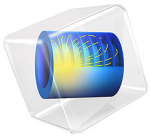

Heat Sink with Surface-to-Surface Radiation

This model is licensed under the COMSOL Software License Agreement 5.6.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This application extends the Heat Sink model by taking surface-to-surface radiation into account. For a detailed description of the application, see Heat Sink.

Application Library path: `Heat_Transfer_Module/Tutorials, _Forced_and_Natural_Convection/heat_sink_surface_radiation`

Modeling Instructions

ROOT

In this second part you modify and solve the model to study the effects of surface-to-surface radiation between the heat sink and the channel walls.

1. From the **File** menu, choose **Open**.
2. Browse to the model's Application Libraries folder and double-click the file `heat_sink.mph`.

COMPONENT 1 (COMP1)

Now modify the model to include surface-to-surface radiation effects. First you need to enable the surface-to-surface radiation property.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

By default, the radiation direction is controlled by the opacity of the domains. The solid parts are automatically defined as opaque while the fluid parts are transparent. You can change these settings by modifying the **Opacity** subnode under **Solid** and **Fluid** features.

When the **Diffuse Surface** boundary condition defines the radiation direction as opacity controlled (default) the selected boundaries should be located between an opaque and a transparent domain. The exterior is defined as transparent by default. Change this setting to make the exterior opaque and have the radiation direction automatically defined on the channel walls.

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>Radiation>Surface-to-Surface Radiation (rad)**.

4 Click **Add to Component 1** in the window toolbar.

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

SURFACE-TO-SURFACE RADIATION (RAD)

Now you can add a surface-to-surface boundary condition to the model.

1 In the **Settings** window for **Surface-to-Surface Radiation**, locate the **Boundary Selection** section.

2 From the **Selection** list, choose **Exterior Walls**.

Diffuse Surface 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Surface-to-Surface Radiation (rad)** click **Diffuse Surface 1**.

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

3 Find the **Ambient temperature** subsection. From the T_{amb} list, choose **Ambient temperature (ampr1)**.

4 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.85`.

Opacity 1

1 In the **Physics** toolbar, click **Domains** and choose **Opacity**.

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

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

ADD MULTIPHYSICS

1 In the **Physics** toolbar, click **Add Multiphysics** to open the **Add Multiphysics** window.

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

3 In the tree, select **No Predefined Multiphysics Available for the Selected Physics Interfaces**.

4 Find the **Select the physics interfaces you want to couple** subsection. In the table, clear the **Couple** check box for **Laminar Flow (spf)**.

5 In the tree, select **Heat Transfer>Radiation>Heat Transfer with Surface-to-Surface Radiation**.

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

7 In the **Physics** toolbar, click **Add Multiphysics** to close the **Add Multiphysics** window.

COMPONENT 1 (COMP1)

Hide the boundaries on the top and fronts to see the interior of the channel and the heat sink.

1 Click the **Click and Hide** button in the **Graphics** toolbar.

2 Click the **Select Boundaries** button in the **Graphics** toolbar.

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

4 Select Boundaries 1, 2, and 4 only.

ROOT

In order to keep the previous solution and to be able to compare it with this version of the model, create a new stationary study.

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 1 - WITHOUT RADIATION

1 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Study 1 - without radiation` in the **New label** text field.

3 Click **OK**.

STUDY 2 - WITH RADIATION

1 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Study 2 - with radiation` in the **New label** text field.

3 Click **OK**.

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

RESULTS

Temperature (ht) 1

The same default plots as before are generated automatically. Modify the temperature plot to compare both cases.

Arrow Volume 1

1 In the **Temperature (ht) 1** toolbar, click  **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Laminar Flow>Velocity and pressure>u,v,w - Velocity field**.

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

4 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `40`.

5 Find the **y grid points** subsection. In the **Points** text field, type `20`.

6 Find the **z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

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

Color Expression 1

1 In the **Temperature (ht) 1** toolbar, click  **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude - m/s**.

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

The plot in the **Graphics** window should look like that in the figure below.