WORKSHOP 1.3

Component design

Contents

Introduction

In this workshop, we will learn how to interface a component or subsystem. By adding connector and parameter interfaces to a component we will be able to use it in other models. We will create a test rig, to verify the components behavior. And optionally add an icon and some documentation.

Loading TrainingPack

In this workshop, we will utilize a training package prepared in advance. It is called **TrainingPack.zip** and should have been provided to you by your course leader. Follow the instructions below to upload it.

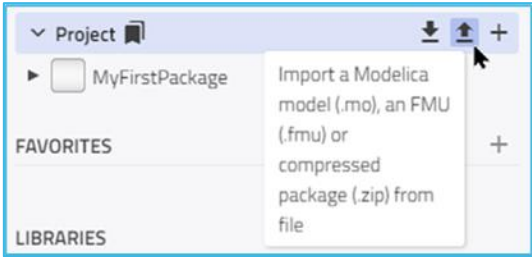- Start the import by clicking the upload button in the *Library Browser*:

Figure 1 Import modelica package

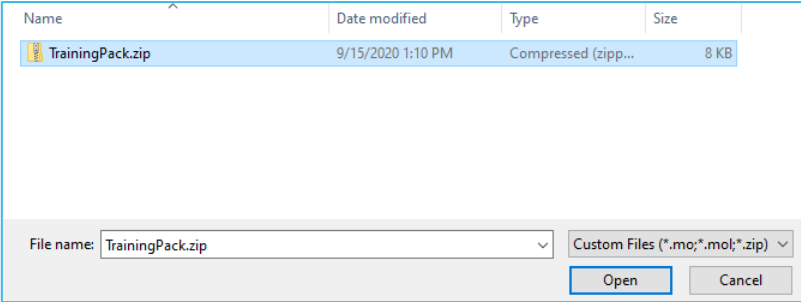- Select the supplied **TrainingPack.zip** file.

Figure 2 Load TrainingPack
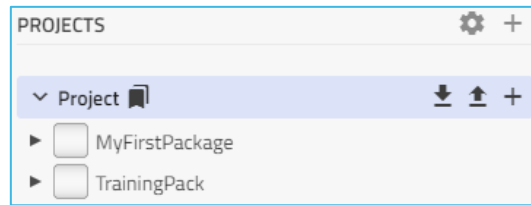
- The package should now be ready to use.

Figure 3 TrainingPack successfully loaded

Designing a component interface

To properly reuse subsystem components, you must consider the following:
- Parameter interface, choosing what data to expose and how to present it.
- Connector Interface, how the component can interact with other components.

In the following part of the workshop, you will reuse the simple double inertia model created in workshop 1 and create an elastic shaft. You will propagate the necessary parameters and create a connector interface.

1. Create a new sub-package called **Components** in **TrainingPack.W3.ComponentDesign**
2. Create a new sub-package called **Experiments** in **TrainingPack.W3.ComponentDesign**
3. Create a new model called **ElasticShaft** in **TrainingPack.W3.ComponentDesign.Components**
4. Go back to **MyFirstModel** created in Workshop 1, select all components, and copy them into the **ElasticShaft** class.
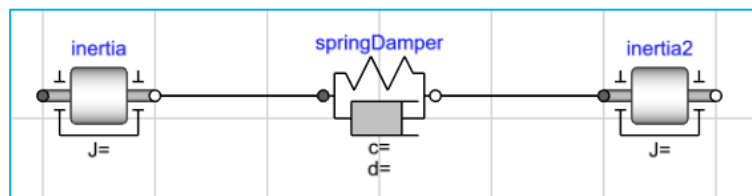

