

Heat Conduction with a Localized Heat Source on a Disk

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This classical verification example solves the steady-state temperature distribution in a plane disk heated by a localized heat source at its center. It shows and compares different ways to define a heat source localized on a small domain by representing it either as a geometrical point or as a small disk.

Both modelings have analytical solutions to which the obtained numerical results can be compared. The results bring guidelines to select the suitable option depending on the ratio of source to surrounding geometry typical size.

Model Definition

The model computes the temperature field on a cork disk of radius $R_{\text{disk}} = 0.1 \text{ m}$. A fixed temperature $T_0 = 300 \text{ K}$ is set on the disk boundary, and a heat source of total power P = 1 W is applied on a small circular area (radius $R_{\text{source}} = 10^{-2} \text{ m}$) centered at the origin.



Figure 1: Geometry and boundary conditions.

The model configuration shows an axial symmetry which implies that the temperature profile is only a function of the distance to the center of the disks. In cylindrical coordinates it means that the temperature profile is a function of r only: $T(r, \theta) = T(r, 0)$. Despite the fact that this tutorial can be set up using a 1D axisymmetric geometry, defining it in 2D makes its extension to non-axial symmetric cases easier. Recall that Cartesian coordinates (x, y) and cylindrical coordinates (r, θ) are related by:

$$r^{2} = x^{2} + y^{2}$$
$$\theta = \operatorname{atan}\left(\frac{y}{x + \sqrt{x^{2} + y^{2}}}\right)$$

In this document both coordinate systems are used jointly.

PUNCTUAL HEAT SOURCE MODEL

In order to simplify the geometry and to avoid high aspect ratio when R_{source} is significantly smaller than R_{disk} , represent the source as a punctual source applied on the origin point. This model corresponds to the following equation with a singular source term, with the following formal formulation:

$$\begin{cases} \nabla \cdot (-k\nabla T) = Q\delta & \text{in the disk domain} \\ T = T_0 & \text{on the disk boundary} \end{cases}$$

where k is the thermal conductivity, $Q = P/d_z$ is the volumetric heat source, d_z is the outof-plane thickness and δ is the Dirac distribution centered at the origin. The solution of this equation is:

$$T(r) = T_0 - \frac{Q}{2\pi k} \ln\left(\frac{r}{R_{\text{disk}}}\right)$$
(1)

According to Equation 1, the temperature goes to $\pm \infty$ when approaching the origin (r = 0). This singularity is illustrated on Figure 2 where the temperature value increases indefinitely when refining the mesh.



Figure 2: Distribution of relative temperature on the disk with a point heat source at the center.

VOLUME HEAT SOURCE

You can also model the heat source explicitly and apply it on a disk of radius R_{source} around the origin. Then, the formulation of the problem to solve is:

$$\begin{cases} \nabla \cdot (-k\nabla T) = f & \text{in the disk domain} \\ T = T_0 & \text{on the disk boundary} \end{cases}$$

where f is a smoothed heat source distribution defined by

$$f(r) = \begin{cases} \frac{Q}{\pi R_{\text{source}}^2} & \text{if } r < R_{\text{source}} \\ 0 & \text{if } r \ge R_{\text{source}} \end{cases}$$

The analytical solution in this case is:

$$T(r) = \begin{cases} T_0 - \frac{Q}{2\pi k} \left(\frac{1}{2} \left(\frac{r^2}{R_{\text{source}}^2} - 1 \right) + \ln \left(\frac{R_{\text{source}}}{R_{\text{disk}}} \right) \right) & \text{if } r < R_{\text{source}} \\ T_0 - \frac{Q}{2\pi k} \ln \left(\frac{r}{R_{\text{disk}}} \right) & \text{if } r \ge R_{\text{source}} \end{cases}$$
(2)

The spatial extension of the heat source has a smoothing effect that removes the singularity at the origin, as shown on Figure 3 for $R_{\text{source}} = 10^{-2} \text{ m}$:



Figure 3: Distribution of relative temperature on the disk with a volume heat source.

Equation 1 and Equation 2 show that the temperature profiles are identical for $r \ge R_{\text{source}}$. The only difference is observed inside the source disk ($r < R_{\text{source}}$), as shown on Figure 4:



Figure 4: Analytical temperature distribution along a disk radius for punctual vs volume source, for a source radius of 10^{-2} m.

Notes about the COMSOL implementation

COMSOL Multiphysics provides different options to model a localized heat source.

