Created in COMSOL Multiphysics 6.2

Isothermal Box

Introduction

This example solves the heat transfer in an isothermal box aimed at transporting refrigerated articles such as medical materials for 24 hours. In this case, the box not only has to keep the content cold over a long period of time, but also has to respect a storage temperature restriction. In this model, the restriction interval is between 2°C and 8°C.

The box needs an insulating material such as foam to separate the content from the exterior environment. A cold source, for instance ice at nearly –5°C, is then added. However, to stay above the lower temperature bound of 2°C, foam is also placed between the ice and the contents.

A thickening agent often increases the water viscosity before freezing it. Once melt, this eutectic mixture is meant to avoid convective motion that may accelerate warming.

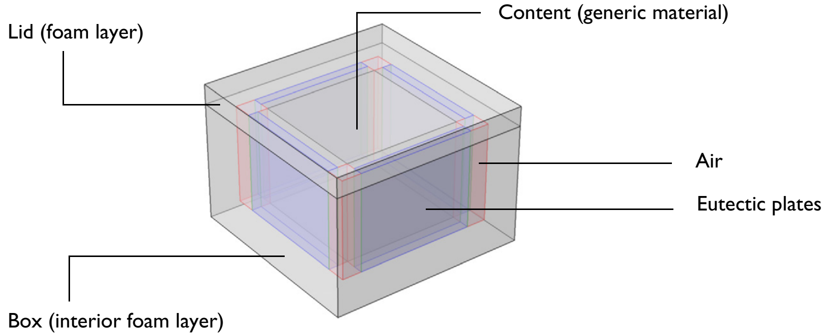


Figure 1: Geometry and material distribution of the isothermal box.

Model Definition

Figure 1 illustrates the geometry and material distribution of the isothermal box. Foam isolates the box from the exterior environment. The content is surrounded by four eutectic plates behind foam layers at the vertical boundaries. The remaining space at the corners of the content is filled with air. The content is at an initial temperature of 5°C.

AMBIENT TEMPERATURE

In this model, the ambient temperature follows the last climate data from ASHRAE. The Sevilla weather station was chosen for this simulation (see Figure 2), with a typical temperature profile of June 1st, starting at 6 a.m.

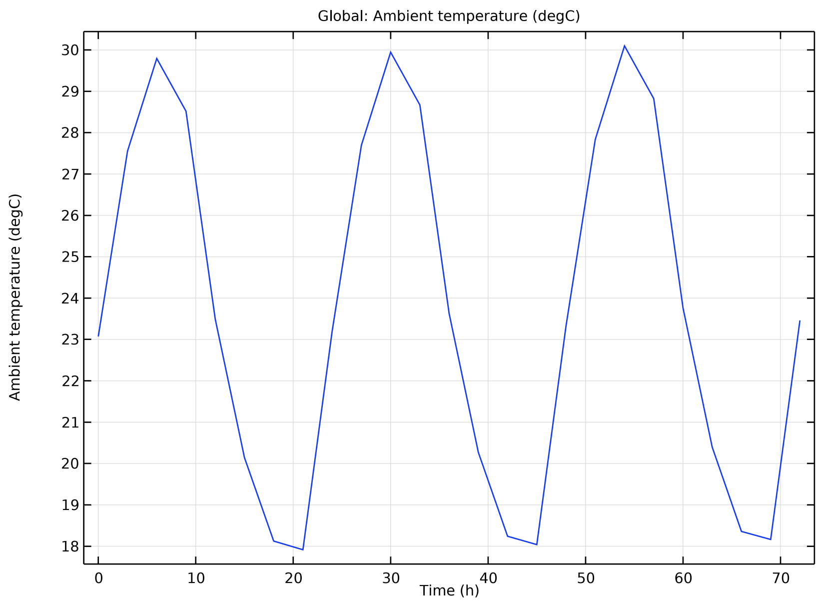


Figure 2: Typical ambient temperature profile according to ASHRAE climate data, given by the Sevilla weather station from June 1st at 6 a.m. to June 4th at 6 a.m.

Convective cooling conditions with this time-dependent temperature profile apply on the exterior boundaries of the box.

GENERIC MATERIAL FOR THE CONTENT

To simplify the model, you can consider that all the available space for the content is used. This assumption corresponds to cases where overpackaging fills the remaining empty space to attenuate shocks caused by transportation. The content may also change depending on the situations. Hence, a generic material with thermophysical properties stated in Table 1 is used for this simulation.