Figure 4 Connected components

5. Now we will create the connector interface. Drag *Modelica.Mechanics.Rotational.Interfaces.Flange_a* and (twice!) into the model. Name them as *flange_a* and *flange_b*, then connect them to the inertias, as shown in **Figure** 5 below.
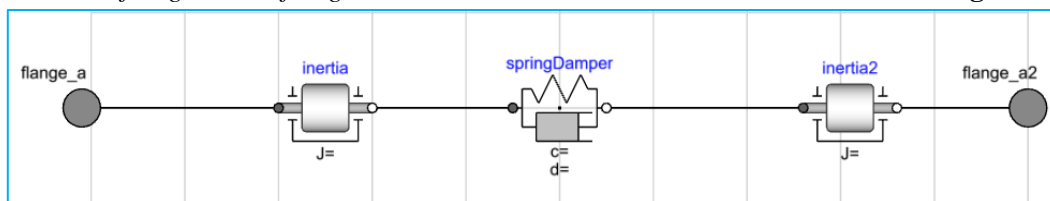

Figure 5 Connected components with flange

6. For clarity, rename **inertia** to **inertia1**.

Note: If not done already, reset the start attribute of *inertia1.phi* to 0 radians, otherwise the shaft will initialize in a skewed state.

7. We need to decide what data needs to be exposed when using the component. In this case, we will expose *J1*, *J2*, *c* and *d*. This is done in two steps:

a. Create a new parameter in the component.
b. Propagate the new parameter down into the instance as a modifier.

8. Open the **Properties** tab of the *Details Panel*. On the top right, there is a ⊞ button.
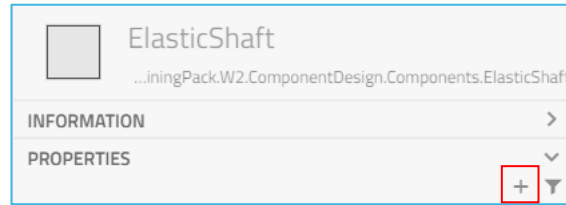

Figure 6 Finding Add Variable icon

9. Click the button to open the create new parameter dialog. Create the four parameters needed:

Figure 7 Creating new parameter for J1

Types for the coefficients are:

Coefficient	Type	Expression	Description	Group
J1	MomentOfInertia	0.6 [kg m²]	Moment of inertia	Inertia
J2	MomentOfInertia	0.4 [kg m²]	Moment of inertia	Inertia
c	RotationalSpringConstant	1000 [N m/rad]	Rotational spring constant	Spring Damper
d	RotationalDampingConstant	10 [N m s/rad]	Rotational damping constant	Spring Damper

10. Now all we need to do is to place the modifiers on the components. Click on *intertia1*, and go to the parameter *J*.
11. The idea is to modify *J* with the new parameter *J1* that we created. If you click the parameter value box, you can autocomplete the content of the instance tree; start typing *J* and choose *J1* from the list.
12. Similarly, select *J2* for *inertia2.J*, *c* for *springDamper.c* and *d* for *springDamper.d*.
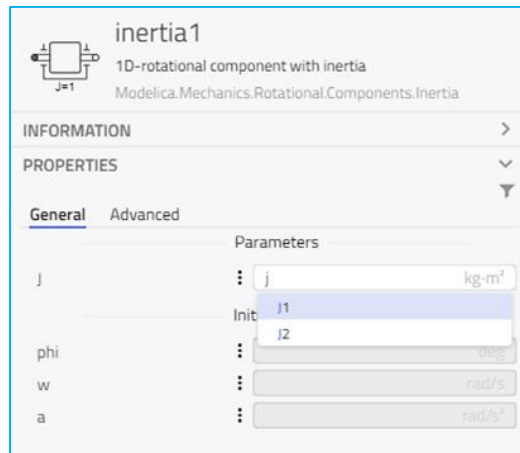
Figure 8 Modifying J with the new parameter

Test Rig

Next step is to test the behavior of the component. You will create a test rig, mount the component, and simulate it.

