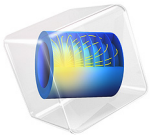

Model created in COMSOL Multiphysics 6.4

Air-Cooled Battery Energy Storage System

Introduction

This model investigates the thermal behavior of a battery energy storage system (BESS) designed to store electrical energy using batteries for later use. Such systems typically include cooling devices to regulate the temperature of battery modules, ensuring optimal battery performance. Analyzing the cooling system involves solving a conjugate heat transfer problem, which accounts for turbulent forced convection within the BESS.

Model Definition

In this example, the BESS model consists of the cabinet containing ten battery modules, each of which comprises sixteen battery units. Four extracting fans generate airflow within the cabinet to cool internal battery modules with the fresh air supplied through four wired gauzes, or grilles at ambient temperature. Additionally, every battery module has its own extracting fan, which enhances forced convection inside the module enclosure. Air enters each module enclosure through a perforated plate and side openings. The battery units act as heat sources, each generating 12 W of heat power. Symmetry reduces the modeling geometry of the BESS to half its size, as shown in Figure 1.

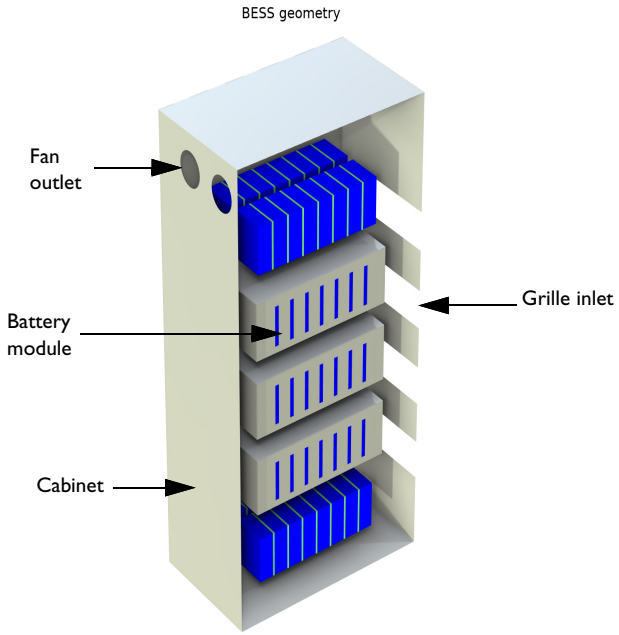

Figure 1: The modeling geometry of the BESS is reduced to half its size due to symmetry.

The air extracted by a fan is influenced by the static pressure, which is the pressure difference between each side of the fan. This information is typically provided by fan manufacturers as a curve representing the volume flow rate as a function of static pressure. Figure 2 shows such curves used for the cabinet and module fans.

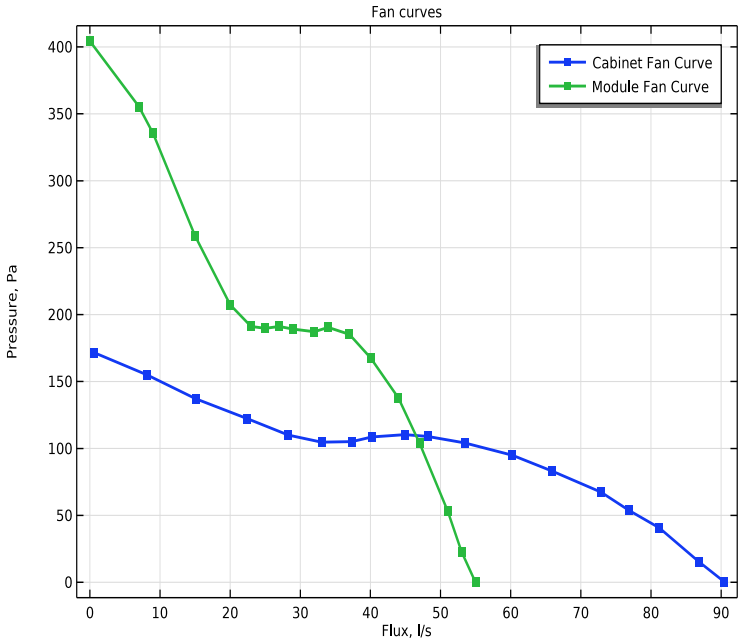

Figure 2: Static pressure curves of the cabinet and module fans.

The ambient air temperature is 20°C. The inlet wired gauze solidity is 0.5, and the wire diameter is 1 mm. The solidity of the module perforated plate is 0.36, and its refraction coefficient is 1. The cabinet and modules enclosures are made of aluminum with the wall thickness of 2 mm. The solid blocks separating the battery units in the modules are made of acrylic plastic. Table 1 summarizes the effective thermal properties of the battery units used in this model.

TABLE 1: EFFECTIVE THERMAL PROPERTIES OF THE BATTERY UNITS.

Property	Value	Description
k_{tl}	1 W/(m·K)	Through-layer thermal conductivity
k_{il}	30 W/(m·K)	In-layer thermal conductivity
ρ_{eff}	2000 kg/(m^3)	Density
$C_{p,\mathrm{eff}}$	1400 J/(kg·K)	Heat capacity

Modeling Considerations

The flow is simulated using the Algebraic yPlus turbulence model. It is less sensitive to mesh resolution compared to transport-equation models like Spalart–Allmaras or the k-ε model, and does not require the use of wall functions.

The Algebraic yPlus turbulence model is consistent with no-slip boundary condition. To enforce the low-Reynolds-number formulation of the Algebraic yPlus model, the **Wall Treatment** setting of the **Turbulent Flow, Algebraic yPlus (spf)** interface is set to **Low Re**.

To apply no-slip boundary conditions on the interior boundaries representing the module walls, the **Interior Wall** feature is used.

The inlet air temperature is set to ambient, as the air is assumed to come from outside the cabinet. The inlet boundaries are configured using multiple **Grill** boundary features, which account for head loss caused by air entering the cabinet enclosure through the wire gauze. The perforated plates at the inlets of the battery modules are simulated using **Screen** boundary features.

COMSOL Multiphysics provides a dedicated boundary condition for modeling fan behavior. The outlet boundaries are configured using **Fan** boundary features with provided static pressure curve data. For the module fans located inside the computational domain, the **Interior Fan** features are used.