TABLE 1: THERMOPHYSICAL PROPERTIES OF THE GENERIC MATERIAL.

Thermophysical Property	**Value**
Thermal conductivity	1 W/(m·K)
Density	2000 kg/m^3
Heat capacity at constant pressure	800 J/(kg·K)

THIN THERMALLY RESISTIVE LAYERS

The eutectic mixture is generally contained in thin plastic packages that behave as resistive layers. The thermal conductivity is set to 0.30 W/(m·K) in this model and the thickness to 1 mm. Similarly, thin air layers between foam, lid, and content (see Figure 3) can be modeled as a thin film of conductivity 0.025 W/(m·K) and thickness 50 μm.

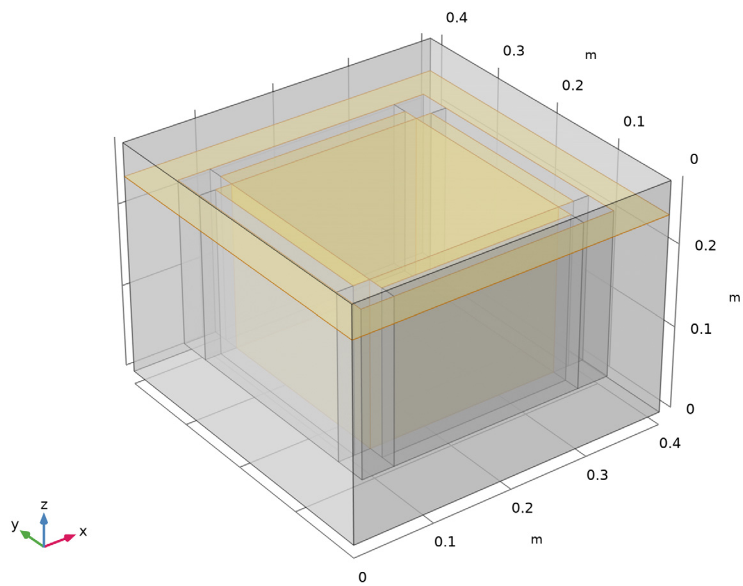


Figure 3: Selection corresponding to the air thin thermally resistive layers.

ICE-TO-WATER PHASE CHANGE

To keep the medical content refrigerated during transportation, the four lateral eutectic plates act as energy storage that maintain the box at low temperature. Phase change occurs at around 0°C. As long as the ice is not completely melt, ideally all along the transportation time, the temperature in the eutectic plates remain stable near 0°C which protects the content from exceeding the critical temperature.

Results and Discussion

Figure 4 shows several curves representing the evolution of temperature in the box for 72 hours, together with a temperature plot of the eutectic plates.

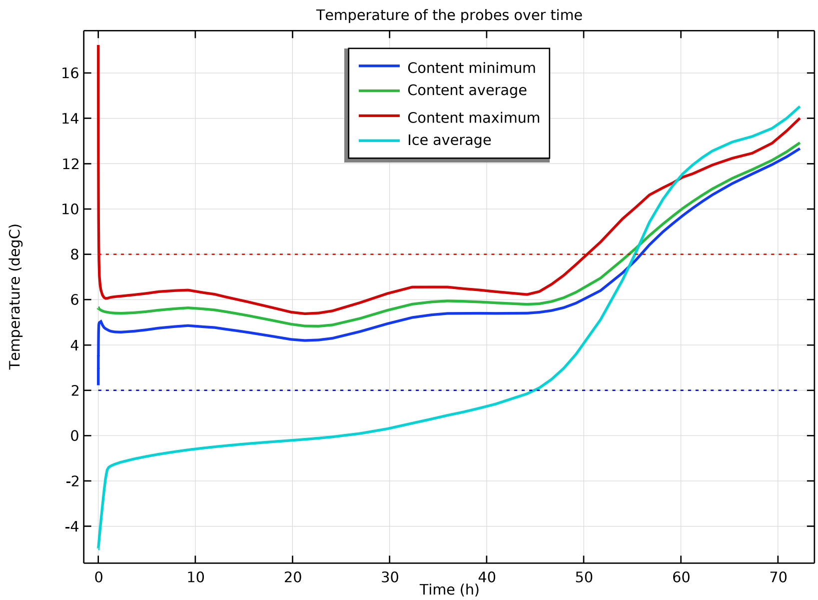


Figure 4: Temperature variations of the contents (red, green, and blue), average temperature of the ice blocks (cyan) and critical values (dotted lines), during 72 hours.

