



Battery Pack Discharge Control with Thermal Analysis

Introduction

Thermal management is important in battery modeling. This example computes the temperature distribution in a battery pack during a 4C discharge. To ensure a constant output power and prevent extreme battery usage condition, the multiphysics model is coupled to a control diagram in Simulink. There, the current is automatically adjusted based on output power and the battery voltage. The maximum temperature in the battery pack is also monitored.

Note: This model requires licenses for both the Battery Design Module and LiveLink™ for Simulink®.

Model Definition

The COMSOL model includes the electrochemical analysis of a battery cell coupled to a 3D thermal analysis of the battery stack. For a detailed description of the battery pack and the underlying COMSOL model, see the *Thermal Distribution in a Pack of Cylindrical Batteries* tutorial in the Battery Design Module Application Library.

The control system implemented in Simulink ensures a constant usage power and prevents possible battery damage. The control system consists of a PI controller to adjust the battery usage power to 180 W. The applied current in the battery is limited to a minimum value of -16 A and is stopped when the voltage reaches a minimum value of 12 V.

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 the applied current, provided by Simulink. The block has three outputs: the stack voltage, the temperature in the middle cell, and the power usage.

[Figure 1](#) shows the full simulation diagram in Simulink.

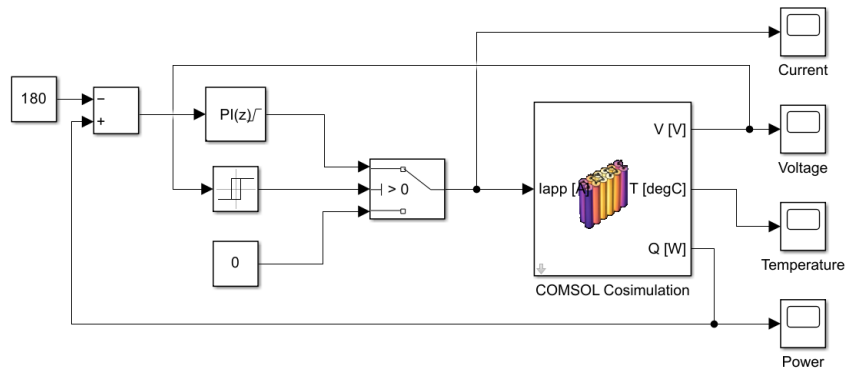


Figure 1: Battery control diagram.

Results and Discussion

Figure 2 below shows the temperature in the battery stack. The maximum temperature is located in the middle of the stack, which corresponds to the third cell in the COMSOL model.

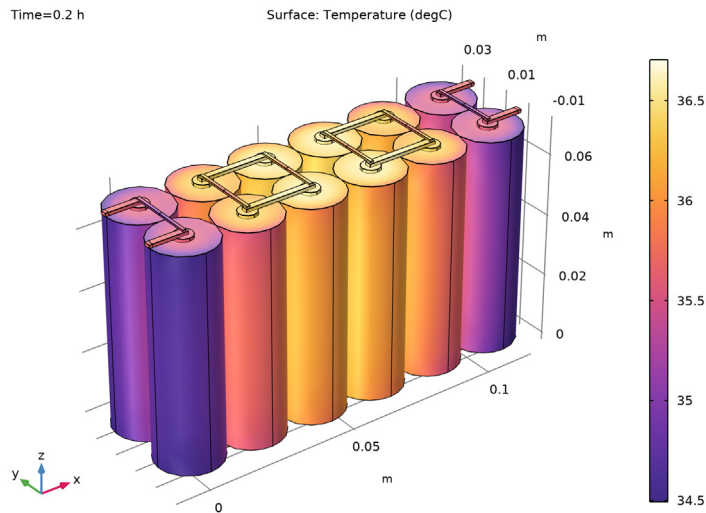


Figure 2: Temperature distribution in battery stack.

Figure 3 below shows the voltage of battery during usage. The voltage drops rapidly when the battery is fully charged and almost discharged; these are the regions that can cause most damage. Because it is not recommended to leave the battery in this region the applied current is cut when the voltage is below 10.5 V.

