Created in COMSOL Multiphysics 6.2

Heat Conduction with a Localized Heat Source on a Disk

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 $R_{\mathrm{disk}} = 0.1$ m. A fixed temperature $T_0 = 300$ 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 $R_{\mathrm{source}} = 10^{-2}$ m) centered at the origin.

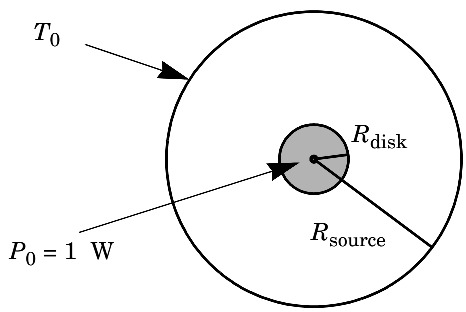

Figure 1: Geometry and boundary conditions.

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 nonaxial symmetric cases easier. Recall that Cartesian coordinates (x, y) and cylindrical coordinates (r, θ) are related by:

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

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 out-of-plane thickness and δ is the Dirac distribution centered at the origin. The solution of this equation is:

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

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.

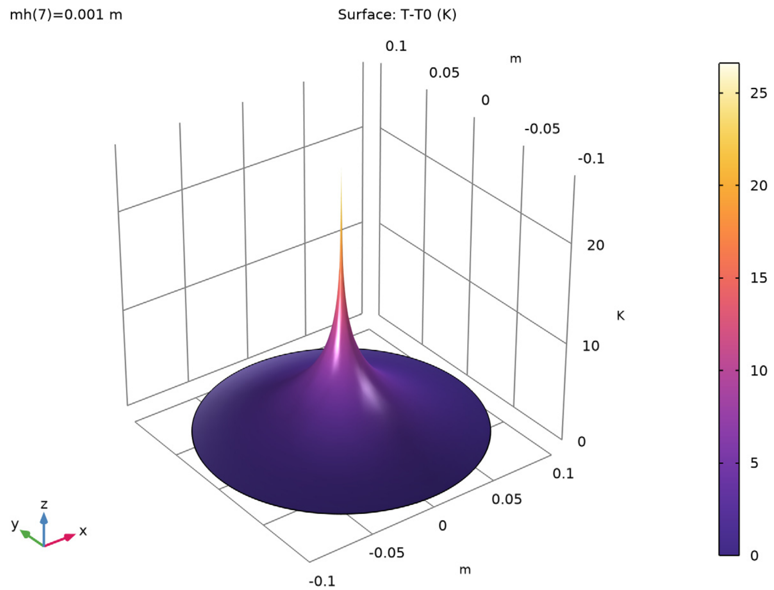

Figure 2: Distribution of relative temperature on the disk with a point heat source at the center.

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

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

The analytical solution in this case is:

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

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

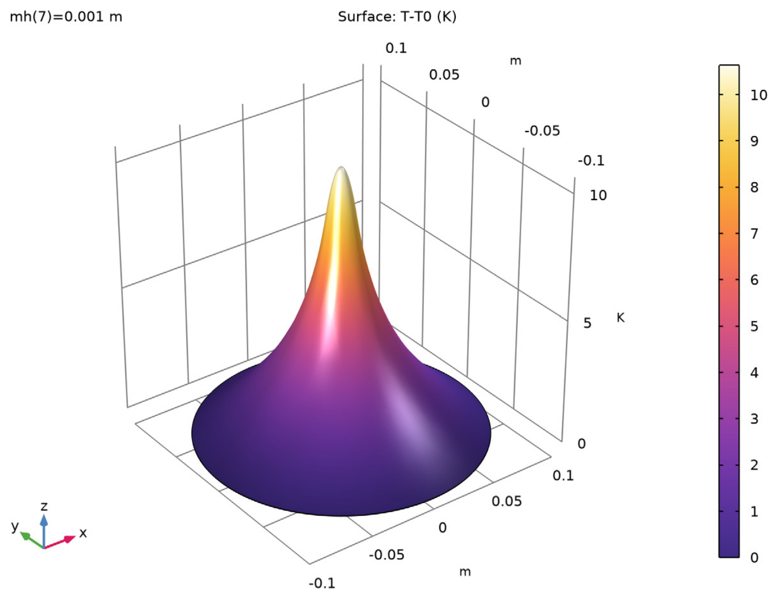

Figure 3: Distribution of relative temperature on the disk with a volume heat source.

