Chapter 7: Modeling Radiation and Natural Convection

This tutorial is divided into the following sections:

7.1. 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.

7.2. Prerequisites

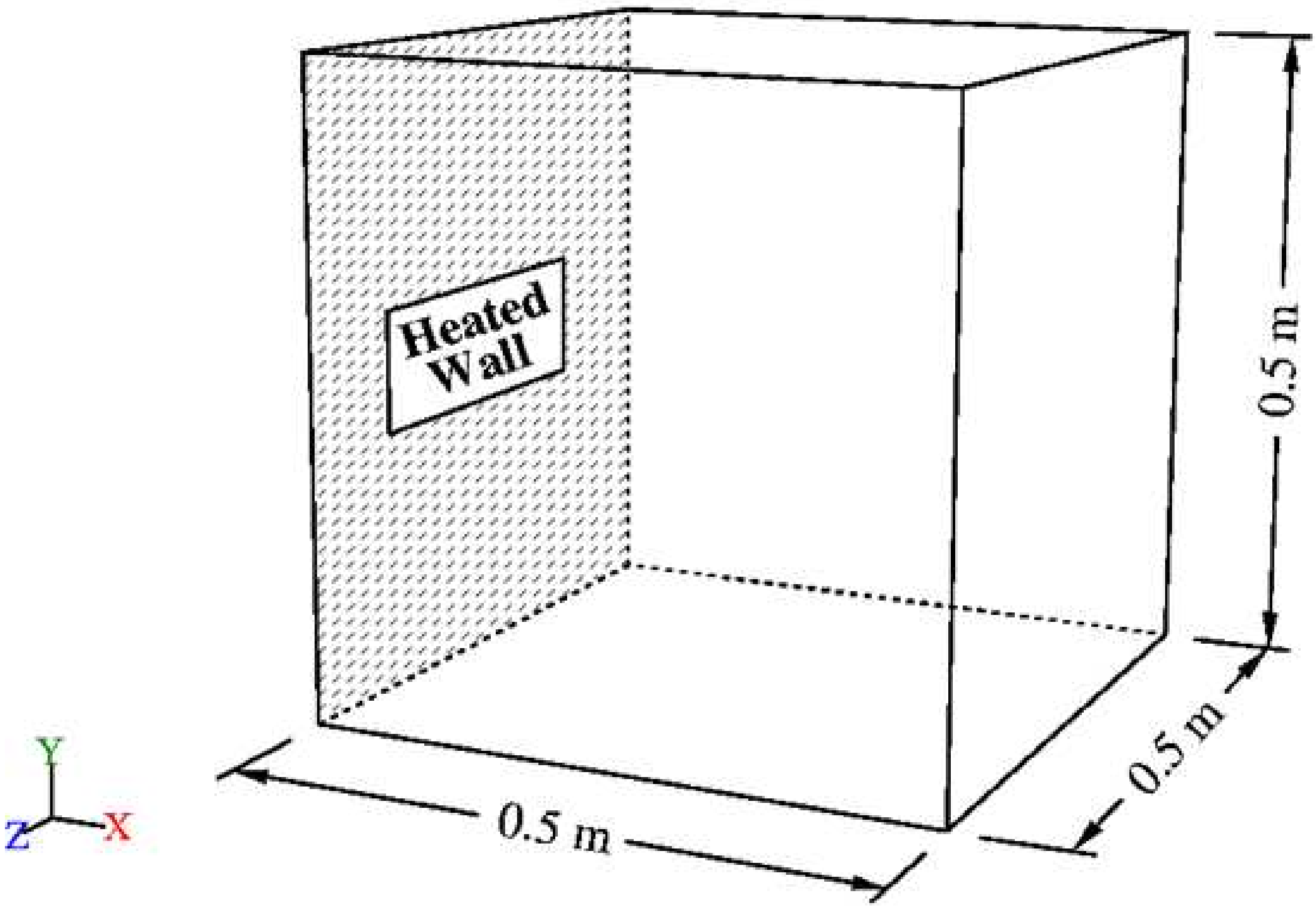
This tutorial is written with the assumption that you have completed *Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 111), and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

7.3. Problem Description

The problem to be considered is shown schematically in *Figure 7.1* (p. 278). A three-dimensional box $(0.5 \times 0.5 \times 0.5)$ 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 5×10^8. This means the flow is most likely laminar. The Planck number $k / \left(4 \sigma L T_0^3\right)$ is 0.003, and measures the relative importance of conduction to radiation.

Figure 7.1 Schematic of the Problem



7.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

7.4.1. Preparation

1. Download `radiation_natural_convection.zip` from the ANSYS Customer Portal or the User Services Center to your working folder (as described in *Preparation* (p. 4) of *Introduction to Using ANSYS FLUENT in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 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 Starting ANSYS FLUENT Using FLUENT Launcher in the 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.

7.4.2. Step 1: Mesh

1. Read the mesh file `rad.msh.gz`.

   File → **Read** → **Mesh...**

   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.

7.4.3. Step 2: General Settings

General

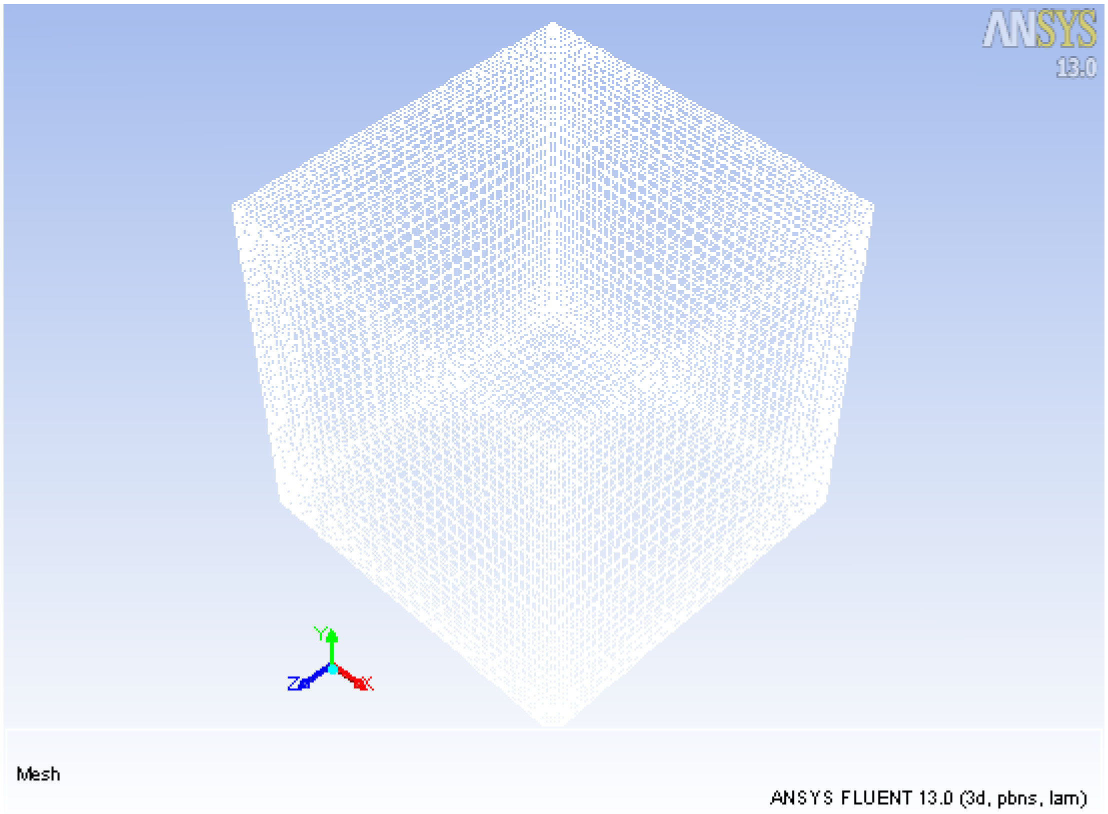
1. Check the mesh.

   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.