- I The source support can be defined as a geometrical point. In this case, use the Line Heat Source (2D and 2D axisymmetric), Point Heat Source on Axis (2D axisymmetric) or Point Heat Source (3D) features. This leads to a singularity in the temperature field at the point where the source is applied. Numerically, the finer the mesh, the larger the temperature variation. In general, the two alternatives described below should be considered instead of this option, except for cases where a singular source is needed.
- 2 The heat source definition described above can be modified so that COMSOL Multiphysics accounts for the source size without needing a mesh nor a geometry change. In Line Heat Source (2D and 2D axisymmetric), Point Heat Source on Axis (2D axisymmetric) or Point Heat Source (3D) features, select the Specify heat source radius check box and set the Heat source radius to R_{source}. Then the heat source is automatically distributed over a disk in 2D (a torus in 2D axisymmetric or a sphere in

3D) as illustrated by the blue circle on the right image of Figure 5, even if the mesh elements size is larger than R_{source} .

3 If the size of the source is not too small compared to the surrounding geometry details, then a domain representing the heat source can be drawn (see the disk of radius R_{source} on the left image of Figure 5) and a **Heat Source** feature can be defined there. This option can be considered when the increase of the number of mesh elements induced by the geometry change can be afforded.



Figure 5: Different mesh configurations: elements size smaller (left) and larger (right) than the source radius. The source domain is delimited by a blue circle at the center.

Results and Discussion

In this section, first take advantage of the simple geometrical configuration to verify the accuracy of the different methods by comparing the numerical results with the analytical solutions in the Numerical accuracy of the different methods subsection.

However in many practical cases, the use of the **Heat Source** feature is not an option because of the prohibitive computational cost induced by the meshing of the heat source domain. The accuracy of the **Line Heat Source** feature, with or without the **Specify heat source radius** option is analyzed in the Coarse mesh case subsection.

Finally the results are summarized to define Guidelines for modeling a heat source localized on a small domain.

NUMERICAL ACCURACY OF THE DIFFERENT METHODS

In order to check the accuracy of the **Line Heat Source** feature for a punctual heat source, the mesh is gradually refined around the heat source, by lowering the maximum size mh of the elements in the neighborhood of the origin from 10^{-2} m to 10^{-6} m.



Figure 6: Relative L2 error (punctual analytical vs. numerical solution with a Line Heat Source feature) as a function of mesh refinement.

Figure 6 shows that the relative L2 error diminishes with mesh refinement, which validates the use of the **Line Heat Source** for this kind of problems.

The maximum temperature value obtained with different meshes, shown in Figure 7, illustrates the temperature amplitude increase as the mesh size decreases.



Figure 7: Maximum temperature as a function of mesh refinement (numerical solution for a punctual heat source).

A model with a volume source $(R_{\text{source}} = 10^{-2} \text{ m})$ is now considered and the convergence of the numerical solution as the mesh is refined is investigated.

First, the **Heat Source** feature is used on the domain of radius R_{source} , which has to be explicitly drawn in the geometry. Figure 8 shows the very good agreement between the computed temperature and the analytical solution.



Figure 8: Temperature distribution along a disk radius for a volume source, analytical and numerical computations (Heat Source feature).

The accuracy of the **Line Heat Source** feature with **Specify heat source radius** check box selected is verified using comparable mesh configurations on a geometry representing the source as a point.



Figure 9: Temperature distribution along a disk radius for a volume source, analytical and numerical solutions (Line Heat Source with the Specify heat source radius check box selected).

Figure 9 shows the very good agreement between the analytical solution and the temperature computed using the Line Heat Source feature with Specify heat source radius check box selected.



Figure 10: Relative L2 error (volume source analytical vs. numerical solution using a Line Heat Source with the Specify heat source radius check box selected) as a function of mesh refinement.

The convergence of the relative L2 error for fine mesh cases $(mh \le 10^{-2} m)$ is shown on Figure 10. This validates the accuracy of the Line Heat Source feature with Specify heat source radius check box selected.

COARSE MESH CASE

When the meshing of the heat source domain is not affordable, the **Heat Source** feature is not applicable any more. Low mesh resolution configurations are now considered to compare the accuracy of the **Line Heat Source** feature, with or without the **Specify heat source radius**.



Figure 11: Relative L2 error as a function of mesh size, using a Line Heat Source feature with the Specify heat source radius check box selected.

The case mh = 0.06 m (first point in the upper-left corner) corresponds to the mesh displayed on the right column of Figure 5, for which the circle of radius R_{source} is much smaller than the mesh element size. Even for this case, the relative L2 error remains in an acceptable range (relative error less than 0.03).

To go further, the error on this very coarse mesh is compared for the two versions of the **Line Heat Source** feature, namely with and without the **Specify heat source radius** check box selected.



Figure 12: Temperature distribution along a disk radius for the two versions of the Line Heat Source feature (with and without specified radius), on a coarse mesh (mh = 0.06 m)

Figure 12 shows that the error close to the heat source is greatly reduced by selecting the **Specify heat source radius** check box.

GUIDELINES

This tutorial brings the following conclusions regarding the modeling of localized heat sources with COMSOL Multiphysics.

