Model created in COMSOL Multiphysics 6.4

Magnetic Brake — LiveLink™ *for* Simulink® Simulation

Introduction

A magnet brake in its simplest form consists of a disc of conductive material and a permanent magnet, as in Figure 1 below, represented in orange and light gray, respectively. The magnet generates a constant magnetic field, in which the disc is rotating. When a conductor moves in a magnetic field it induces currents, and the Lorentz forces from the currents slow the disc. This model is variant of the Magnetic Brake model from the AC/DC Module Application Library, where the angular velocity is computed in Simulink based on the induced torque and the disk moment of inertia.

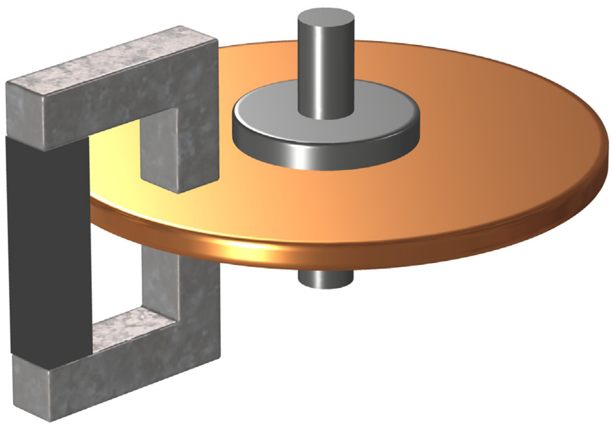


Figure 1: Conceptual geometry of a magnet brake.

Note: This models requires licenses for both the AC/DC Module and LiveLink™ for Simulink®.

Model Definition

The relation between the induced torque T_z and the angular velocity ω can be described by an ordinary differential equation (ODE):

$$\frac{d\omega}{dt} = \frac{T_z}{I}$$

Here, I is the moment of inertia of the disk.

The induced torque is computed in COMSOL Multiphysics as a 3D electromagnetic stationary problem.

The cosimulation with COMSOL Multiphysics and Simulink is set up by exporting a COMSOL Cosimulation file from the COMSOL model and then adding this to the COMSOL Cosimulation block in the Simulink simulation diagram. The input of the block consists of the angular velocity ω, provided by Simulink using an integrator block. The block output is the ratio T_z/I.

Figure 2 shows the full simulation diagram in Simulink.

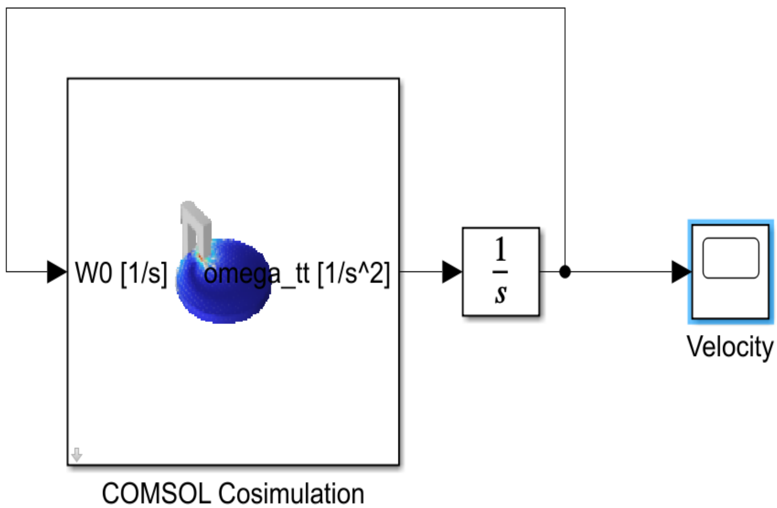


Figure 2: The simulation control diagram.

The initial velocity of the disc is 1000 rpm. To see the effect of the remanent flux in the magnet, a second input is set to the COMSOL block; the focus in this simulation is a constant value of 1 T.

Results and Discussion

Figure 3 below shows the current density in the disk at initial rotating velocity (1000 rpm).

