Created in COMSOL Multiphysics 6.0

Axisymmetric Transient Heat Transfer

Introduction

This example shows an axisymmetric transient thermal analysis with a step change to 1000°C at time 0. The example is taken from a NAFEMS benchmark collection (Ref. 1).

Model Definition

This example considers the 0.3 m-by- 0.4 m domain. For the boundary conditions, assume the following:

- The left boundary is the symmetry axis.
- The other boundaries have a temperature of 1000°C. The entire domain is at 0°C at the start, which represents a step change in temperature at the boundaries.

In the domain use the following material properties:

- The density, ρ, is 7850 kg/m^3
- The heat capacity is 460 J/(kg·°C)
- The thermal conductivity is 52 W/(m·°C).

The benchmark case is described with a simulation time of 190 s.

This models doubles the simulation with two scenarios:

1. the temperature condition of 1000°C is maintained during all the simulation.
2. at t = 190 s, the temperature condition is replaced by a thermal insulation condition.

Results

The following revolved surface plot shows the temperature distribution inside the cylinder after 190 seconds:

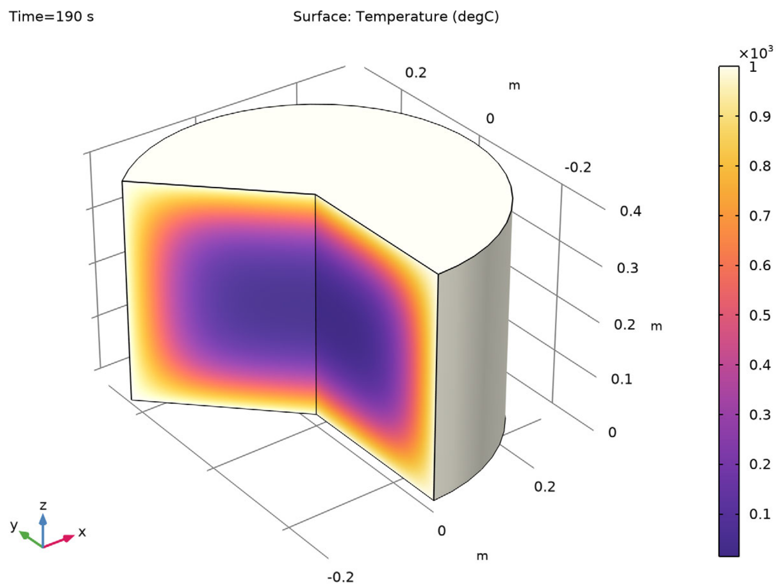


Figure 1: Temperature distribution after 190 seconds.

The benchmark result for the target location ($t = 190$ s, $r = 0.1$ m and $z = 0.3$ m) is a temperature of 186.5°C. The COMSOL Multiphysics model, using a default mesh with about 430 elements, gives a temperature close to 186.5°C.

The line graph below shows the temperature variation during 380 s at the target location ($r = 0.1$ m and $z = 0.3$ m) for the two scenarios.

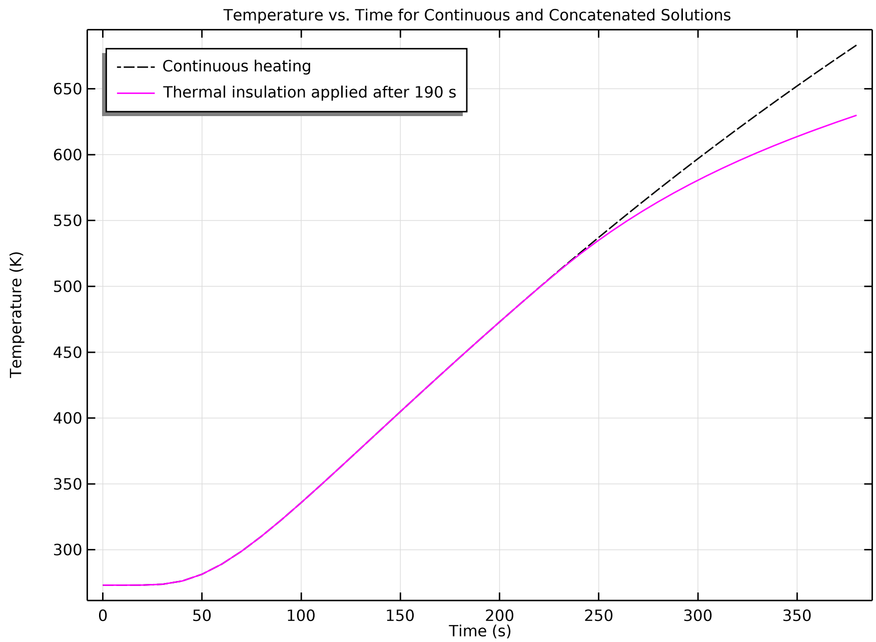


Figure 2: Temperature variation at r = 0.1 m and z = 0.3 m for continuous heating and for thermal insulation after 190 s.

