Created in COMSOL Multiphysics 6.2

Spacecraft Thermal Analysis

Introduction

This model demonstrates how to compute satellite temperature over multiple orbit periods by coupling Orbital Thermal Loads to Heat Transfer in Solids. The direct solar, albedo, and Earth infrared thermal loads are computed over a single orbit, and are periodically repeated over multiple orbits.

A 1U CubeSat in a circular 400 km altitude orbit is rotating about its nadir-pointing axis. A geometric model of the satellite considers the frame, boards, sensor, interior instruments, and solar cells.

The solar cells are modeled as having zero thickness to reduce the geometric complexity, and are modeled via a boundary condition that considers the thickness and material properties. The conversion of the incident light into electricity is modeled via an additional heat load that accounts for the nonthermal absorption.

The instruments within the satellite generate heat, and one of the instruments switches to a higher power mode during part of the eclipse period. The objective of the simulation is to predict the temperature over time.

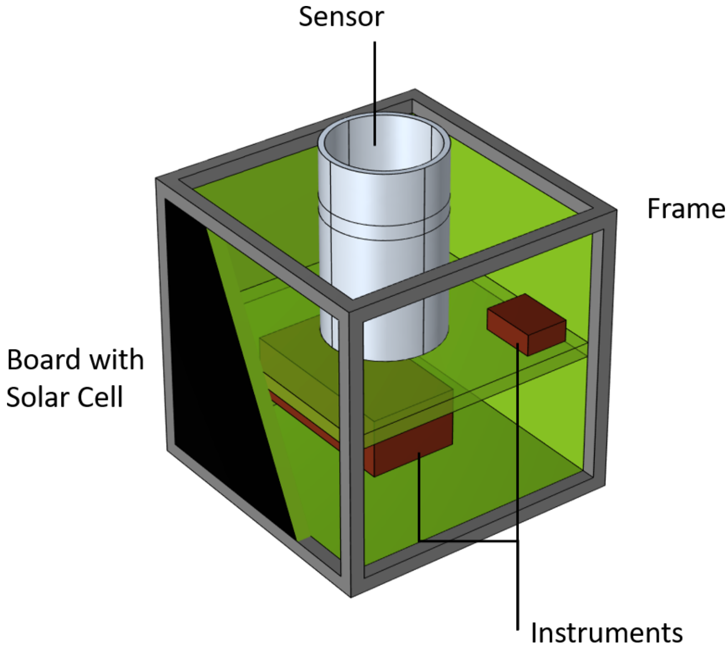

Figure 1: Cutout view of the satellite geometry.

Figure 1 shows the geometry used to model the satellite. An aluminum frame has circuit boards mounted on the sides and within. The outside boards have solar cells on the surface, but these are not explicitly represented in the geometry. Instead, they will be modeled via a boundary condition. The board on the inside has instruments mounted on both sides, as well as a sensor that protrudes through the board on the top. Although the geometry here is simplified, and contains less components than a real satellite, the overall modeling workflow is similar regardless of the complexity of the geometry.

The model uses a combination of the Heat Transfer in Solids interface, which computes the temperature within the solid structure, and the Orbital Thermal Loads, which computes the environmental radiative loads and the heat exchange between surfaces. The environmental loads are computed based upon the orbital parameters as well as the satellite orientation. This is demonstrated in the example Orbit Calculation.

There are three sets of radiative boundary conditions used. For the surfaces facing the exterior of the satellite, the emissivities are specified via the two-band solar and ambient model. The solar cells have an emissivity in the solar band of 0.99 and 0.95 in the ambient band. The solar cells absorb light very well, but convert a fraction of the incident light in the solar band into electrical energy rather than heat, this is accounted for via a boundary condition in the Heat Transfer in Solids interface. The remaining exterior surfaces have an emissivity of 0.2 in the solar band, and 0.85 in the ambient. This represents a surface coating that reflects well in the solar band, and emits well in the ambient band, to keep the satellite as cool as possible. For radiative heat exchange within the interior of the satellite, a constant emissivity of 0.8 is used for all surfaces to model radiative heat exchange within the interior.