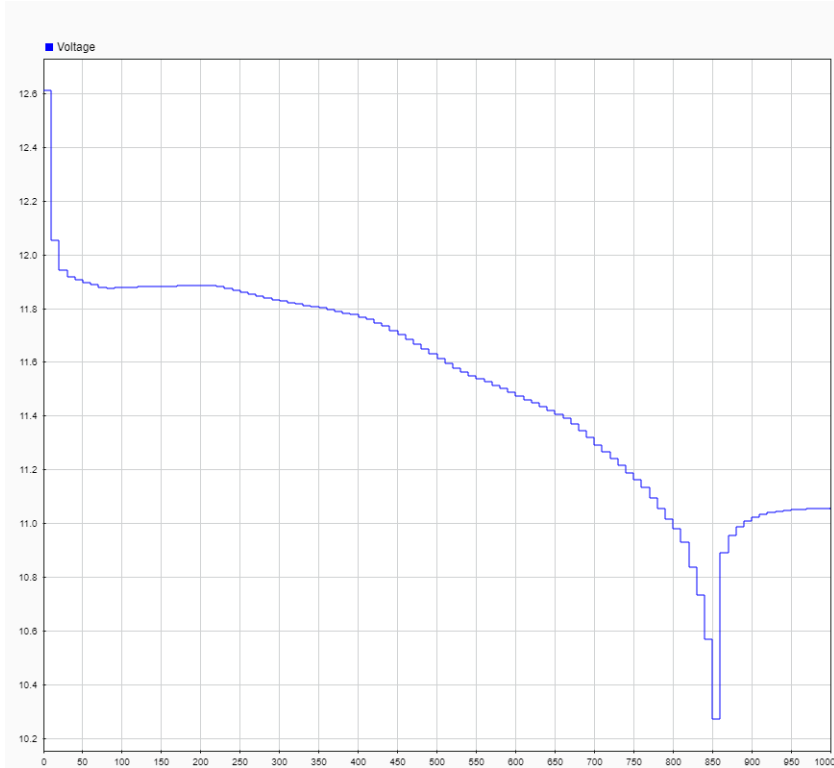


Figure 3: Voltage in battery stack.

Figure 4 below shows the applied voltage to ensure a usage power of 180 W. The current keep decreasing as the battery get discharged. The current is limited to -16 A to not damage the battery; this limit is reached after almost 700 s at full power. When the minimum allowed voltage is reached, the current is cut, here around 860 s.

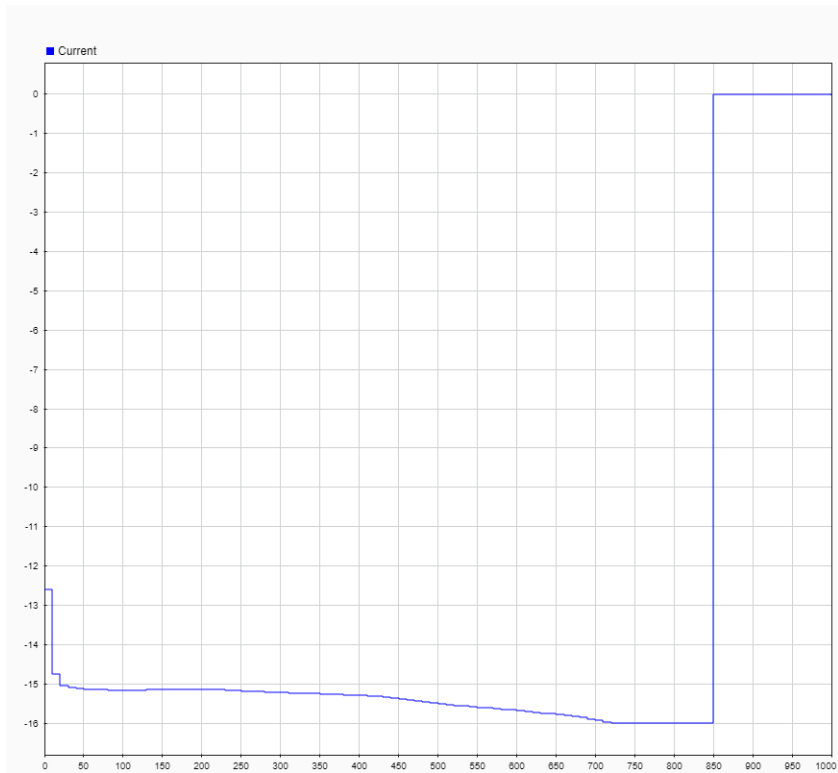


Figure 4: Electric current in battery stack.