If the heat source radius is large enough so that it can be drawn and meshed without prohibitive computational cost, then the **Heat Source** feature is the best option. In other cases, the **Line Heat Source** (2D and 2D axisymmetric), **Point Heat Source on axis** (2D axisymmetric) or **Point Heat Source** (3D) features with **Specify heat source radius** check box selected should be preferred. The **Specify heat source radius** check box is left cleared only for cases where the source is intended to be singular.

The **Line Heat Source** with specified radius option appears therefore as an accurate alternative to the punctual approach. In particular the temperature at the heat source location converges to a finite value when the mesh is refined.

Application Library path: Heat_Transfer_Module/Verification_Examples/ localized_heat_source

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🙆 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click **2D**.
- 2 In the Select Physics tree, select Heat Transfer>Heat Transfer in Solids (ht).
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click **M** Done.

GLOBAL DEFINITIONS

Parameters 1

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

Name	Expression	Value	Description
R_disk	0.1[m]	0.1 m	Disk radius
R_source	0.01[m]	0.01 m	Source radius
Т0	300[K]	300 K	Disk boundary temperature
k_cork	0.042[W/(m*K)]	0.042 W/(m·K)	Thermal conductivity, cork

Name	Expression	Value	Description
cp_cork	1.88[kJ/(kg*K)]	1880 J/(kg·K)	Heat capacity at constant pressure, cork
rho_cork	150[kg/m^3]	150 kg/m³	Density, cork
mh	0.01[m]	0.01 m	Mesh size parameter

Define an analytic function for the solution of the problem with a punctual heat source.

Analytic I (an I)

- I In the Home toolbar, click f(X) Functions and choose Global>Analytic.
- 2 In the Settings window for Analytic, locate the Definition section.
- 3 In the Expression text field, type -1/(2*pi*k_cork)*log(sqrt(x^2+y^2)/R_disk)
 + T0.
- 4 In the **Arguments** text field, type x, y.
- **5** Locate the **Units** section. In the table, enter the following settings:

Argument	Unit	
x	m	

6 In the Function text field, type K.

7 Locate the Plot Parameters section. In the table, enter the following settings:

Argument	Lower limit	Upper limit	Unit
x	-R_disk	R_disk	m
у	-R_disk	R_disk	

8 Click 💽 Plot.

Define an analytic function for the solution of the problem with a volume heat source.

Analytic 2 (an2)

- I In the Home toolbar, click f(X) Functions and choose Global>Analytic.
- 2 In the Settings window for Analytic, locate the Definition section.
- 3 In the Expression text field, type if (sqrt(x^2+y^2)>R_source, (-1/(2*pi* k_cork)*log(sqrt(x^2+y^2)/R_disk) + T0), (1/(2*pi*k_cork)*(-(x^2+ y^2)/(2*R_source^2)+0.5-log(R_source/R_disk)) + T0)).
- **4** In the **Arguments** text field, type x, y.

5 Locate the Plot Parameters section. In the table, enter the following settings:

Argument	Lower limit	Upper limit	Unit
x	-R_disk	R_disk	m
у	-R_disk	R_disk	

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

Argument	Unit		
x	m		

7 In the Function text field, type K.

8 Click 💽 Plot.

The geometry consists of a circle and a point at the origin of the circle. This geometry is suitable for the definition of either a punctual or a volume heat source with the **Line Heat Source** feature.

GEOMETRY I

Circle I (c1)

- I In the **Geometry** toolbar, click \bigcirc **Circle**.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type R_disk.

Point I (ptl)

- I In the **Geometry** toolbar, click **Point**.
- 2 In the Settings window for Point, click 📳 Build All Objects.

Next, create a new material (Cork) for the disk, and define the needed properties.

MATERIALS

Cork

- I In the Materials toolbar, click 🚦 Blank Material.
- 2 In the Settings window for Material, type Cork in the Label text field.

3 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	k_cork	W/(m·K)	Basic
Density	rho	rho_cork	kg/m³	Basic
Heat capacity at constant pressure	Ср	cp_cork	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

- I In the Model Builder window, under Component I (compl) 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 T0.

As a first step, consider the case of a punctual heat source. Use the **Line Heat Source** feature with default settings for that. You will disable this branch later when defining a volume heat source.

Line Heat Source 1

- I In the Physics toolbar, click 💭 Points and choose Line Heat Source.
- 2 Select Point 3 only.
- 3 In the Settings window for Line Heat Source, locate the Line Heat Source section.
- 4 From the Heat source list, choose Heat rate.
- **5** In the P_1 text field, type 1.

Next, define a parameterized mesh that can be refined around the origin of the disk. This way you can study the effect of mesh refinement on the heat source computation without increasing too much the mesh size.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Sequence Type section.
- 3 From the list, choose User-controlled mesh.

Size

- I In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- 3 From the **Predefined** list, choose **Coarse**.