Note that **Grill**, **Screen**, **Fan**, and **Interior Fan** features are not intended for use on disjoint boundaries; therefore, a separate feature is applied to each boundary.

For modeling the heat transfer in cabinet and module walls, it is strongly recommended to use the **Thin Layer** boundary feature, which is specifically designed for thin geometries and significantly reduces the number of degrees of freedom in the model.

The heat transfer in the battery units is modeled using the tailored **Battery Layers** feature. It allows to specify the density, heat capacity, and anisotropic thermal conductivity of each unit.

A convective heat flux boundary condition is applied to the top and side surfaces of the cabinet, correlating the inward heat flux, q, with the temperature difference between the wall and the surrounding atmosphere:

$$q = h(T_{\text{ext}} - T)$$

Here, h represents the heat transfer coefficient. The Heat Transfer Module includes a library of heat transfer coefficient functions, which can be easily accessed via the **Heat Flux** boundary feature.

Beside the symmetry plane, all remaining exterior boundaries are thermally insulated walls.

Results and Discussion

The most interesting aspect of this simulation is to locate which components are subject to overheating. Figure 3 clearly shows that the temperature distribution is not homogeneous.

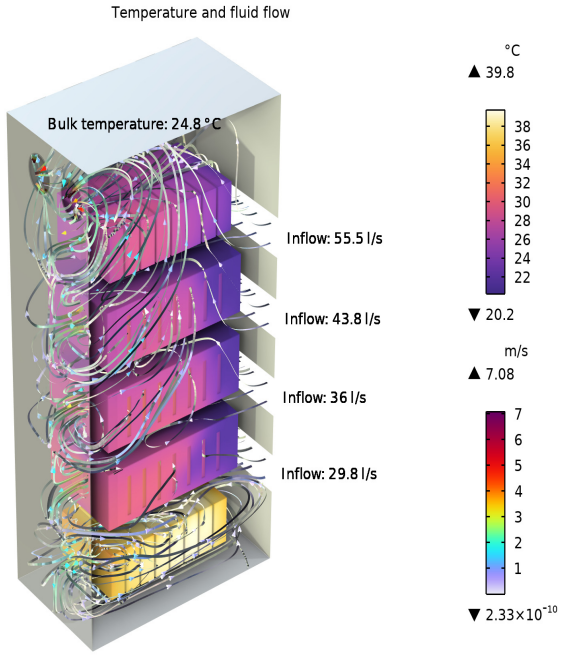

Figure 3: Temperature and fluid flow inside the BESS cabinet.

The maximum temperature is about 40°C and is located at one of the bottom battery units. The maximum air velocity is about 7 m/s at the suction side of the cabinet fans.

Figure 3 clearly shows that the volumetric flow is unevenly distributed between the cabinet inlets. The top cabinet grille receives more air, while the flow rate decreases further down. The top inlet is closest to the outlet fans, and the flow resistance is lowest over this inlet and through the cabinet.

The uneven flow distribution results in the battery modules at the bottom receiving the least cooling, leading to the highest temperatures in this area. Additionally, poor cooling of the bottom module is caused by hot air recirculation inside the cabinet enclosure. The

heated air extracted by the bottom module fan is partially sucked back through the module's side openings, as shown in Figure 4.

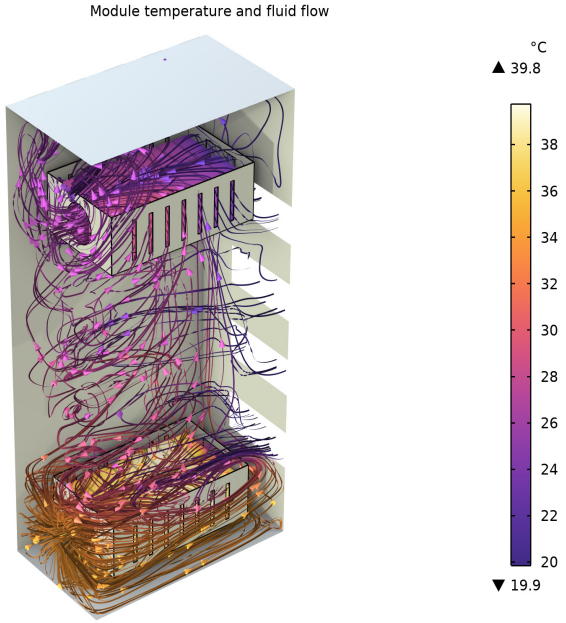

Figure 4: Temperature distribution around the top and bottom battery modules.

Notes About the COMSOL Implementation

A value of –1 m/s is specified for the initial value of the u-component of the velocity field. Although this value does not approximate the actual flow field, it provides a good reference for the velocity scale and direction, and improve the convergence.

To save computational time, a one-way coupling approach is used in this model. This approach is applicable when the temperature dependence of the fluid's physical properties can be neglected. In such cases, the fluid-flow equations can be solved independently.

To set up the one-way coupled nonisothermal flow model, the tailored **Stationary, One-Way with Initialization NITF** preset study is selected. Note that the one-way coupled approach is only suitable when the temperature differences in the model are not significant, and variations in air density and viscosity are negligible. Otherwise, a fully coupled approach should be used, and gravity effects must be taken into account. For more details, refer to the *Heat Transfer Module User's Guide*.

Application Library path: `Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/air_cooled_bess`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Nonisothermal Flow** > **Turbulent Flow** > **Turbulent Flow, Algebraic yPlus**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Stationary, One-Way with Initialization NITF**.

6 Click **Done**.

GEOMETRY 1

Note that the **Geometry representation** must be set to **COMSOL kernel** to fit with the numbering of the boundaries used in these instructions.

The model geometry is available as a parameterized geometry sequence in a separate MPH-file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions. Otherwise, insert the geometry sequence as follows:

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `air_cooled_bess_geom_sequence.mph`.

The application's Application Library folder is shown in the **Application Library path** section immediately before the current section. Note that the path given there is relative to the COMSOL Application Library root.

3 In the **Geometry** toolbar, click **Build All**.

Next, load parameter definitions and material properties.

4 In the **Model Builder** window, collapse the **Geometry 1** node.

DEFINITIONS

Change the view settings to improve the rendering of 3D plots.

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

View 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** > **View 1** node, then click **View 1**.

