

WORKSHOP 1.1

Opening Modelon Impact, simulating a model and post-processing

Contents

Introduction	1
Creating a workspace	1
Simulating a model	1
Visualizing the results	4
Create a plot from the results	4
Create a simple model	5

Introduction

In this workshop, you will become familiar with the Modelon Impact workflow. You will simulate a model from the Hydraulics library and learn how to plot variables. As a second task, a simple model will be built using drag-and-drop components from the Modelica Standard Library.

Creating a workspace

1. Open *<Modelon Impact URL>* in your browser (currently Chromium based browsers such as Chrome, Edge and Opera are supported).
2. On the landing page, click **New** and then **Create New Workspace**; name the workspace and click **Create**. This workspace will be the one you use throughout the training.

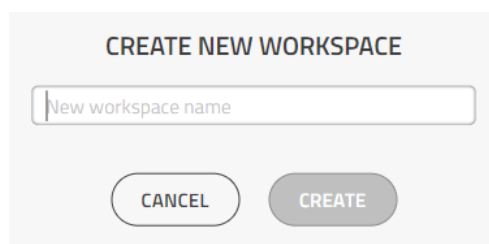




Figure 1 Creating a new workspace

Simulating a model

1. Once you are in your new workspace, go to the top right corner and click on the “Apps” icon , and launch the *Workspace Configuration* app.
2. The *Workspace Configuration* app will open in a new chrome tab. Click on the workspace tab.
3. In workspace tab click on “edit” icon  to show the libraries available in Modelon Impact as shown in [Figure 2 Workspace Configuration](#)
4. From here drag and drop Modelon Base Library (MBL) and Hydraulics Library (HL) as dependencies shown in [Figure 3](#).

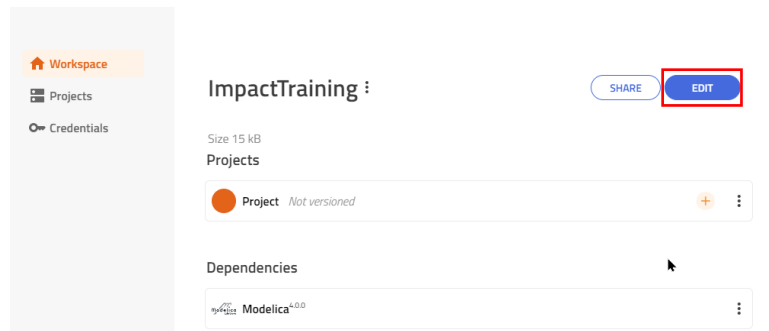


Figure 2 Workspace Configuration

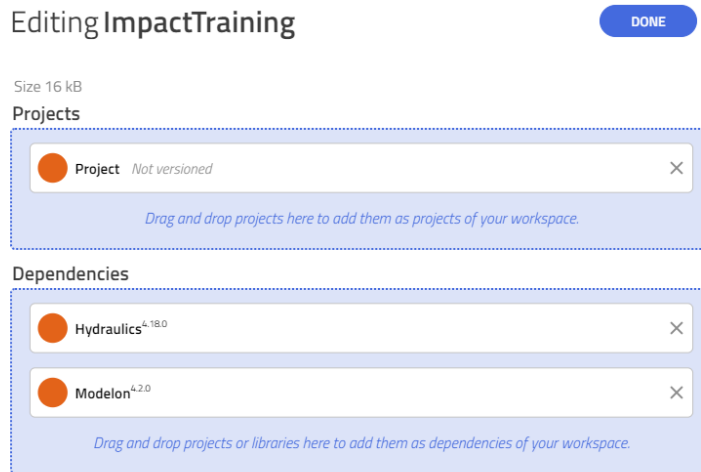




Figure 3 Adding libraries as dependencies

5. After moving libraries to dependencies click on “done” icon . Now switch to Impact tab an automatic reload pop-up will appear, click reload so that the libraries will appear in *Library Browser*.
6. Open the *Library Browser* by clicking on the  button on the left-side of the model canvas.
7. In the *Library Browser*, browse to the model: **Hydraulics.Examples.HydraulicLift**, and double-click to open it.