Size 1

- I In the Model Builder window, right-click Free Triangular I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Point**.
- **4** Select Point **3** only.
- 5 Locate the Element Size section. From the Predefined list, choose Coarse.
- 6 Click the **Custom** button.
- 7 Locate the Element Size Parameters section.
- 8 Select the Maximum element size check box. In the associated text field, type mh.
- 9 Click 📗 Build All.

Now, define integration and maximum operators on the whole domain, for postprocessing.

DEFINITIONS

Integration 1 (intop1)

- I In the Definitions toolbar, click 🖉 Nonlocal Couplings and choose Integration.
- 2 In the Settings window for Integration, locate the Source Selection section.
- **3** From the Selection list, choose All domains.

Maximum I (maxopI)

- I In the Definitions toolbar, click *N*onlocal Couplings and choose Maximum.
- 2 In the Settings window for Maximum, locate the Source Selection section.
- **3** From the Selection list, choose All domains.

STUDY I: POINT SOURCE

- I In the Model Builder window, click Study I.
- 2 In the Settings window for Study, type Study 1: Point Source in the Label text field.

Parametric Sweep

I In the Study toolbar, click **Parametric Sweep**.

Define a parametric sweep on the maximum size of mesh elements in order to study the effects of mesh refinement on the heat source computation.

- 2 In the Settings window for Parametric Sweep, locate the Study Settings section.
- 3 Click + Add.
- 4 Select mh from the list.
- 5 Click Range.
- 6 In the Range dialog box, type -6 in the Start text field.
- 7 In the **Step** text field, type 0.5.
- 8 In the **Stop** text field, type -2.
- 9 From the Function to apply to all values list, choose explo(x) –
 Exponential function (base 10).
- IO Click Replace.
- II In the Study toolbar, click **=** Compute.

RESULTS

Temperature - Study 1

The first default plot shows the temperature distribution in a 2D plot for $mh = 10^{-2}m$. Proceed as follows to reproduce the plot of Figure 2 that corresponds to $mh = 10^{-3}m$.

- I In the Settings window for 2D Plot Group, type Temperature Study 1 in the Label text field.
- 2 Locate the Data section. From the Parameter value (mh (m)) list, choose 0.001.
- 3 In the Temperature Study I toolbar, click 💿 Plot.

Surface

- I In the Model Builder window, expand the Temperature Study I node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type T-TO.

Height Expression 1

- I In the Temperature Study I toolbar, click 💧 Height Expression.
- 2 Click 💽 Plot.

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

Then, proceed to reproduce the plot of Figure 6, by plotting the relative L2 error between numerical and analytical solutions as a function of 1/mh, in order to check numerical convergence.

- L2 Error from Analytical Solution Study I
- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type L2 Error from Analytical Solution
 Study 1 in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study I: Point Source/ Parametric Solutions I (sol2).

Line Graph 1

- I In the L2 Error from Analytical Solution Study I toolbar, click 📐 Line Graph.
- 2 In the Settings window for Line Graph, locate the Selection section.
- 3 From the Selection list, choose All boundaries.
- 4 Locate the y-Axis Data section. In the Expression text field, type sqrt(intop1((T-an1(x,y))^2))/sqrt(intop1((T-T0)^2)).
- 5 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 6 In the **Expression** text field, type 1/mh.
- 7 Click to expand the Coloring and Style section. From the Color list, choose Blue.
- 8 From the Width list, choose 5.
- 9 In the L2 Error from Analytical Solution Study I toolbar, click 💿 Plot.
- **IO** Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

Next, plot the maximum temperature as a function of 1/mh, as in the plot of Figure 7.

Maximum Temperature - Study I

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Maximum Temperature Study 1 in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study 1: Point Source/ Parametric Solutions 1 (sol2).

Line Graph 1

- I In the Maximum Temperature Study I toolbar, click 📐 Line Graph.
- 2 In the Settings window for Line Graph, locate the Selection section.

- 3 From the Selection list, choose All boundaries.
- 4 Locate the y-Axis Data section. In the Expression text field, type maxop1(T).
- 5 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 6 In the Expression text field, type 1/mh.
- 7 Locate the Coloring and Style section. From the Color list, choose Blue.
- 8 From the Width list, choose 5.
- 9 In the Maximum Temperature Study I toolbar, click 💿 Plot.

IO Click the **X-Axis Log Scale** button in the **Graphics** toolbar.

HEAT TRANSFER IN SOLIDS (HT)

As a second step, consider the case of a volume heat source. Use the **Line Heat Source** feature with the **Specify heat source radius** option selected, and the **Heat source radius** set to a positive value.

Line Heat Source 2

- I In the Physics toolbar, click 💮 Points and choose Line Heat Source.
- **2** Select Point 3 only.
- 3 In the Settings window for Line Heat Source, locate the Line Heat Source section.
- 4 From the Heat source list, choose Heat rate.
- **5** In the P_1 text field, type 1.
- 6 Locate the Heat Source Radius section. Select the Specify heat source radius check box.
- 7 In the *R* text field, type R_source.