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

3 Clear the **Show grid** checkbox.

4 Clear the **Show axis orientation** checkbox.

5 Click to expand the **Visual Effects** section. Select the **Ambient occlusion** checkbox.

6 Set the **Shadow strength** value to **0.7**.

7 In the **Shadow strength** text field, type `0.75`.

8 Select the **Direct shadows** checkbox.

9 In the **Shadow softness** text field, type `0.2`.

10 In the **Shadow strength** text field, type `0.25`.

11 From the **Quality preset** list, choose **High quality**.

12 Click to expand the **Environment** section. From the **Environment map** list, choose **Outdoor**.

Headlight 4

In the **View 1** toolbar, click **Headlight**.

GLOBAL DEFINITIONS

Geometry Parameters

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

2 In the **Settings** window for **Parameters**, type `Geometry Parameters` in the **Label** text field.

Model Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.

2 In the **Settings** window for **Parameters**, type `Model Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `air_cooled_bess_model_parameters.txt`.

DEFINITIONS

Add a node for the ambient properties in the model. The default temperature value is 293.15 K. It is possible to edit the ambient temperature value or to define it using the meteorological data, which gives access to climate data from more than 8600 stations in the world.

Ambient Properties 1 (ampr1)

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

2 In the **Settings** window for **Ambient Properties**, locate the **Ambient Conditions** section.

3 In the T_{amb} text field, type `T_amb`.

MATERIALS

Next, add materials.

ADD MATERIAL

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

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

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

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

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Air**.

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Acrylic plastic**.

3 Click the **Add to Component** button in the window toolbar.

MATERIALS

Acrylic plastic (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Gaps**.

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Aluminum**.

3 Click the **Add to Component** button in the window toolbar.

4 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat3)

1 Click the **Wireframe Rendering** button in the **Graphics** toolbar to see the interior walls selection in the next steps.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All Aluminum Walls**.

TURBULENT FLOW, ALGEBRAIC YPLUS (SPF)

Now set up the fluid flow simulation. Start by specifying a selection for the fluid domain and setting up the model parameters.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, Algebraic yPlus (spf)**.

2 In the **Settings** window for **Turbulent Flow, Algebraic yPlus**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

4 Locate the **Turbulence** section. From the **Wall treatment** list, choose **Low Re**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, Algebraic yPlus (spf)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

u_init	x

Interior Wall 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Wall**.

2 In the **Settings** window for **Interior Wall**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Module Walls**.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

Fan 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fan**.

2 Select Boundary 6 only.

3 In the **Settings** window for **Fan**, locate the **Flow Direction** section.

4 From the **Flow direction** list, choose **Outlet**.

5 Locate the **Parameters** section. From the **Flow condition** list, choose **Static pressure curve, data**.

6 Locate the **Static Pressure Curve Data** section. Click **Load from File**.

7 Browse to the model's Application Libraries folder and double-click the file `air_cooled_bess_cabinet_fan_curve.txt`.

8 Locate the **Units** section. From the **Flux** list, choose **dm^3/s**.

Fan 2

1 Right-click **Fan 1** and choose **Duplicate**.

2 In the **Settings** window for **Fan**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 5 only.

Fan 1, Fan 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, Algebraic yPlus (spf)**, Ctrl-click to select **Fan 1** and **Fan 2**.

2 Right-click and choose **Group**.

Cabinet Fans

In the **Settings** window for **Group**, type `Cabinet Fans` in the **Label** text field.

Grille 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Grille**.
2. Select Boundary 882 only.
3. In the **Settings** window for **Grille**, locate the **Parameters** section.
4. From the **Flow condition** list, choose **Grille type**.

Grille 2

1. Right-click **Grille 1** and choose **Duplicate**.
2. In the **Settings** window for **Grille**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 881 only.

Grille 3

1. Right-click **Grille 2** and choose **Duplicate**.
2. In the **Settings** window for **Grille**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 880 only.

Grille 4

1. Right-click **Grille 3** and choose **Duplicate**.
2. In the **Settings** window for **Grille**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 879 only.

Grille 1, Grille 2, Grille 3, Grille 4

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, Algebraic yPlus (spf)**, Ctrl-click to select **Grille 1**, **Grille 2**, **Grille 3**, and **Grille 4**.
2. Right-click and choose **Group**.

Cabinet Grilles

In the **Settings** window for **Group**, type `Cabinet Grilles` in the **Label** text field.

Interior Fan 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Interior Fan**.
2. Select Boundary 32 only.

3 In the **Settings** window for **Interior Fan**, locate the **Parameters** section.

4 From the **Flow condition** list, choose **Static pressure curve, data**.

5 Locate the **Static Pressure Curve Data** section. Click **Load from File**.

6 Browse to the model's Application Libraries folder and double-click the file `air_cooled_bess_module_fan_curve.txt`.

7 Locate the **Units** section. From the **Flux** list, choose **dm^3/s**.

Interior Fan 2

1 Right-click **Interior Fan 1** and choose **Duplicate**.

2 In the **Settings** window for **Interior Fan**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 31 only.

Interior Fan 3

1 Right-click **Interior Fan 2** and choose **Duplicate**.

2 In the **Settings** window for **Interior Fan**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 30 only.

Interior Fan 4

1 Right-click **Interior Fan 3** and choose **Duplicate**.

2 In the **Settings** window for **Interior Fan**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 29 only.

Interior Fan 5

1 Right-click **Interior Fan 4** and choose **Duplicate**.

2 In the **Settings** window for **Interior Fan**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 28 only.

Interior Fan 1, Interior Fan 2, Interior Fan 3, Interior Fan 4, Interior Fan 5

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, Algebraic yPlus (spf)**, Ctrl-click to select **Interior Fan 1**, **Interior Fan 2**, **Interior Fan 3**, **Interior Fan 4**, and **Interior Fan 5**.

2 Right-click and choose **Group**.

Module Fans

In the **Settings** window for **Group**, type `Module Fans` in the **Label** text field.

Screen 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Screen**.
2. Select Boundary 877 only.
3. In the **Settings** window for **Screen**, locate the **Screen Type** section.
4. From the **Screen type** list, choose **Perforated plate**.
5. Locate the **Parameters** section. In the σ_s text field, type `sigma`.
6. From the **Refraction** list, choose **User defined**. In the η text field, type `1`.