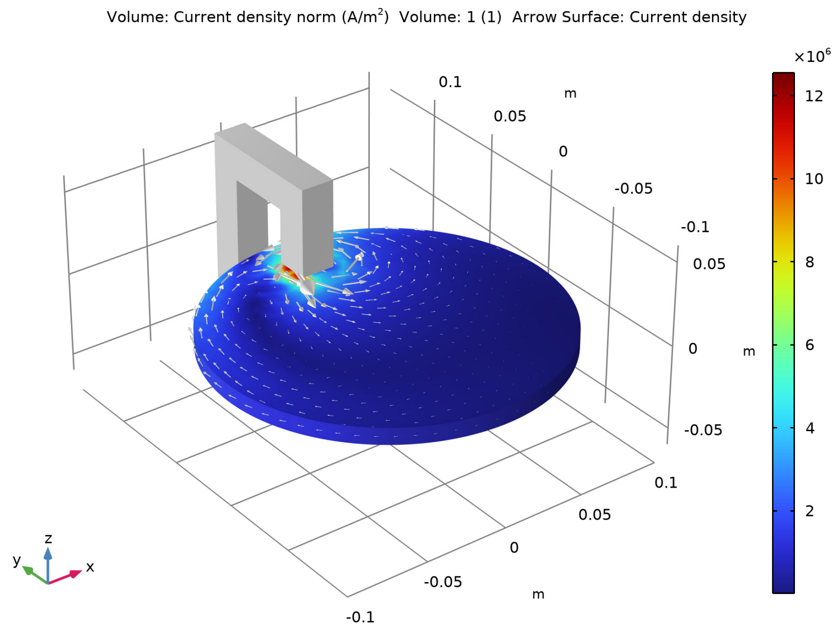


Figure 3: Norm of the current density in the disk at maximal rotating velocity.

Figure 4 shows the decrease of rotational velocity with a 1 T remanent flux density norm.

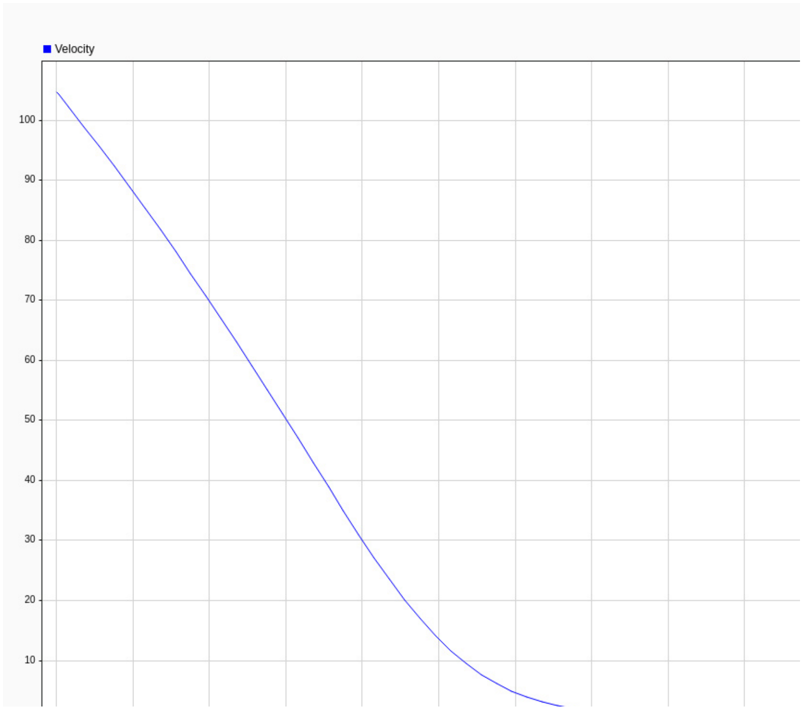


Figure 4: Angular velocity versus time.

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/magnetic_brake_llsimulink`

Modeling Instructions — COMSOL Desktop

In this example you will start from an existing model which is an example described in the AC/DC Module Application Libraries.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **ACDC Module** > **Devices, Transducers and Actuators** > **magnetic_brake** in the tree.

3 Click **Open**.

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	2*pi*dV0	104.72 1/s	Angular velocity

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

MAGNETIC AND ELECTRIC FIELDS (MEF)

Velocity (Lorentz Term) 2

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Magnetic and Electric Fields (mef)** node.

2 Right-click **Component 1 (comp1)** > **Magnetic and Electric Fields (mef)** > **Velocity (Lorentz Term) 1** and choose **Duplicate**.