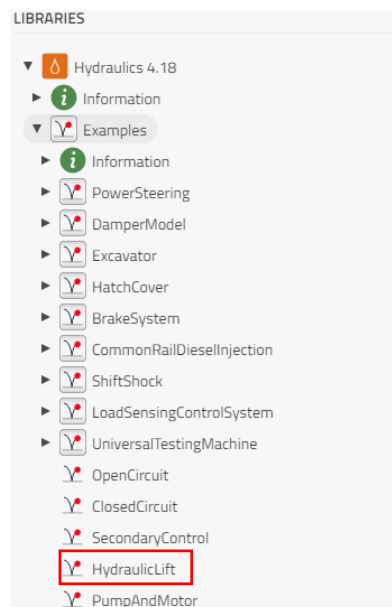



Figure 4 Library Browser

- Open the *Details Panel* (can be expanded from the right-side of the model canvas by clicking on the  button) and expand the **Information** tab to read the documentation of the model.

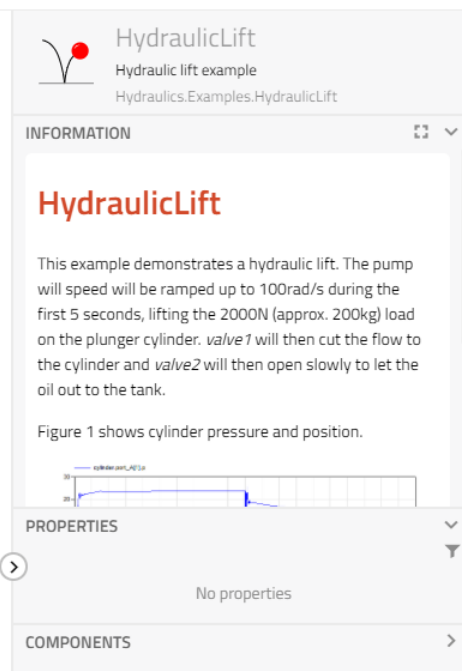



Figure 5 Information tab of Details Panel

- Switch to **Experimentation** mode by clicking the button  in the *App Bar* on the top of your window. You can now safely experiment with your model, without changing the original model. In the *Details Panel*, expand the **Experiment** tab, and under **Analysis**, go to **Advanced** and see the numerical settings (solver and tolerance). See [Figure 6](#).

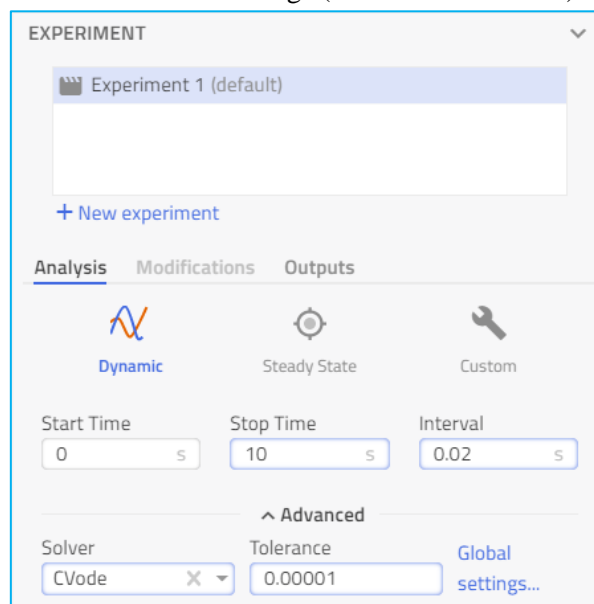



Figure 6 Simulation setup

- Simulate the model by clicking the Simulate icon . Once the simulation finishes, the application will automatically switch to the *Result* mode in the *App Bar*.

Visualizing the results

Some models contain 2D graphical information that can be viewed as a result of the simulation.