Screen 2

1. Right-click **Screen 1** and choose **Duplicate**.
2. In the **Settings** window for **Screen**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 876 only.

Screen 3

1. Right-click **Screen 2** and choose **Duplicate**.
2. In the **Settings** window for **Screen**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 875 only.

Screen 4

1. Right-click **Screen 3** and choose **Duplicate**.
2. In the **Settings** window for **Screen**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 874 only.

Screen 5

1. Right-click **Screen 4** and choose **Duplicate**.
2. In the **Settings** window for **Screen**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 873 only.

Screen 1, Screen 2, Screen 3, Screen 4, Screen 5

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, Algebraic yPlus (spf)**, Ctrl-click to select **Screen 1**, **Screen 2**, **Screen 3**, **Screen 4**, and **Screen 5**.

2 Right-click and choose **Group**.

Module Screens

In the **Settings** window for **Group**, type `Module Screens` in the **Label** text field.

TURBULENT FLOW, ALGEBRAIC YPLUS (SPF)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Turbulent Flow, Algebraic yPlus (spf)** node.

HEAT TRANSFER IN FLUIDS (HT)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Fluids (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 From the T list, choose **Ambient temperature (ampr1)**.

Solid 1

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

2 In the **Settings** window for **Solid**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Gaps**.

Now define thermal properties of the battery units.

Battery Layers 1

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

2 In the **Settings** window for **Battery Layers**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Cells**.

4 Locate the **Battery Layers** section. In the k_{tl} text field, type `k_tl`.

5 In the k_{il} text field, type `k_il`.

6 In the ρ_{eff} text field, type `rho_eff`.

7 In the $C_{p,eff}$ text field, type `Cp_eff`.

Next, use the `Q0` parameter to define the heat source in the battery units.

Heat Source 1

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

2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Cells**.

4 Locate the **Heat Source** section. In the Q_0 text field, type `Q0`.

Inflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 In the **Settings** window for **Inflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cabinet Grills (Work Plane 3)**.

4 Locate the **Upstream Properties** section. From the T_{ustr} list, choose **Ambient temperature (ampr1)**.

The ambient temperature is defined in the **Ambient Properties** node under **Shared Properties**.

Outflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cabinet Fans (Work Plane 3)**.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cabinet Side Walls**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 From the **Heat transfer coefficient** list, choose **External natural convection**.

6 In the L text field, type `H_cabinet`.

7 From the p_{A} list, choose **Ambient absolute pressure (ampr1)**.

8 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Heat Flux 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cabinet Top**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 From the **Heat transfer coefficient** list, choose **External natural convection**.

6 From the list, choose **Horizontal plate, upside**.

7 In the L text field, type `L_cabinet*W_cabinet/(2*(L_cabinet+W_cabinet))`.

8 From the p_A list, choose **Ambient absolute pressure (ampr1)**.

9 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

Thin Layer 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for **Thin Layer**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All Aluminum Walls**.

4 Locate the **Layer Model** section. From the **Layer type** list, choose **General**.

Initial Values 1a

1 In the **Model Builder** window, click **Initial Values 1a**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 From the T list, choose **Ambient temperature (ampr1)**.

MATERIALS

Aluminum (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Aluminum (mat3)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thickness	lth	2[mm]	m	Shell

Now create the mesh. Use manual settings to create a proper mesh for this model.

MESH 1

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Battery Module**.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Module Fan (Work Plane 1)**.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Finer**.

Free Tetrahedral 2

In the **Mesh** toolbar, click  **Free Tetrahedral**.

Size 1

1 Right-click **Free Tetrahedral 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Cabinet Selection**.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Fine**.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Cabinet Fans (Work Plane 3)**.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Extra fine**.

Boundary Layers 1

1 In the **Mesh** toolbar, click  **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Air**.

5 Click to expand the **Corner Settings** section. From the **Handling of sharp edges** list, choose **Trimming**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All Walls**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `3`.

5 In the **Thickness adjustment factor** text field, type `0.05`.

6 Click  **Build All**.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

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

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

4 In the **Study** toolbar, click  **Compute**.

RESULTS

Preferred Units 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results** and choose **Preferred Units**.

3 In the **Settings** window for **Preferred Units**, locate the **Units** section.

4 Click  **Add Physical Quantity**.

5 In the **Physical Quantity** dialog, type `temperature` in the text field.

6 In the tree, select **General** > **Temperature (K)**.

7 Click **OK**.

8 In the **Settings** window for **Preferred Units**, locate the **Units** section.

9 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Temperature	K	°C

Layered Material 1

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **More Datasets** > **Layered Material**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Middle Modules**.

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.

3 From the **Plane** list, choose **xy-planes**.

4 In the **z-coordinate** text field, type `Gap_bottom_modules+H_module/2`.

Cabinet Fan Curve

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Cabinet Fan Curve` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `air_cooled_bess_cabinet_fan_curve.txt`.

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	`Flux, l/s`
2	`Pressure, Pa`

Module Fan Curve

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Module Fan Curve` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `air_cooled_bess_module_fan_curve.txt`.

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	`Flux, l/s`
2	`Pressure, Pa`

Fan Curves

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Fan Curves` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Fan curves`.

Cabinet Fan Curve

1 Right-click **Fan Curves** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, type `Cabinet Fan Curve` in the **Label** text field.

3 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

5 Click to expand the **Legends** section. Find the **Include** subsection. Select the **Label** checkbox.

6 Clear the **Headers** checkbox.

7 Select the **Show legends** checkbox.

Module Fan Curve

1 Right-click **Cabinet Fan Curve** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, type `Module Fan Curve` in the **Label** text field.

3 Locate the **Data** section. From the **Table** list, choose **Module Fan Curve**.

4 In the **Fan Curves** toolbar, click **Plot**.

BESS Geometry

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `BESS Geometry` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `BESS geometry`.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 Right-click **BESS Geometry** and choose **Surface**.

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

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

Material Appearance 1

1 Right-click **Surface 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material** list, choose **Aluminum (mat3)**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Cabinet Walls**.

Surface 2

Right-click **Surface 1** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Middle Modules**.

Surface 3

1 In the **Model Builder** window, right-click **BESS Geometry** and choose **Surface**.

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

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

Material Appearance 1

1 Right-click **Surface 3** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material** list, choose **Acrylic plastic (mat2)**.

