Created in COMSOL Multiphysics 5.6

Control of an Inverted Pendulum

Introduction

An inverted pendulum is a pivot-mounted structure for which the position of the center gravity is placed above the pivot point. The structure in its original position is unstable, and without exterior force it will always tend to fall down. This is a classical problem within control systems engineering — keeping the structure in equilibrium by applying an external load on the cart. The control load depends on the pendulum tilting angle, the cart position, and the cart velocity. The equilibrium is assumed when all control parameters are zero.

Note: This models requires licenses for the Structural Mechanics Module and LiveLink™ for Simulink®.

Model Definition

The mechanical model of the pendulum is implemented in COMSOL Multiphysics. The pendulum system consists of a 50 cm x 0.5 cm x 0.5 cm rod and a 2 kg cart. The rod is pivot mounted on the cart, and the last one is free to move horizontally. In COMSOL Multiphysics, the rod is modeled using a 2D geometry and as a rigid-domain feature, while the cart is only represented with its mass.

The control system is implemented in Simulink using three PID controllers that provide the force to apply to the cart in order to hold the pendulum in balance. The rod is considered in equilibrium when the tilting angle, the cart position, and the cart velocity are zero. These are the values returned by the COMSOL simulation used by the PID controllers.

The cosimulation with COMSOL Multiphysics and Simulink is set up by exporting a COMSOL Cosimulation file from the COMSOL model and then adding this file to the COMSOL Cosimulation block in the Simulink simulation diagram. The input of the block consists of perturbation and control forces provided by Simulink. The block has three outputs: the tilting rotation of the rod, the displacement, and the velocity of the cart.

The rod is dropped with an initial tilting angle of 1 degree, and a perturbation force is applied as a 0.1 s pulse with a 1 N amplitude.

Figure 1 shows the full control simulation diagram in Simulink.

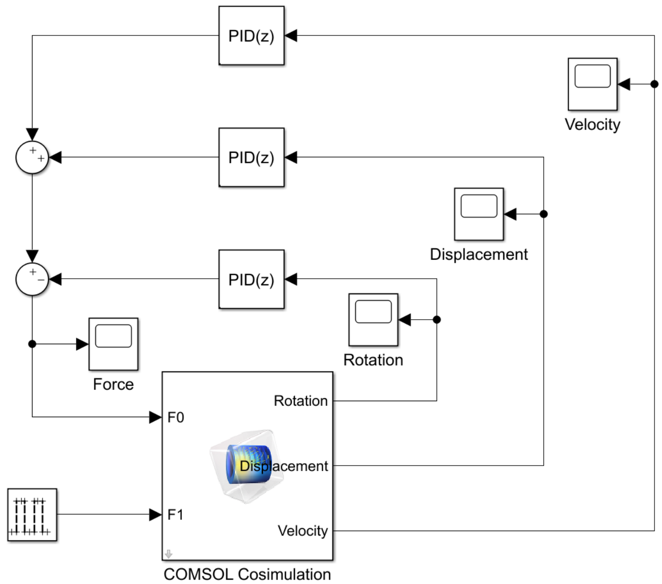

Figure 1: Simulink diagram of the inverted pendulum cosimulation.

The simulation diagram is made so that you can modify the perturbation force or set a different initial tilting angle.

Results and Discussion

Figure 2 shows the tilting of the pendulum. After the perturbation with a maximum angle of around 22 degrees, the pendulum is stabilized around a vertical position.