The two instruments mounted on the board within the satellite are modeled as solid copper, with a heat load uniformly distributed over the volume. The smaller instrument dissipates 1 W continuously, and the larger dissipates 0.5 W, but during the eclipse switches to a higher-power mode, dissipating 5 W. The higher-power mode begins 20 minutes after entering eclipse, and lasts for 15 minutes. The switching of magnitude is controlled via the Events interface.

The solar cells are modeled via the Thin Layer boundary condition which accounts for heat transfer through the thickness as well as along the surface. To account for the conversion of light into electric energy rather than thermal energy, a heat load of negative magnitude is applied, which is equivalent to reducing the absorbed environmental heat load in the solar band at those surfaces.

The instruments and sensors are mounted onto the interior board, and this mounting is approximated via a surface resistance specified in a Thermal Contact boundary condition. This introduces a jump in temperature across the boundary. At all other mating boundaries the temperature field is continuous between parts.

The solution procedure involves two study steps. In the first step, the environmental loads are computed over a single orbit, since it is assumed that these loads will not change significantly over several orbits. In the second step, the loads are periodically repeated over four orbits.

Figure 2 displays the temperature of the satellite after 4 orbit periods.

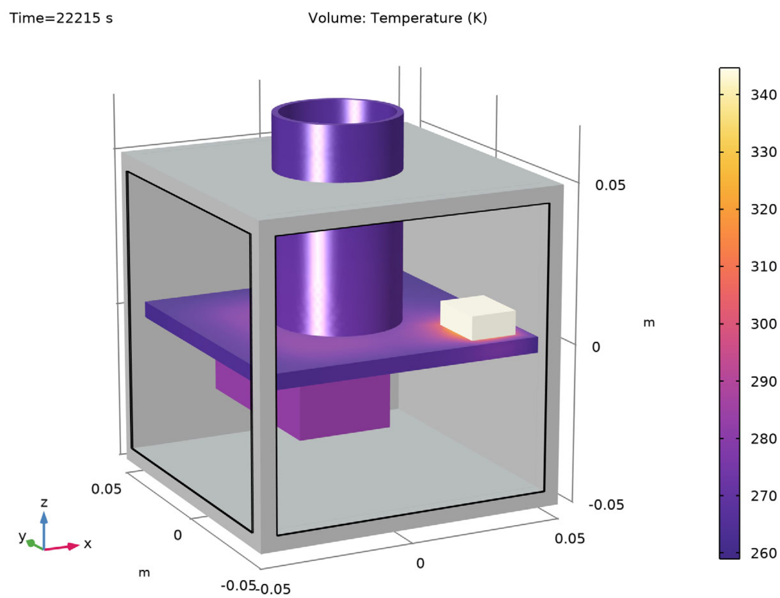

Figure 2: Temperature field on the sensor, instruments, and interior board.

[Figure 3](#) shows the evolution of the maximum and minimum temperature over time. It shows that it takes several orbits for the solution to become periodic.

