility is to connect two shaft coaxially by using couplings and a coupling connection. This action creates then coaxial shafts, but they are still treated separately in their respective calculation.

The first step is to add a coupling on each shaft you want to connect. These couplings will be added at a default position on the shaft, extreme right or left, where there is no element already defined (by default on the right). For example here on "Shaft2" and "SunShaft":
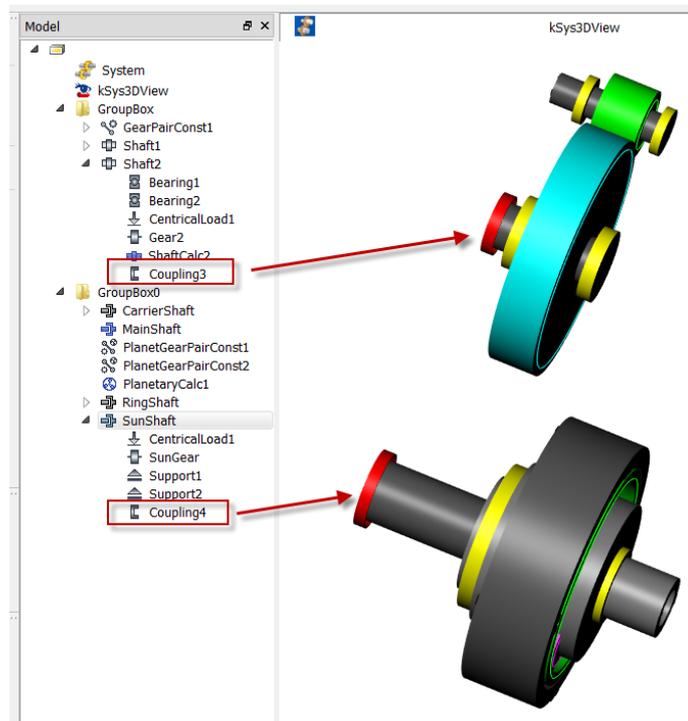

Then, when adding a coupling constraint connecting these two couplings, the shaft will be positioned so that the couplings are overlapping each other, with automatically the best direction for the shafts so that they are not overlapping each other. If the user wants to modify the position of the couplings, it is then recommended to put them as close as possible from the edge of a shaft, and to do that before adding the constraint.
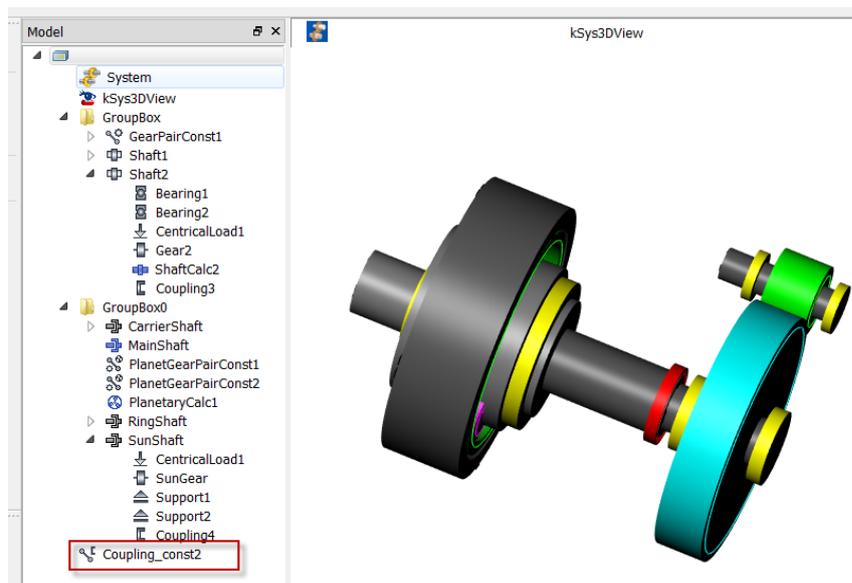
When calling the function "Dialog" of the constraint, the shafts will anyway always be repositioned if the user keeps the selection "yes" for the coaxial positioning. He can then also add an offset between the two shafts to move them without moving the couplings independently (negative possible if he wants that the shafts get closer to each other).