11. To visualize the 2D animation of the Hydraulic lift, slide the time slider, located at the bottom of the screen, and check the model behavior from 0 sec to 10 sec. The 2000N (approx. 200kg) load on the plunger cylinder is initially lifted as the pump speed is ramped up to 100 rad/s during the first 5 seconds. valve1 will then cut the flow to the cylinder and valve2 will then open slowly to let the oil out to the tank as shown in [Figure 7](#).

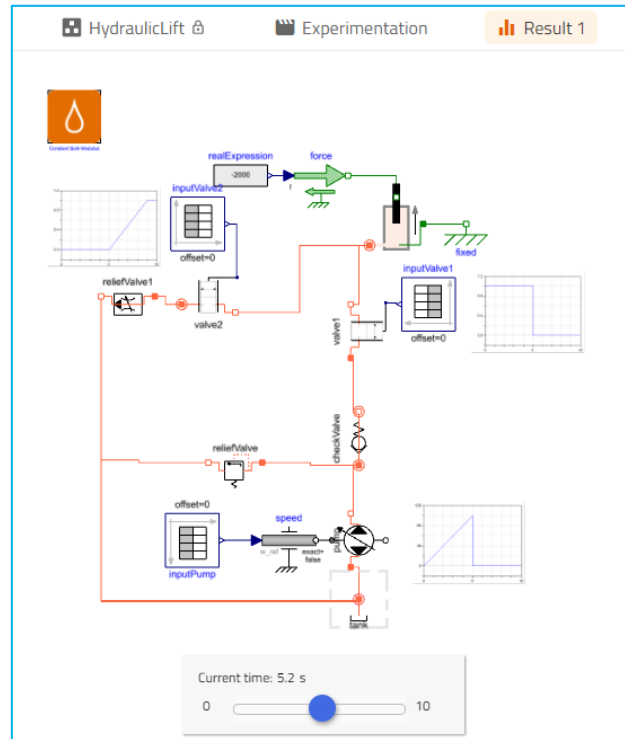


Figure 7 Animate model

Create a plot from the results

It is very easy to create plots from the results found in *Details Panel* under the *Calculated Values* tab.

12. To create the plots, you will drag and drop the cylinder pressure (*cylinder.port_A[1].p*) and cylinder position (*cylinder.flange_b.s*) variables from the *Calculated values* tab in the *Details Panel*.

To quickly find the variables: (see [Figure 8](#) below).

1. Select the **cylinder** component on the Canvas.
2. Type the variable name in *Filter...* text box. Try to filter on pressure variable “*p*” in the cylinder to find *cylinder.port_A[1].p*
3. Drag and drop the *p* variable to the *Model canvas* to plot it. Similarly, find and plot the cylinder flange position (*cylinder.flange_b.s*) as shown in [Figure 9](#) .

