

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/m³
- 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:

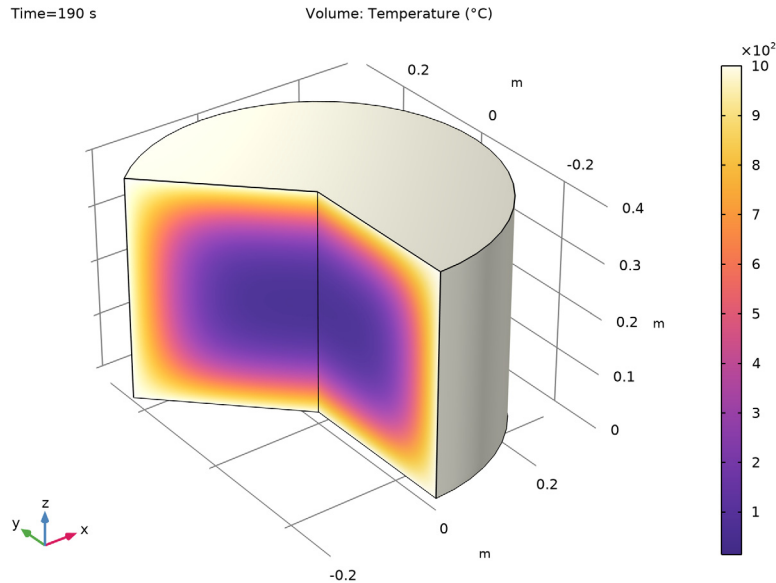


Figure 1: Temperature distribution after 190 seconds.

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.

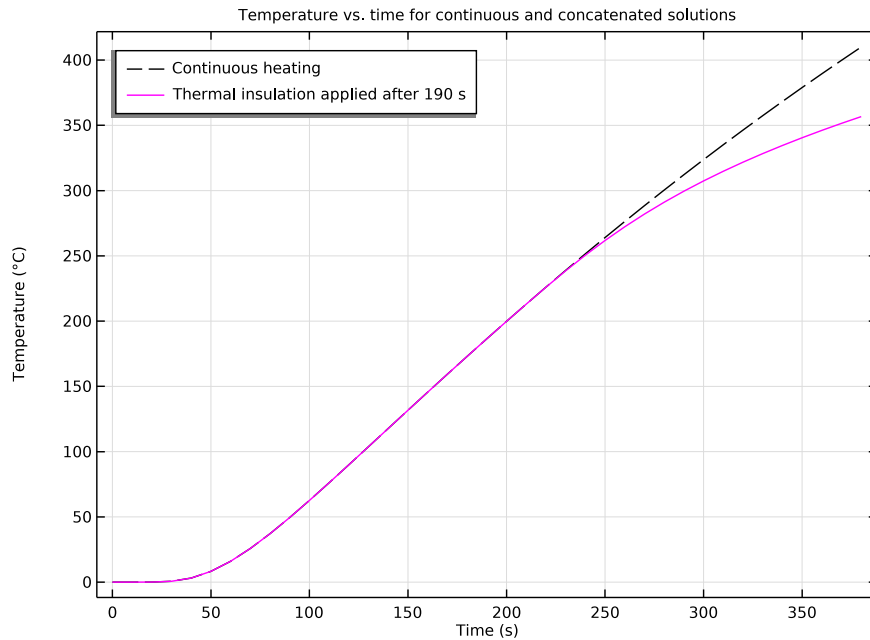


Figure 2: Temperature variation at $r = 0.1$ m and $z = 0.3$ m for continuous heating and for thermal insulation after 190 s.

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 I

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 **Physics** toolbar, click  **Boundaries** 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 I

- 1 In the **Model Builder** window, click **Initial Values I**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the T text field, type 0[degC].

STUDY I

Time Dependent - Continuous Simulation (with Heating)

- 1 In the **Model Builder** window, under **Study I** 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 **Study** toolbar, click  **Compute**.

Change the unit of the temperature results to degrees Celsius.

RESULTS

Preferred Units I

- 1 In the **Results** toolbar, click  **Configurations** and choose **Preferred Units**.
- 2 In the **Settings** window for **Preferred Units**, locate the **Units** section.
- 3 Click  **Add Physical Quantity**.
- 4 In the **Physical Quantity** dialog, select **General > Temperature (K)** in the tree.
- 5 Click **OK**.
- 6 In the **Settings** window for **Preferred Units**, locate the **Units** section.
- 7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Temperature	K	°C

- 8 Click  **Apply**.

Temperature (ht)

The default plot shows the 2D distribution of temperature.

- 1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.
- 2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **190**.

To get the plot shown in [Figure 1](#), add a result template for 3D Temperature distribution:

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) > Heat Transfer in Solids > Temperature (ht)**.
- 4 Click the **Add Result Template** button in the window toolbar.

RESULTS

Temperature 3D

- 1 In the **Settings** window for **3D Plot Group**, type **Temperature 3D** in the **Label** text field.
- 2 Locate the **Data** section. From the **Time (s)** list, choose **190**.