In the next steps, configure **Study I** to use **Line Heat Source I** and create a second study that uses **Line Heat Source 2**.

STUDY I: POINT SOURCE

Step 1: Stationary

- I In the Model Builder window, under Study I: Point Source click Step I: Stationary.
- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- **3** Select the Modify model configuration for study step check box.
- 4 In the tree, select Component I (compl)>Heat Transfer in Solids (ht)>Line Heat Source 2.
- 5 Right-click and choose **Disable**.

ADD STUDY

- I In the Home toolbar, click 2 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>Stationary.
- 4 Click Add Study in the window toolbar.
- 5 In the Home toolbar, click Add Study to close the Add Study window.

STUDY 2

Step 1: Stationary

- I In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 2 Select the Modify model configuration for study step check box.
- 3 In the tree, select Component I (compl)>Heat Transfer in Solids (ht)>Line Heat Source I.
- 4 Right-click and choose Disable.

Again, define a parametric sweep on the maximum size of mesh elements in order to study the effects of mesh refinement on the heat source computation.

- 5 In the Model Builder window, click Study 2.
- 6 In the Settings window for Study, type Study 2: Point Source with Radius in the Label text field.

Parametric Sweep

- I In the Study toolbar, click **Parametric Sweep**.
- 2 In the Settings window for Parametric Sweep, locate the Study Settings section.
- 3 Click + Add.
- 4 Select mh from the list.
- 5 Click Range.
- 6 In the Range dialog box, type -6 in the Start text field.
- 7 In the **Step** text field, type 0.5.
- 8 In the **Stop** text field, type -2.
- 9 From the Function to apply to all values list, choose explo(x) Exponential function (base 10).
- IO Click Replace.
- II In the Study toolbar, click **=** Compute.

RESULTS

Temperature - Study 2

The first default plot shows the temperature distribution in a 2D plot for $mh=10^{-2}m$. Proceed as follows to reproduce the plot in Figure 3.

- I In the Settings window for 2D Plot Group, type Temperature Study 2 in the Label text field.
- 2 Locate the Data section. From the Parameter value (mh (m)) list, choose 0.001.
- 3 In the Temperature Study 2 toolbar, click 🗿 Plot.

Surface

- I In the Model Builder window, expand the Temperature Study 2 node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type T-TO.

Height Expression 1

- I In the Temperature Study 2 toolbar, click 🔺 Height Expression.
- 2 Click 💿 Plot.
- **3** Click the **Comextents** button in the **Graphics** toolbar.

Plot the analytical solutions of the two problems (punctual and volume heat sources) along a disk radius, to reproduce the plot of Figure 4.

Cut Line 2D I

- I In the **Results** toolbar, click \frown **Cut Line 2D**.
- 2 In the Settings window for Cut Line 2D, locate the Data section.
- 3 From the Dataset list, choose Study 2: Point Source with Radius/ Parametric Solutions 2 (sol13).
- 4 Locate the Line Data section. In row Point 2, set X to R_disk.
- 5 Click 💽 Plot.

Analytical Solutions, Point Source with/without Radius

- I In the Results toolbar, click \sim ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Analytical Solutions, Point Source with/without Radius in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Cut Line 2D I.
- 4 From the Parameter selection (mh) list, choose First.

- 5 Click to expand the Title section. From the Title type list, choose Manual.
- 6 In the Title text area, type Temperature comparison with different point sources.
- 7 Locate the Plot Settings section.
- 8 Select the y-axis label check box. In the associated text field, type Temperature (K).

Line Graph I

- I In the Analytical Solutions, Point Source with/without Radius toolbar, click Line Graph.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type an1(x,y).
- 4 Locate the Coloring and Style section. From the Width list, choose 3.
- 5 Click to expand the Legends section. Select the Show legends check box.
- 6 From the Legends list, choose Manual.
- 7 In the table, enter the following settings:

Legends

Without specified radius

Analytical Solutions, Point Source with/without Radius

In the Model Builder window, click Analytical Solutions, Point Source with/without Radius.

Line Graph 2

- I In the Analytical Solutions, Point Source with/without Radius toolbar, click C Line Graph.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type an2(x,y).
- 4 Locate the Coloring and Style section. From the Width list, choose 3.
- 5 Click to expand the Title section. From the Title type list, choose None.
- 6 Locate the Legends section. Select the Show legends check box.
- 7 From the Legends list, choose Manual.
- 8 In the table, enter the following settings:

Legends

With specified radius

9 In the Analytical Solutions, Point Source with/without Radius toolbar, click 🗿 Plot.

Next, proceed to reproduce the plot of Figure 9, by plotting the temperature distribution for numerical and analytical solutions along a disk radius, for $mh=10^{-2}m$.

Temperature vs. Radius - Study 2

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Temperature vs. Radius Study 2 in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Cut Line 2D I.
- 4 From the Parameter selection (mh) list, choose Last.