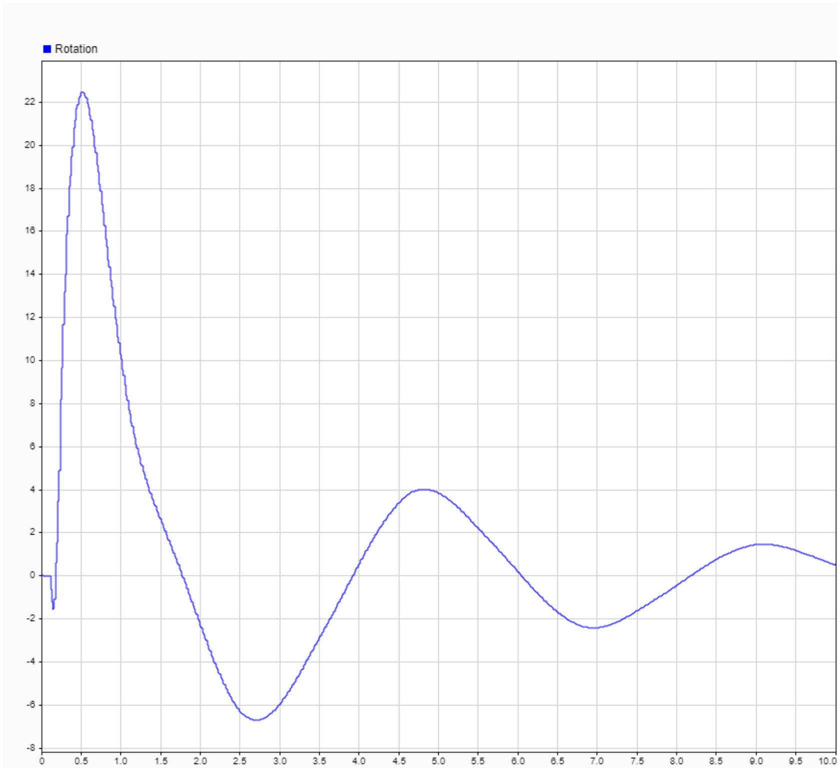

Figure 2: Tilting angle of the pendulum versus time.

Figure 3 shows the chart displacement, which never exceeds 1 m and gets stabilized at the original position.

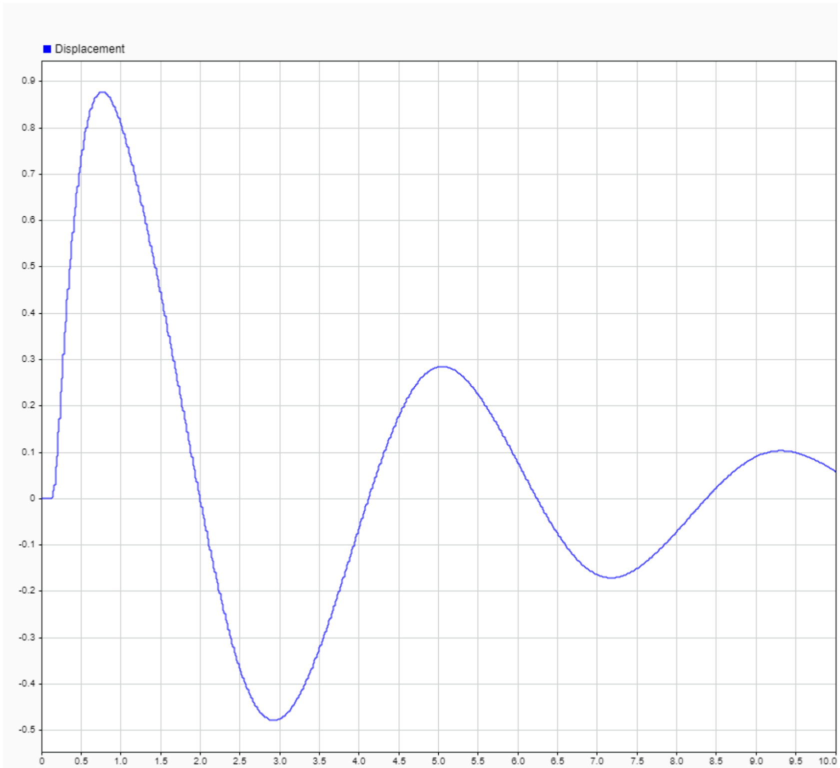

Figure 3: Chart displacement versus time.

Figure 4 shows the translational velocity of the pendulum that quickly decreases and approaches 0.