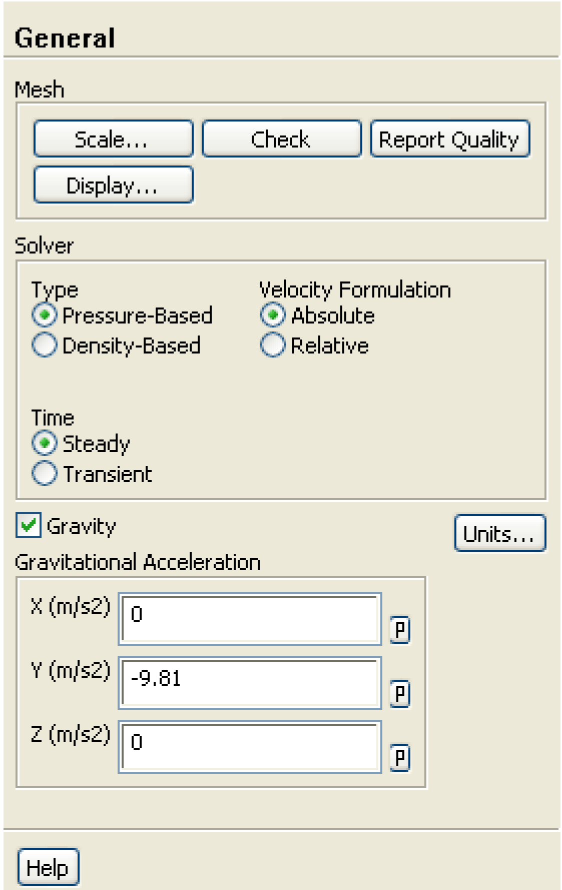
2. Examine the mesh.

Figure 7.2 Graphics Display of Mesh



3. Retain the default solver settings.

General



4. Enable **Gravity**.

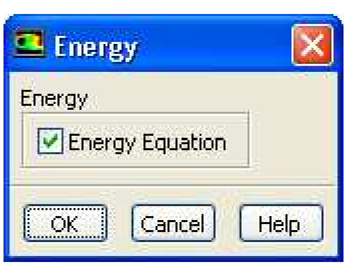
   a. Enter `-9.81` m/s^2 for **Gravitational Acceleration** in the **Y** direction.

7.4.4. Step 3: Models

Models

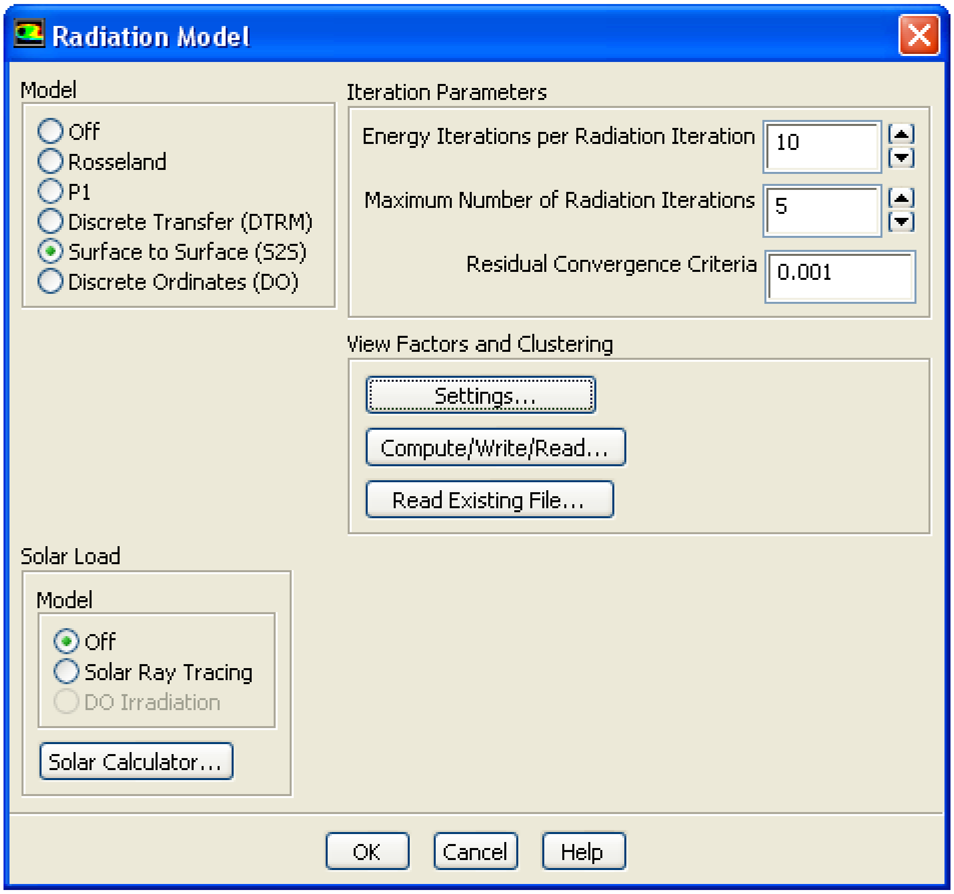
1. Enable the energy equation.

   Models → **Energy** → **Edit...**



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

   Models → **Radiation** → **Edit...**



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".

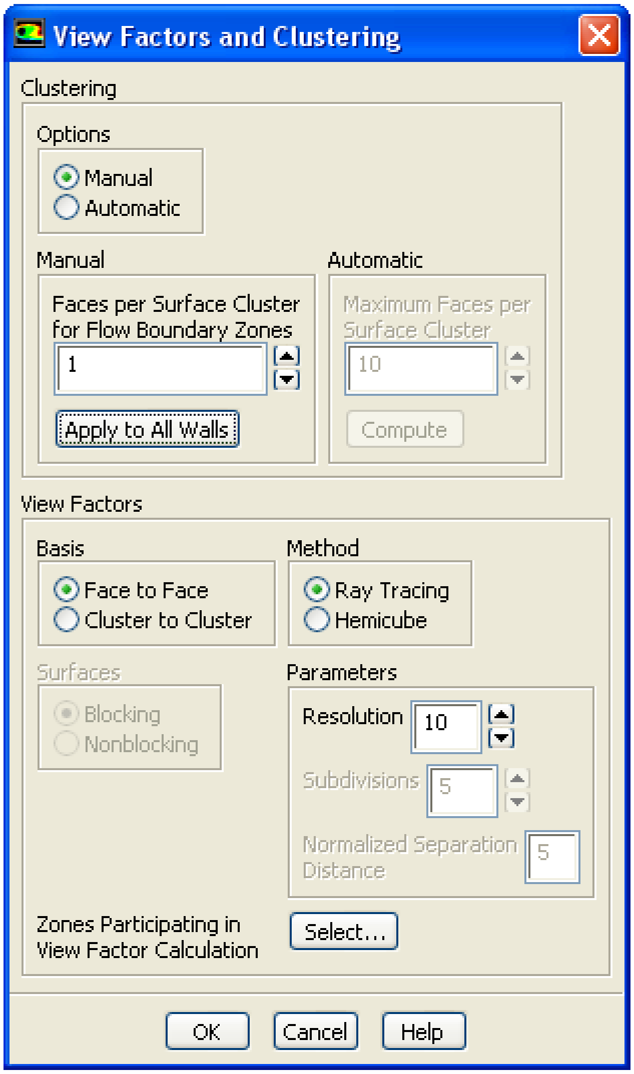
The S2S model assumes that all surfaces are gray and diffuse. Thus according to the gray-body model, if a certain amount of radiation is incident on a surface, then a fraction is reflected, a fraction is absorbed, and a fraction is transmitted. The main assumption of the S2S model is that any absorption, 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 **Settings...** button to open the **View Factors and Clustering** dialog box.

You will define the view factor and cluster parameters.



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

ii. 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. Ensure **Ray Tracing** is selected from the **Method** list in the **View Factors** group box.

iv. Click **OK** to close the **View Factors and Clustering** dialog box.

c. Click **Compute/Write/Read...** in the **View Factors and Clustering** 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.

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.