Line Graph I

- I In the Temperature vs. Radius Study 2 toolbar, click 📐 Line Graph.
- 2 In the Settings window for Line Graph, locate the Coloring and Style section.
- 3 Find the Line style subsection. From the Line list, choose None.
- 4 Find the Line markers subsection. From the Marker list, choose Circle.
- **5** From the **Positioning** list, choose **Interpolated**.
- 6 In the Number text field, type 25.
- 7 Locate the Legends section. Select the Show legends check box.
- 8 From the Legends list, choose Manual.
- **9** In the table, enter the following settings:

Legends

Numerical, mh = 0.01

Temperature vs. Radius - Study 2

In the Model Builder window, click Temperature vs. Radius - Study 2.

Line Graph 2

- I In the Temperature vs. Radius Study 2 toolbar, click 📐 Line Graph.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- 3 In the **Expression** text field, type an2(x,y).
- 4 Locate the Coloring and Style section. From the Width list, choose 3.
- 5 Locate the Legends section. Select the Show legends check box.
- 6 From the Legends list, choose Manual.

7 In the table, enter the following settings:

Legends Analytical

8 In the Temperature vs. Radius - Study 2 toolbar, click 🗿 Plot.

Proceed to reproduce the plot of Figure 10, by plotting the relative L2 error between numerical and analytical solutions as a function of mesh size parameter 1/mh, to check numerical convergence.

- L2 Error from Analytical Solution Study 2
- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type L2 Error from Analytical Solution
 Study 2 in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study 2: Point Source with Radius/ Parametric Solutions 2 (sol13).

Line Graph I

- I In the L2 Error from Analytical Solution Study 2 toolbar, click 📐 Line Graph.
- 2 In the Settings window for Line Graph, locate the Selection section.
- **3** From the Selection list, choose All boundaries.
- 4 Locate the y-Axis Data section. In the Expression text field, type sqrt(intop1((T-an2(x,y))^2))/sqrt(intop1((T-T0)^2)).
- 5 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 6 In the **Expression** text field, type 1/mh.
- 7 Locate the Coloring and Style section. From the Color list, choose Blue.
- 8 From the Width list, choose 5.
- 9 In the L2 Error from Analytical Solution Study 2 toolbar, click 💿 Plot.

IO Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

Next, proceed to reproduce the plot of Figure 12, by plotting the absolute value of the error along a disk radius between each numerical solution and the analytical one for a volume heat source, for $mh=10^{-2}m$.

Cut Line 2D 2

- I In the **Results** toolbar, click **Cut Line 2D**.
- 2 In the Settings window for Cut Line 2D, locate the Data section.

- 3 From the Dataset list, choose Study 1: Point Source/Parametric Solutions 1 (sol2).
- 4 Locate the Line Data section. In row Point 2, set X to R_disk.
- 5 Click 🗿 Plot.
- LI Error from Analytical Solutions Study I and Study 2
- I In the Results toolbar, click \sim ID Plot Group.
- 2 In the Settings window for ID Plot Group, type L1 Error from Analytical Solutions Study 1 and Study 2 in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Cut Line 2D 2.
- 4 From the Parameter selection (mh) list, choose Last.

Line Graph I

- I In the LI Error from Analytical Solutions Study I and Study 2 toolbar, click Line Graph.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- 3 In the **Expression** text field, type abs(T-an2(x,y)).
- 4 Locate the Coloring and Style section. From the Width list, choose 3.
- 5 Locate the Legends section. Select the Show legends check box.
- 6 From the Legends list, choose Manual.
- 7 In the table, enter the following settings:

Legends

Point source without specified radius

LI Error from Analytical Solutions - Study I and Study 2

In the Model Builder window, click LI Error from Analytical Solutions - Study I and Study 2.

Line Graph 2

- I In the LI Error from Analytical Solutions Study I and Study 2 toolbar, click Line Graph.
- 2 In the Settings window for Line Graph, locate the Data section.
- 3 From the Dataset list, choose Cut Line 2D I.
- **4** From the **Parameter selection (mh)** list, choose **Last**.
- 5 Locate the y-Axis Data section. In the Expression text field, type abs(T-an2(x,y)).
- 6 Locate the Title section. From the Title type list, choose None.
- 7 Locate the Coloring and Style section. From the Width list, choose 3.

- 8 Locate the Legends section. Select the Show legends check box.
- 9 From the Legends list, choose Manual.
- **IO** In the table, enter the following settings:

Legends

Point source with specified radius

II In the LI Error from Analytical Solutions - Study I and Study 2 toolbar, click 🗿 Plot.

Now, define a new mesh that is not refined any more at the origin of the disk, but is parameterized to be coarsened instead, with parameter mh.

MESH 2

- I In the Mesh toolbar, click Add Mesh and choose Add Mesh.
- 2 In the Settings window for Mesh, locate the Sequence Type section.
- 3 From the list, choose User-controlled mesh.

