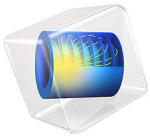

Heat Sink with Surface-to-Surface Radiation

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 this setting using the **Opacity** feature in the **Surface-to-Surface Radiation** interface.

When the **Diffuse Surface** boundary condition defines **Emitted radiation direction** as **Opacity controlled** (the default setting), the selected boundaries should be located between an opaque and a transparent domain. The exterior is defined as transparent by default. Change the default 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 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 In the **Model Builder** window, click **Component 1 (comp1)**.

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

3 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. Edit the first study to exclude Surface-to-Surface Radiation and make sure the same solution will be computed in case it is solved again.

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**.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1 - without radiation** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Surface-to-Surface Radiation (rad)**.

4 In the table, clear the **Solve for** check box for **Heat Transfer with Surface-to-Surface Radiation 1 (htrad1)**.

STUDY 2 - WITH RADIATION

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - with radiation` in the **Label** text field.

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

RESULTS

Some of the default plot groups of the new study will have similar names to the ones that already exist. To avoid confusion, organize plots in two groups.

Energy Balance (ht), Pressure (spf), Temperature (ht), Temperature and Fluid Flow (nitf1), Velocity (spf)

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Temperature (ht)**, **Velocity (spf)**, **Pressure (spf)**, **Temperature and Fluid Flow (nitf1)**, and **Energy Balance (ht)**.
2. Right-click and choose **Group**.

Without radiation

In the **Settings** window for **Group**, type `Without radiation` in the **Label** text field.

Pressure (spf) 1, Surface Radiosity (rad), Temperature (ht) 1, Temperature and Fluid Flow (nitf1) 1, Velocity (spf) 1

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Temperature (ht) 1**, **Velocity (spf) 1**, **Pressure (spf) 1**, **Surface Radiosity (rad)**, and **Temperature and Fluid Flow (nitf1) 1**.
2. Right-click and choose **Group**.

With radiation

In the **Settings** window for **Group**, type `With radiation` in the **Label** text field.

Transparency 1

1. In the **Model Builder** window, expand the **Results>With radiation>Temperature (ht) 1** node.
2. Right-click **Volume 1** and choose **Transparency**.

Temperature and Fluid Flow, with Radiation

The same default plot for temperature and fluid flow as before is generated automatically. Modify this plot to compare both case with and without radiation.

1. In the **Settings** window for **3D Plot Group**, type `Temperature and Fluid Flow, with Radiation` in the **Label** text field.
2. In the **Model Builder** window, expand the **Temperature and Fluid Flow, with Radiation** node.

Fluid Flow

1. In the **Model Builder** window, expand the **Results>With radiation> Temperature and Fluid Flow, with Radiation>Fluid Flow** node, then click **Fluid Flow**.
2. In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** text field, type `40`.

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

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

6 In the **Coordinates** text field, type `5`.

Filter 1

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

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `spf.U>0.25*nitf1.Uave`.

4 In the **Temperature and Fluid Flow, with Radiation** toolbar, click **Plot**.

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