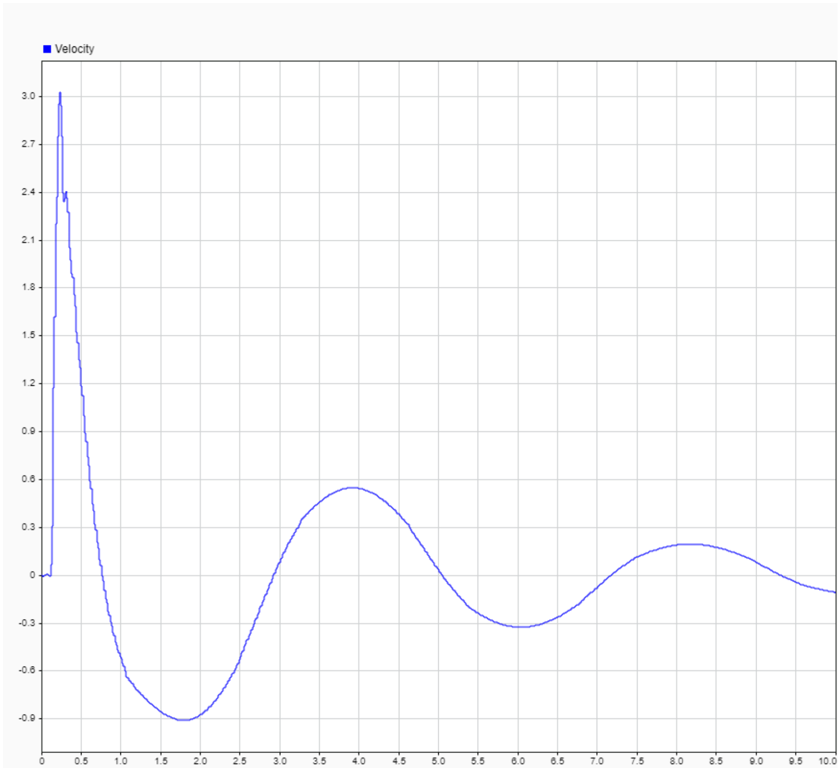

Figure 4: Chart velocity versus time.

Figure 5 shows the input force. Just after the perturbation, the control force is quickly adjusted. After 1 s, the force varies smoothly until the pendulum is fully stabilized.

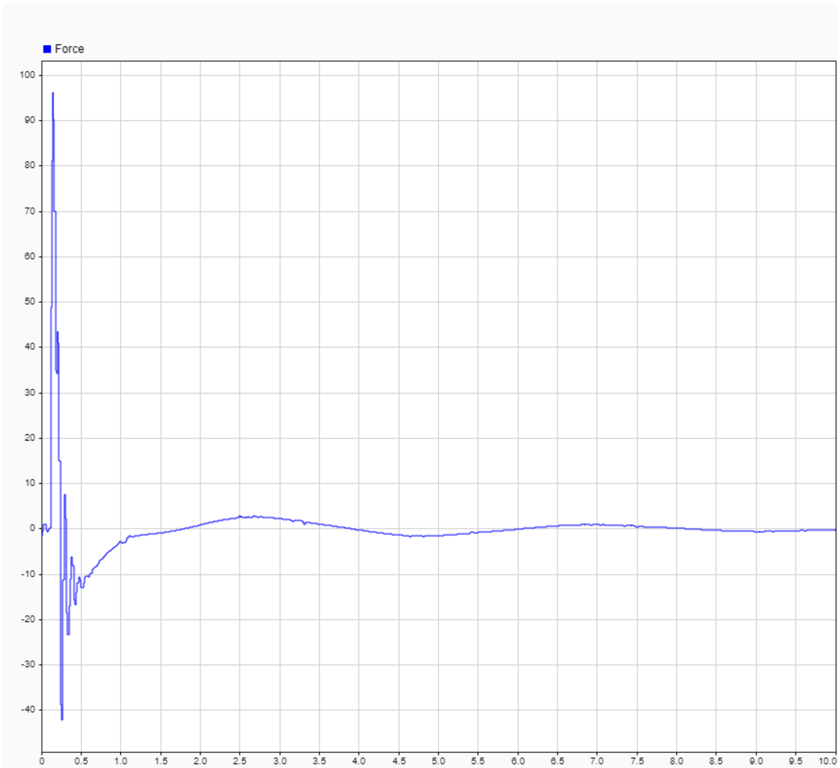

Figure 5: Control force applied on the chart along the time.