The average temperature of the ice (cyan curve) quickly falls from –5°C to –1°C but remains close to 0°C for nearly 24 hours. The plain green, blue, and red curves show the evolution of average, minimum, and maximum temperature of the content, respectively. They remain in the range 2°C to 8°C (dotted lines) during the first 24 hours. Between 24 and 48 hours, just after the end of the melting process, the content comes increasingly closer to the critical temperature of 8°C. After 48 hours, the whole box is too close or above 8°C.

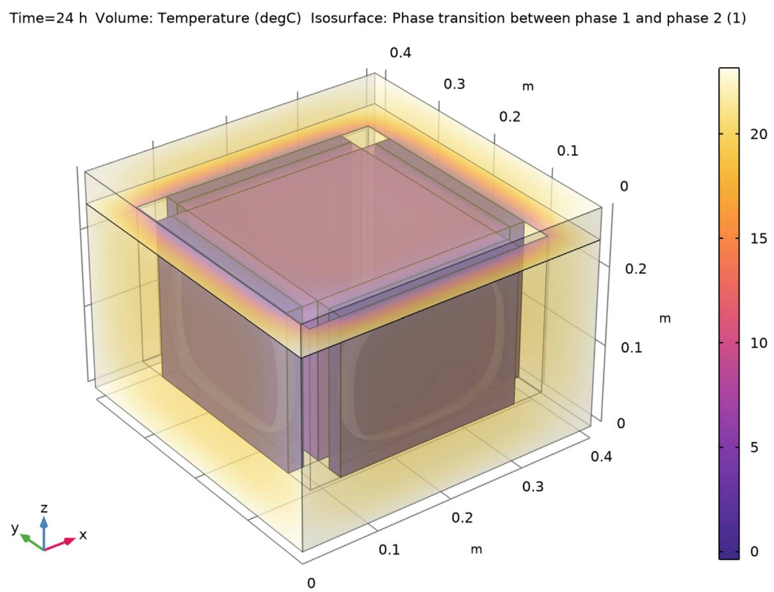


Figure 5: Temperature field in the box at time 24 hours.

Figure 5 shows the temperature profile in the box after 24 hours of transportation. Sharp temperature gradients can be observed in the foam insulating layers, which protects the content from outside warmth.

Application Library path: `Heat_Transfer_Module/Medical_Technology/isothermal_box`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Time Dependent**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Start by entering the model parameters.

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

Name	Expression	Value	Description
box_w	40[cm]	0.4 m	Box width
box_d	40[cm]	0.4 m	Box depth
box_h	24[cm]	0.24 m	Box height
content_w	24[cm]	0.24 m	Content width
content_d	24[cm]	0.24 m	Content depth
content_h	20[cm]	0.2 m	Content height
ice_t	2[cm]	0.02 m	Eutectic plates thickness
foam_t	4[cm]	0.04 m	Foam layers thickness
lid_t	4[cm]	0.04 m	Box lid thickness

The next steps create the geometry with cumulative selections.

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 box_w.
4 In the **Depth** text field, type box_d.

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

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

Block 2 (blk2)

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 `content_w`.

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

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

6 Locate the **Position** section. In the **x** text field, type `(box_w-content_w)/2`.

7 In the **y** text field, type `(box_d-content_d)/2`.

8 In the **z** text field, type `box_h-content_h`.

9 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

10 In the **New Cumulative Selection** dialog box, type `Content` in the **Name** text field.

11 Click **OK**.

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

13 Click the **Transparency** button in the **Graphics** toolbar.

The **Transparency** functionality is convenient here to display the interior of the box.

Block 3 (blk3)

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 `ice_t`.