Surface 4

1 In the **Model Builder** window, right-click **BESS Geometry** and choose **Surface**.

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 **Blue**.

Selection 1

1 Right-click **Surface 4** and choose **Selection**.

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

3 From the **Selection** list, choose **Cells**.

4 In the **BESS Geometry** toolbar, click **Plot**.

RESULTS

BESS Geometry

In the **Model Builder** window, collapse the **Results** > **BESS Geometry** node.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Turbulent Flow, Algebraic yPlus** > **Wall Resolution (spf)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Wall Resolution (spf)

1 In the **Settings** window for **3D Plot Group**, locate the **Title** section.

2 From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Wall resolution in viscous units`.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

5 In the **Wall Resolution (spf)** toolbar, click **Plot**.

Surface 1

1 In the **Model Builder** window, expand the **Wall Resolution (spf)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **Prism**.

Temperature and Fluid Flow

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Temperature and Fluid Flow` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Temperature and fluid flow`.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

7 Select the **Show units** checkbox.

8 From the **Position** list, choose **Right double**.

Volume 1

1 Right-click **Temperature and Fluid Flow** and choose **Volume**.

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

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.

Selection 1

1 Right-click **Volume 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Cells**.

Volume 2

1 In the **Model Builder** window, right-click **Temperature and Fluid Flow** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Data** section.

3 From the **Dataset** list, choose **Layered Material 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `T`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.

Temperature and Fluid Flow

Click the  **Zoom Extents** button in the **Graphics** toolbar.

Streamline 1

1 Right-click **Temperature and Fluid Flow** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 In the **Number** text field, type `30`.

4 Locate the **Selection** section. From the **Selection** list, choose **Cabinet Grills (Work Plane 3)**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Ribbon**.

6 In the **Width expression** text field, type `1[mm]`.

7 Select the **Width scale factor** checkbox. In the associated text field, type `5`.

8 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

9 Select the **Scale factor** checkbox. In the associated text field, type `0.02`.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **Prism**.

4 From the **Color table transformation** list, choose **Nonlinear**.

5 In the **Color calibration parameter** text field, type `-0.7`.

Material Appearance 1

1 In the **Model Builder** window, right-click **Streamline 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Plastic (shiny)**.

5 Locate the **Color** section. Select the **Use the plot's color** checkbox.

Streamline 2

1 Right-click **Streamline 1** and choose **Duplicate**.

2 In the **Settings** window for **Streamline**, locate the **Selection** section.

3 From the **Selection** list, choose **Module Grill (Work Plane 1)**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Streamline 1**.

Surface 1

1 In the **Model Builder** window, right-click **Temperature and Fluid Flow** and choose **Surface**.

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

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

Material Appearance 1

1 Right-click **Surface 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material** list, choose **Aluminum (mat3)**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Cabinet Frontside Walls**.

Surface 2

Right-click **Surface 1** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Cabinet Backside**.

Transparency 1

1 In the **Model Builder** window, right-click **Surface 2** and choose **Transparency**.

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

3 Find the **Transparency** subsection. Set the **Transparency** value to **0.7**.

Annotation 1

1 In the **Model Builder** window, right-click **Temperature and Fluid Flow** and choose **Annotation**.

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

3 In the **Text** text field, type `Bulk temperature: eval(ht.ofl1.Tave,°C,3) °C`.

4 Locate the **Position** section. In the **y** text field, type `W_cabinet/2-W_cabinet/(Ny_cabinet_fans+1)`.

5 In the **z** text field, type `H_cabinet-0.1*R_fan_cabinet`.

6 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

7 From the **Anchor point** list, choose **Lower left**.

8 In the **Temperature and Fluid Flow** toolbar, click **Plot**.

Annotation 2

1 Right-click **Temperature and Fluid Flow** and choose **Annotation**.

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

3 In the **Text** text field, type `Inflow: eval(abs(spf.grille4.volumeFlowRate),l/s, 3) l/s`.

4 Locate the **Position** section. In the **x** text field, type `L_cabinet`.

5 In the **y** text field, type `0`.

6 In the **z** text field, type `Gap_bottom_modules+H_module+Gap_z_modules/2`.

7 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

8 From the **Anchor point** list, choose **Middle left**.

9 In the **Temperature and Fluid Flow** toolbar, click **Plot**.

Annotation 3

1 Right-click **Annotation 2** and choose **Duplicate**.

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

3 In the **Text** text field, type `Inflow: eval(abs(spf.grille3.volumeFlowRate),l/s, 3) l/s`.

4 Locate the **Position** section. In the **z** text field, type `Gap_bottom_modules+2*H_module+1.5*Gap_z_modules`.

5 In the **Temperature and Fluid Flow** toolbar, click **Plot**.

Annotation 4

1 Right-click **Annotation 3** and choose **Duplicate**.

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

3 In the **Text** text field, type `Inflow: eval(abs(spf.grille2.volumeFlowRate),l/s, 3) l/s`.

4 Locate the **Position** section. In the **z** text field, type `Gap_bottom_modules+3*H_module+2.5*Gap_z_modules`.

5 In the **Temperature and Fluid Flow** toolbar, click **Plot**.

Annotation 5

1 Right-click **Annotation 4** and choose **Duplicate**.

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

3 In the **Text** text field, type `Inflow: eval(abs(spf.grille1.volumeFlowRate),l/s, 3) l/s`.

4 Locate the **Position** section. In the **z** text field, type `Gap_bottom_modules+4*H_module+3.5*Gap_z_modules`.

5 In the **Temperature and Fluid Flow** toolbar, click **Plot**.

RESULTS

Temperature and Fluid Flow

In the **Model Builder** window, collapse the **Results** > **Temperature and Fluid Flow** node.

Module Temperature and Fluid Flow

1. In the **Results** toolbar, click  **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Module Temperature and Fluid Flow` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `Module temperature and fluid flow`.
5. Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
6. Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.
7. Select the **Show units** checkbox.

Volume 1

1. Right-click **Module Temperature and Fluid Flow** and choose **Volume**.
2. In the **Settings** window for **Volume**, locate the **Expression** section.
3. In the **Expression** text field, type `T`.
4. Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.

Selection 1

1. Right-click **Volume 1** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Bottom Cells**.

Surface 1

1. In the **Model Builder** window, right-click **Module Temperature and Fluid Flow** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `1`.