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

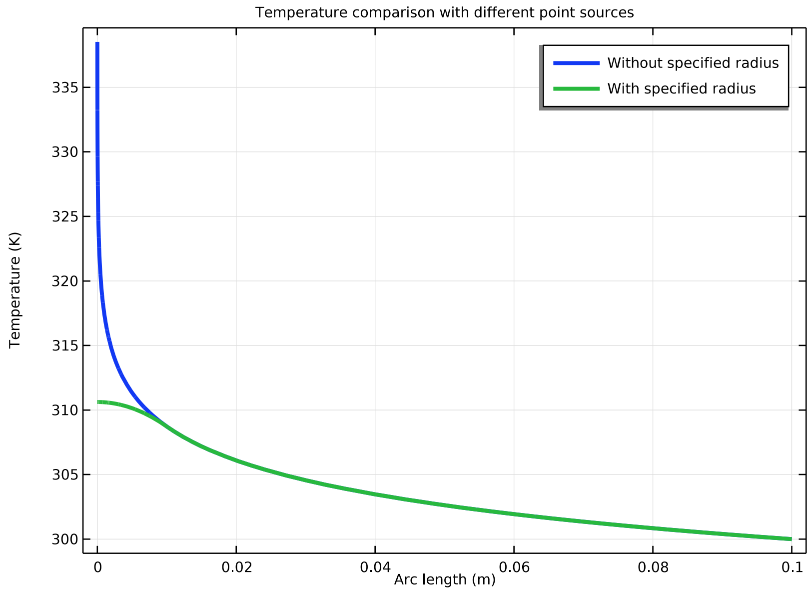

Figure 4: Analytical temperature distribution along a disk radius for punctual vs volume source, for a source radius of 10^{-2} m.

Notes about the COMSOL implementation

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

1. 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.

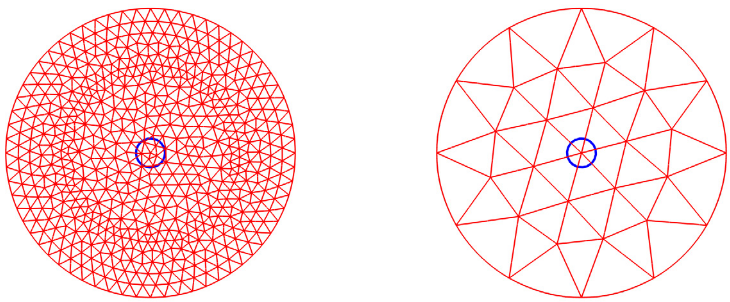

Figure 5: Different mesh configurations: elements size smaller (left) and larger (right) than the source radius. The source domain is delimited by a blue circle at the center.

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.