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

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

6 Locate the **Position** section. In the **x** text field, type `box_w-foam_t-ice_t`.

7 In the **y** text field, type `(box_d-content_d)/2`.

8 In the **z** text field, type `box_h-content_h`.

9 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

10 In the **New Cumulative Selection** dialog box, type `Ice` in the **Name** text field.

11 Click **OK**.

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

13 Right-click **Block 3 (blk3)** and choose **Duplicate**.

Block 4 (blk4)

1 In the **Model Builder** window, click **Block 4 (blk4)**.

2 In the **Settings** window for **Block**, locate the **Position** section.

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

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

Block 5 (blk5)

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 `content_w`.

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

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

6 Locate the **Position** section. In the **x** text field, type `(box_w-content_w)/2`.

7 In the **y** text field, type `foam_t`.

8 In the **z** text field, type `box_h-content_h`.

9 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Ice**.

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

11 Right-click **Block 5 (blk5)** and choose **Duplicate**.

Block 6 (blk6)

1 In the **Model Builder** window, click **Block 6 (blk6)**.

2 In the **Settings** window for **Block**, locate the **Position** section.

3 In the **y** text field, type `box_d-foam_t-ice_t`.

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

Block 7 (blk7)

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 `(box_w-content_w)/2-foam_t`.

4 In the **Depth** text field, type `(box_d-content_d)/2-foam_t`.

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

6. Locate the **Position** section. In the **x** text field, type `foam_t`.
7. In the **y** text field, type `foam_t`.
8. In the **z** text field, type `box_h-content_h`.
9. Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.
10. In the **New Cumulative Selection** dialog box, type `Empty Space` in the **Name** text field.
11. Click **OK**.
12. In the **Geometry** toolbar, click **Build All**.
13. Right-click **Block 7 (blk7)** and choose **Duplicate**.

Block 8 (blk8)

1. In the **Model Builder** window, click **Block 8 (blk8)**.
2. In the **Settings** window for **Block**, locate the **Position** section.
3. In the **x** text field, type `(box_w+content_w)/2`.
4. In the **Geometry** toolbar, click **Build All**.
5. Right-click **Block 8 (blk8)** and choose **Duplicate**.

Block 9 (blk9)

1. In the **Model Builder** window, click **Block 9 (blk9)**.
2. In the **Settings** window for **Block**, locate the **Position** section.
3. In the **y** text field, type `(box_d+content_d)/2`.
4. In the **Geometry** toolbar, click **Build All**.
5. Right-click **Block 9 (blk9)** and choose **Duplicate**.

Block 10 (blk10)

1. In the **Model Builder** window, click **Block 10 (blk10)**.
2. In the **Settings** window for **Block**, locate the **Position** section.
3. In the **x** text field, type `foam_t`.
4. In the **Geometry** toolbar, click **Build All**.

Block 11 (blk11)

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 `box_w`.
4. In the **Depth** text field, type `box_d`.

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

6 Locate the **Position** section. In the **z** text field, type `box_h`.

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

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

After finalizing the geometry, define a few remaining useful selections to be used in the rest of the model.

DEFINITIONS

Foam

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Foam` in the **Label** text field.

3 Select Domains 1 and 2 only.

Thin Air Resistive Layers

1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Thin Air Resistive Layers` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

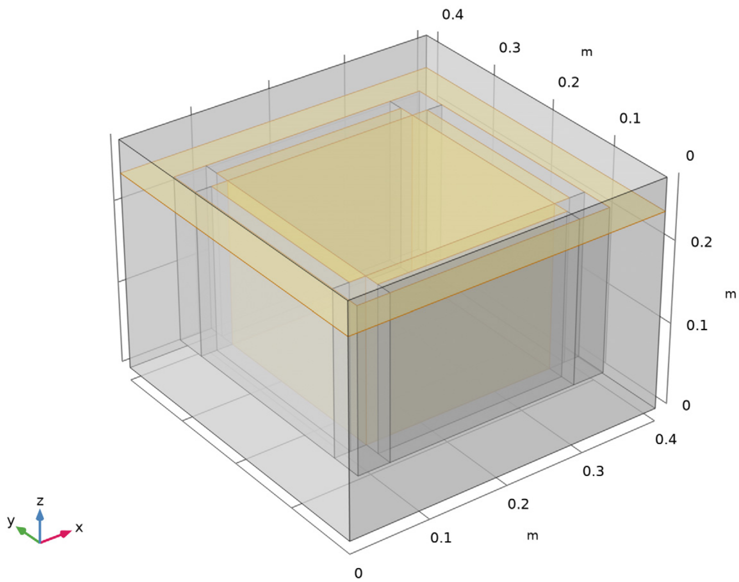
4 In the **Add** dialog box, in the **Input selections** list, choose **Foam** and **Content**.