Setting Up the Cosimulation

Follow the workflow below to set up the cosimulation with COMSOL Multiphysics and Simulink:

1. Set up the COMSOL model and make sure that the study runs. Only studies with a single Stationary or Time Dependent study step are supported for cosimulation.
2. Save the COMSOL model. This step is important because the name of the model is needed to load the cosimulation file in Simulink.
3. Add the Cosimulation for Simulink feature node to the COMSOL model. Use this to define the inputs, outputs, and study for the cosimulation.

4 From the Cosimulation for Simulink feature node, export the file for cosimulation. Any location will work, but it is good practice to export this file to the location where the MPH-file has been saved.

5 Create or load the simulation diagram in Simulink, and add the COMSOL Cosimulation block.

6 Double-click the COMSOL Cosimulation block, and enter the name of the cosimulation file exported from COMSOL Multiphysics.

Application Library path: `LiveLink_for_Simulink/Tutorials/inverted_pendulum`

Modeling Instructions — COMSOL Desktop

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
w0	5[mm]	0.005 m	Pendulum width
l0	50[cm]	0.5 m	Pendulum length
alpha0	1[deg]	0.017453 rad	Initial pendulum angle
M	2[kg]	2 kg	Mass of roller
F0	0[N]	0 N	Correction force
F1	0[N]	0 N	Perturbation force

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `w0`.

4 In the **Height** text field, type `l0`.

5 Locate the **Position** section. In the **x** text field, type `-w0/2`.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `0 0`.

4 In the **y** text field, type `0 l0`.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `alpha0`.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Structural steel**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 In the d text field, type `w0`.

Rigid Domain 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Domain**.

2 Select Domain 1 only.

3 In the **Settings** window for **Rigid Domain**, locate the **Center of Rotation** section.

4 From the list, choose **Centroid of selected entities**.

5 From the **Entity level** list, choose **Point**.

Center of Rotation: Point 1

1 In the **Model Builder** window, click **Center of Rotation: Point 1**.

2 Select Point 5 only.

Rigid Domain 1

In the **Model Builder** window, click **Rigid Domain 1**.

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Displacement/Rotation**.

2 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement at Center of Rotation** section.

3 Select the **Prescribed in y direction** check box.

Rigid Domain 1

In the **Model Builder** window, click **Rigid Domain 1**.

Applied Force 1

1 In the **Physics** toolbar, click **Attributes** and choose **Applied Force**.

2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.

3 Specify the $\mathbf{F}$ vector as

F0	x
0	y

Rigid Domain 1

In the **Model Builder** window, click **Rigid Domain 1**.

Mass and Moment of Inertia 1

1 In the **Physics** toolbar, click **Attributes** and choose **Mass and Moment of Inertia**.

2 In the **Settings** window for **Mass and Moment of Inertia**, locate the **Mass and Moment of Inertia** section.

3 In the m text field, type `M`.

Gravity 1

1 In the **Physics** toolbar, click **Domains** and choose **Gravity**.

2 Select Domain 1 only.

Point Load 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

F1	x
0	y

DEFINITIONS

Variables 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Model Builder** window, right-click **Mapped 1** and choose **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,1e-2,2)`.

4 Select the **Include geometric nonlinearity** check box.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Intermediate**.

5 From the **Maximum step constraint** list, choose **Constant**.

6 In the **Maximum step** text field, type `2e-3`.

7 Click **Compute**.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Stress (solid)

In the **Model Builder** window, expand the **Surface 1** node, then click **Results>Stress (solid)**.