Material Appearance 1

1. Right-click **Surface 1** and choose **Material Appearance**.
2. In the **Settings** window for **Material Appearance**, locate the **Appearance** section.
3. From the **Material** list, choose **Aluminum (mat3)**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Bottom Module**.

Surface 2

Right-click **Surface 1** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Cabinet Frontside Walls**.

4 In the **Module Temperature and Fluid Flow** toolbar, click **Plot**.

Streamline 1

1 In the **Model Builder** window, right-click **Module Temperature and Fluid Flow** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 In the **Number** text field, type `40`.

4 Locate the **Selection** section. From the **Selection** list, choose **Bottom Module Openings**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Ribbon**.

6 In the **Width expression** text field, type `1[mm]`.

7 Select the **Width scale factor** checkbox. In the associated text field, type `3`.

8 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

9 Select the **Scale factor** checkbox. In the associated text field, type `0.02`.

10 Locate the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

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

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

2D Plot Group 6

In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Line 1

1. In the **Model Builder** window, right-click **Module Temperature and Fluid Flow** and choose **Line**.
2. In the **Settings** window for **Line**, 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 **Black**.

Selection 1

1. Right-click **Line 1** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Bottom Module**.
5. In the **Module Temperature and Fluid Flow** toolbar, click **Plot**.

Volume 2

1. In the **Model Builder** window, under **Results** > **Module Temperature and Fluid Flow** right-click **Volume 1** and choose **Duplicate**.
2. In the **Settings** window for **Volume**, click to expand the **Inherit Style** section.
3. From the **Plot** list, choose **Volume 1**.

Selection 1

1. In the **Model Builder** window, expand the **Volume 2** node, then click **Selection 1**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Top Cells**.

Surface 3

In the **Model Builder** window, under **Results** > **Module Temperature and Fluid Flow** right-click **Surface 1** and choose **Duplicate**.

Selection 1

1. In the **Model Builder** window, expand the **Surface 3** node, then click **Selection 1**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Top Module**.

Line 2

In the **Model Builder** window, under **Results** > **Module Temperature and Fluid Flow** right-click **Line 1** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Line 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Top Module**.

Streamline 2

1 In the **Model Builder** window, under **Results** > **Module Temperature and Fluid Flow** right-click **Streamline 1** and choose **Duplicate**.

2 In the **Model Builder** window, click **Streamline 2**.

3 In the **Settings** window for **Streamline**, locate the **Selection** section.

4 From the **Selection** list, choose **Top Module Openings**.

5 In the **Module Temperature and Fluid Flow** toolbar, click **Plot**.

Cross Section of the Bottom Module

1 In the **Model Builder** window, under **Results** click **2D Plot Group 6**.

2 In the **Settings** window for **2D Plot Group**, type `Cross Section of the Bottom Module` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Temperature and velocity field at cross section of the bottom module`.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

6 Select the **Show units** checkbox.

7 From the **Position** list, choose **Right double**.

Surface 1

1 Right-click **Cross Section of the Bottom Module** and choose **Surface**.

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

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.

Arrow Surface 1

1 Right-click **Cross Section of the Bottom Module** and choose **Arrow Surface**.

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

3 Find the **x grid points** subsection. In the **Points** text field, type `30`.

4 Find the **y grid points** subsection. In the **Points** text field, type `30`.

5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

Color Expression 1

1 Right-click **Arrow Surface 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **Prism**.

4 From the **Color table transformation** list, choose **Nonlinear**.

5 In the **Color calibration parameter** text field, type `-0.7`.

6 In the **Cross Section of the Bottom Module** toolbar, click **Plot**.

Geometry Modeling Instructions

Follow these steps to create the geometry for the air cooled BESS model.

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

NEW

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

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

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. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `air_cooled_bess_geom_parameters.txt`.

GEOMETRY 1

Block 1 (blk1)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Width** text field, type `L_module`.
4. In the **Depth** text field, type `W_module`.
5. In the **Height** text field, type `H_module`.
6. Click **Build Selected**.

Work Plane 1 (wp1)

1. In the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3. From the **Plane type** list, choose **Face parallel**.
4. On the object **blk1**, select Boundary 2 only.
5. Locate the **Unite Objects** section. Clear the **Unite objects** checkbox.

6 Locate the **Selections of Resulting Entities** section. Find the **Selections from plane geometry** subsection. Select the **Show in physics** checkbox.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Module Grill

1 In the **Work Plane** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Module Grill` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `W_screen`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

7 From the **Show in 3D** list, choose **All levels**.

Module Fan

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, type `Module Fan` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `R_fan_module`.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** node.

2 In the **Model Builder** window, click **Work Plane 1 (wp1)**.

3 In the **Settings** window for **Work Plane**, click **Build Selected**.

Move 1 (mov1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms** > **Move**.

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Module Grill (Work Plane 1)**.

4 Locate the **Displacement** section. In the **x** text field, type `L_module`.

5 Click **Build Selected**.

Module Side Openings

1 In the **Geometry** toolbar, click  **Work Plane**.

2 In the **Settings** window for **Work Plane**, type `Module Side Openings` in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane type** list, choose **Face parallel**.

4 On the object **blk1**, select Boundary 3 only.

5 Click to expand the **Local Coordinate System** section. From the **Origin** list, choose **Vertex projection**.

6 On the object **blk1**, select Point 1 only.

7 In the **Rotation** text field, type `-90`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

Module Side Openings (wp2) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Module Side Openings (wp2) > Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click  **Rectangle**.

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

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

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

5 Locate the **Position** section. In the **xw** text field, type `(L_module-(L_cell+Gap_x_cells)*(Nx_cells-2)-W_hole)/2`.

6 In the **yw** text field, type `Gap_y_hole+W_hole/2`.

Module Side Openings (wp2) > Array 1 (arr1)

1 In the **Work Plane** toolbar, click  **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **All objects**.

4 Locate the **Size** section. From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `Nx_cells-1`.

6 Locate the **Displacement** section. In the **xw** text field, type `L_cell+Gap_x_cells`.

7 Click  **Build Selected**.

GEOMETRY 1

Module Side Openings (wp2)

1 In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** > **Module Side Openings (wp2)** node.

2 In the **Model Builder** window, click **Module Side Openings (wp2)**.

3 In the **Settings** window for **Work Plane**, click **Build Selected**.

Copy 1 (copy1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms** > **Copy**.