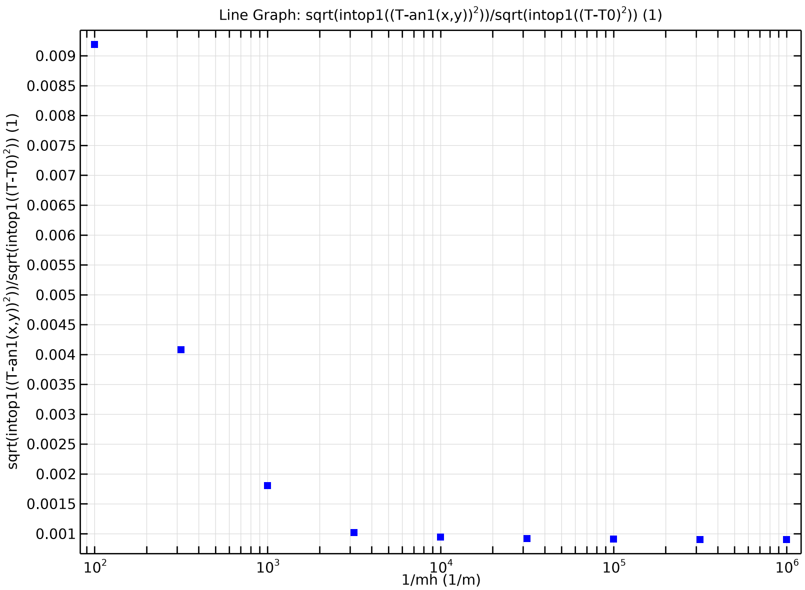

Figure 6: Relative L2 error (punctual analytical vs. numerical solution with a Line Heat Source feature) as a function of mesh refinement.

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.

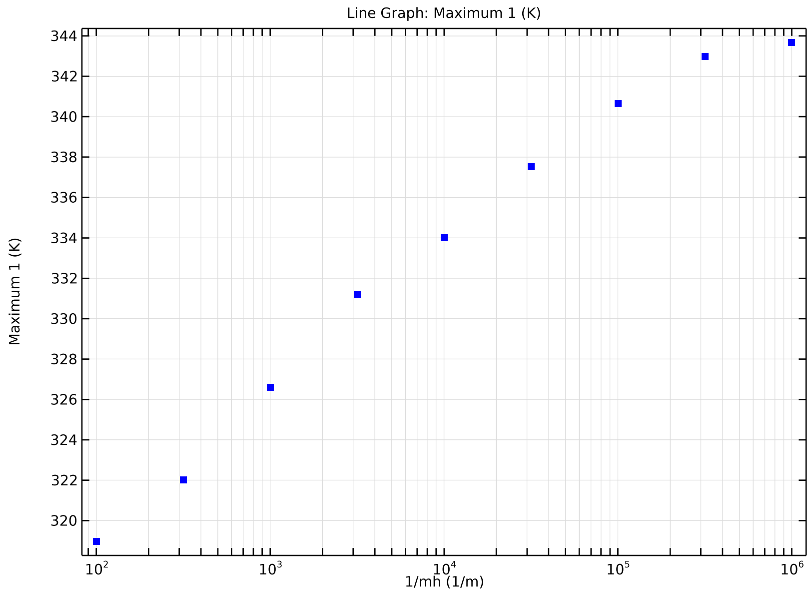

Figure 7: Maximum temperature as a function of mesh refinement (numerical solution for a punctual heat source).

A model with a volume source ($R_{source} = 10^{-2}$ 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.

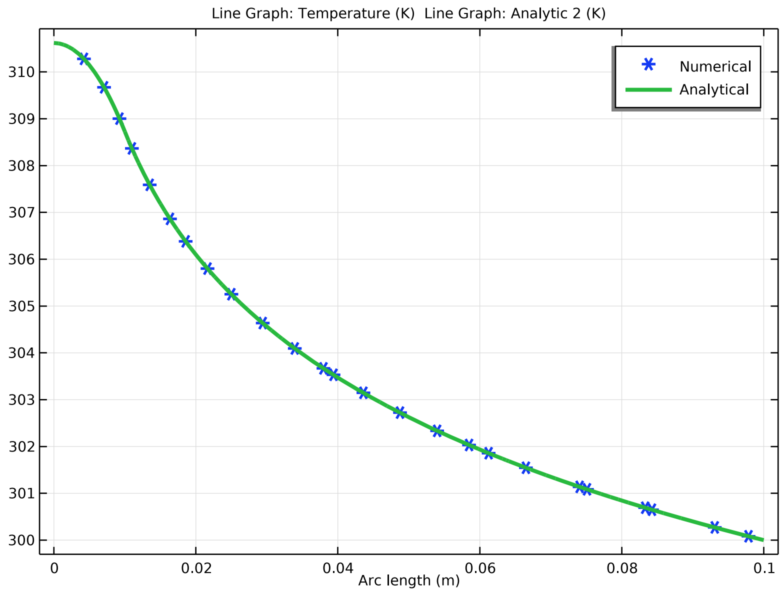

Figure 8: Temperature distribution along a disk radius for a volume source, analytical and numerical computations (Heat Source feature).

