Tutorial 5. Modeling Radiation and Natural Convection

Introduction

In this tutorial combined radiation and natural convection are solved in a three-dimensional square box on a mesh consisting of hexahedral elements.

This tutorial demonstrates how to do the following:

- Use the surface-to-surface (S2S) radiation model in ANSYS FLUENT.
- Set the boundary conditions for a heat transfer problem involving natural convection and radiation.
- Calculate a solution using the pressure-based solver.
- Display velocity vectors and contours of wall temperature, surface cluster ID, and radiation heat flux.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1, and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

Problem Description

The problem to be considered is shown schematically in Figure 5.1. A three-dimensional box (0.5 x 0.5 x 0.5) has a hot wall at 473 K and all other walls at 293 K. Gravity acts downwards. The medium contained in the box is assumed to be absorbing and emitting, so that the radiant exchange between the walls is attenuated by absorption and augmented by emission in the medium. All walls are black. The objective is to compute the flow and temperature patterns in the box, as well as the wall heat flux, using the surface-to-surface (S2S) model available in ANSYS FLUENT.

The working fluid has a Prandtl number of approximately 0.71, and the Rayleigh number based on $L (0.5)$ is $5 \times 10^8$. This means the flow is most likely laminar. The Planck number $k/(4\sigma LT_0^3)$ is 0.003, and measures the relative importance of conduction to radiation.
Setup and Solution

Preparation

1. Download `radiation_natural_convection.zip` from the User Services Center to your working folder (as described in Tutorial 1).

2. Unzip `radiation_natural_convection.zip`.
   The mesh file `rad.msh.gz` can be found in the `radiation_natural_convection` folder created after unzipping the file.

3. Use FLUENT Launcher to start the 3D version of ANSYS FLUENT.
   For more information about FLUENT Launcher, see Section 1.1.2 in the separate User’s Guide.

**Note:** The Display Options are enabled by default. Therefore, after you read the mesh, it will be displayed in the embedded graphics window.

Step 1: Mesh

1. Read the mesh file `rad.msh.gz`.
   ![Mesh Reading Process]
   As the mesh is read, messages will appear in the console reporting the progress of the reading. The mesh size will be reported as 64,000 cells.
Step 2: General Settings

1. Check the mesh.

   **General** → **Check**

   ANSYS FLUENT will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

2. Examine the mesh.

   ![Figure 5.2: Graphics Display of Mesh](image)

   Figure 5.2: Graphics Display of Mesh
3. Retain the default solver settings.

4. Enable Gravity.
   
   (a) Enter \(-9.81 \, \text{m/s}^2\) for Gravitational Acceleration in the \(Y\) direction.

**Step 3: Models**

1. Enable the energy equation.
2. Enable the **Surface to Surface (S2S)** radiation model.

![Radiation Model dialog box]

(a) Select **Surface to Surface (S2S)** in the **Model** list.

*The Radiation Model* dialog box will expand to show additional inputs for the S2S model.

The surface-to-surface (S2S) radiation model can be used to account for the radiation exchange in an enclosure of gray-diffuse surfaces. The energy exchange between two surfaces depends in part on their size, separation distance, and orientation. These parameters are accounted for by a geometric function called a “view factor”.

The S2S model assumes that all surfaces are gray and diffuse. Thus according to the gray-body model, if a certain amount of radiation is incident on a surface, then a fraction is reflected, a fraction is absorbed, and a fraction is transmitted. The main assumption of the S2S model is that any absorption, emission, or scattering of radiation by the medium can be ignored. Therefore only “surface-to-surface” radiation is considered for analysis.

For most applications the surfaces in question are opaque to thermal radiation (in the infrared spectrum), so the surfaces can be considered opaque. For gray, diffuse, and opaque surfaces it is valid to assume that the emissivity is equal to the absorptivity and that reflectivity is equal to 1 minus the emissivity.
When the S2S model is used, you also have the option to define a “partial enclosure” which allows you to disable the view factor calculation for walls with negligible emission/absorption or walls that have uniform temperature. The main advantage of this option is to speed up the view factor calculation and the radiosity calculation.