2 In the **Settings** window for **Copy**, locate the **Input** section.

3 From the **Input objects** list, choose **Module Side Openings**.

4 Clear the **Keep input objects** checkbox.

5 Locate the **Displacement** section. In the **y** text field, type `0 W_module`.

6 Click **Build Selected**.

Module Housing

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, type `Module Housing` in the **Label** text field.

3 Locate the **Union** section. From the **Input objects** list, choose **All objects**.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

5 Click **Build Selected**.

Cells

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

2 In the **Settings** window for **Block**, type `Cells` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `L_cell`.

4 In the **Depth** text field, type `W_cell`.

5 In the **Height** text field, type `H_cell`.

6 Locate the **Position** section. In the **x** text field, type `(L_module-(L_cell+Gap_x_cells)*Nx_cells+Gap_x_cells)/2`.

7 In the **y** text field, type `(W_module-(W_cell+Gap_y_cells)*Ny_cells+Gap_y_cells)/2`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

9 From the **Show in physics** list, choose **All levels**.

10 Click **Build Selected**.

Gaps

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

2 In the **Settings** window for **Block**, type `Gaps` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `Gap_x_cells`.

4 In the **Depth** text field, type `W_cell`.

5 In the **Height** text field, type `H_cell`.

6 Locate the **Position** section. In the **x** text field, type `(L_module-(L_cell+Gap_x_cells)*Nx_cells+Gap_x_cells)/2+L_cell`.

7 In the **y** text field, type `(W_module-(W_cell+Gap_y_cells)*Ny_cells+Gap_y_cells)/2`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

9 From the **Show in physics** list, choose **All levels**.

10 Click **Build Selected**.

Array 1 (arr1)

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

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Cells**.

4 Locate the **Size** section. In the **x size** text field, type `Nx_cells`.

5 In the **y size** text field, type `Ny_cells`.

6 Locate the **Displacement** section. In the **x** text field, type `L_cell+Gap_x_cells`.

7 In the **y** text field, type `W_cell+Gap_y_cells`.

8 Click **Build Selected**.

Array 2 (arr2)

1 Right-click **Array 1 (arr1)** and choose **Duplicate**.

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Gaps**.

4 Locate the **Size** section. In the **x size** text field, type `Nx_cells-1`.

5 Click **Build Selected**.

Module Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, type `Module Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Module Housing**, **Cells**, and **Gaps**.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Module Grill (Work Plane 1)**, **Module Fan (Work Plane 1)**, and **Module Side Openings**.

10 Click **OK**.

Module Aluminum Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, type `Module Aluminum Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Module Walls** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Cells** and **Gaps**.

10 Click **OK**.

11 In the **Settings** window for **Difference Selection**, click **Build Selected**.

Battery Module

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `Battery Module` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Module Housing**, **Cells**, and **Gaps**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, click **Build Selected**.

Array 1 (arr1), Array 2 (arr2), Battery Module (unisel1), Block 1 (blk1), Cells (blk2), Copy 1 (copy1), Gaps (blk3), Module Aluminum Walls (difsel2), Module Housing (uni1), Module Side Openings (wp2), Module Walls (difsel1), Move 1 (mov1), Work Plane 1 (wp1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, Ctrl-click to select **Block 1 (blk1)**, **Work Plane 1 (wp1)**, **Move 1 (mov1)**, **Module Side Openings (wp2)**, **Copy 1 (copy1)**, **Module Housing (uni1)**, **Cells (blk2)**, **Gaps (blk3)**, **Array 1 (arr1)**, **Array 2 (arr2)**, **Module Walls (difsel1)**, **Module Aluminum Walls (difsel2)**, and **Battery Module (unisel1)**.

2 Right-click and choose **Group**.

Battery Module

1 In the **Settings** window for **Group**, type `Battery Module` in the **Label** text field.

2 Click **Build Selected**.

3 In the **Model Builder** window, collapse the **Battery Module** node.

Move 2 (mov2)

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

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Battery Module**.

4 Locate the **Displacement** section. In the **x** text field, type `Gap_front_modules`.

5 In the **y** text field, type `Gap_side_modules-W_cabinet/2`.

6 In the **z** text field, type `Gap_bottom_modules`.

7 Click **Build Selected**.

Array 3 (arr3)

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

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Battery Module**.

4 Locate the **Size** section. In the **y size** text field, type `Ny_modules`.

5 In the **z size** text field, type `Nz_modules`.

6 Locate the **Displacement** section. In the **y** text field, type `W_module+Gap_y_modules`.

7 In the **z** text field, type `H_module+Gap_z_modules`.

8 Click **Build Selected**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Cabinet Housing

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

2 In the **Settings** window for **Block**, type `Cabinet Housing` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `L_cabinet`.

4 In the **Depth** text field, type `W_cabinet`.

5 In the **Height** text field, type `H_cabinet`.

6 Locate the **Position** section. In the **y** text field, type `-W_cabinet/2`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

Work Plane 3 (wp3)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **blk4**, select Boundary 2 only.

5 Locate the **Local Coordinate System** section. In the **Rotation** text field, type `180`.

6 Locate the **Unite Objects** section. Clear the **Unite objects** checkbox.

7 Locate the **Selections of Resulting Entities** section. Find the **Selections from plane geometry** subsection. Select the **Show in physics** checkbox.

Work Plane 3 (wp3) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Cabinet Grills

1 In the **Work Plane** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Cabinet Grills` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `W_grille`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **yw** text field, type `Gap_bottom_modules+H_module+Gap_z_modules/2-H_cabinet/2`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

8 From the **Show in 3D** list, choose **All levels**.

Work Plane 3 (wp3) > Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Cabinet Grills**.

4 Locate the **Size** section. From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `Nz_modules-1`.

6 Locate the **Displacement** section. In the **yw** text field, type `H_module+Gap_z_modules`.

7 Click **Build Selected**.

Cabinet Fans

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, type `Cabinet Fans` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `R_fan_cabinet`.

4 Locate the **Position** section. In the **xw** text field, type `W_cabinet/(Ny_cabinet_fans+1)-W_cabinet/2`.

5 In the **yw** text field, type `H_cabinet/2-2*R_fan_cabinet`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

7 From the **Show in 3D** list, choose **All levels**.

8 Click **Build Selected**.

Work Plane 3 (wp3) > Array 2 (arr2)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Cabinet Fans**.