Use the parameter mh to control the maximum and minimum size of the mesh elements.

Size

- I In the Model Builder window, under Component I (compl)>Meshes>Mesh 2 click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- 3 Click the **Custom** button.
- **4** Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type mh.
- 5 In the Minimum element size text field, type mh.
- 6 Click 📗 Build All.

Define a new study corresponding to the problem with a volume source on a coarse mesh.

ADD STUDY

- I In the Home toolbar, click ~ 2 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>Stationary.
- 4 Click Add Study in the window toolbar.
- 5 In the Home toolbar, click Add Study to close the Add Study window.

STUDY 3

Step 1: Stationary

- I In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 2 Select the Modify model configuration for study step check box.
- 3 In the tree, select Component I (compl)>Heat Transfer in Solids (ht)>Line Heat Source I.
- 4 Right-click and choose **Disable**.

Define a parametric sweep on the size of mesh elements in order to check that the heat source is still well approximated on coarse meshes.

- 5 In the Model Builder window, click Study 3.
- 6 In the Settings window for Study, type Study 3: Point Source with Radius, Coarse Mesh in the Label text field.

Parametric Sweep

- I In the Study toolbar, click **Parametric Sweep**.
- 2 In the Settings window for Parametric Sweep, locate the Study Settings section.
- 3 Click + Add.
- 4 Select mh from the list.
- 5 Click Range.
- 6 In the Range dialog box, type 0.01 in the Start text field.
- 7 In the **Step** text field, type 0.012.
- 8 In the **Stop** text field, type 0.06.
- 9 Click Replace.
- **IO** In the **Study** toolbar, click **= Compute**.

RESULTS

Mesh Resolution - Study 3

The first default plot shows the temperature distribution in a 2D plot for $mh=10^{-2}m$. Proceed as follows to reproduce the plots for mesh configurations in Figure 5.

I In the Settings window for 2D Plot Group, type Mesh Resolution - Study 3 in the Label text field.

Surface

I In the Model Builder window, expand the Mesh Resolution - Study 3 node, then click Surface.

- 2 In the Settings window for Surface, locate the Coloring and Style section.
- 3 From the Coloring list, choose Uniform.
- 4 Select the Wireframe check box.

Mesh Resolution - Study 3

In the Model Builder window, click Mesh Resolution - Study 3.

Contour I

- I In the Mesh Resolution Study 3 toolbar, click 🛞 Contour.
- 2 In the Settings window for Contour, locate the Expression section.
- **3** In the **Expression** text field, type $sqrt(x^2+y^2)$.
- 4 Locate the Levels section. From the Entry method list, choose Levels.
- 5 In the Levels text field, type R_source.
- 6 Locate the Coloring and Style section. From the Contour type list, choose Tube.
- 7 From the Coloring list, choose Uniform.
- 8 From the Color list, choose Blue.
- 9 Clear the Color legend check box.

IO In the Mesh Resolution - Study 3 toolbar, click **O** Plot.

II Click the **Comextents** button in the **Graphics** toolbar.

Now change the mesh resolution to a finer configuration.

Mesh Resolution - Study 3

- I In the Model Builder window, click Mesh Resolution Study 3.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Parameter value (mh (m)) list, choose 0.01.
- 4 In the Mesh Resolution Study 3 toolbar, click 🗿 Plot.

Next, proceed to reproduce the plot of Figure 11, by plotting the relative L2 error between numerical and analytical solutions as a function of 1/mh, to check numerical convergence.

L2 Error from Analytical Solution - Study 3

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type L2 Error from Analytical Solution
 - Study 3 in the Label text field.

3 Locate the Data section. From the Dataset list, choose Study 3: Point Source with Radius, Coarse Mesh/Parametric Solutions 3 (sol24).

Line Graph 1

- I In the L2 Error from Analytical Solution Study 3 toolbar, click 📐 Line Graph.
- 2 In the Settings window for Line Graph, locate the Selection section.
- 3 From the Selection list, choose All boundaries.
- 4 Locate the y-Axis Data section. In the Expression text field, type sqrt(intop1((T-an2(x,y))^2))/sqrt(intop1((T-T0)^2)).
- 5 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 6 In the **Expression** text field, type 1/mh.
- 7 Locate the Coloring and Style section. From the Color list, choose Blue.
- 8 From the Width list, choose 5.
- 9 In the L2 Error from Analytical Solution Study 3 toolbar, click 💿 Plot.

As a last step, you can check that the computation of a volume heat source can also be performed by using the **Heat Source** feature. In order to do that, you need to include a smaller circle of radius R_source into the geometry, and to change the physics branch. Define a new component to include these changes and keep this step separated from the previous ones.

ADD COMPONENT

In the Model Builder window, right-click the root node and choose Add Component>2D.

GEOMETRY 2

Import the previously defined geometry, and complete it with the circle of radius R_source.