(b) Click the Set... button in the Parameters group box to open the View Factor and Cluster Parameters dialog box.

You will define the view factor and cluster parameters.

1. Retain the value of 1 for Faces per Surface Cluster for Flow Boundary Zones in the Parameters group box.

2. Click Apply to All Walls.

The S2S radiation model is computationally very expensive when there are a large number of radiating surfaces. The number of radiating surfaces is reduced by clustering surfaces into surface “clusters”. The surface clusters are made by starting from a face and adding its neighbors and their neighbors until a specified number of faces per surface cluster is collected.
For a small 2D problem, the default value of 1 for Faces per Surface Cluster for Flow Boundary Zones is acceptable. For a large problem you can increase this number to reduce the memory requirement for the view factor file that is saved in a later step. This may also lead to some reduction in the computational expense. However, this is at the cost of some accuracy. This tutorial illustrates the influence of clusters.

iii. Select Ray Tracing in the Method list in the View Factor group box.
iv. Click OK to close the View Factor and Cluster Parameters dialog box.

(c) Click Compute/Write... for Methods in the View Factors group box to open the Select File dialog box and to compute the view factors.

The file created in this step will store the cluster and view factor parameters.

You need to perform this step if the problem is being solved for the first time. For subsequent calculations you can read the view factor and cluster information from an existing file (by clicking Read... instead of Compute/Write...).

i. Enter rad_1.s2s.gz as the file name for S2S File.
ii. Click OK in the Select File dialog box.

Note: The size of the view factor file can be very large if not compressed. It is highly recommended to compress the view factor file by providing .gz or .Z extension after the name (i.e. rad_1.gz or rad_1.Z). For small files, you can provide the .s2s extension after the name.

ANSYS FLUENT will print an informational message describing the progress of the view factor calculation in the console.

(d) Click OK to close the Radiation Model dialog box.
Step 4: Materials

1. Set the properties for air.

   (a) Select \textit{incompressible-ideal-gas} from the \textit{Density} drop-down list.
   
   (b) Enter 1021 J/kg-K for \textit{Cp (Specific Heat)}.
   
   (c) Enter 0.0371 W/m-K for \textit{Thermal Conductivity}.
   
   (d) Enter 2.485e-05 kg/m-s for \textit{Viscosity}.
   
   (e) Retain the default value of 28.966 kg/kgmol for \textit{Molecular Weight}.
   
   (f) Click \textit{Change/Create} and close the \textit{Create/Edit Materials} dialog box.
2. Define the new material, insulation.

- Navigate to Materials → Solid → Create/Edit...

(a) Enter insulation for Name and delete the entry in the Chemical Formula field.
(b) Enter 50 kg/m$^3$ for Density.
(c) Enter 800 J/kg-K for Cp (Specific Heat).
(d) Enter 0.09 W/m-K for Thermal Conductivity.
(e) Click Change/Create.

A Question dialog box will open, asking if you want to overwrite aluminum.

(f) Click No in the Question dialog box to retain aluminum and add the new material (insulation) to the materials list.
The Create/Edit Materials dialog box will be updated to show the new material, insulation, in the FLUENT Solid Materials drop-down list.

(g) Close the Create/Edit Materials dialog box.

Step 5: Boundary Conditions

1. Set the boundary conditions for the front wall (w-high-x).
(a) Click the Thermal tab and select Mixed in the Thermal Conditions group box.
(b) Select insulation from the Material Name drop-down list.
(c) Enter 5 W/m$^2$ – K for Heat Transfer Coefficient.
(d) Enter 293.15 K for both Free Stream Temperature and External Radiation Temperature.
(e) Enter 0.75 for External Emissivity.
(f) Enter 0.95 for Internal Emissivity.
(g) Enter 0.05 m for Wall Thickness.
(h) Click OK to close the Wall dialog box.
2. Copy boundary conditions to define the side walls \( w\text{-high-z} \) and \( w\text{-low-z} \).