Here example if "Coupling3" moved to the other side of "Shaft2", and with an offset of 20 mm.
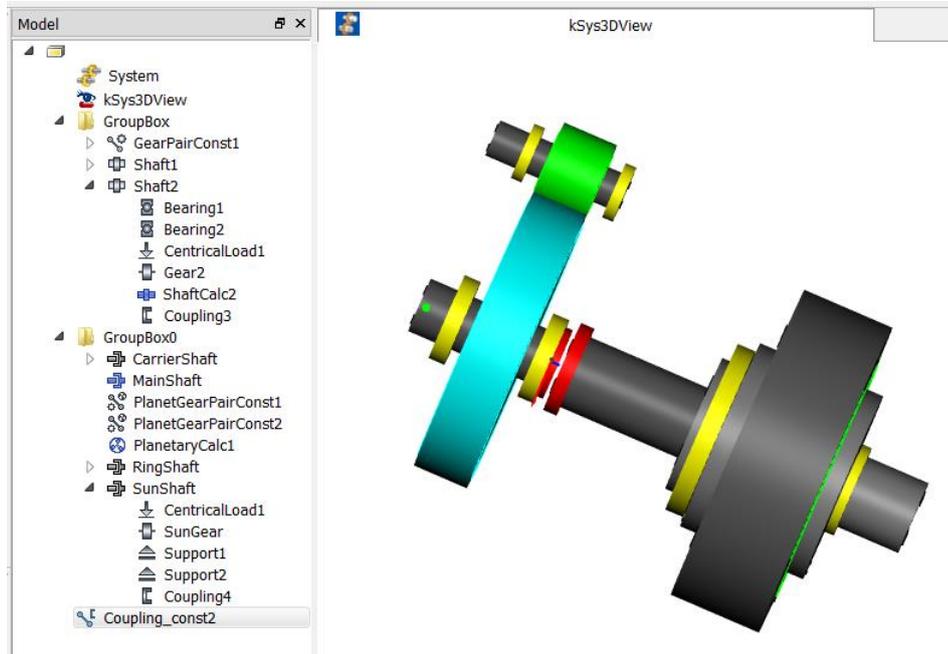

After calling the "Dialog" function of the constraint, the "SunShaft" is then repositioned on the other side, with the opposite direction (possible to change in the dialog function of the shaft), and the positive offset.

5. Connection and positioning by shaft merging

Finally, the last possibility is to merge two shafts. This means that one shaft is going to be extended with the geometry and the elements of the other one, which is going to be removed afterwards.

For that you can just select a shaft with the right click and then call the function "Merge". This is the shaft that will be extended. Then a dialog comes up to select the shaft that will be added to the previous one (and then removed), on which side, and with what direction (in the local coordinates system of the shaft).
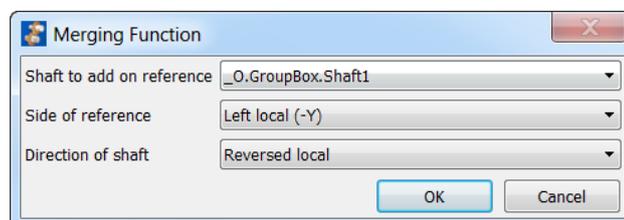
Important note: when merging to coaxial shafts, all the other shafts coaxial to the removed one are moved in the calculation file of the extended one, as well as the connection elements. If you want to merge a coaxial shaft and a parallel shaft, you have to extend the coaxial shaft, not the parallel one (to avoid moving all coaxial shafts in this case). Everything else is not moved (constraints, gear calculations...). It can then happen that a group box remains empty after the merge. The user can then decide what to do with it.

It can also happen that it is not possible to reposition the shafts automatically because of the referenced shaft used for positioning the other shafts in the model. In this case either a dialog function from each shaft to reposition comes up, either the user has to change the positioning strategy of its shafts (referenced shaft are going in two directions instead of one).