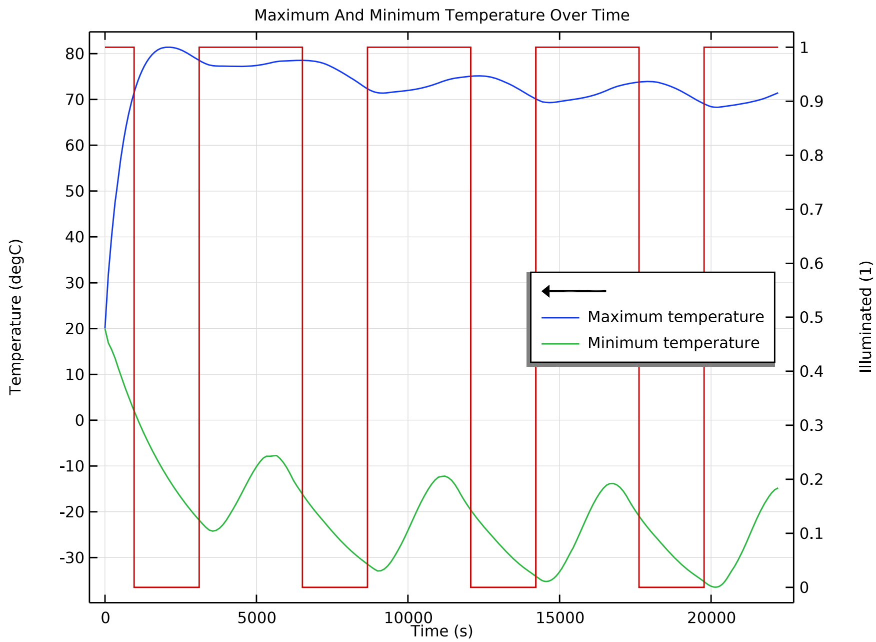

Figure 3: Maximum and minimum temperature on the satellite boundaries over time. The red line indicates when the satellite is exposed to the Sun.

Application Library path: `Heat_Transfer_Module/Orbital_Thermal_Loads/spacecraft_thermal_analysis`

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>Radiation>Heat Transfer with Orbital Thermal Loads**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Events (ev)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Orbital Thermal Loads>Orbital Temperature**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`Efficiency`	`0.3`	0.3	`Solar cell efficiency`

GEOMETRY 1

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

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

2 Right-click **Geometry 1** and choose **Insert Sequence**.

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

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

Now that the geometry is imported, define selections to reuse for the material and feature selections.

DEFINITIONS

Define a selection consisting of all exterior boundaries of all domains. Radiation from all of these faces will be considered.

All Radiative Boundaries

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

2 Right-click **Definitions** and choose **Selections>Explicit**.

3 In the **Settings** window for **Explicit**, type `All Radiative Boundaries` in the **Label** text field.

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

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

Define a selection consisting of all boundaries that are facing the exterior void. These are the boundaries that can be exposed to environmental loads.

Space-Facing Boundaries

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `0` in the **Selection** text field.

5 Click **OK**.

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

7 From the **Output entities** list, choose **Adjacent boundaries**.

8 In the **Label** text field, type `Space-Facing Boundaries`.

Solar Cells

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

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

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5 12 39 83` in the **Selection** text field.

6 Click **OK**.

Add operators to evaluate maximum and minimum temperature over the model.

Maximum on All Domains

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

2 In the **Settings** window for **Maximum**, type `Maximum on All Domains` in the **Label** text field.

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

Minimum on All Domains

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Minimum**.

2 In the **Settings** window for **Minimum**, type `Minimum on All Domains` in the **Label** text field.

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

ADD MATERIAL

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

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

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

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

5 In the tree, select **Built-in>Copper**.

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

7 In the tree, select **Built-in>FR4 (Circuit Board)**.

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

9 In the tree, select **Built-in>Silicon**.

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

11 In the tree, select **Built-in>Titanium beta-21S**.

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

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat2)**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `8 10` in the **Selection** text field.

5 Click **OK**.

FR4 (Circuit Board) (mat3)

1 In the **Model Builder** window, click **FR4 (Circuit Board) (mat3)**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2-7, 11` in the **Selection** text field.

5 Click **OK**.

Silicon (mat4)

1 In the **Model Builder** window, click **Silicon (mat4)**.
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 **Solar Cells**.

Titanium beta-21S (mat5)

1 In the **Model Builder** window, click **Titanium beta-21S (mat5)**.
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `9` in the **Selection** text field.
5 Click **OK**.

The larger instrument dissipates 0.5 W, but it switches to a higher-power mode for 15 minutes during the eclipse, dissipating 5 W.

HEAT TRANSFER IN SOLIDS (HT)

Heat Source 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Heat Transfer in Solids (ht)** and choose **Heat Source**.
2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `8` in the **Selection** text field.
5 Click **OK**.

HS1 is an indicator controlled by the Events interface. It's enabled 20 minutes after entering eclipse, and lasts for 15 minutes.

6 In the **Settings** window for **Heat Source**, locate the **Heat Source** section.
7 From the **Heat source** list, choose **Heat rate**.
8 In the P_0 text field, type `0.5[W]+4.5[W]*HS1`.