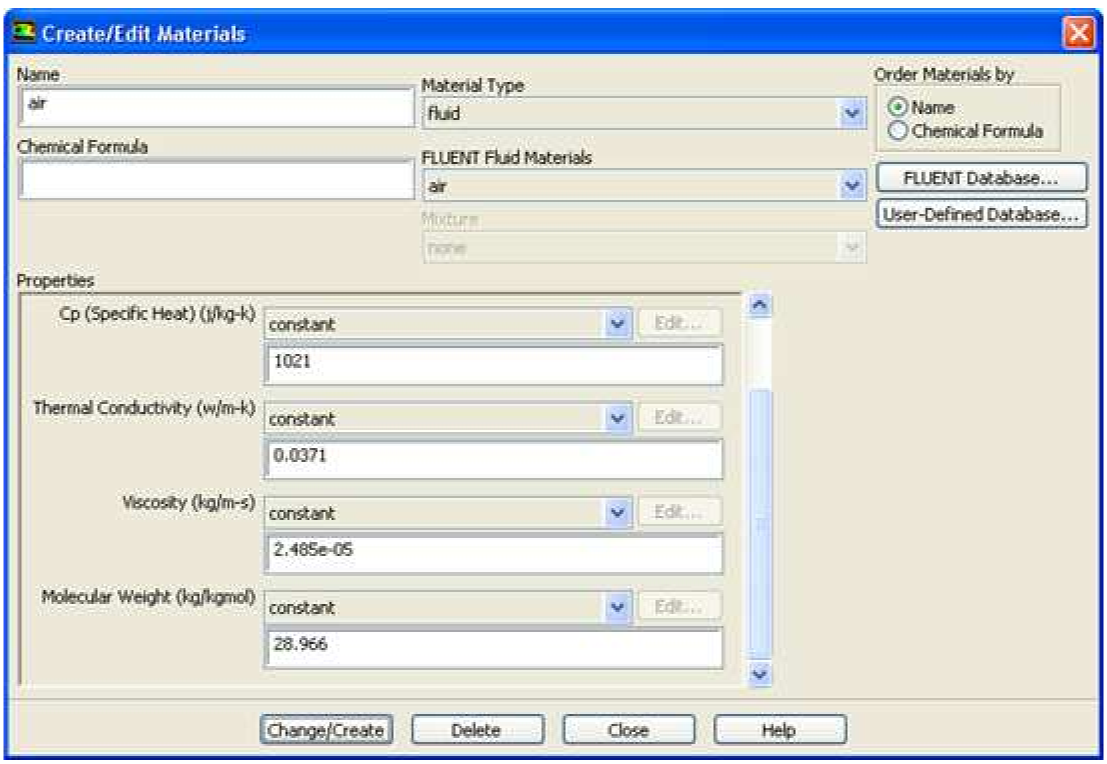
d. Click **OK** to close the **Radiation Model** dialog box.

7.4.5. Step 4: Materials

Materials

1. Set the properties for air.

Materials → **air** → **Create/Edit...**



a. Select **incompressible-ideal-gas** from the **Density** drop-down list.

b. Enter `1021` J/kg-K for **Cp (Specific Heat)**.

c. Enter `0.0371` W/m-K for **Thermal Conductivity**.

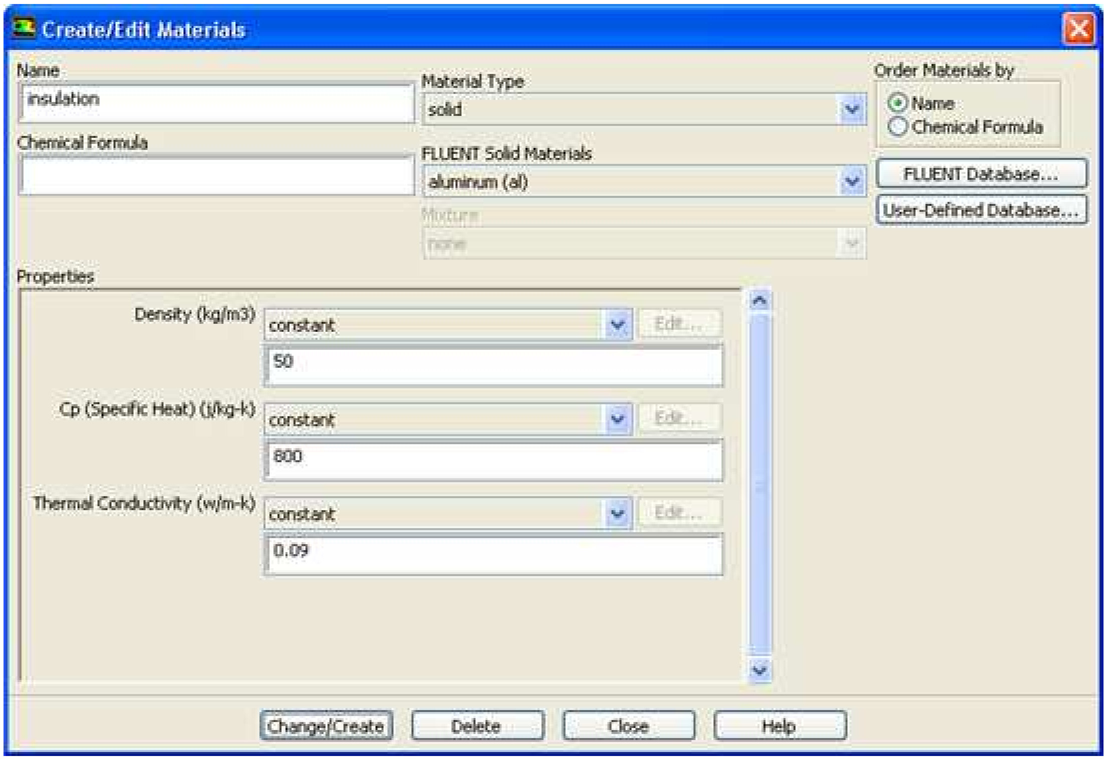
d. Enter `2.485e-05` kg/m-s for **Viscosity**.

e. Retain the default value of `28.966` kg/kmol for **Molecular Weight**.

f. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

2. Define the new material, **insulation**.

Materials → **Solid** → **Create/Edit...**



a. Enter `insulation` for **Name** and delete the entry in the **Chemical Formula** field.

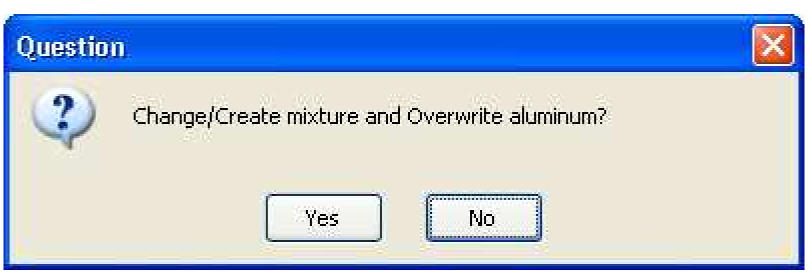
b. Enter `50` kg/m^3 for **Density**.

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**.*



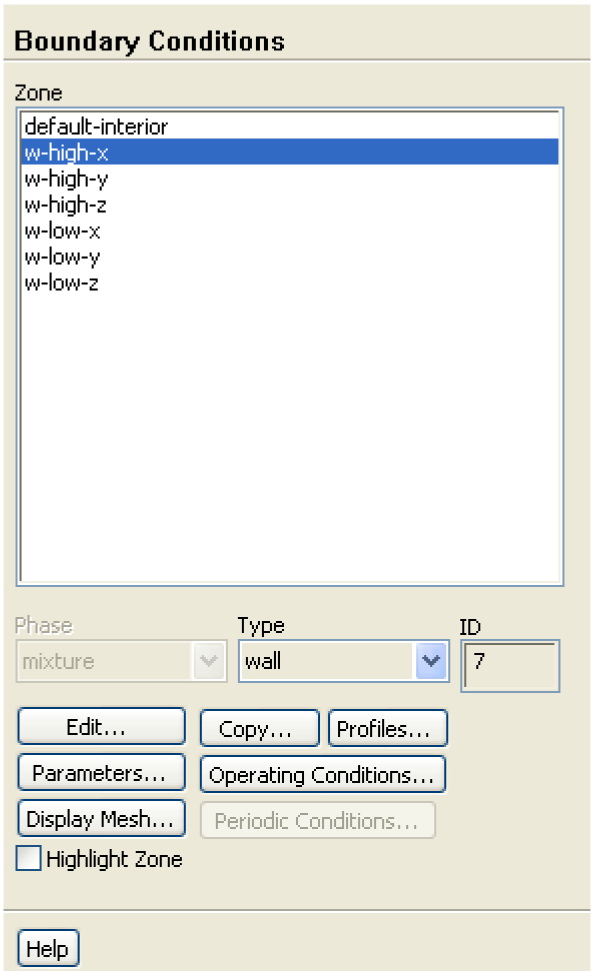
f. Click **No** in the **Question** dialog box to retain **aluminum** and add the new material (**insulation**) to the materials list.