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.

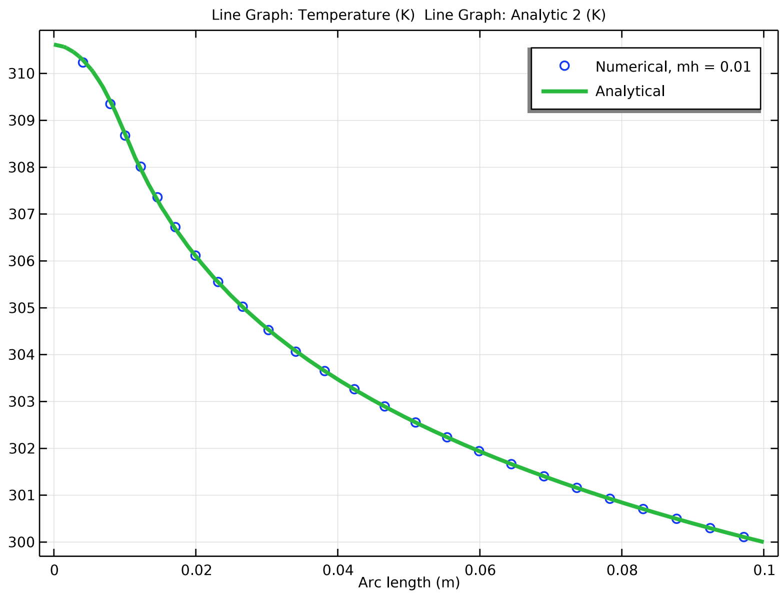

Figure 9: Temperature distribution along a disk radius for a volume source, analytical and numerical solutions (Line Heat Source with the Specify heat source radius check box selected).

Figure 9 shows 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.

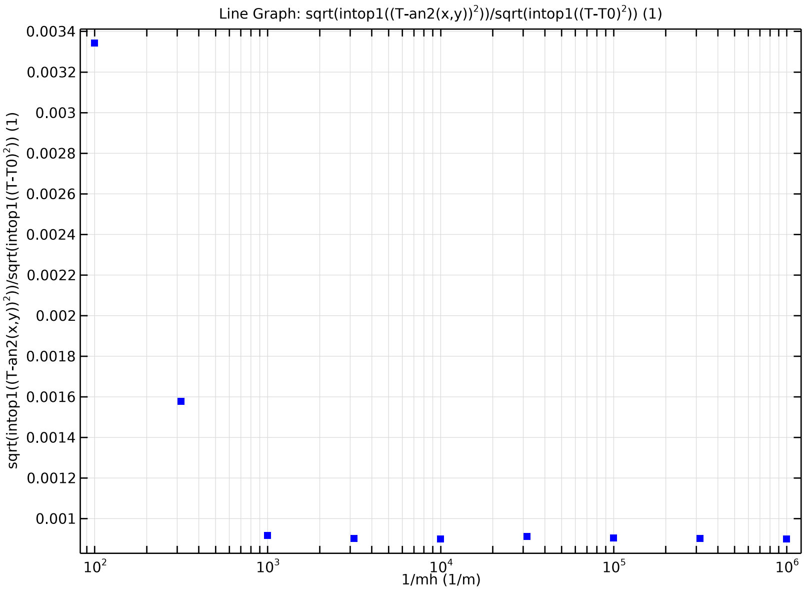

Figure 10: Relative L2 error (volume source analytical vs. numerical solution using a Line Heat Source with the Specify heat source radius check box selected) as a function of mesh refinement.

The convergence of the relative L2 error for fine mesh cases (`mh` $\leq 10^{-2}$ m) is shown in 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**.