   - **Boundary Conditions** \( \rightarrow \) **Copy…**

   ![Copy Conditions dialog box]

   (a) Select \( w\text{-high-x} \) from the **From Boundary Zone** selection list.

   (b) Select \( w\text{-high-z} \) and \( w\text{-low-z} \) from the **To Boundary Zones** selection list.

   (c) Click **Copy**.

   *A Warning dialog box will open, asking if you want to copy the boundary conditions of \( w\text{-high-x} \) to \( w\text{-high-z} \) and \( w\text{-low-z} \).*

   ![Warning dialog box]

   (d) Click **OK** in the **Warning** dialog box.

   (e) Close the **Copy Conditions** dialog box.
3. Set the boundary conditions for the heated wall (w-low-x).

(a) Click the Thermal tab and select Temperature in the Thermal Conditions group box.
(b) Retain the default selection of aluminum from the Material Name drop-down list.
(c) Enter 473.15 K for Temperature.
(d) Enter 0.95 for Internal Emissivity.
(e) Click OK to close the Wall dialog box.
4. Set the boundary conditions for the top wall (w-high-y).

(a) Click the **Thermal** tab and select **Mixed** in the **Thermal Conditions** group box.

(b) Select **insulation** from the **Material Name** drop-down list.

(c) Enter $3 \text{ w/m}^2 \cdot \text{K}$ for **Heat Transfer Coefficient**.

(d) Enter 293.15 K for both **Free Stream Temperature** and **External Radiation Temperature**.

(e) Enter 0.75 for **External Emissivity**.

(f) Enter 0.95 for **Internal Emissivity**.

(g) Enter 0.05 m for **Wall Thickness**.

(h) Click **OK** to close the **Wall** dialog box.
5. Copy boundary conditions to define the bottom wall (w-low-y).

(a) Select w-high-y from the From Boundary Zone selection list.
(b) Select w-low-y from the To Boundary Zones selection list.
(c) Click Copy.

A Warning dialog box will open, asking if you want to copy the boundary conditions of w-high-y to w-low-y.

(d) Click OK in the Warning dialog box.
(e) Close the Copy Conditions dialog box.
Step 6: Solution

1. Set the solution parameters.

   (a) Select Body Force Weighted from the Pressure drop-down list in the Spatial Discretization group box.

   (b) Retain the default selection of First Order Upwind from the Momentum and Energy drop-down lists.
2. Set the under-relaxation factors.

![Solution Controls](image)

(a) Enter 0.4 for Momentum.

*Buoyancy driven cases will need stiffer relaxation for better results. A good starting point for momentum would be 0.4.*
3. Initialize the solution.

(a) Enter 450 K for Temperature.

(b) Click Initialize.
4. Create the new surface, \( zz_{\text{center}}.z \).

   Surface \( \rightarrow \) Iso-Surface...

   (a) Select Mesh... and Z-Coordinate from the Surface of Constant drop-down lists.
   (b) Click Compute and retain the value 0 in the Iso-Values field.
   (c) Enter \( zz_{\text{center}}.z \) for New Surface Name.
   (d) Click Create and close the Iso-Surface dialog box.

5. Save the case file (\texttt{rad.a.1.cas.gz})

   File \( \rightarrow \) Write \( \rightarrow \) Case...

6. Start the calculation by requesting 100 iterations Figure 5.3.

   \( \text{Run Calculation} \)
Figure 5.3: Scaled Residuals

(a) Enter 100 for Number of Iterations.
(b) Click Calculate.

An inspection of the residual plot at this stage suggests that the solution is not converging in a stable manner. This can be a common problem with natural convection (buoyancy driven) flows which tend to be unstable in their physical nature.
7. Display contours of static temperature.