*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.

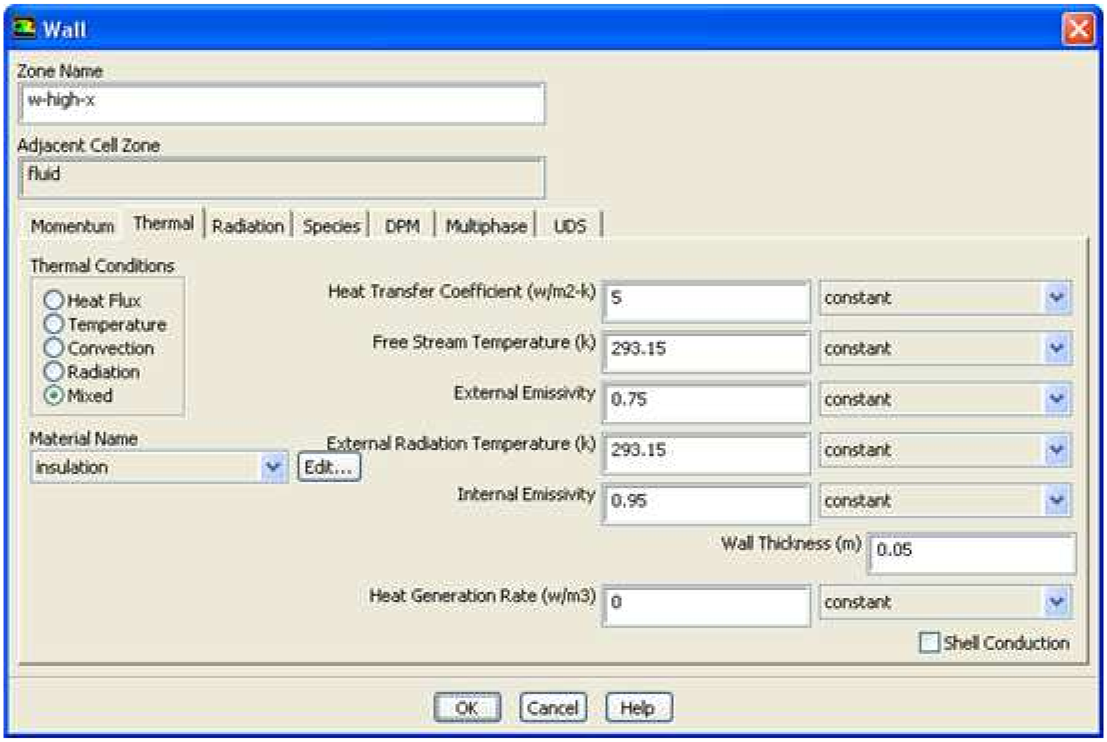
7.4.6. Step 5: Boundary Conditions

Boundary Conditions



1. Set the boundary conditions for the front wall (**w-high-x**).

   Boundary Conditions → **w-high-x** → **Edit...**



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**.

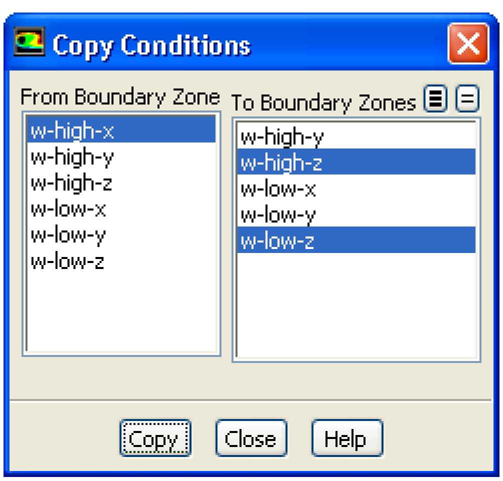
f. Enter `0.95` for **Internal Emissivity**.

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-high-z** and **w-low-z**.

Boundary Conditions → **Copy...**

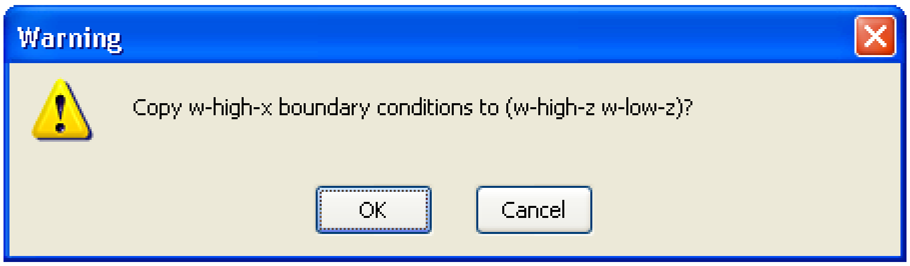


a. Select **w-high-x** from the **From Boundary Zone** selection list.

b. Select **w-high-z** and **w-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-high-x** to **w-high-z** and **w-low-z**.*

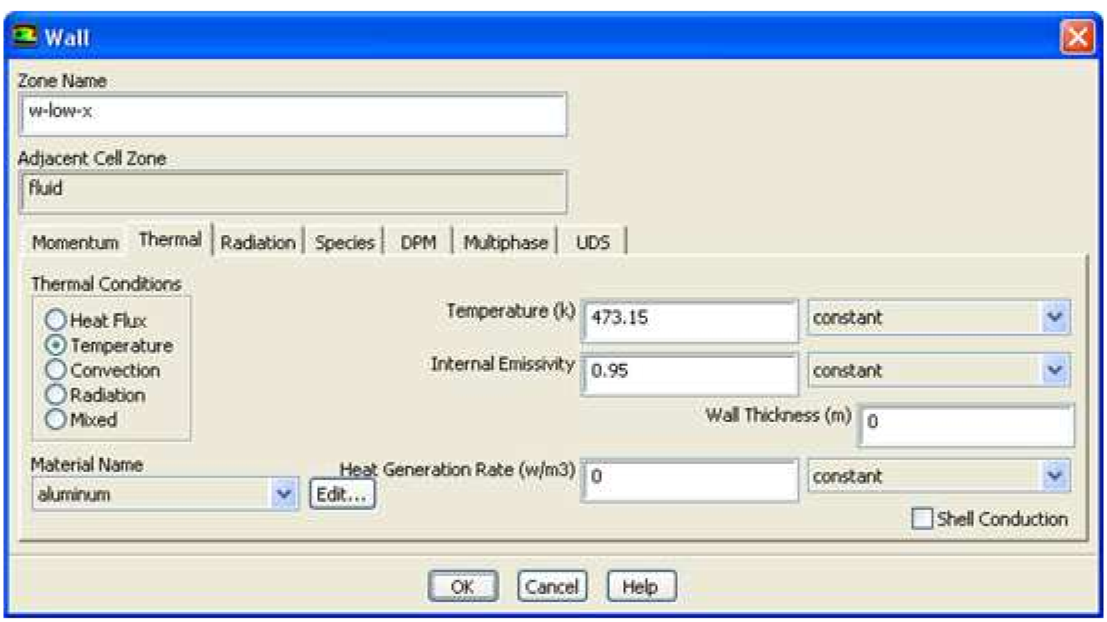


d. Click **OK** in the **Warning** dialog box.

e. Close the **Copy Conditions** dialog box.

3. Set the boundary conditions for the heated wall (**w-low-x**).

Boundary Conditions → **w-low-x** → **Edit...**



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.