Reference

1. A.D. Cameron, J.A. Casey, and G.B. Simpson, *NAFEMS Benchmark Tests for Thermal Analysis (Summary)*, NAFEMS, Glasgow, 1986.

Application Library path: `COMSOL_Multiphysics/Heat_Transfer/heat_transient_axi`

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 **2D Axisymmetric**.

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**.

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.3`.

4 In the **Height** text field, type `0.4`.

5 Click **Build All Objects**.

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

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

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

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

4 Locate the **Temperature** section. In the T_0 text field, type `1000[degC]`.

Solid 1

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

2 In the **Settings** window for **Solid**, locate the **Heat Conduction, Solid** section.

3 From the k list, choose **User defined**. In the associated text field, type `52`.

4 Locate the **Thermodynamics, Solid** section. From the ρ list, choose **User defined**. In the associated text field, type `7850`.

5 From the C_p list, choose **User defined**. In the associated text field, type `460`.

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 `0[degC]`.

STUDY 1

Time Dependent - Continuous Simulation (with Heating)

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

2 In the **Settings** window for **Time Dependent**, type `Time Dependent - Continuous Simulation (with Heating)` in the **Label** text field.

3 Locate the **Study Settings** section. In the **Output times** text field, type `range(0,10,380)`.

To improve time accuracy, lower the default solver tolerance:

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

5 In the **Relative tolerance** text field, type `1e-5`.

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

RESULTS

Temperature, 3D (ht)

To get the plot shown in Figure 1, just change the unit as follows:

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

2 From the **Time (s)** list, choose **190**.

Surface

1 In the **Model Builder** window, expand the **Temperature, 3D (ht)** node, then click **Surface**.

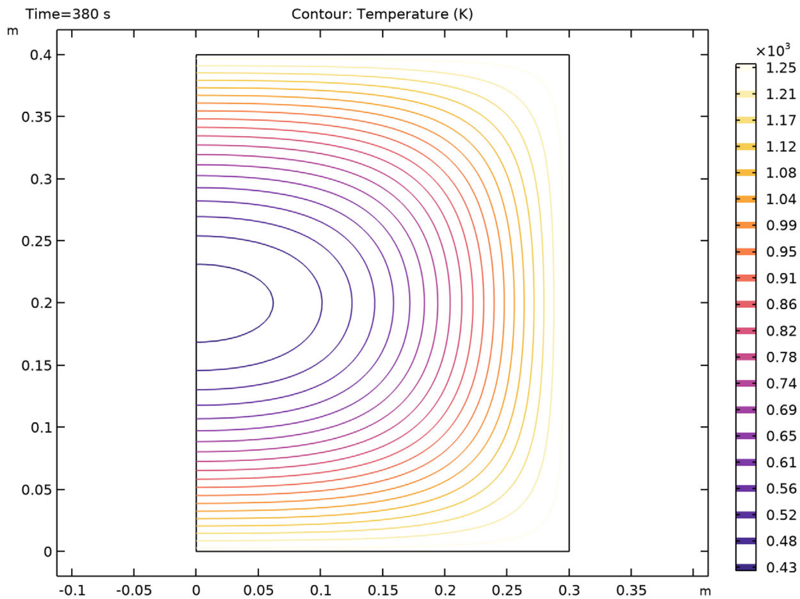
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

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

Isothermal Contours (ht)

The second default plot group visualizes the temperature field using a contour plot.



The benchmark value for the temperature at t = 190 s, r = 0.1 m and z = 0.3 m is 186.5°C. To compare the value from the simulation, evaluate the temperature at this position.

Cut Point 2D 1

1. In the **Results** toolbar, click **Cut Point 2D**.
2. In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.
3. In the **R** text field, type `0.1`.
4. In the **Z** text field, type `0.3`.

Point Evaluation 1

1. In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.
2. In the **Settings** window for **Point Evaluation**, locate the **Data** section.
3. From the **Dataset** list, choose **Cut Point 2D 1**.
4. From the **Time selection** list, choose **From list**.
5. In the **Times (s)** list, select **190**.

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

Expression	Unit	Description
`T`	`degC`	`Temperature`

7 Click **Evaluate**.

As an optional extension of the model, you can add a study sequence where, starting from 190 s, the boundaries are thermally insulated.

ADD STUDY

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

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

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

4 Click **Add Study** in the window toolbar.

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

STUDY 2

Time Dependent - First Part (with Heating)

1 In the **Settings** window for **Time Dependent**, type `Time Dependent - First Part (with Heating)` in the **Label** text field.

2 Locate the **Study Settings** section. In the **Output times** text field, type `range(0,10,190)`.

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

4 In the **Relative tolerance** text field, type `1e-5`.

Time Dependent - Second Part (with Insulation)

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent> Time Dependent**.