(a) Enable Filled in the Options group box.
(b) Select Temperature... and Static Temperature from the Contours of drop-down lists.
(c) Select zz_center_z from the Surfaces selection list.
(d) Enable Draw Mesh in the Options group box to open the Mesh Display dialog box.
   i. Select Outline in the Edge Type list.
   ii. Click Display and close the Mesh Display dialog box.
(e) Disable Auto Range.
(f) Enter 421 for Min and 473.15 for Max.
(g) Click Display and rotate the view as shown in Figure 5.4.
(h) Close the Contours dialog box. (Figure 5.4).

A regular check for most buoyant cases is to look for evidence of stratification in the temperature field, near horizontal bands of similar temperature. These may be broken or disturbed by buoyant plumes. For this case you can expect reasonable stratification with some disturbance at the vertical walls where the air is driven...
round. However, the results show very little evidence of this. This is most likely due to the physical instability of the flow process. To help overcome this, make use of relaxation to damp out the instabilities.

8. Change the under-relaxation factor for **Momentum**.

   Solution Controls
   
   (a) Enter 0.1 for **Momentum**.

   The relaxation factor on momentum was already reduced to 0.4 before solving. We shall now drop it even further to 0.1. In general, avoid this type of stiff relaxation as it will slow down the solution speed, but in cases like this it is necessary. However, avoid reducing the relaxation factor much further.

9. Request 100 more iterations.

   Run Calculation

Figure 5.4: Contours of Static Temperature
Step 7: Postprocessing

1. Create the new surface, \(zz_x\_side\).

   ![Surface button](Image)

   - Enter \((-0.25, 0, 0.25)\) for \((x0, y0, z0)\) respectively.
   - Enter \((0.25, 0, 0.25)\) for \((x1, y1, z1)\) respectively.
   - Enter \(zz_x\_side\) for New Surface Name.
   - Click Create and close the Line/Rake Surface dialog box.
2. Display contours of wall temperature (outer surface).

- **Graphics and Animations** → **Contours** → **Set Up...**

(a) Make sure that Filled is enabled in the Options group box.

(b) Disable Node Values.

(c) Select Temperature... and Wall Temperature (Outer Surface) from the Contours of drop-down lists.

(d) Select all surfaces except default-interior and zz_x_side.

(e) Disable Auto Range and Draw Mesh.

(f) Enter 413 for Min and 473.15 for Max.

(g) Click Display and rotate the view as shown in Figure 5.5.
Figure 5.5: Contours of Wall Temperature

3. Display contours of static temperature.

   (a) Make sure that Filled is enabled in the Options group box.
   (b) Select Temperature... and Static Temperature from the Contours of drop-down lists.
(c) Deselect all surfaces and select `zz_center.z` from the Surfaces selection list.

(d) Enable Draw Mesh in the Options group box to open the Mesh Display dialog box.

   i. Make sure that Outline in the Edge Type list is selected.

   ii. Click Display and close the Mesh Display dialog box.

(e) Enable Node Values.

(f) Disable Auto Range.

(g) Enter 421 for Min and 473.15 for Max.

(h) Click Display and rotate the view as shown in Figure 5.6.

Figure 5.6: Contours of Static Temperature

The temperature field now ties in with expectations, displaying good stratification with disturbance at the walls.

- **Graphics and Animations** ➔ **Contours** ➔ **Set Up...**

(a) Make sure that **Filled** is enabled in the **Options** group box.
(b) Disable both **Node Values** and **Draw Mesh** in the **Options** group box.
(c) Select **Wall Fluxes...** and **Radiation Heat Flux** from the **Contours of** drop-down list.
(d) Select all surfaces except **default-interior** and **zz_x_side**.
(e) Click **Display** and rotate the view as shown in Figure 5.7.
(f) Close the **Contours** dialog box.

*Figure 5.7 shows the radiating wall (w-low-x) with positive heat flux and all other walls with negative heat flux.*
5. Display vectors of velocity magnitude.

(a) Retain the default selection of Velocity from the Vectors of drop-down list.
(b) Retain the default selection of *Velocity...* and *Velocity Magnitude* from the *Color by* drop-down list.