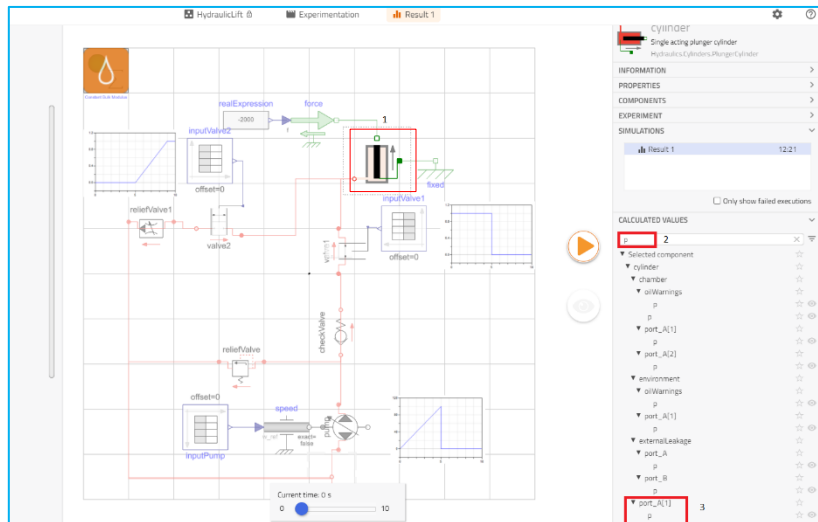


Figure 8 Plotting result variables

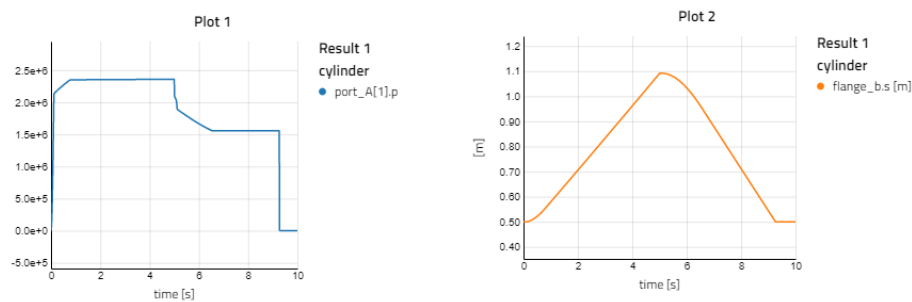


Figure 9 Plotting the cylinder pressure and flange position


You have accomplished the following:

1. Opened a model.
2. Simulated it
3. Reviewed the results, through animation and plotting.

Well done!

Create a simple model

In the following section, you will create a simple model of a mechanical oscillatory system using components in the Modelica Standard Library. First you create the model class, then drag the components onto the canvas and connect them.

1. Open the *Library Browser* (left sidebar).
2. Use default project or create a new project using this icon  PROJECTS
3. Once the project is created you can rename if necessary, by right clicking over the project.
4. Click on the + button in the next to project (default/created) to bring up the **Create New Class** menu and create a new top level *Modelica Library/Package* and name it as **MyFirstPackage**.
5. Give the new model the name **MyFirstModel** and press **Create**. By convention class names should start with an upper-case letter as shown in **Figure 10**. Note that the default package is selected to be **Workspace**.

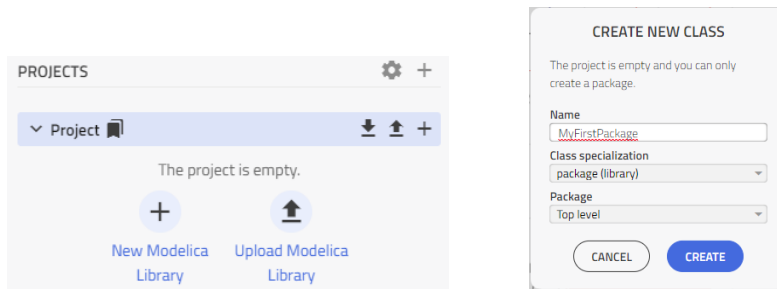


Figure 10 Creating a model

6. The empty model canvas for the model **MyFirstModel** should now be open. Let us now drag the needed components on to the canvas.
7. Navigate to *Modelica.Mechanics.Rotational.Components*.
8. Drag-and-drop two **Inertia** models and a **SpringDamper** model. Connect the components as shown in [Figure 11](#).