Figure 5 shows the usage power provided by the battery. You can notice the response time to reach full power caused by the PID controller, which is about 50 sec. Once the current is restricted to its maximum value, the usage power can no longer be enforced to a constant value and starts to drop until the current is cut.

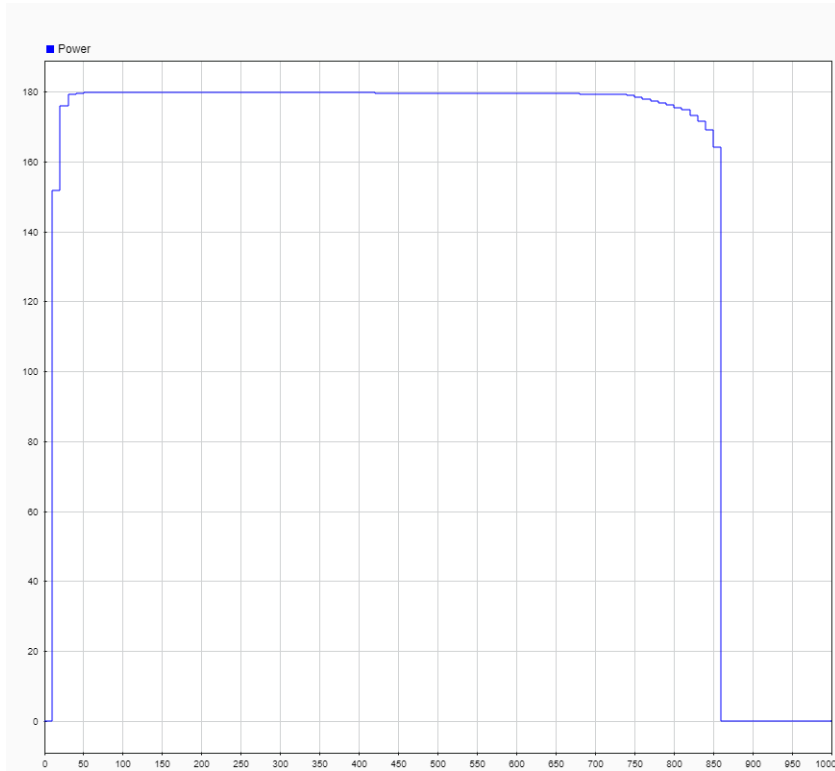


Figure 5: Battery power.

Figure 6 shows the variation of the temperature in the third cell of the COMSOL model, which is used as output of the COMSOL Cosimulation block. The temperature keeps increasing until the power starts to drop, but the rise suddenly before the current is cut.