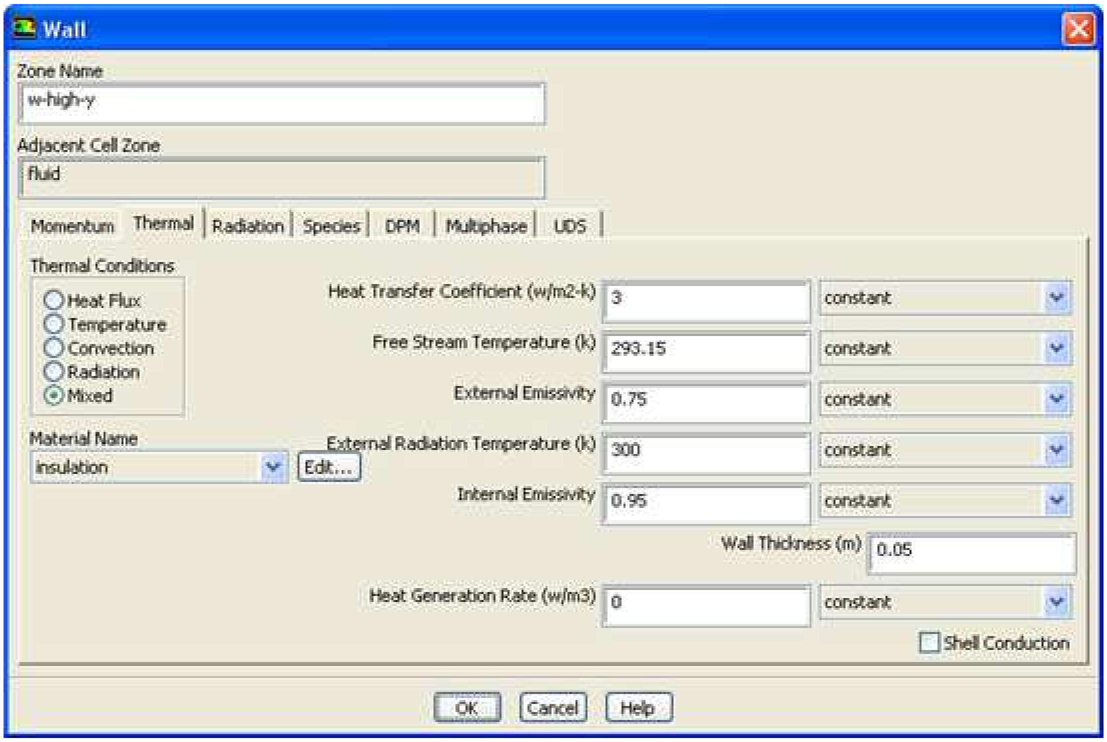
c. Enter `473.15` K for **Temperature**.

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**).

Boundary Conditions → **w-high-y** → **Edit...**



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 $w/m^2 - K$ 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**.

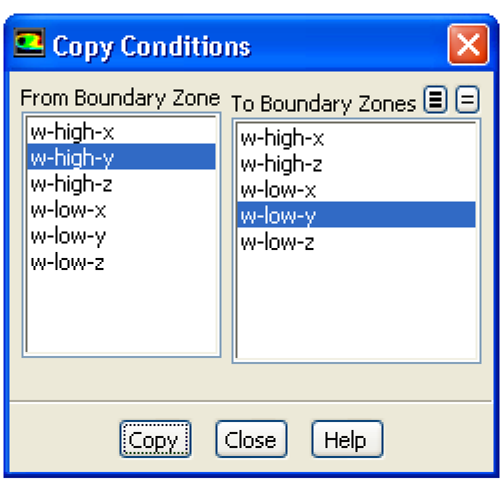
f. Enter 0.95 for **Internal Emissivity**.

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

h. Click **OK** to close the **Wall** dialog box.

5. Copy boundary conditions to define the bottom wall (**w-low-y**).

Boundary Conditions → **Copy...**

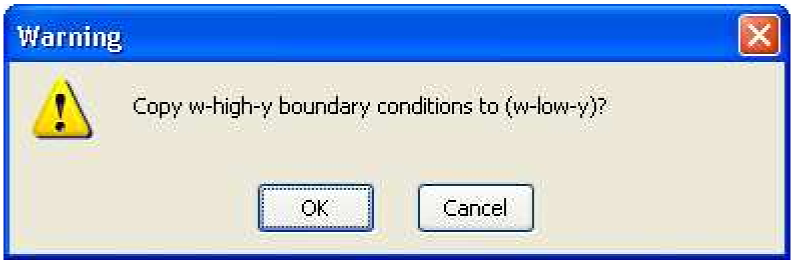


a. Select **w-high-y** from the **From Boundary Zone** selection list.

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**.*



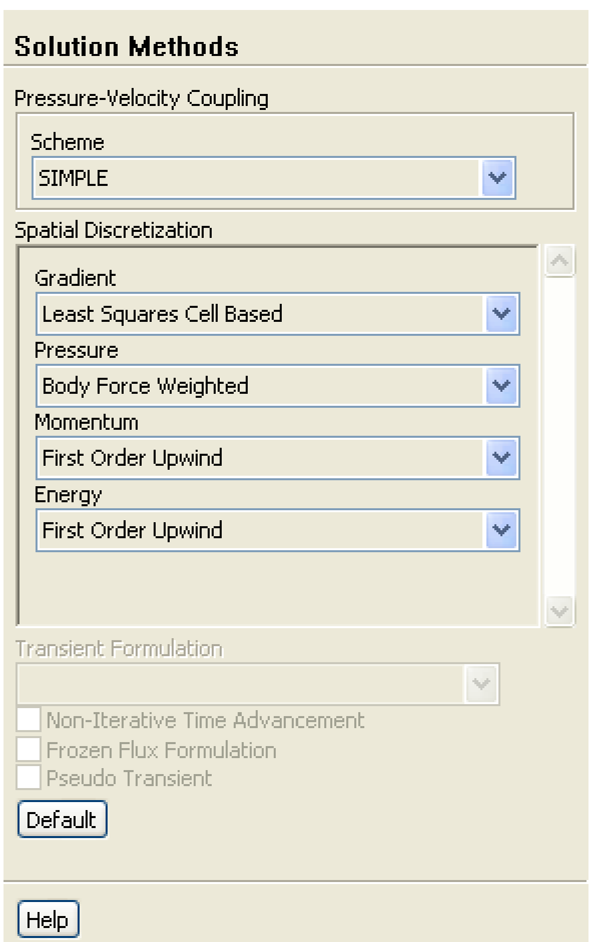
d. Click **OK** in the **Warning** dialog box.

e. Close the **Copy Conditions** dialog box.

7.4.7. Step 6: Solution

1. Set the solution parameters.

Solution Methods

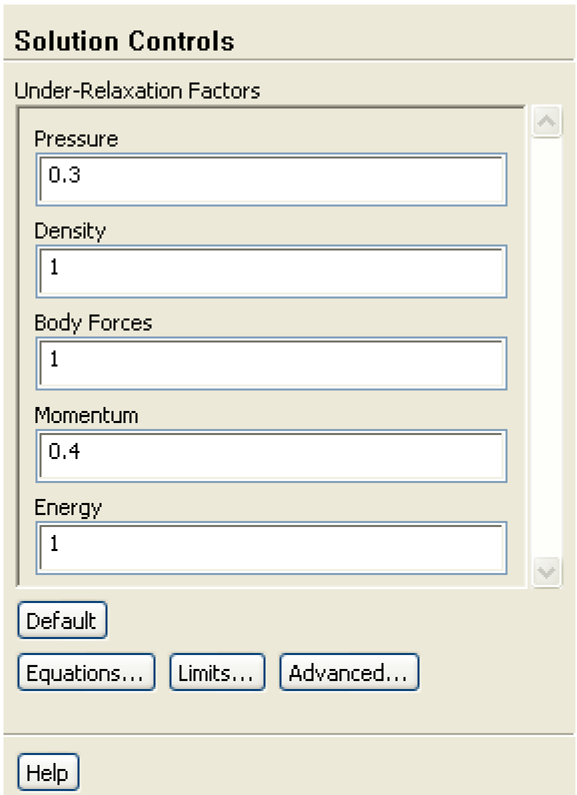


a. Select **Body Force Weighted** from the **Pressure** drop-down list in the **Spatial Discretization** group box.