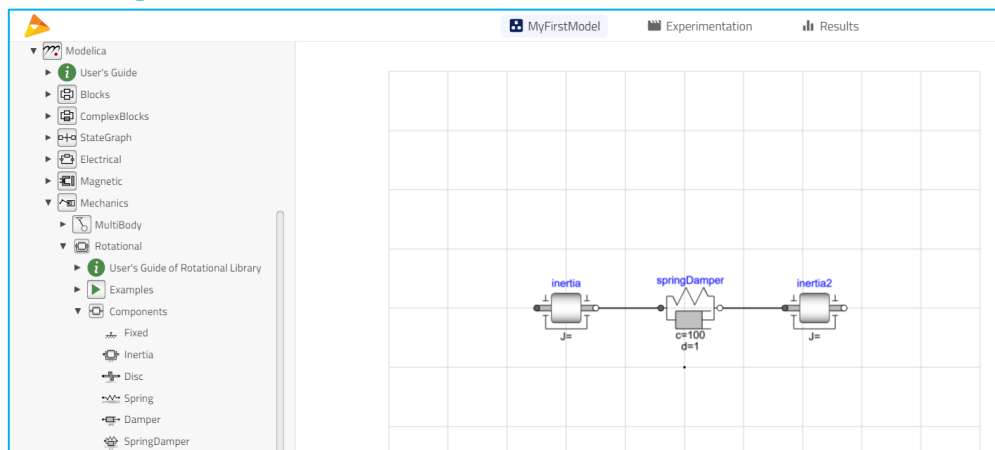


Figure 11 Connected components

Note: Changes made to models in Impact are saved automatically; there is no **Save** button that you need to click to save progress.

9. Open the *Details Panel*. You can alternatively double-click on a component to open the *Details Panel* with the **Properties** tab expanded for that component.
 - a. Select the **springDamper** component to set its parameters (See [Figure 12](#)):
 - i. $c=100$ Nm/rad
 - ii. $d=1$ Nms/rad

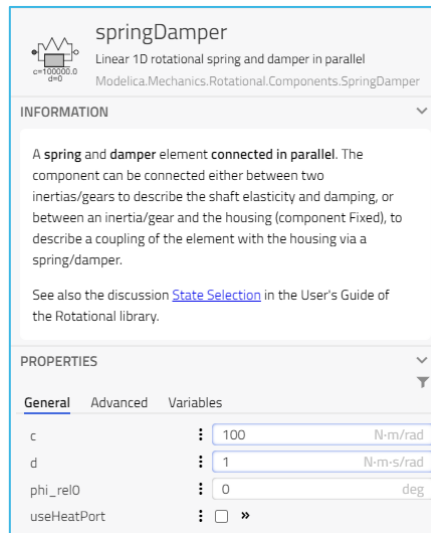


Figure 12 Parameter settings for *springDamper*

- b. Select the **inertia** component to set its parameters.
 - i. $J=0.1$
 Set the initial value for *phi* variable as shown in Figure 13. Make sure to select the checkbox adjacent to the *start* value to set fixed attribute to true.

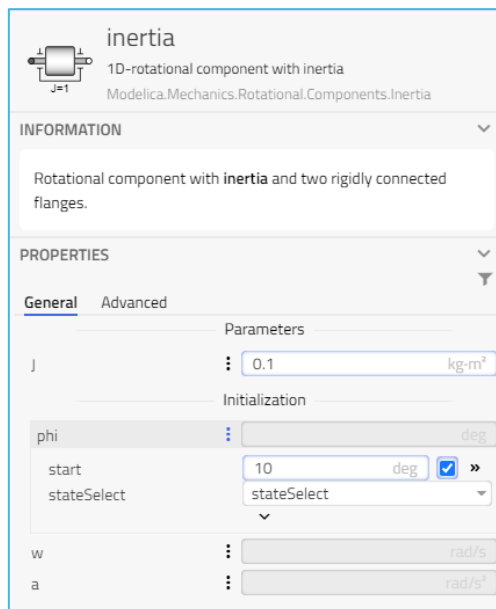


Figure 13 Parameter settings for *inertia*

- c. Select the **inertia2** component to set its parameters. The parameter sidebar for inertia should look like Figure 14.

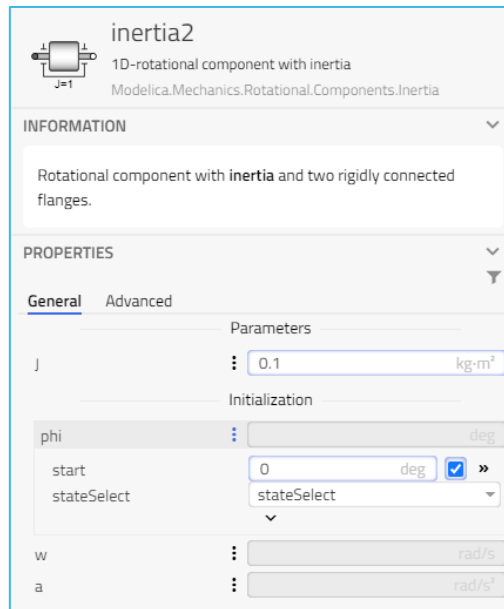


Figure 14 Parameter settings for *inertia2*

Note: It is possible to rename the components in the *Details Panel*, see Figure 15.