13. Create a test experiment for the elastic shaft component.
    a. Create a new model **TestShaft** in **TrainingPack.W3.ComponentDesign.Experiments**.
    b. Drag in the **ElasticShaft** component.
    c. Drag the following:
        1. ***Modelica.Mechanics.Rotational.Components.Fixed***
        2. ***Modelica.Mechanics.Rotational.Sources.Torque***
        3. ***Modelica.Blocks.Sources.Sine***
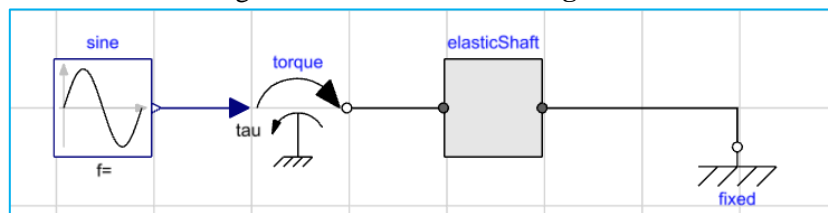   d. Connect them according to the schema below in **Figure** 9:

Figure 9 ElasticShaft with its components

    e. Parametrize the model according to the data given below:

Parameter:	Value:
sine.amplitude	100
sine.freqHz	2 [Hz]
elasticShaft.J1	0.6 [kg m²]
elasticShaft.J2	0.4 [kg m²]
elasticShaft.c	100000 [N m/rad]
elasticShaft.d	0 [N m s/rad]

    f. Simulate the model for 1 second. Plot graphs for:
       • *springDamper.tau*[N_m]
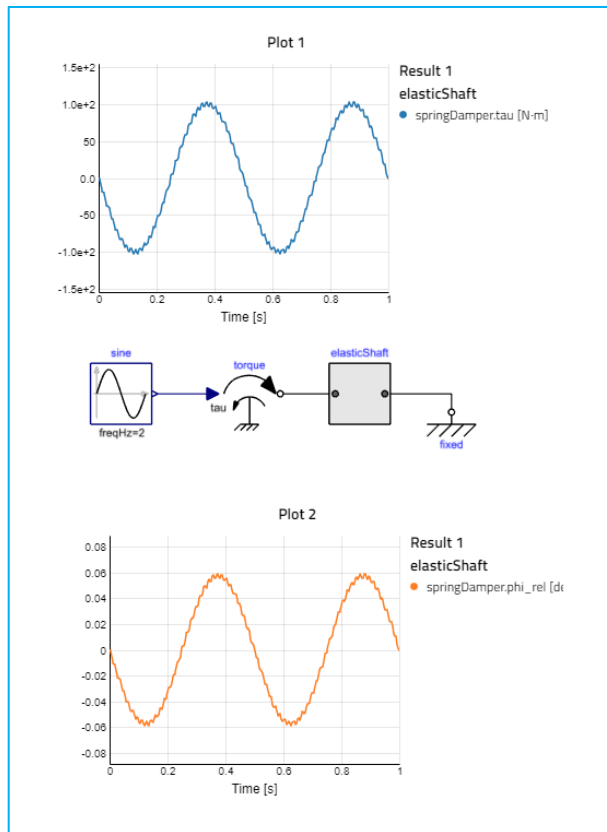   • *springDamper.phi_rel*[deg]

Copyright © 2022 Modelon

Figure 10 Results of elastic shaft experiment

Create an icon and add documentation

Use the icon editor to create a nice icon for the elasticShaft.

Use the documentation editor to add information about the component.

This can be done by right-clicking on the **ElasticShaft** class and selecting *Edit Icon* and *Show documentation*, respectively. See figure below.
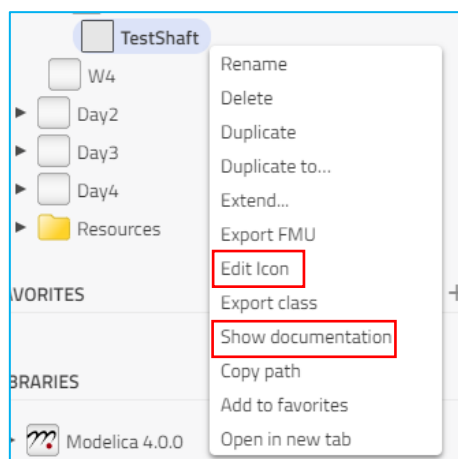
This concludes workshop 1.3. Well done!