Point Trajectories 1

1 In the **Stress (solid)** toolbar, click **More Plots** and choose **Point Trajectories**.

2 In the **Settings** window for **Point Trajectories**, locate the **Trajectory Data** section.

3 From the **Plot data** list, choose **Points**.

4 In the **X-expression** text field, type `x`.

5 In the **Y-expression** text field, type `y`.

6 Select Point 2 only.

7 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Point**.

8 In the **Point radius expression** text field, type `0.005`.

Animation 1

1 In the **Stress (solid)** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 In the **Number of frames** text field, type `250`.

SAVE THE COMSOL MODEL

1 From the **File** menu, choose **Save As**.

2 Browse to a suitable folder, enter the filename `inverted_pendulum.mph`, and then click **Save**.

Exporting the Cosimulation File

In the following configure the cosimulation, and export the file for cosimulation that will be loaded into Simulink.

GLOBAL DEFINITIONS

Cosimulation for Simulink 1

1 In the **Study** toolbar, click **Cosimulation for Simulink**.

2 In the **Settings** window for **Cosimulation for Simulink**, locate the **Filename** section.

3 In the **Filename** text field, type `inverted_pendulum`.

4 Locate the **Inputs** section. Click **Add**.

5 In the table, enter the following settings:

Parameter name	Initial value	Unit
F0 (Correction force)	0	N

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Initial value	Unit
F1 (Perturbation force)	0	N

8 Click to expand the **Block Parameters** section. Click **Add**.

9 In the table, enter the following settings:

Parameter name	Initial value	Unit
alpha0 (Initial pendulum angle)	`1[deg]`	rad

10 Locate the **Outputs** section. In the table, enter the following settings:

Expression	Unit	Name
solid.rd1.phi*180/pi+alpha0		Rotation
solid.rd1.u		Displacement
solid.rd1.u_tx		Velocity

11 Click **Export**.

Modeling Instructions — Simulink

Once you have created the COMSOL model and saved the cosimulation file you can start Simulink to continue with the setup there.

1. Start COMSOL with Simulink.
2. In MATLAB enter the command `mphapplicationlibraries` to start the GUI for viewing models from the LiveLink for Simulink application library.
3. Browse to the folder LiveLink_for_Simulink/Tutorials, and select inverted_pendulum.slx.
4. Click Open to get the simulation diagram as in Figure 1.

   The included COMSOL Cosimulation block is already configured with a cosimulation file based on the model from the COMSOL Application Library and ready to run. If you want to run the simulation directly, go to Step 7 below. Else, if you want to use the model file and cosimulation file you have created by following the steps in the section Modeling Instructions — COMSOL Desktop, you can continue with Step 5 below.
5. Double-click the COMSOL Cosimulation block.
6. In the COMSOL Cosimulation window settings, in the Filename edit field enter the name of the file for cosimulation for Simulink as created in the section Exporting File for Cosimulation for Simulink.

Note: In case the folder path of the file for cosimulation for Simulink is not set in MATLAB enter the full filename.

   For this simulation the stop time is set to 10 s and the communication step size is set to 10^{-2} s.

7. To run the simulation, click Run.

If you want to run another simulation with a different initial tilting angle, double-click the COMSOL Cosimulation block, then click Block parameters button. In the alpha0 edit field, enter the desired initial tilting angle in radian and click OK.

POSTROCESSING THE SOLUTION IN THE COMSOL DESKTOP

Once you have run the simulation in Simulink you can postprocess the computed solution stored in the COMSOL model, for instance the steps below show how to generate an animation of the pendulum.

1 Once you have run the simulation in Simulink, go to the MATLAB prompt and enter:

   ```
   mphlaunch
   ```

   This will start a COMSOL Desktop with the model used to run the cosimulation.

2 In the **Model Builder**, expand the **Results** node, and then expand the Export node.

3 Select the **Animation 1** node, and click **Show Frame**.

Note: Close the COMSOL Desktop before running a new simulation in Simulink.