2 In the **Settings** window for **Time Dependent**, type `Time Dependent - Second Part (with Insulation)` in the **Label** text field.

3 Locate the **Study Settings** section. In the **Output times** text field, type `range(190,10,380)`.

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

5 In the **Relative tolerance** text field, type `1e-5`.

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

7 In the tree, select **Component 1 (Comp1)>Heat Transfer in Solids (Ht)>Temperature 1**.

8 Click **Disable**.

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

To combine the two time-dependent simulations, add a **Combine Solutions** study step. This concatenates the two solutions and makes it possible to treat the output as a single continuous time-dependent solution.

Combine Solutions

1 In the **Study** toolbar, click **Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 From the **First solution** list, choose **Study 2/Solution Store 1 (sol3)**.

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

RESULTS

Surface

1 In the **Model Builder** window, expand the **Temperature, 3D (ht) 1** node, then click **Surface**.

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

3 From the **Unit** list, choose **degC**.

4 In the **Temperature, 3D (ht) 1** toolbar, click **Plot**.

Cut Point 2D - Continuous Heating

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Point 2D 1**.

2 In the **Settings** window for **Cut Point 2D**, type `Cut Point 2D - Continuous Heating` in the **Label** text field.

Cut Point 2D - Combined Solutions

1 In the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, type `Cut Point 2D - Combined Solutions` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Point Data** section. In the **R** text field, type `0.1`.

5 In the **Z** text field, type `0.3`.

Join - Temperature Difference

1 In the **Results** toolbar, click **More Datasets** and choose **Join**.

2 In the **Settings** window for **Join**, type `Join - Temperature Difference` in the **Label** text field.

3 Locate the **Data 1** section. From the **Data** list, choose **Cut Point 2D - Continuous Heating**.

4 Locate the **Data 2** section. From the **Data** list, choose **Cut Point 2D - Combined Solutions**.

Temperature, 1D

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

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

5 In the **Title** text area, type `Temperature vs. Time for Continuous and Concatenated Solutions`.

Point Graph 1

1 Right-click **Temperature, 1D** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Cut Point 2D - Continuous Heating**.

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

5 From the **Color** list, choose **Black**.

Point Graph 2

1 In the **Model Builder** window, right-click **Temperature, 1D** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Cut Point 2D - Combined Solutions**.

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

Point Graph 1

1 In the **Model Builder** window, click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Continuous heating`

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

2 In the **Settings** window for **Point Graph**, locate the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Thermal insulation applied after 190 s`

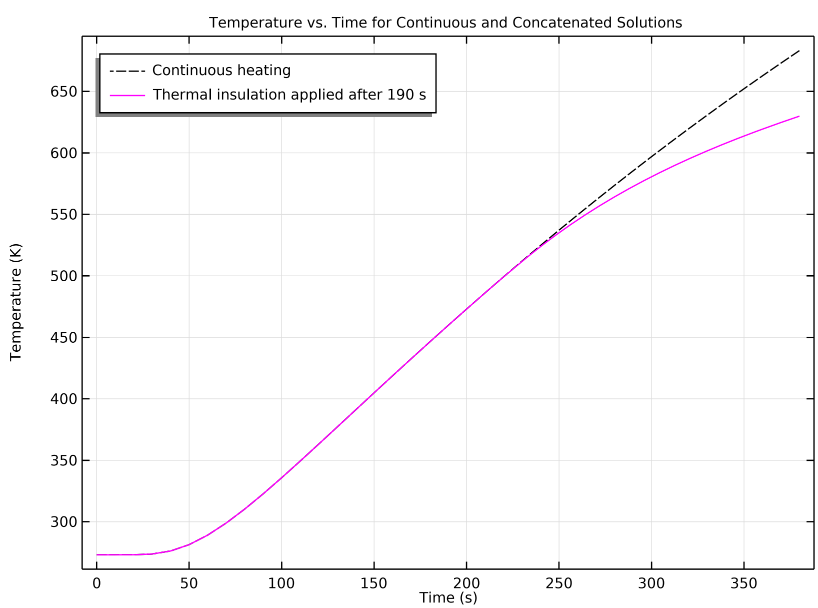
Temperature, 1D

1 In the **Model Builder** window, click **Temperature, 1D**.

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

3 From the **Position** list, choose **Upper left**.

4 In the **Temperature, 1D** toolbar, click **Plot**.



Temperature Difference, 1D

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Temperature Difference, 1D` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Join - Temperature Difference**.
4. Locate the **Title** section. From the **Title type** list, choose **Manual**.
5. In the **Title** text area, type `Temperature Difference`.

Point Graph 1

1. Right-click **Temperature Difference, 1D** and choose **Point Graph**.
2. In the **Temperature Difference, 1D** toolbar, click **Plot**.