Add a heat source on the smaller instrument that dissipates 1 W continuously.

Heat Source 2

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2 Select Domain 10 only.
3 In the **Settings** window for **Heat Source**, locate the **Heat Source** section.

4 From the **Heat source** list, choose **Heat rate**.

5 In the P_0 text field, type `1[W]`.

The solar cells are modeled via the Thin Layer boundary condition, of type General, which accounts for heat transfer through the thickness as well as along the surface.

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 **Solar Cells**.

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

MATERIALS

Silicon (mat4)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Silicon (mat4)** node, then click **Silicon (mat4)**.

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	0.5[mm]	m	Shell

4 Locate the **Orientation and Position** section. From the **Position** list, choose **Bottom side on boundary**.

The instruments and sensors are mounted onto the interior board, and this mounting is approximated via a surface resistance specified in a Thermal Contact boundary condition. This introduces a jump in temperature across the boundary.

HEAT TRANSFER IN SOLIDS (HT)

Thermal Contact 1

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

2 Select Boundaries 43, 47, and 67 only.

3 In the **Settings** window for **Thermal Contact**, locate the **Thermal Contact** section.

4 From the **Contact model** list, choose **Equivalent thin resistive layer**.

5 In the R_{eq} text field, type `0.01[K*m^2/W]`.

6 In the text field, type `0[W/(m^2*K)]`.

To account for the conversion of light into electric energy rather than thermal energy. The corresponding energy loss is modeled using a heat flux condition on the solar cells boundaries.

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 **Solar Cells**.

4 Locate the **Heat Flux** section. In the q_0 text field, type `-otl.Grad_band1*Efficiency`.

ORBITAL THERMAL LOADS (OTL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Orbital Thermal Loads (otl)**.

2 In the **Settings** window for **Orbital Thermal Loads**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All Radiative Boundaries**.

Sun Properties 1

By default, the Sun vector and solar flux are set to correspond to the winter solstice.

Planet Properties 1

1 In the **Model Builder** window, click **Planet Properties 1**.

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

3 Find the **Planet initial position** subsection. From the **Planet longitude at start time** list, choose **Longitude at subspacecraft point**.

The Earth properties are used to define the planet.

4 Locate the **Radiative Properties** section. From the **Albedo** list, choose **User-defined distribution**.

The proportion of solar radiation incident onto Earth reflected diffusely back toward the satellite, or albedo, is set to 0.3 everywhere on the planet.

5 In the $\alpha_{0,\lambda}$ text field, type `0.3`.

Earth is relatively warm compared to background deep space and is modeled as a distributed emitter having a uniform flux of 225 W/m^2.

6 From the **Planet radiative flux** list, choose **User-defined for each band**.

7 In the table, enter the following settings:

Spectral band	**Planet radiative flux (W/m^2)**
Ambient: [2.5[um], +infinity[	`225[W/m^2]`

Orbital Parameters 1

A circular orbit is defined at 400 km altitude, inclination of 50°, and local time at ascending node set to 15 h.

1. In the **Model Builder** window, click **Orbital Parameters 1**.
2. In the **Settings** window for **Orbital Parameters**, locate the **Orbital Parameters** section.
3. From the **Orbit type** list, choose **Circular**.
4. In the R text field, type `otl.R_planet+400[km]`.
5. In the i text field, type `50[deg]`.
6. From the **Ascending node** list, choose **Local time at ascending node**.
7. In the t_Ω text field, type `15[h]`.

Spacecraft Orientation 1

The orientation of the satellite is such that the primary direction points toward Earth. The satellite is slowly rotating about its primary axis, so the secondary axis can be any vector that is not parallel to nadir. In this case the default setting, of the +X direction corresponding to the direction of travel, is used.

1. In the **Model Builder** window, click **Spacecraft Orientation 1**.
2. In the **Settings** window for **Spacecraft Orientation**, locate the **Spacecraft Orientation** section.
3. Find the **Rotations** subsection. From the **Rotation about primary axis** list, choose **Angular rate**.
4. In the ω text field, type `2*360[deg]/otl.T_orbit`.

Diffuse Surface, Inside

Now, define the radiative properties of the different boundaries.