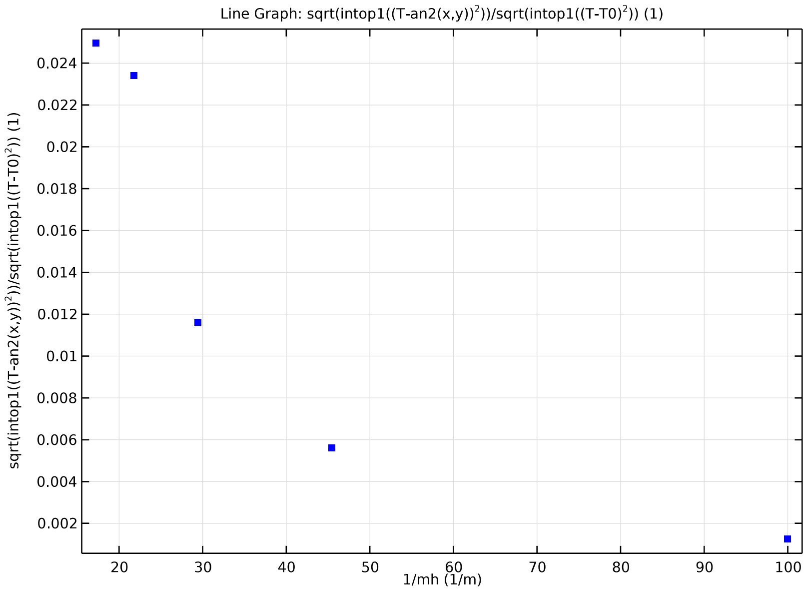

Figure 11: Relative L2 error as a function of mesh size, using a Line Heat Source feature with the Specify heat source radius check box selected.

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.

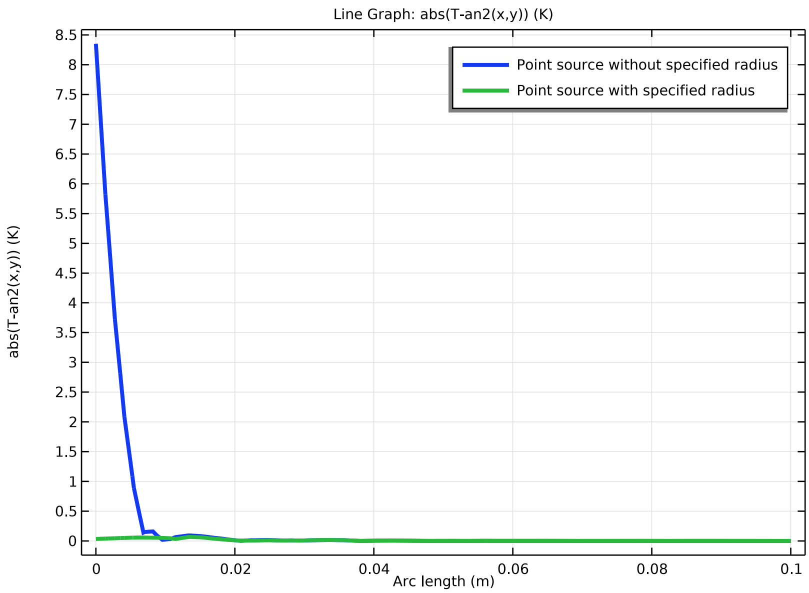

Figure 12: Temperature distribution along a disk radius for the two versions of the Line Heat Source feature (with and without specified radius), on a coarse mesh (mh = 0.06 m).

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

1 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 **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 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
R_disk	0.1[m]	0.1 m	Disk radius
R_source	0.01[m]	0.01 m	Source radius
T0	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 1 (an1)

1 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:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	-R_disk	R_disk	0	m
√	y	-R_disk	R_disk	0	

8 Click **Plot**.

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

Analytic 2 (an2)

1 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:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	-R_disk	R_disk	0	m
√	y	-R_disk	R_disk	0	

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 1

Circle 1 (c1)

1 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_disk`.