(c) Deselect all surfaces and select `zz_center_z` from the *Surfaces* selection list.

(d) Enable *Draw Mesh* in the *Options* group box to open the *Mesh Display* dialog box.
   
i. Make sure that *Outline* is selected in the *Edge Type* list.
   
ii. Click *Display* and close the *Mesh Display* dialog box.

(e) Enter 7 for *Scale*.

(f) Click *Display* (Figure 5.8) and rotate the view as shown in Figure 5.8.

(g) Close the *Vectors* dialog box.

Figure 5.8: Vectors of Velocity Magnitude

![Velocity Vectors Colored By Velocity Magnitude](Image)
6. Compute view factors and radiation emitted from the front wall (w-high-x) to all other walls.

Report S2S Information...

(a) Make sure that View Factors is enabled in the Report Options group box.
(b) Enable Incident Radiation.
(c) Select w-high-x from the From selection list.
(d) Select all zones except w-high-x from the To selection list.
(e) Click Compute and close the S2S Information dialog box.

The computed values of the Views Factors and Incident Radiation are displayed in the console. A view factor of approximately 0.2 for each wall is a good value for the square box.
7. Compute the total heat transfer rate.

(a) Select **Total Heat Transfer Rate** in the **Options** group box.
(b) Select all boundary zones except default-interior from the **Boundaries** selection list.
(c) Click **Compute**.

**Note:** *The energy imbalance is approximately 0.08%.*

8. Compute the total heat transfer rate for **w-low-x**.

(a) Select **Total Heat Transfer Rate** in the **Options** group box.
(b) Select all boundary zones except default-interior from the **Boundaries** selection list.
(c) Click **Compute**.
(a) Retain the selection of **Total Heat Transfer Rate** in the **Options** group box.

(b) Deselect all boundary zones and select **w-low-x** from the **Boundaries** selection list.

(c) Click **Compute**.

**Note:** *The net heat load is approximately 251.55 W*

9. Compute the radiation heat transfer rate.

   ![Flux Reports](image)

(a) Select **Radiation Heat Transfer Rate** in the **Options** group box.

(b) Select all boundary zones except **default-interior** from the **Boundaries** selection list.

(c) Click **Compute**.

**Note:** *The net heat load is approximately -0.12 W.*
10. Compute the radiation heat transfer rate for w-low-x.

- Reports → Fluxes → Set Up...

(a) Retain the selection of Radiation Heat Transfer Rate in the Options group box.
(b) Deselect all boundary zones and select w-low-x from the Boundaries selection list.
(c) Click Compute and close the Flux Reports dialog box.

The net heat load is approximately 208.08 W. After comparing the total heat transfer rate and radiation heat transfer rate, it can be concluded that radiation is the dominant mode of heat transfer.
11. Display temperature profile for the side wall.

(a) Select Temperature... and Wall Temperature (Outer Surface) from the Y Axis Function drop-down lists.
(b) Retain the default selection of Direction Vector from the X Axis Function drop-down list.
(c) Select zz_x.side from the Surfaces selection list.
(d) Click Plot (Figure 5.9).
(e) Enable Write to File and click the Write... button to open the Select File dialog box.
   i. Enter tp_1.xy for XY File.
   ii. Click OK in the Select File dialog box.
(f) Disable the Write to File option.
(g) Close the Solution XY Plot dialog box.
12. Save the case and data files (rad_b_1.cas.gz and rad_b_1.dat.gz).

File → Write → Case & Data...

Step 8: Compare the Contour Plots after Varying Radiating Surfaces

1. Increase the number of faces per cluster to 10.

Models → Radiation → Edit...

(a) Click the Set... button in the Parameters group box to open the View Factor and Cluster Parameters dialog box.

i. Enter 10 for Faces per Surface Cluster for Flow Boundary Zones in the Parameters group box.

ii. Click Apply to All Walls and close the View Factor and Cluster Parameters dialog box.
(b) Click Compute/Write... for Methods in the View Factors group box to open the Select File dialog box and to compute the view factors.