1. In the **Model Builder** window, under **Component 1 (comp1)>Orbital Thermal Loads (otl)** click **Diffuse Surface 1**.
2. In the **Settings** window for **Diffuse Surface**, type `Diffuse Surface, Inside` in the **Label** text field.
3. Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.8`.

Diffuse Surface, Outside

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

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

3 From the **Selection** list, choose **Space-Facing Boundaries**.

4 In the **Label** text field, type `Diffuse Surface, Outside`.

5 Locate the **Surface Emissivity** section. From the ε list, choose **User-defined for each band**.

6 In the table, enter the following settings:

Spectral band	Emissivity (1)
Solar: [0, 2.5[um][	0.2
Ambient: [2.5[um], +infinity[	0.85

Diffuse Surface, Solar Cells

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

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

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

4 In the **Label** text field, type `Diffuse Surface, Solar Cells`.

5 Locate the **Surface Emissivity** section. From the ε list, choose **User-defined for each band**.

6 In the table, enter the following settings:

Spectral band	Emissivity (1)
Solar: [0, 2.5[um][	0.99
Ambient: [2.5[um], +infinity[	0.95

The following features need to be added to control the switching of magnitude of the heat source in the larger instrument when entering the eclipse.

EVENTS (EV)

In the **Model Builder** window, under **Component 1 (comp1)** click **Events (ev)**.

Instrument 1, State Variable

1 In the **Physics** toolbar, click **Global** and choose **Discrete States**.

2 In the **Settings** window for **Discrete States**, type `Instrument 1, State Variable` in the **Label** text field.

3 Locate the **Discrete States** section. In the table, enter the following settings:

Name	Initial value (u0)	Description
HS1	0	Heat Source Indicator

Instrument 1, Turn On

1 In the **Physics** toolbar, click **Global** and choose **Explicit Event**.

2 In the **Settings** window for **Explicit Event**, type `Instrument 1, Turn On` in the **Label** text field.

3 Locate the **Event Timings** section. In the t_i text field, type `otl.t_inEclipse+20[min]`.

4 In the T text field, type `otl.T_eclipse`.

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

Variable	Expression
HS1	1

Instrument 1, Turn Off

1 In the **Physics** toolbar, click **Global** and choose **Explicit Event**.

2 In the **Settings** window for **Explicit Event**, type `Instrument 1, Turn Off` in the **Label** text field.

3 Locate the **Event Timings** section. In the t_i text field, type `otl.t_inEclipse+20[min]+15[min]`.

4 In the T text field, type `otl.T_eclipse`.

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

Variable	Expression
HS1	0

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra coarse**.

4 Click **Build All**.

The solution procedure involves two study steps. In the first step, the environmental loads are computed over a single orbit, since it is assumed that these loads will not change significantly over several orbits. In the second step, the loads are periodically repeated over four orbits.

STUDY 1

Step 1: Orbit Thermal Loads

1 In the **Model Builder** window, under **Study 1** click **Step 1: Orbit Thermal Loads**.

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

3 In the **Output orbit times** text field, type `range(0,0.02,1)`.

Step 2: Orbital Temperature

1 In the **Model Builder** window, click **Step 2: Orbital Temperature**.

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

3 In the **Output orbit times** text field, type `range(0,0.02,4)`.

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

RESULTS

Selection 1

1 In the **Model Builder** window, expand the **Temperature (ht)** node.

2 Right-click **Domain** and choose **Selection**.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5 8 9 10` in the **Selection** text field.

6 Click **OK**.

Layered Shell

In the **Model Builder** window, under **Results>Temperature (ht)** right-click **Layered Shell** and choose **Delete**.

Surface 1

1 In the **Model Builder** window, right-click **Temperature (ht)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Selection 1

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1, 4, 6, 7, 11` in the **Selection** text field.

6 Click **OK**.

Material Appearance 1

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

Temperature (ht)

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

2 Clear the **Plot dataset edges** check box.

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

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

5 Click **Plot**.

Finally, plot the evolution of the minimum and maximum temperature of the satellite over time.