Point 1 (pt1)

1 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

1 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	Cp	`cp_cork`	J/(kg·K)	Basic

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 `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

1. 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 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
3. From the list, choose **User-controlled mesh**.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Coarse**.

Size 1

1. In the **Model Builder** window, right-click **Free Triangular 1** 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)

1. 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 1 (maxop1)

1. In the **Definitions** toolbar, click **Nonlocal 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 1: POINT SOURCE

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

Parametric Sweep

1 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 **exp10(x)** – **Exponential function (base 10)**.

10 Click **Replace**.

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

RESULTS

Temperature - Study 1

The 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.

1 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 1** toolbar, click **Plot**.

Surface 1

1 In the **Model Builder** window, expand the **Temperature - Study 1** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `T-T0`.

Height Expression 1

1 In the **Temperature - Study 1** 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 1

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

2 In the **Settings** window for **1D 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 1: Point Source/ Parametric Solutions 1 (sol2)**.

Line Graph 1

1 In the **L2 Error from Analytical Solution - Study 1** 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 1** toolbar, click **Plot**.

10 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 1

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

2 In the **Settings** window for **1D 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

1 In the **Maximum Temperature - Study 1** 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 1** toolbar, click **Plot**.

10 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

1 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 1** to use **Line Heat Source 1** and create a second study that uses **Line Heat Source 2**.

STUDY 1: POINT SOURCE

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1: Point Source** click **Step 1: 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 1 (comp1)>Heat Transfer in Solids (ht)>Line Heat Source 2**.

5 Right-click and choose **Disable**.

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>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: POINT SOURCE WITH RADIUS

1 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 1 (comp1)>Heat Transfer in Solids (ht)>Line Heat Source 1**.

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

1 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 **exp10(x)** – **Exponential function (base 10)**.

10 Click **Replace**.

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

RESULTS

Temperature - Study 2

The 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.

1 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 1

1 In the **Model Builder** window, expand the **Temperature - Study 2** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `T-T0`.

Height Expression 1

1 In the **Temperature - Study 2** toolbar, click **Height Expression**.

2 Click **Plot**.

3 Click the **Zoom Extents** 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 1

1 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 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

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

2 In the **Settings** window for **1D 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 1**.

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 1

1 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

1 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 `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

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 vs. Radius - Study 2` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.

4 From the **Parameter selection (mh)** list, choose **Last**.

Line Graph 1

1 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

1 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

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

2 In the **Settings** window for **1D 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 1

1 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**.

10 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

1 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**.

L1 Error from Analytical Solutions - Study 1 and Study 2

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

2 In the **Settings** window for **1D 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 1

1 In the **L1 Error from Analytical Solutions - Study 1 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`

L1 Error from Analytical Solutions - Study 1 and Study 2

In the **Model Builder** window, click **L1 Error from Analytical Solutions - Study 1 and Study 2**.

Line Graph 2

1 In the **L1 Error from Analytical Solutions - Study 1 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 1**.

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**.

10 In the table, enter the following settings:

Legends
`Point source with specified radius`

11 In the **L1 Error from Analytical Solutions - Study 1 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

1 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

1 In the **Model Builder** window, under **Component 1 (comp1)>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

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>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

1 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 1 (comp1)>Heat Transfer in Solids (ht)>Line Heat Source 1**.

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

1 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**.

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

RESULTS

Mesh Resolution - Study 3

The 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.

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

Surface 1

1 In the **Model Builder** window, expand the **Mesh Resolution - Study 3** node, then click **Surface 1**.

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 1

1 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.

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

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

Now change the mesh resolution to a finer configuration.

Mesh Resolution - Study 3

1 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

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

2 In the **Settings** window for **1D 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

1 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 1 (imp1)

1 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 1**.

5 Click **Import**.

Circle 1 (c1)

1 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

1 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 1

1 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

1 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

1 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	Cp	`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

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>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 **Add Study** to close the **Add Study** window.

STUDY 4: SURFACE SOURCE

1 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

1 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 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

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D 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 1

1 In the **Temperature vs. Radius - Study 4** 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 **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

1 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 **Plot**.