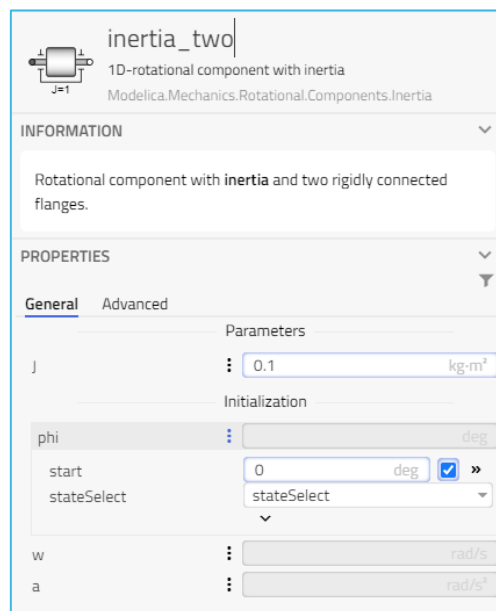



Figure 15 Renaming a component

10. Switch to **Experiment** mode  to go to the simulation environment.
11. We are interested in analyzing the transient behavior (Dynamic Analysis) of the system. Under **Analysis** options in the **Experiment** tab, click on the **Dynamic** icon to activate Dynamic Analysis.
12. Set the simulation stop time to 2 seconds as shown in Figure 16. This will update the default experiment *Experiment 1*.

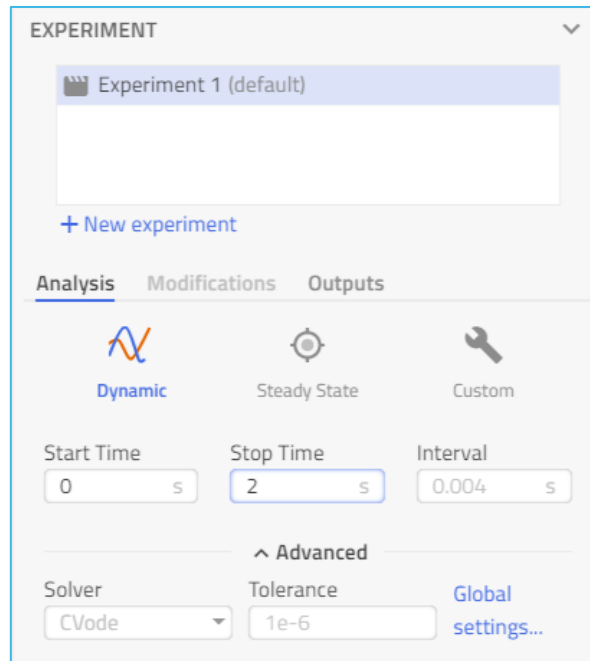



Figure 16 Simulation settings browser

13. Click the Run  button to start the simulation.
14. Select the **inertia** component and drag and drop the ϕ variable on the *Model canvas* to plot the angle ϕ for the **inertia**. Add the angle ϕ for the **inertia2** to the same plot by selecting **inertia2** and dragging and dropping the ϕ result variable to the plot window.
15. Verify whether the model's initial values are as seen in Figure 17. Analyze the results.