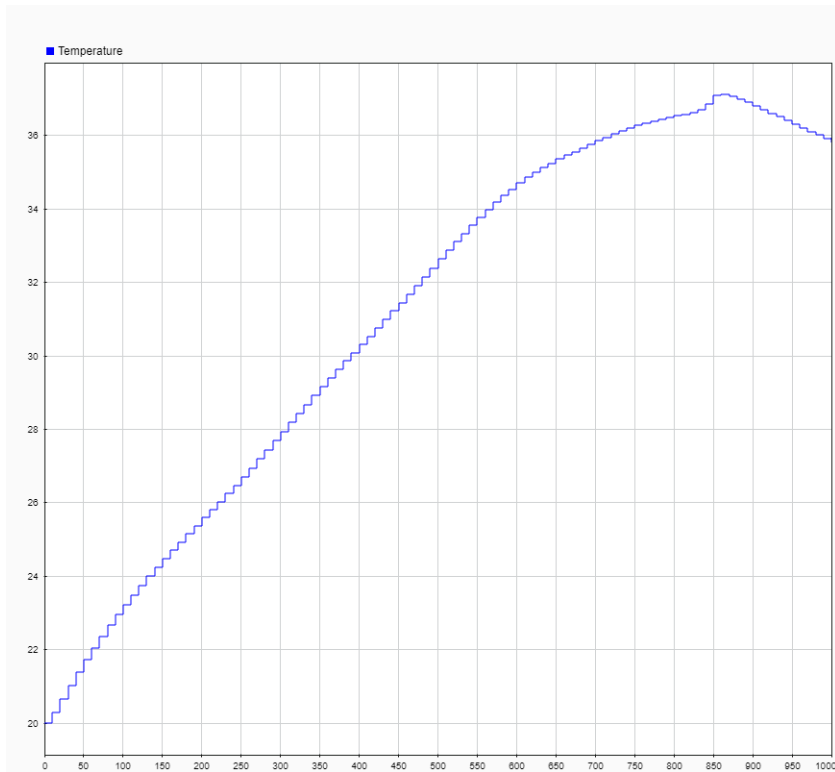


Figure 6: Maximum temperature in battery stack.

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/
battery_control_thermal

Modeling Instructions — COMSOL Desktop

Start this tutorial by opening the model *Thermal Distribution in a Pack of Cylindrical Batteries* from the *Battery Design Module Application Library*.

APPLICATION LIBRARIES

- 1 From the **File** menu, choose **Application Libraries**.
- 2 In the **Application Libraries** window, select **Battery Design Module>Thermal Management>battery_pack_6s2p** in the tree.
- 3 Click  **Open**.

GLOBAL DEFINITIONS

Battery Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Battery Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
Iapp	-10[A]	-10 A	Applied current

COMPONENT 1 (COMP1)

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

DEFINITIONS (COMP1)

E_OCP (int1)

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **E_OCP (int1)**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 Click  **Import**.

dEdT (int2)

- 1 In the **Model Builder** window, click **dEdT (int2)**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 Click  **Import**.

BATTERY PACK (BP)

In the **Model Builder** window, expand the **Component 1 (comp1)>Battery Pack (bp)** node.

Current 1

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Battery Pack (bp)>Current Conductors** node, then click **Current 1**.
- 2 In the **Settings** window for **Current**, locate the **Electrode Current** section.
- 3 In the $I_{s,total}$ text field, type I_{app} .

DEFINITIONS (COMP1)

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

Global Variable Probe 7 (var7)

- 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 voltage in the **Variable name** text field.
- 3 Locate the **Expression** section. In the **Expression** text field, type $bp.E_{cell_1} + bp.E_{cell_2} + bp.E_{cell_3}$.

Global Variable Probe 8 (var8)

- 1 Right-click **Global Variable Probe 7 (voltage)** and choose **Duplicate**.
- 2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.
- 3 In the **Expression** text field, type $abs(I_{app} * (bp.E_{cell_1} + bp.E_{cell_2} + bp.E_{cell_3}))$.

4 In the **Variable name** text field, type power.

STUDY 1

Step 1: Time Dependent

- 1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 From the **Time unit** list, choose s.
- 4 In the **Output times** text field, type 0 1e3.
Run the study so that you can ensure the model is well set.
- 5 In the **Home** toolbar, click  **Compute**.

SAVE THE COMSOL MODEL

- 1 From the **File** menu, choose **Save As**.
- 2 Browse to a suitable folder, enter the filename battery_control_thermal.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 battery_control_thermal.
- 4 Locate the **Inputs** section. Click  **Add**.
- 5 In the table, enter the following settings:

Parameter name	Initial value	Unit
lapp (Applied current)	- 10[A]	A

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

Expression	Unit	Name
comp1.voltage	V	V
comp1.Temp3	degC	T
comp1.power	W	Q

7 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.

8 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 `battery_control_thermal.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 [LiveLink_for_Simulink/Tutorials/battery_control_thermal](#), 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 1000 s and the communication step size is set to 10 s.

7 To run the simulation, click Run.

POSTPROCESSING 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 temperature distribution on the battery pack.

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 select the **Temperature** node.

3 In the **Temperature (ht)** toolbar, click  **Animation**, and choose **Player**.

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