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

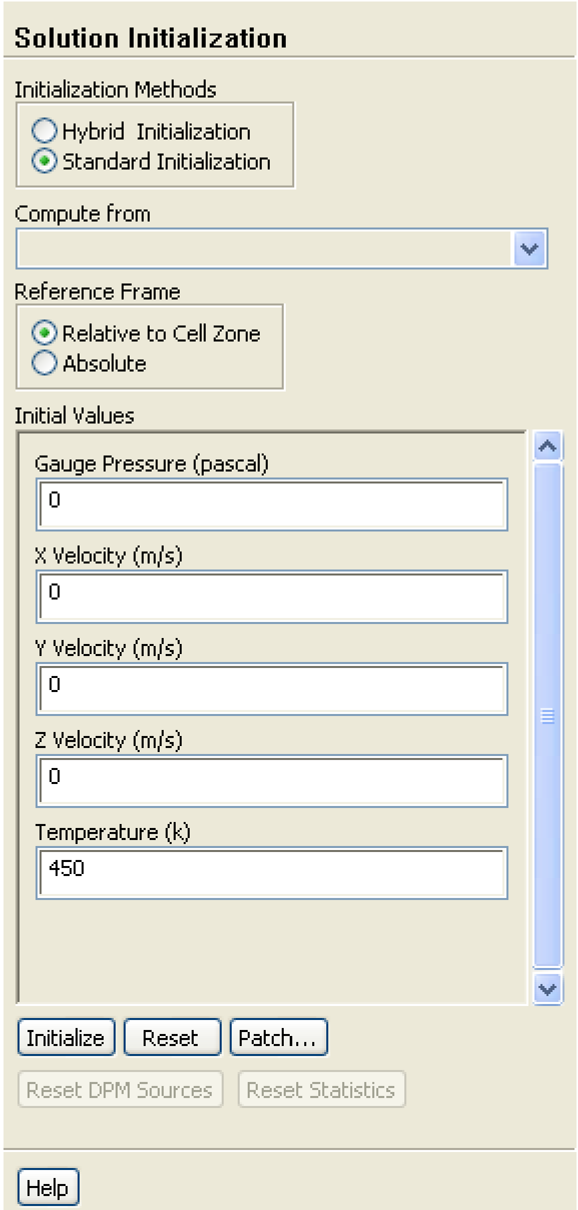


a. Enter `0.4` for **Momentum**.

Buoyancy driven cases will need stiffer relaxation for better results. A good starting point for momentum would be 0.4.

3. Initialize the solution.

Solution Initialization



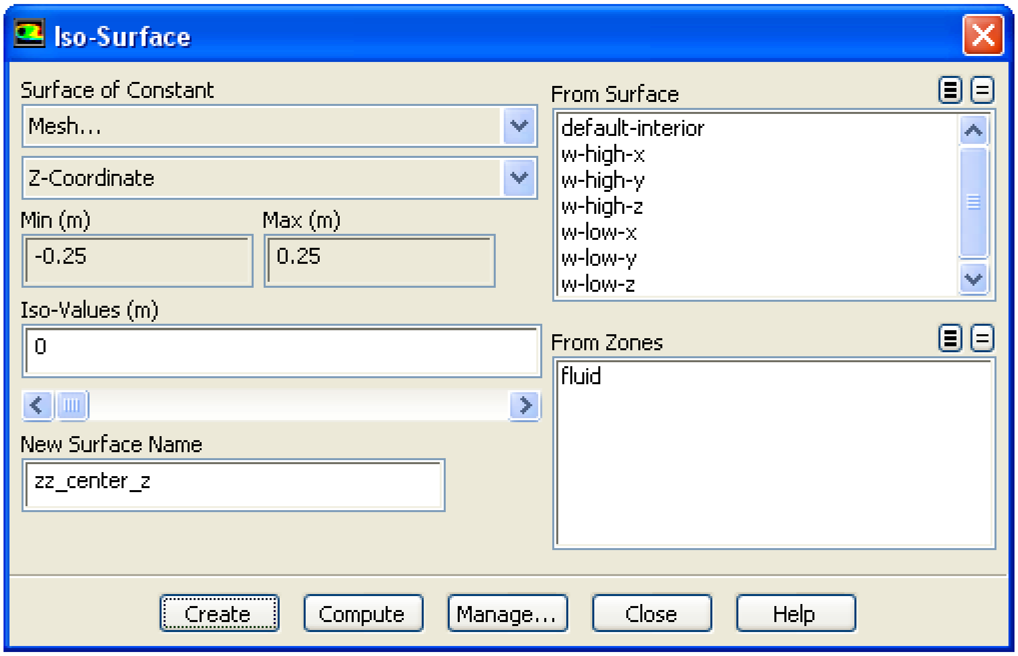
a. Ensure that **Standard Initialization** is selected from the **Initialization Methods** group box.

b. Enter `450` for **Temperature**.

c. Click **Initialize**.

4. Create the new surface, **zz_center_z**.

Surface → **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_center_z` for **New Surface Name**.

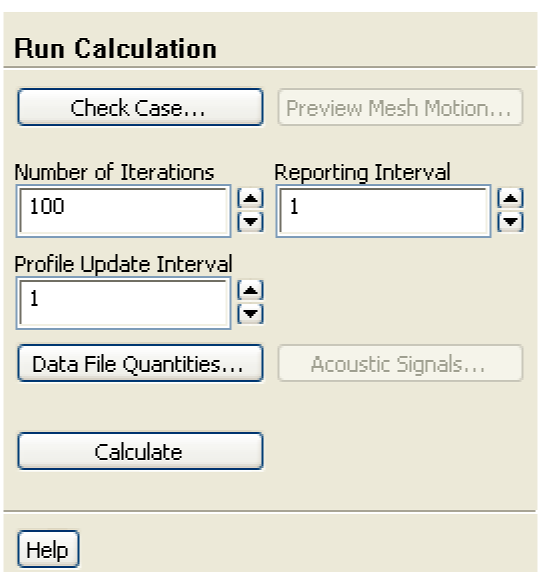
d. Click **Create** and close the **Iso-Surface** dialog box.