Specify a file name where the cluster and view factor parameters will be stored.

i. Enter rad_10.s2s.gz for S2S File.

ii. Click OK in the Select File dialog box.

(c) Click OK to close the Radiation Model dialog box.

2. Initialize the solution.

   Solution Initialization

3. Start the calculation by requesting 650 iterations.

   Run Calculation

   The solution will converge in approximately 612 iterations.

4. Save the case and data files (rad_10.cas.gz and rad_10.dat.gz).

   File → Write → Case & Data...

5. In a manner similar to the steps described in Step 7: 11. (a)–(g), display the temperature profile for the side wall and write it to a file named tp_10_xy.

6. Repeat the procedure outlined in Step 8: 1.–5. for 100, 400, 800, and 1600 faces per surface cluster and save the respective case and data files (e.g., rad_100.cas.gz) and temperature profile files (e.g., tp_100_xy).

7. Display contours of wall temperature (outer surface) for all six cases, in the manner described in Step 7: 2.

   Graphics and Animations → Contours → Set Up...

Figure 5.10: Contours of Wall Temperature (Outer Surface): 1 Face per Surface Cluster
Figure 5.11: Contours of Wall Temperature (Outer Surface): 10 Faces per Surface Cluster

Figure 5.12: Contours of Wall Temperature (Outer Surface): 100 Faces per Surface Cluster

Figure 5.13: Contours of Wall Temperature (Outer Surface): 400 Faces per Surface Cluster
Figure 5.14: Contours of Wall Temperature (Outer Surface): 800 Faces per Surface Cluster

Figure 5.15: Contours of Wall Temperature (Outer Surface): 1600 Faces per Surface Cluster
8. Display contours of surface cluster ID for 1600 faces per surface cluster (Figure 5.16).

(a) Make sure that Filled is enabled in the Options group box.
(b) Make sure that Node Values is disabled.
(c) Select Radiation... and Surface Cluster ID from the Contours of drop-down lists.
(d) Select all surfaces except default-interior and zz.x_side.
(e) Click Display and rotate the figure as shown in Figure 5.16.
(f) Close the Contours dialog box.
9. Read `rad_400.cas.gz` and `rad_400.dat.gz` and, in a similar manner to the previous step, display contours of surface cluster ID (Figure 5.17).

Figure 5.17 shows contours of **Surface Cluster ID** for 400 FPSC. This case shows better clustering compared to all of the other cases.

10. Display the temperature profile plot for 400 FPSC on a plot that includes the temperature profile plots for 1, 10, 100, 800, and 1600 FPSC.

   ![Temperature Profile Plot](https://example.com/temperature_plot.png)

   (a) Make sure that **Write to File** in the Options group is disabled.

   (b) Make sure that **Temperature...** and **Wall Temperature (Outer Surface)** are selected from the **Y Axis Function** drop-down lists.

   (c) Retain the default selection of **Direction Vector** from the **X Axis Function** drop-down list.
(d) Make sure that \texttt{zz\_x\_side} is selected from the \textit{Surfaces} selection list.

(e) Click \textbf{Plot} (Figure 5.18).

(f) Click the \textbf{Load File...} button to open the \textit{Select File} dialog box.
   
   i. Select \texttt{tp\_1.xy}.

   ii. Click \textbf{OK} to close the \textit{Select File} dialog box.

(g) Click \textbf{Plot}.

(h) In a similar manner, click the \textbf{Load File...} button to read the files \texttt{tp\_10.xy}, \texttt{tp\_100.xy}, \texttt{tp\_800.xy}, and \texttt{tp\_1600.xy}, and plot the temperature profiles.

(i) Close the \textit{Solution XY Plot} dialog box.