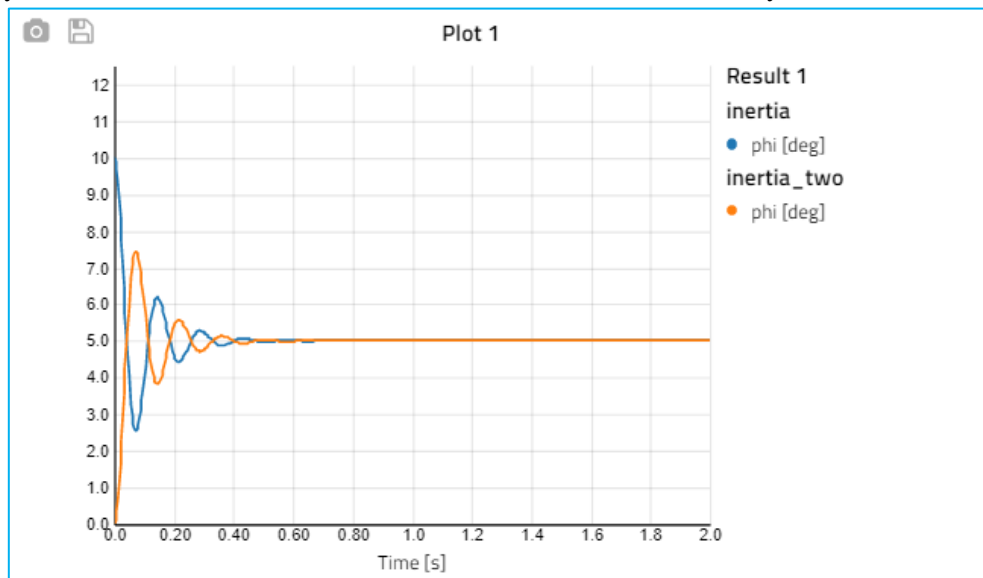


Figure 17 Verify initial values

16. To study the impact of a certain parameter on the system behavior, simply change a parameter in the **Properties** tab of the *Details panel* and hit the simulation button again; the plot will be updated with the new results. In Figure 18, the damping coefficient d for **springDamper** has been changed to 0.5 instead.

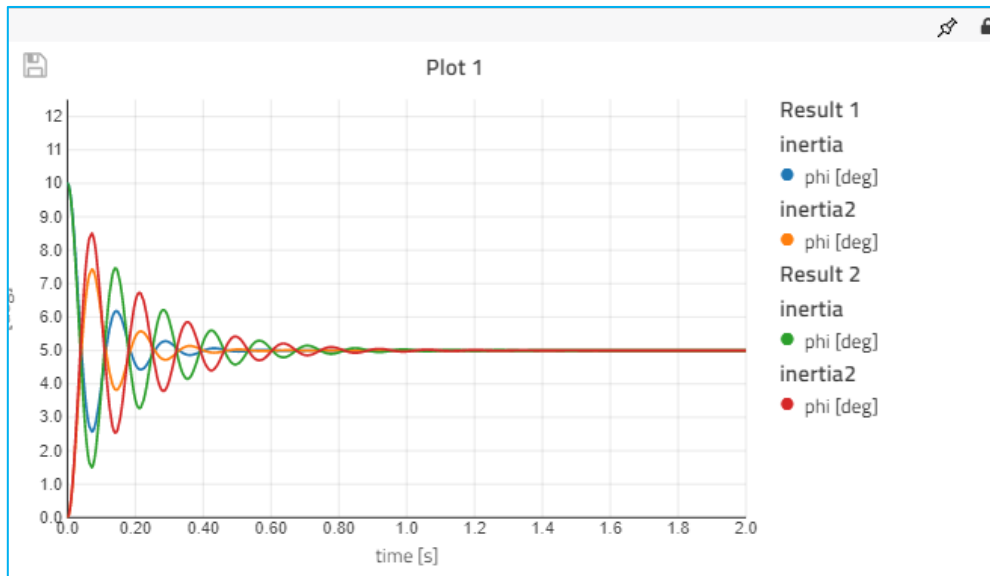


Figure 18 Re-simulate with parameter change

Note: Individual variables from individual simulations can be deleted from a plot by clicking on the variable in the plot legend. For example, to remove *inertia_two.phi* from *Result 2* from the plot shown in Figure 19, hover the mouse cursor over the ● icon next to the variable in the plot legend. This will change the icon to ✂; click on it and select **Delete** to remove the variable from the plot. It is also possible to customize trajectory colors this way.

17. It is also possible to create parametric curves or x-y plots. Let us look at the phase-space for the **inertia** component.