Maximum And Minimum Temperature Over Time

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Label**.

4 In the **Label** text field, type `Maximum And Minimum Temperature Over Time`.

5 Locate the **Plot Settings** section. Select the **Two y-axes** check box.

6 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

7 Locate the **Plot Settings** section.

8 Select the **y-axis label** check box. In the associated text field, type `Temperature (degC)`.

Global 1

1 Right-click **Maximum And Minimum Temperature Over Time** and choose **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**
maxop1(T)	degC	Maximum temperature
minop1(T)	degC	Minimum temperature

Global 2

1. In the **Model Builder** window, right-click **Maximum And Minimum Temperature Over Time** and choose **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
`otl.isIlluminated`	1	Illuminated

4. Locate the **y-Axis** section. Select the **Plot on secondary y-axis** check box.
5. Click to expand the **Legends** section. Clear the **Show legends** check box.

Maximum And Minimum Temperature Over Time

1. In the **Model Builder** window, click **Maximum And Minimum Temperature Over Time**.
2. In the **Maximum And Minimum Temperature Over Time** toolbar, click **Plot**.

Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

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 `10[cm]`.
4. In the **Depth** text field, type `10[cm]`.
5. In the **Height** text field, type `10[cm]`.
6. Locate the **Position** section. From the **Base** list, choose **Center**.

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 `9[cm]`.
4. In the **Depth** text field, type `9[cm]`.
5. In the **Height** text field, type `9[cm]`.
6. Locate the **Position** section. From the **Base** list, choose **Center**.

Difference 1 (dif1)

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

2 Select the object **blk1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **blk2** only.

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 `9[cm]`.

4 In the **Depth** text field, type `9[cm]`.

5 In the **Height** text field, type `5[mm]`.

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

7 In the **z** text field, type `4.75[cm]`.

Mirror 1 (mir1)

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

2 Select the object **blk3** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

Rotate 1 (rot1)

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

2 Select the objects **blk3** and **mir1** only.

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

4 Select the **Keep input objects** check box.

5 Locate the **Rotation** section. From the **Axis type** list, choose **y-axis**.

6 In the **Angle** text field, type `90`.

Rotate 2 (rot2)

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

2 Select the objects **rot1(1)** and **rot1(2)** only.

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

4 Select the **Keep input objects** check box.

5 Locate the **Rotation** section. In the **Angle** text field, type `90`.

Difference 2 (dif2)

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

2 Select the object **dif1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **blk3**, **mir1**, **rot1(1)**, **rot1(2)**, **rot2(1)**, and **rot2(2)** only.

6 Select the **Keep objects to subtract** check box.

Block 4 (blk4)

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 `9[cm]`.

4 In the **Depth** text field, type `9[cm]`.

5 In the **Height** text field, type `5[mm]`.

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

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 `3[cm]`.

4 In the **Depth** text field, type `6[cm]`.

5 In the **Height** text field, type `2[cm]`.

6 Locate the **Position** section. In the **x** text field, type `-3[cm]`.

7 In the **y** text field, type `-3.5[cm]`.

8 In the **z** text field, type `-2.25[cm]`.

Block 6 (blk6)

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 `1.5[cm]`.

4 In the **Depth** text field, type `2[cm]`.

5 In the **Height** text field, type `0.75[cm]`.

6 Locate the **Position** section. In the **x** text field, type `2.5[cm]`.

7 In the **y** text field, type `-4[cm]`.

8 In the **z** text field, type `0.25[cm]`.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `2[cm]`.

4 In the **Height** text field, type `6.5[cm]`.

5 Locate the **Position** section. In the **z** text field, type `2.5[mm]`.

Difference 3 (dif3)

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

2 Select the object **blk3** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **cyl1** only.

6 Select the **Keep objects to subtract** check box.

Cylinder 2 (cyl2)

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

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

3 In the **Radius** text field, type `1.8[cm]`.

4 In the **Height** text field, type `6.5[cm]`.

5 Locate the **Position** section. In the **z** text field, type `2[cm]`.

Difference 4 (dif4)

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

2 Select the object **cyl1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **cyl2** only.

Form Union (fin)

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