5. Save the case file (`rad_a_1.cas.gz`)

   File → **Write** → **Case...**

6. Start the calculation by requesting 100 iterations (*Figure 7.3* (p. 294)).

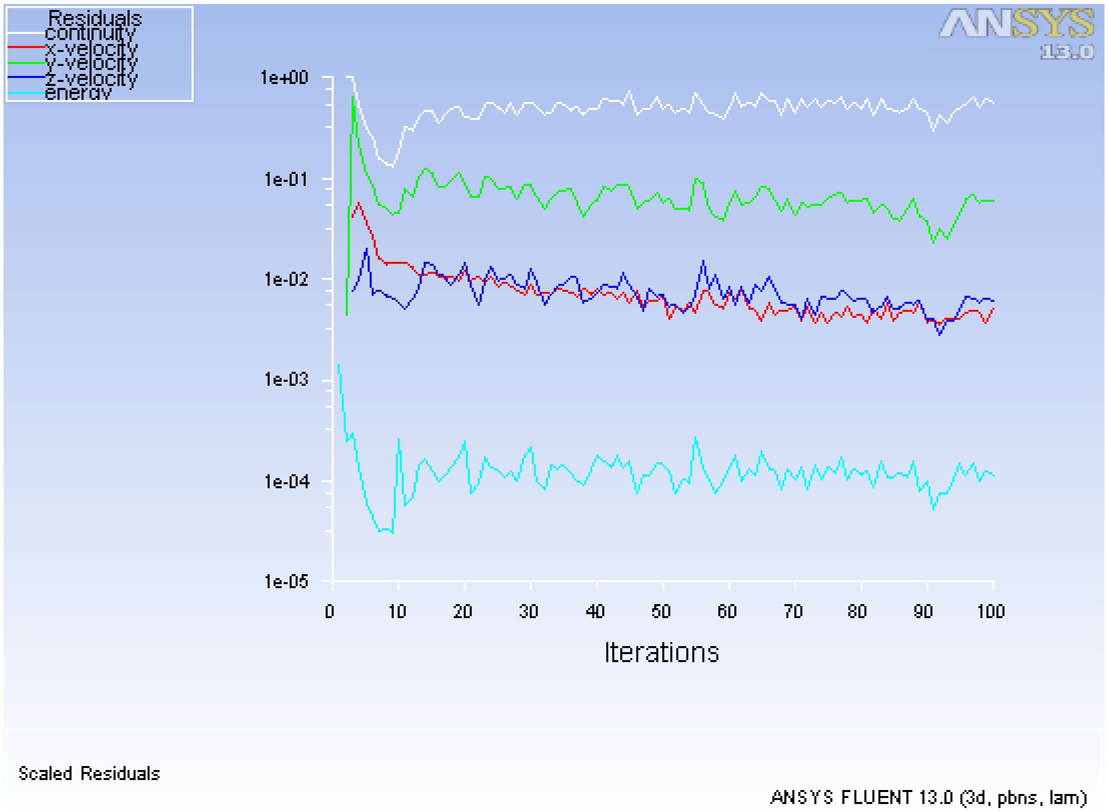
   Run Calculation



a. Enter `100` for **Number of Iterations**.

b. Click **Calculate**.

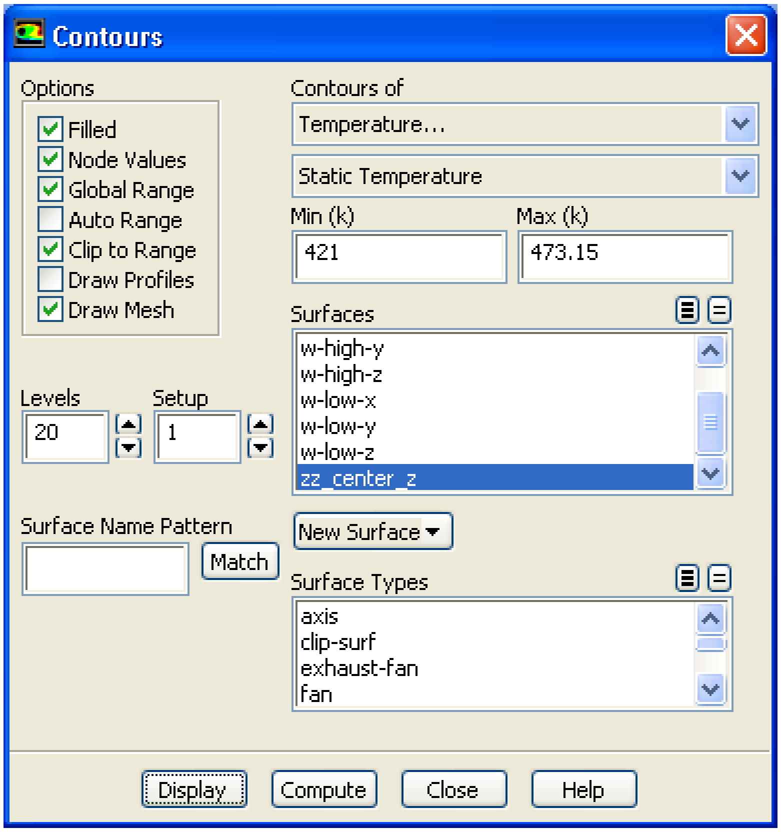
Figure 7.3 Scaled Residuals



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.

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



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.

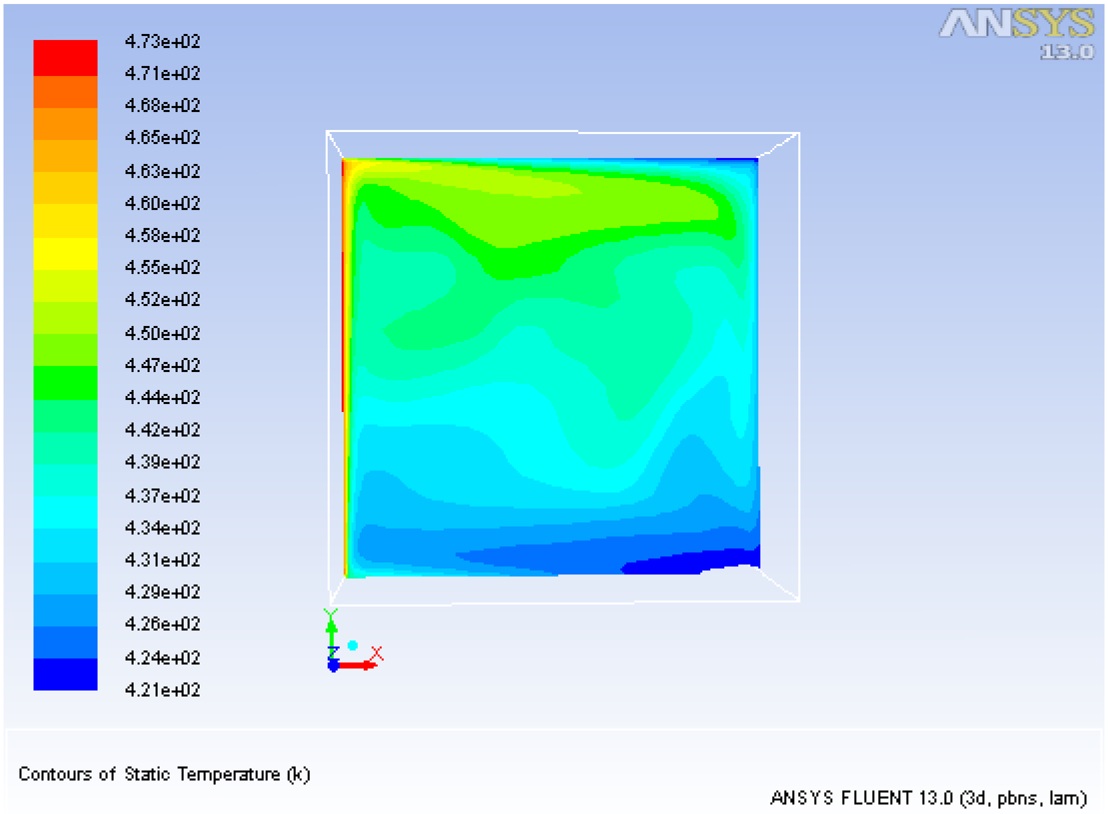
e. Disable **Auto Range**.

f. Enter `421` for **Min** and `473.15` for **Max**.

g. Click **Display** and rotate the view as shown in *Figure 7.4* (p. 296).

h. Close the **Contours** dialog box.

Figure 7.4 Contours of Static Temperature



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.

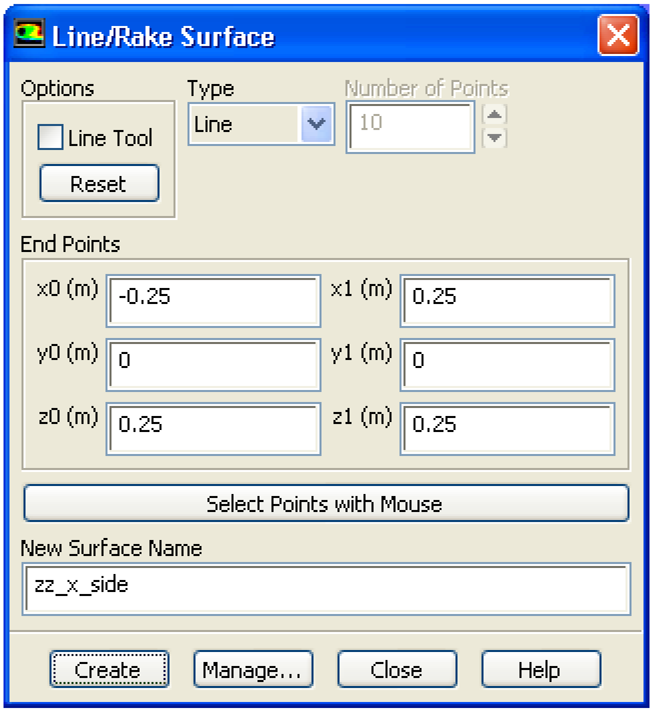
9. Request 100 more iterations.