Here for example "SunShaft" merged with "Shaft1" according to the settings from the previous dialog:
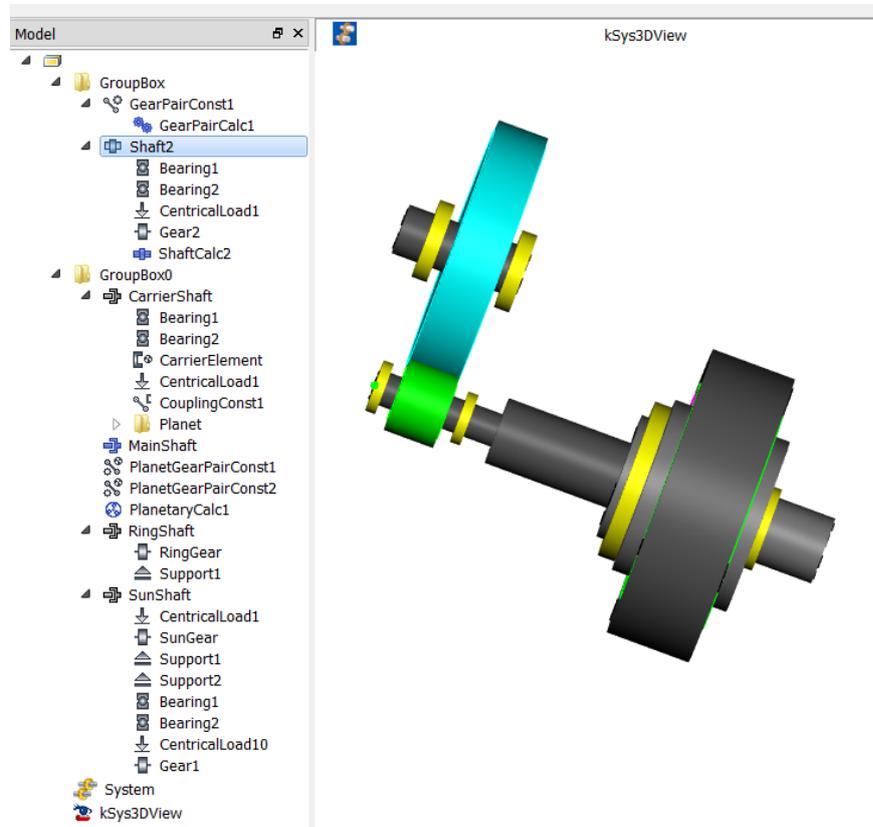
The user can also change the direction of "Shaft2" in the dialog of the positioned group or shaft (here "GroupBox0" or "Shaft2").

And the shaft calculation looks like this before and after the merge:
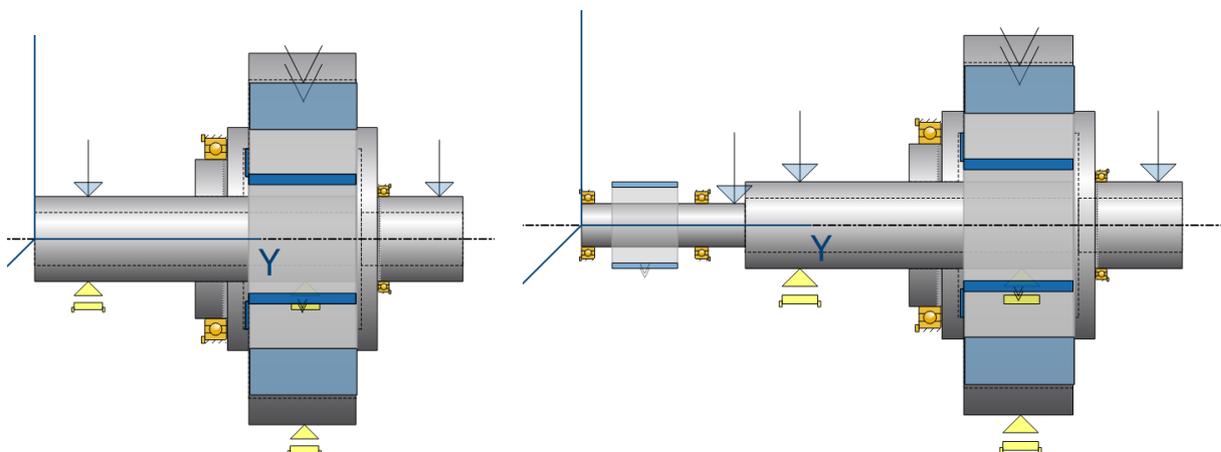
The user might then have to rearrange some elements to be able to run the calculation.