**Figure 5.18: A Comparison of Temperature Profiles along the Side Wall**

\textbf{Note:} \textit{The legend entries in Figure 5.18 have been changed for display purposes. You will see similar changes in Figure 5.19. You do not need to make these changes.}
Step 9: S2S Definition, Solution and Postprocessing with Partial Enclosure

As mentioned previously, when the S2S model is used, you also have the option to define a “partial enclosure”; i.e., you can disable the view factor calculation for walls with negligible emission/absorption, or walls that have uniform temperature. Even though the view factor will not be computed for these walls, they will still emit radiation at a fixed temperature called the “partial enclosure temperature”. The main advantage of this is to speed up the view factor and the radiosity calculation.

In the steps that follow, you will specify the radiating wall ($w$-low-$x$) as a boundary zone that is not participating in the S2S radiation model. Consequently, you will specify the partial enclosure temperature for the wall. The partial enclosure option may not yield accurate results in cases that have multiple wall boundaries that are not participating in S2S radiation and that each have different temperatures. This is because the a single partial enclosure temperature is applied to all of the non-participating walls.

1. Read the case file saved previously for the S2S model (rad_b_1.cas.gz).

2. Set the partial enclosure parameters for the S2S model.

   (a) Click the Radiation tab.

   (b) Disable Participates in S2S Radiation in the S2S Parameters group box.

   (c) Click OK to close the Wall dialog box.
3. Compute the view factors for the S2S model.

   ![Radiation Model](image)

   (a) Enter 473 K for Temperature in the Partial Enclosure group box.

   (b) Click Compute/Write... for Methods in the View Factors group box to open the Select File dialog box and to compute the view factors.

   The view factor file will store the view factors for the radiating surfaces only. This may help you control the size of the view factor file as well as the memory required to store view factors in ANSYS FLUENT. Furthermore, the time required to compute the view factors will reduce as only the view factors for radiating surfaces will be calculated.

   **Note:** You should compute the view factors only after you have specified the boundaries that will participate in the radiation model using the Boundary Conditions dialog box. If you first compute the view factors and then make a change to the boundary conditions, ANSYS FLUENT will use the view factor file stored previously for calculating a solution, in which case, the changes that you made to the model will not be used for the calculation. Therefore, you should recompute the view factors and save the case file whenever you modify the number of objects that will participate in radiation.

   i. Enter `rad_partial.s2s.gz` as the file name for S2S File.

   ii. Click OK in the Select File dialog box.
(c) Click OK to close the Radiation Model dialog box.

4. Initialize the solution.
   Solution Initialization

5. Start the calculation by requesting 650 iterations.
   Run Calculation
   The solution will converge in approximately 631 iterations.

6. Save the case and data files (rad_partial.cas.gz and rad_partial.dat.gz).
   File→Write→Case & Data...

7. Compute the radiation heat transfer rate.
   Reports→Fluxes→Set Up...

   (a) Make sure that Radiation Heat Transfer Rate is selected in the Options group box.

   (b) Select all boundary zones except default-interior from the Boundaries selection list.

   (c) Click Compute and close the Flux Reports dialog box.

8. Compare the temperature profile for the side wall to the profile saved in tp_1.xy.
   Plots→XY Plot→Set Up...

   (a) Select all of items in the File Data selection list and click Free Data.
(b) Display the temperature profile and write it to a file named `tp_partial.xy`, in a manner similar to the instructions shown in Step 7: 11. (a)–(f).

(c) Read and display the temperature profile saved in `tp_1.xy`, in a manner similar to the instructions shown in Step 8: 10. (f)–(g).

(d) Close the Solution XY Plot dialog box.

![Diagram of temperature profile](image)

**Figure 5.19: Temperature Profile—With and Without Partial Enclosure (1 FPSC)**

**Summary**

In this tutorial you studied combined natural convection and radiation in a three-dimensional square box and compared the performance of surface-to-surface (S2S) radiation models in ANSYS FLUENT for various radiating surfaces. The S2S radiation model is appropriate for modeling the enclosure radiative transfer without participating media whereas the methods for participating radiation may not always be efficient.

For more information about the surface-to-surface (S2S) radiation model, see Section 13.3 in the separate User’s Guide.

**Further Improvements**

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Tutorial 1.