Volume 1

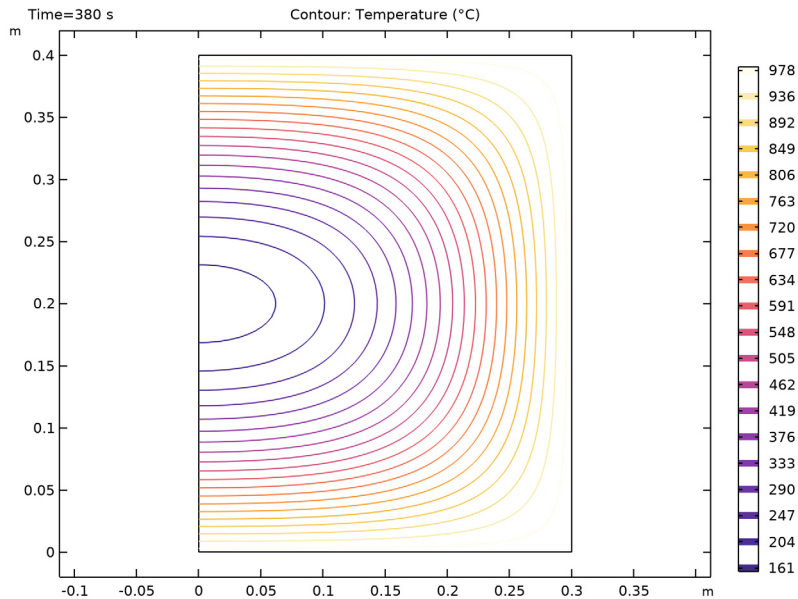
Add another result template to visualize the temperature field using a contour plot.

RESULT TEMPLATES

- 1 Go to the **Result Templates** window.
- 2 In the tree, select **Study 1/Solution 1 (sol1) > Heat Transfer in Solids > Isothermal Contours (ht)**.
- 3 Click the **Add Result Template** button in the window toolbar.
- 4 From the **Results** menu, choose **Result Templates**.

RESULTS

Isothermal Contours (ht)



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 I

- 1 In the **Model Builder** window, expand the **Temperature 3D** node.
- 2 Right-click **Results** > **Datasets** and choose **Cut Point 2D**.
- 3 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.
- 4 In the **R** text field, type 0.1.
- 5 In the **Z** text field, type 0.3.

Point Evaluation I

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

- 6 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 the **Add Study** button 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** checkbox.
- 7 In the tree, select **Component 1 (comp1)** > **Heat Transfer in Solids (ht)** > **Temperature 1**.
- 8 Click  **Disable**.

Temperature (ht) 1

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.

Step 3: 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

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, ID

- 1 In the **Results** toolbar, click  **ID Plot Group**.

- 2 In the **Settings** window for **ID 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 **From theme**.

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

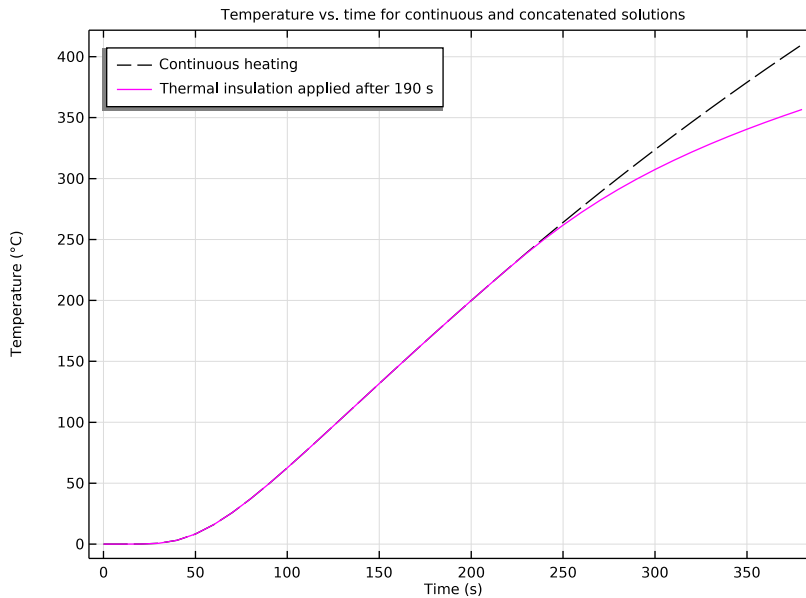
5 In the table, enter the following settings:

Legends

Thermal insulation applied after 190 s

Temperature, 1D

- 1 In the **Model Builder** window, click **Temperature, 1D**.
- 2 In the **Settings** window for **ID 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 **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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, ID** and choose **Point Graph**.
- 2 In the **Temperature Difference, ID** toolbar, click  **Plot**.
- 3 In the **Model Builder** window, click **Point Graph 1**.