5 Click **OK**.

6 In the **Settings** window for **Adjacent**, locate the **Output Entities** section.

7 Clear the **Exterior boundaries** check box.

8 Select the **Interior boundaries** check box.



Exterior Surfaces

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Exterior Surfaces` in the **Label** text field.

3 Locate the **Input Entities** section. Select the **All domains** check box.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

In order to monitor the temperature field inside the box, define now a few probes that would display temperature values during computation.

Content Minimum

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `Content Minimum` in the **Label** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Minimum**.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Content**.

5 Locate the **Expression** section. From the **Table and plot unit** list, choose **degC**.

Content Average

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `Content Average` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **Content**.

4 Locate the **Expression** section. From the **Table and plot unit** list, choose **degC**.

Content Maximum

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `Content Maximum` in the **Label** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Content**.

5 Locate the **Expression** section. From the **Table and plot unit** list, choose **degC**.

Ice Average

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `Ice Average` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **Ice**.

4 Locate the **Expression** section. From the **Table and plot unit** list, choose **degC**.

Before setting up the material properties, specify the boundaries which are modeled as thin layers or as thin films, and which domains that are solid or changing phase. Using this information, COMSOL Multiphysics can detect which material properties are needed.

HEAT TRANSFER IN SOLIDS (HT)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Heat Transfer in Solids (ht)** and choose **Fluid**.

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

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

Phase Change Material 1

In the **Physics** toolbar, click **Attributes** and choose **Phase Change Material**.

Thin Film 1

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

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

3 From the **Selection** list, choose **Thin Air Resistive Layers**.

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 **Ice**.

MATERIALS

You can now define the material properties.

Foam

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Foam` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Foam**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.03	W/(m·K)	Basic
Density	rho	25	kg/m³	Basic
Heat capacity at constant pressure	Cp	2	J/(kg·K)	Basic

Content Material

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Content Material` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Content**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	1	W/(m·K)	Basic
Density	rho	2000	kg/m³	Basic
Heat capacity at constant pressure	Cp	800	J/(kg·K)	Basic

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

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

MATERIALS

Air (mat3)

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

2 From the **Selection** list, choose **Empty Space**.

Water

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Water` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Ice**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.6	W/(m·K)	Basic
Density	rho	1000	kg/m³	Basic
Heat capacity at constant pressure	Cp	4200	J/(kg·K)	Basic

Ice

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Ice` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Ice**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	2.3	W/(m·K)	Basic
Density	rho	1000	kg/m³	Basic
Heat capacity at constant pressure	Cp	2050	J/(kg·K)	Basic

Eutectic Plates Package

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Eutectic Plates Package` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

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

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.3	W/(m·K)	Basic
Density	rho	1050	kg/m³	Basic
Heat capacity at constant pressure	Cp	20	J/(kg·K)	Basic
Thickness	lth	300[um]	m	Shell

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

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

MATERIALS

Air Boundaries

1 In the **Settings** window for **Material**, type `Air Boundaries` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

3 From the **Selection** list, choose **Thin Air Resistive Layers**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thickness	lth	50[um]	m	Shell

In the subsequent instructions you will define the ambient temperature using the ASHRAE climate data at a given weather station and set up the domain and boundary conditions.

DEFINITIONS (COMP1)

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 Settings** section.
3. From the **Ambient data** list, choose **Meteorological data (ASHRAE 2021)**.
4. Locate the **Location** section. Click **Set Weather Station**.
5. In the **Weather Station** dialog box, select **Europe>Spain>SEVILLA AP (083910)** in the tree.
6. Click **OK**.
7. In the **Settings** window for **Ambient Properties**, locate the **Time** section.
8. Find the **Local time** subsection. In the table, enter the following settings:

Hour	Minute	Second
10	00	00

HEAT TRANSFER IN SOLIDS (HT)

Phase Change Material 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)>Fluid 1** click **Phase Change Material 1**.
2. In the **Settings** window for **Phase Change Material**, locate the **Phase Change** section.
3. In the $\Delta T_{1 \to 2}$ text field, type `3.5`.
4. Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Ice (mat5)**.
5. Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Water (mat4)**.