- a. Create a new plot by dragging and dropping the variable *inertia.w* on to the canvas.
- b. Change the independent variable of the plot to *inertia.phi* by dragging and dropping it from the **Calculated Values** tab to the x-axis of the plot.

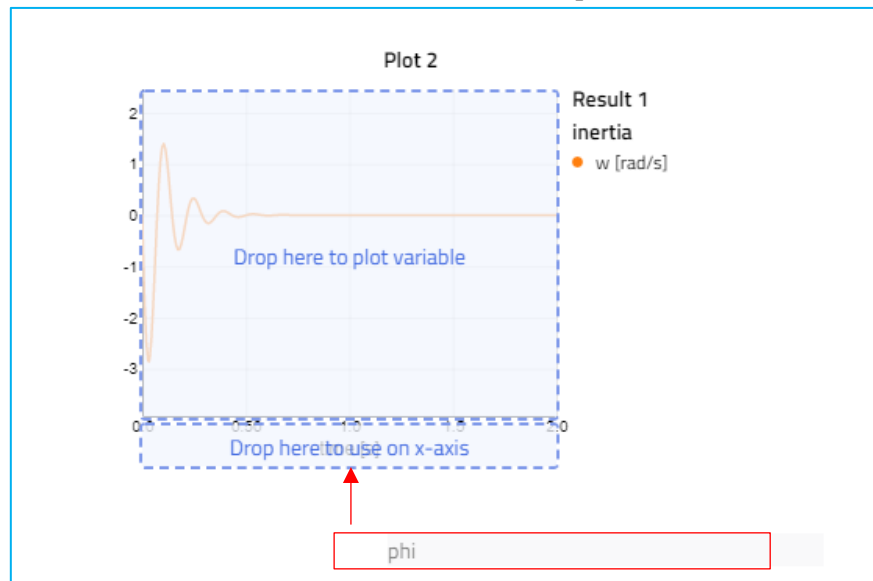


Figure 19 Changing the independent variable of a plot

- c. You should see a phase-space diagram similar to Figure 20 .

Note: Results or parametric curves from different simulations can be compared by selecting the relevant results from the **Simulation** tab.

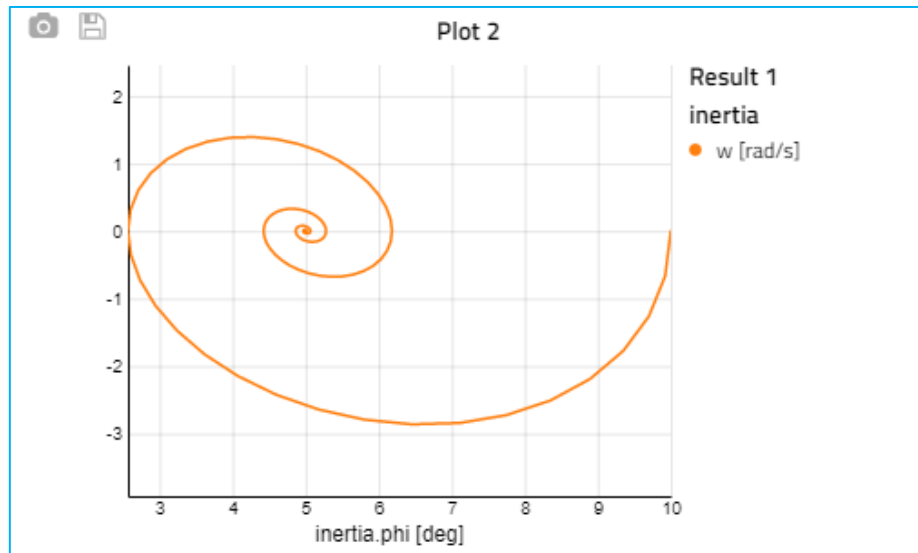


Figure 20 Phase-space diagram

d. Questions:

- i. In the last exercise, you can see that the curve spirals into a single point, what does this tell you about the system?
- ii. Is it possible to modify any parameters in the model such that the curve would become circular?

This concludes workshop 1.1. Well done!