Import I (imp1)

- I In the **Home** toolbar, click **Import**.
- 2 In the Settings window for Import, locate the Import section.
- **3** From the **Source** list, choose **Geometry sequence**.
- 4 From the Geometry list, choose Geometry I.
- 5 Click Import.

Circle 1 (c1)

- I In the **Geometry** toolbar, click 💽 **Circle**.
- 2 In the Settings window for Circle, locate the Size and Shape section.

3 In the Radius text field, type R_source.

4 Click 🟢 Build All Objects.

This time, use the **Heat Source** feature to apply the source in the domain delimited by the circle of radius R_source. This replaces the use of the **Line Heat Source** feature in previous steps.

ADD PHYSICS

- I In the Home toolbar, click 🙀 Add Physics to open the Add Physics window.
- 2 Go to the Add Physics window.
- 3 In the tree, select Heat Transfer>Heat Transfer in Solids (ht).
- 4 Find the Physics interfaces in study subsection. In the table, clear the Solve check boxes for Study 1: Point Source, Study 2: Point Source with Radius, and Study 3: Point Source with Radius, Coarse Mesh.

5 Click Add to Component 2 in the window toolbar.

6 In the Home toolbar, click 🙀 Add Physics to close the Add Physics window.

HEAT TRANSFER IN SOLIDS 2 (HT2)

Temperature I

- I Right-click Component 2 (comp2)>Heat Transfer in Solids 2 (ht2) and choose Temperature.
- 2 Select Boundaries 1, 2, 5, and 8 only.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the T_0 text field, type T0.

Heat Source 1

- I In the Physics toolbar, click 🔵 Domains and choose Heat Source.
- **2** Select Domain 2 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.

Define the same material as before.

MATERIALS

Cork

I In the Materials toolbar, click 🚦 Blank Material.

2 In the Settings window for Material, type Cork in the Label text field.

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

Property	Variable	Value	Unit	P roperty group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	k_cork	W/(m·K)	Basic
Density	rho	rho_cork	kg/m³	Basic
Heat capacity at constant pressure	Ср	cp_cork	J/(kg·K)	Basic

You can visualize the mesh generated by default for this new geometry.

MESH 3

In the Model Builder window, under Component 2 (comp2) right-click Mesh 3 and choose Build All.

Add a new study for the computation of the volume heat source with the **Heat Source** feature.

ADD STUDY

- I In the Home toolbar, click $\sim\sim$ 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>Stationary.
- 4 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Heat Transfer in Solids (ht).
- 5 Click Add Study in the window toolbar.
- 6 In the Model Builder window, click the root node.
- 7 In the Home toolbar, click ~ 2 Add Study to close the Add Study window.

STUDY 4: SURFACE SOURCE

- I In the **Settings** window for **Study**, type **Study 4**: **Surface Source** in the **Label** text field.
- **2** In the **Home** toolbar, click **= Compute**.

RESULTS

Temperature - Study 4

In the **Settings** window for **2D Plot Group**, type Temperature - Study 4 in the **Label** text field.

Finally, proceed to reproduce the plot of Figure 8, by plotting the temperature distribution for numerical and analytical solutions along a disk radius.

Cut Line 2D 3

- I In the **Results** toolbar, click \square **Cut Line 2D**.
- 2 In the Settings window for Cut Line 2D, locate the Data section.
- 3 From the Dataset list, choose Study 4: Surface Source/Solution 30 (8) (sol30).
- 4 Locate the Line Data section. In row Point 2, set X to R_disk.
- 5 Click 🗿 Plot.

Temperature vs. Radius - Study 4

- I In the Results toolbar, click \sim ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Temperature vs. Radius Study 4 in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Cut Line 2D 3.

Line Graph I

- I In the Temperature vs. Radius Study 4 toolbar, click \sim Line Graph.
- 2 In the Settings window for Line Graph, locate the Coloring and Style section.
- 3 Find the Line style subsection. From the Line list, choose None.
- 4 Find the Line markers subsection. From the Marker list, choose Cycle.
- 5 From the **Positioning** list, choose **Interpolated**.
- 6 In the Number text field, type 25.
- 7 Locate the Legends section. Select the Show legends check box.
- 8 From the Legends list, choose Manual.
- **9** In the table, enter the following settings:

Legends

Numerical

Temperature vs. Radius - Study 4

In the Model Builder window, click Temperature vs. Radius - Study 4.

Line Graph 2

- I In the Temperature vs. Radius Study 4 toolbar, click 📐 Line Graph.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type an2(x,y).
- 4 Locate the Coloring and Style section. From the Width list, choose 3.
- 5 Locate the Legends section. Select the Show legends check box.
- 6 From the Legends list, choose Manual.
- 7 In the table, enter the following settings:

Legends

Analytical

8 In the Temperature vs. Radius - Study 4 toolbar, click on Plot.