Isothermal Domain 1

1. In the **Physics** toolbar, click **Domains** and choose **Isothermal Domain**.
2. In the **Settings** window for **Isothermal Domain**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Empty Space**.

Isothermal Domain Interface 1

1. In the **Model Builder** window, click **Isothermal Domain Interface 1**.

2 In the **Settings** window for **Isothermal Domain Interface**, locate the **Isothermal Domain Interface** section.

3 From the **Interface type** list, choose **Convective heat flux**.

4 In the h text field, type `5`.

Initial Values 1

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

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

3 In the T text field, type `20[degC]`.

Initial Values 2

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

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

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

4 Locate the **Initial Values** section. In the T text field, type `5[degC]`.

Initial Values 3

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

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

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

4 Locate the **Initial Values** section. In the T text field, type `-5[degC]`.

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 **Exterior Surfaces**.

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

5 In the h text field, type `5`.

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

MESH 1

For phase change materials, a finer mesh is necessary in order to accurately model the melting front. In the following steps, build a finer mesh within the ice domains compared to the remaining parts of the box.

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 **Ice**.

Size 1

1 In the **Mesh** toolbar, click **Size Attribute** and choose **Extra Fine**.

2 In the **Settings** window for **Size**, click **Build Selected**.

Free Tetrahedral 2

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

2 Click **Build Mesh**.

The model is now ready for computation.

STUDY 1

Step 1: Time Dependent

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

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

3 From the **Time unit** list, choose **h**.

4 Click **Range**.

5 In the **Range** dialog box, type `3` in the **Step** text field.

6 In the **Stop** text field, type `72`.

7 Click **Replace**.

For more robust convergence, tighten the relative tolerance, which controls the size of the time steps taken by the solver.

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

9 From the **Tolerance** list, choose **User controlled**.

10 In the **Relative tolerance** text field, type `1e-3`.

11 In the **Home** toolbar, click **Compute**.

The first default plot shows the temperature values caught by the previously defined probes during computation. Complete it by displaying the temperature restriction interval.

RESULTS

Content Temperature

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.
2 In the **Settings** window for **1D Plot Group**, type `Content Temperature` 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 of the probes over time`.
5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
6 Select the **y-axis label** check box. In the associated text field, type `Temperature (degC)`.
7 Locate the **Data** section. From the **Dataset** list, choose **Probe Solution 2 (sol1)**.
8 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Content Temperature** node, then click **Probe Table Graph 1**.
2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.
3 From the **Width** list, choose **2**.
4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
5 In the table, enter the following settings:

Legends
`Content minimum`
`Content average`
`Content maximum`
`Ice average`

Content Temperature

In the **Model Builder** window, click **Content Temperature**.

Global 1

1 In the **Content Temperature** toolbar, click **Global**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
2[degC]	degC	Temperature restriction, lower bound

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 From the **Color** list, choose **Blue**.

7 Click to expand the **Legends** section. Clear the **Show legends** check box.

8 In the **Content Temperature** toolbar, click **Plot**.

Content Temperature

In the **Model Builder** window, click **Content Temperature**.

Global 2

1 In the **Content Temperature** toolbar, click **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
8[degC]	degC	Temperature restriction, upper bound

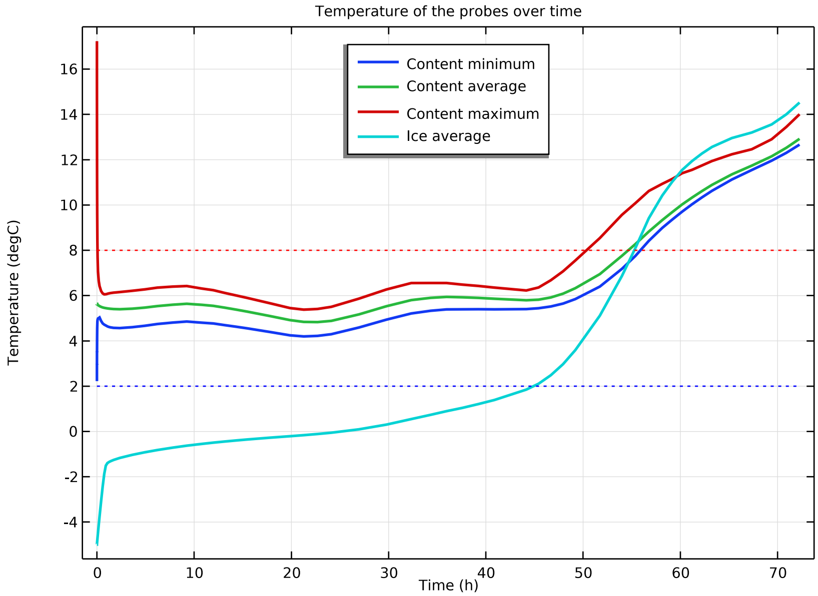
4 Locate the **Title** section. From the **Title type** list, choose **None**.