3 In the **Settings** window for **Velocity (Lorentz Term)**, locate the **Velocity (Lorentz Term)** section.

4 Specify the **v** vector as

-y*W0	x
x*W0	y
0	z

5 Right-click **Velocity (Lorentz Term) 2** and choose **Move Up**.

DEFINITIONS

To define the cosimulation block output you need to create first a global variable probe.

Global Variable Probe 4 (var4)

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

2 In the **Settings** window for **Global Variable Probe**, type `omega_tt` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `-axialTorque/mass1.Izz`.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

4 Right-click and choose **Add Study**.

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

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** checkbox.

3 In the tree, select **Component 1 (comp1)** > **Magnetic and Electric Fields (mef)** > **Velocity (Lorentz Term) 1**.

4 Right-click and choose **Disable**.

5 In the tree, select **Component 1 (comp1)** > **Global ODEs and DAEs (ge)**.

6 Click **Disable in Model**.

7 In the **Model Builder** window, click **Study 2**.

8 In the **Settings** window for **Study**, locate the **Study Settings** section.

9 Clear the **Generate default plots** checkbox.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** node, then click **Iterative 1**.

4 In the **Settings** window for **Iterative**, click to expand the **Error** section.

5 In the **Factor in error estimate** text field, type 1.

6 Click **Compute**.

RESULTS

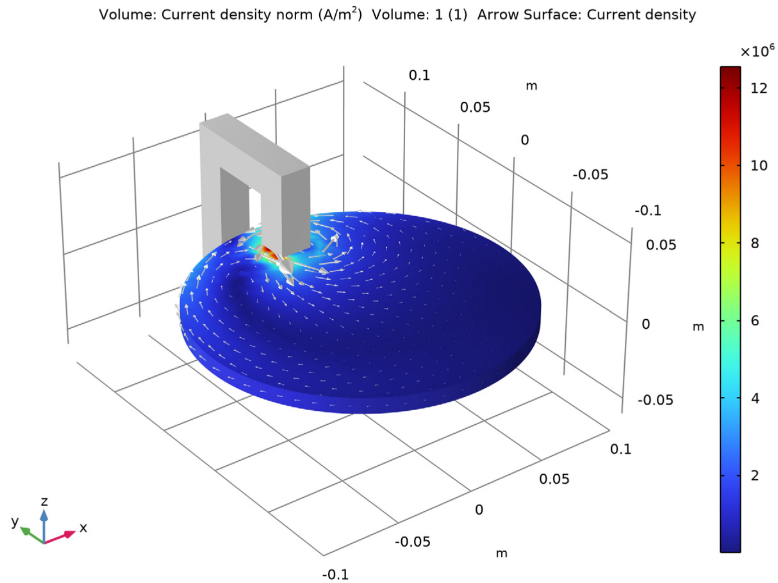
3D Plot Group 6

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 4** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

4 In the **3D Plot Group 6** toolbar, click **Plot**.



SAVE THE COMSOL MODEL

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

2 Browse to a suitable folder, enter the filename `magnetic_brake_llsimulink.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 `magnetic_brake_llsimulink`.

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

5 In the table, enter the following settings:

Parameter name	Initial value	Unit
W0 (Angular velocity)	`2*pi*dV0`	1/s

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

7 In the table, enter the following settings:

Parameter name	Initial value	Unit
mB (Magnet flux)	`1[T]`	T

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

Expression	Unit	Name
`comp1.omega_tt`	`1/s^2`	`omega_tt`

9 Locate the **Image** section. Click **Set from Graphics Window**This sets the current temperature plot (if a solution is available) as the thumbnail used for the COMSOL Cosimulation block inside Simulink.

10 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 magnetic_brake_llsimulink.slx.

4 Click Open to get the simulation diagram in Simulink as in Figure 2.

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.

---

7 To run the simulation, click Run.

The simulation instruction are now over, you can run again a simulation with different settings such as the remanent flux in the magnet or the initial velocity.

If you want to change the remanent flux double-click the COMSOL Cosimulation block, then click Block parameters button. In the mB field, enter the desired value in Tesla and click **OK**.

If you want to change the initial velocity, double-click the Integrator block. In the **Initial condition** field enter the desired initial velocity in radian per seconds.