4 Locate the **Size** section. From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `Ny_cabinet_fans`.

6 Locate the **Displacement** section. In the **xw** text field, type `W_cabinet/(Ny_cabinet_fans+1)`.

7 Click **Build Selected**.

Work Plane 3 (wp3)

1 In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** > **Work Plane 3 (wp3)** node.

2 In the **Model Builder** window, click **Work Plane 3 (wp3)**.

3 In the **Settings** window for **Work Plane**, click **Build Selected**.

Move 3 (mov3)

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

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Cabinet Grills (Work Plane 3)**.

4 Locate the **Displacement** section. In the **x** text field, type `L_cabinet`.

5 Click **Build Selected**.

Cabinet Selection

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `Cabinet Selection` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Cabinet Housing**, **Cabinet Grills (Work Plane 3)**, and **Cabinet Fans (Work Plane 3)**.

6 Click **OK**.

Cabinet

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, type `Cabinet` in the **Label** text field.

3 Locate the **Union** section. From the **Input objects** list, choose **Cabinet Selection**.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

5 Click **Build Selected**.

BESS

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `BESS` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Battery Module** and **Cabinet**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, click **Build Selected**.

Symmetry

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

2 In the **Settings** window for **Block**, type `Symmetry` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `L_cabinet`.

4 In the **Depth** text field, type `W_cabinet/2`.

5 In the **Height** text field, type `H_cabinet`.

6 Locate the **Position** section. In the **y** text field, type `-W_cabinet/2`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

8 From the **Show in physics** list, choose **Boundary selection**.

9 Locate the **Assigned Attributes** section. Select the **Construction geometry** checkbox.

10 Click **Build Selected**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

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

3 From the **Objects to add** list, choose **BESS**.

4 From the **Objects to subtract** list, choose **Symmetry**.

5 Click **Build Selected**.

Cabinet Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, type `Cabinet Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Cabinet** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Cabinet Grills (Work Plane 3)** and **Cabinet Fans (Work Plane 3)**.

10 Click **OK**.

Cabinet Top

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `Cabinet Top` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.
5 Click **Add**.
6 In the **Add** dialog, select **Cabinet Walls** in the **Selections** list.
7 Click **OK**.
8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.
9 In the **z minimum** text field, type `0.99*H_cabinet`.
10 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Cabinet Bottom

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `Cabinet Bottom` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.
5 Click **Add**.
6 In the **Add** dialog, select **Cabinet Walls** in the **Selections** list.
7 Click **OK**.
8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.
9 In the **z maximum** text field, type `0.01*H_cabinet`.
10 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Cabinet Side Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.
2 In the **Settings** window for **Difference Selection**, type `Cabinet Side Walls` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.
5 In the **Add** dialog, select **Cabinet Walls** in the **Selections to add** list.
6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Cabinet Top** and **Cabinet Bottom**.

10 Click **OK**.

All Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `All Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Module Walls** and **Cabinet Walls**.

6 Click **OK**.

All Aluminum Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `All Aluminum Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Module Aluminum Walls** and **Cabinet Walls**.

6 Click **OK**.

Air

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, type `Air` in the **Label** text field.

3 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

4 In the **Add** dialog, select **Cabinet** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

7 Click the **Add** button for **Selections to subtract**.

8 In the **Add** dialog, in the **Selections to subtract** list, choose **Cells** and **Gaps**.

9 Click **OK**.

Middle Modules

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `Middle Modules` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.
5 Click **Add**.
6 In the **Add** dialog, select **Module Aluminum Walls** in the **Selections** list.
7 Click **OK**.
8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.
9 In the **z minimum** text field, type `W_module+Gap_bottom_modules`.
10 In the **z maximum** text field, type `H_cabinet-2*H_module-Gap_bottom_modules`.

Bottom Module

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `Bottom Module` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.
5 Click **Add**.
6 In the **Add** dialog, select **Module Aluminum Walls** in the **Selections** list.
7 Click **OK**.
8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.
9 In the **z maximum** text field, type `0.99*H_module+Gap_bottom_modules`.

Bottom Cells

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `Bottom Cells` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.
4 Click **Add**.
5 In the **Add** dialog, select **Cells** in the **Selections** list.
6 Click **OK**.
7 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.
8 In the **z maximum** text field, type `0.99*H_module+Gap_bottom_modules`.

Bottom Module Openings

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Bottom Module Openings` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, in the **Selections** list, choose **Module Grill (Work Plane 1)** and **Module Side Openings**.

7 Click **OK**.

8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.

9 In the **z maximum** text field, type `0.99*H_module+Gap_bottom_modules`.

Top Module

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Bottom Module (boxsel4)** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Top Module` in the **Label** text field.

3 Locate the **Box Limits** section. In the **z minimum** text field, type `0.99*(Nz_modules-1)*(H_module+Gap_z_modules)+Gap_bottom_modules`.

4 In the **z maximum** text field, type `0.99*(Nz_modules-1)*(H_module+Gap_z_modules)+Gap_bottom_modules+H_module`.

Top Cells

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Bottom Cells (boxsel5)** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Top Cells` in the **Label** text field.

3 Locate the **Box Limits** section. In the **z minimum** text field, type `0.99*(Nz_modules-1)*(H_module+Gap_z_modules)+Gap_bottom_modules`.

4 In the **z maximum** text field, type `0.99*(Nz_modules-1)*(H_module+Gap_z_modules)+Gap_bottom_modules+H_module`.

Top Module Openings

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Bottom Module Openings (boxsel6)** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Top Module Openings` in the **Label** text field.

3 Locate the **Box Limits** section. In the **z maximum** text field, type `0.99*(Nz_modules-1)*(H_module+Gap_z_modules)+Gap_bottom_modules+H_module`.

4 In the **z minimum** text field, type `0.99*(Nz_modules-1)*(H_module+Gap_z_modules)+Gap_bottom_modules`.

Cabinet Backside

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Cabinet Backside` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Cabinet Walls** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.

9 In the **x maximum** text field, type `0.01`.

10 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Cabinet Frontside Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, type `Cabinet Frontside Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Cabinet Walls** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Cabinet Backside** in the **Selections to subtract** list.

10 Click **OK**.