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

6 From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Clear the **Show legends** check box.

8 In the **Content Temperature** toolbar, click **Plot**.



Temperature (ht)

The second default plot shows the temperature in volume.

1. In the **Model Builder** window, under **Results** click **Temperature (ht)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Time (h)** list, choose **24**.

Domain

1. In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Domain**.
2. In the **Settings** window for **Volume**, locate the **Expression** section.
3. From the **Unit** list, choose **degC**.

Layered Shell

1. In the **Model Builder** window, click **Layered Shell**.
2. In the **Settings** window for **Volume**, locate the **Expression** section.
3. From the **Unit** list, choose **degC**.

The following steps add a plot of the phase change front.

Temperature (ht)

In the **Model Builder** window, click **Temperature (ht)**.

Isosurface 1

1 In the **Temperature (ht)** toolbar, click **Isosurface**.

2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Heat Transfer in Solids>Phase change>ht.alpha12 - Phase transition between phase 1 and phase 2 - 1**.

3 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

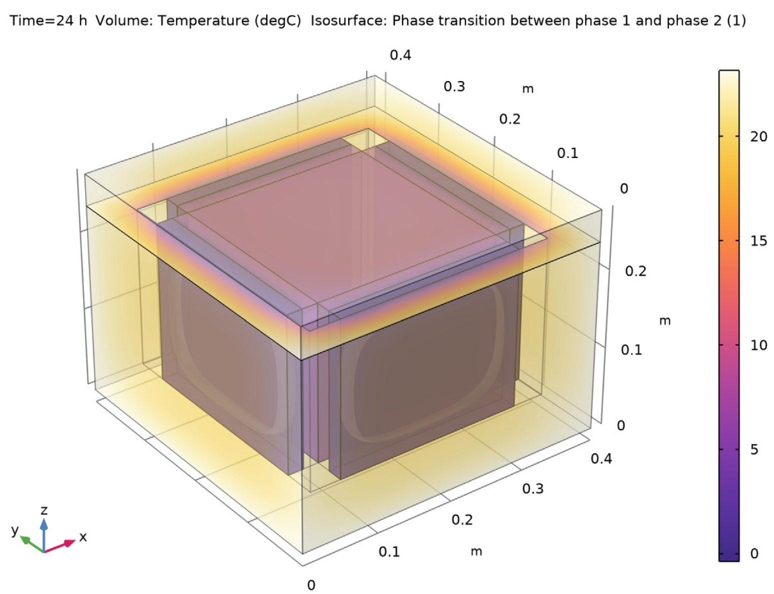
4 In the **Levels** text field, type `0.5`.

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

6 From the **Color** list, choose **White**.

7 Clear the **Color legend** check box.

8 In the **Temperature (ht)** toolbar, click **Plot**.



Volume Average 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Average> Volume Average**.

2 Select Domain 4 only.

3 In the **Settings** window for **Volume Average**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`ht.theta1`	1	

5 Click **Evaluate**.

Ice Fraction

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

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Volume fraction (1)`.

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

6 In the **Title** text area, type `Ice fraction in eutectic plates`.

Table Graph 1

1 Right-click **Ice Fraction** and choose **Table Graph**.

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

3 From the **Table** list, choose **Table 2**.

4 In the **Ice Fraction** toolbar, click **Plot**.

Create a new plot group showing the ambient temperature provided by the built-in climate data.

Ambient Temperature

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

3 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 In the **Ambient Temperature** toolbar, click **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Ambient data>ampr1.T_amb - Ambient temperature - K**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`ampr1.T_amb`	`degC`	`Ambient temperature`

4 In the **Ambient Temperature** toolbar, click **Plot**.