   Run Calculation

7.4.8. Step 7: Postprocessing

1. Create the new surface, **zz_x_side**.

   Surface → **Line/Rake...**



a. Enter `(-0.25, 0, 0.25)` for **(x0, y0, z0)** respectively.

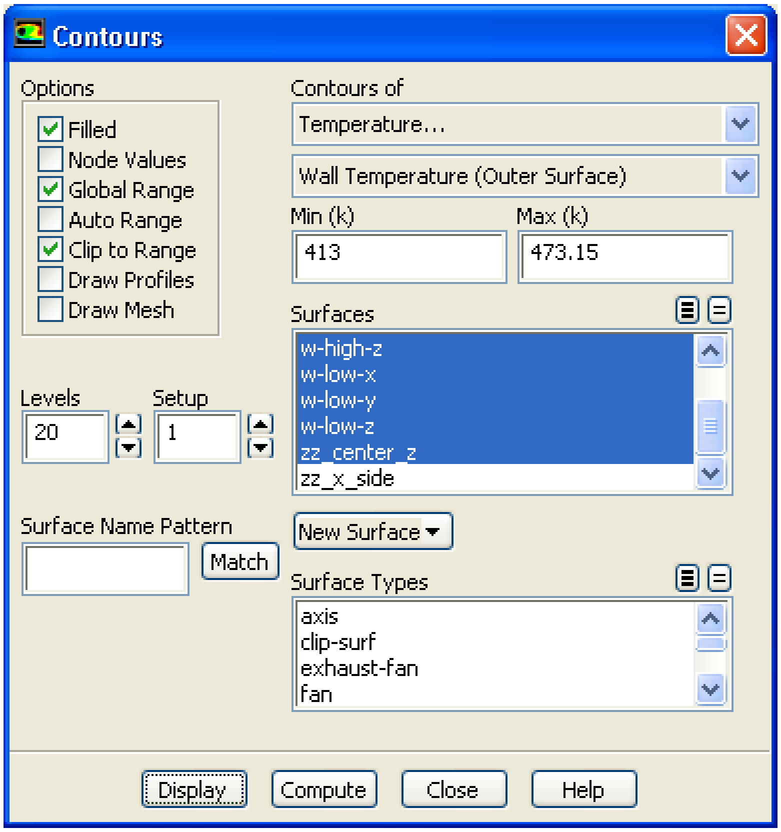
b. Enter `(0.25, 0, 0.25)` for **(x1, y1, z1)** respectively.

c. Enter `zz_x_side` for **New Surface Name**.

d. 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.

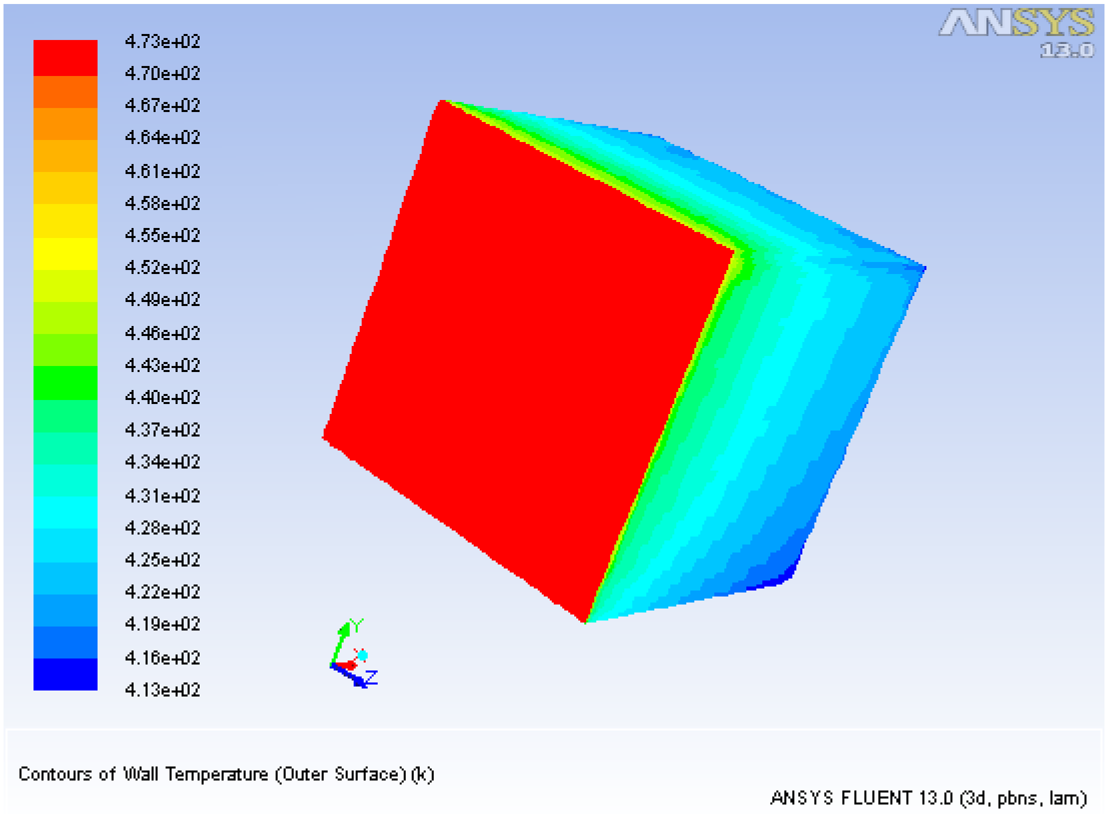
d. Select all surfaces except **default-interior** and **zz_x_side**.

e. Disable **Auto Range** and **Draw Mesh**.

f. Enter `413` for **Min** and `473.15` for **Max**.

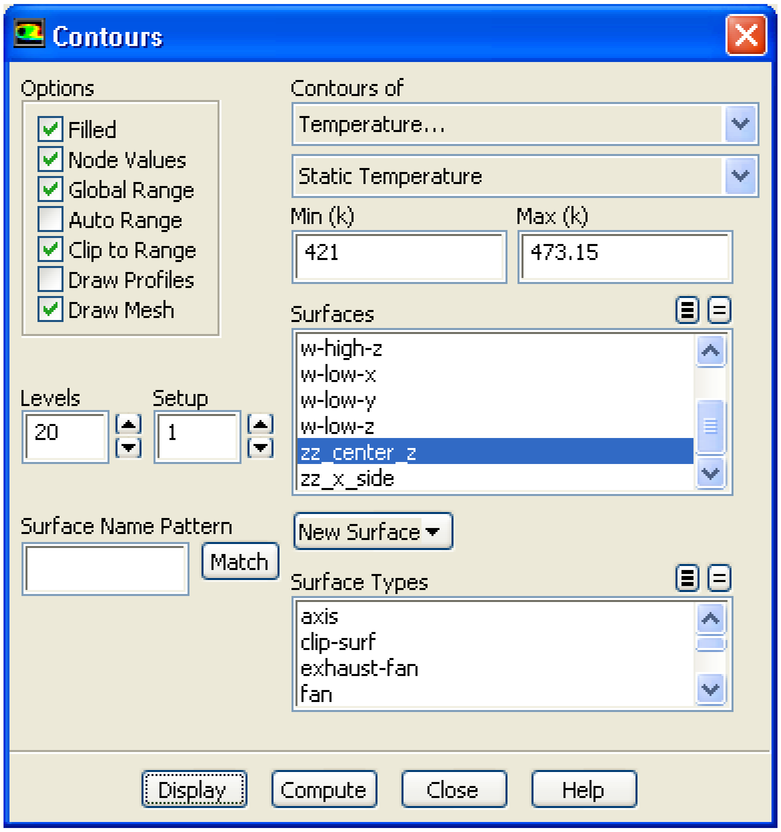
g. Click **Display** and rotate the view as shown in *Figure 7.5* (p. 299).

Figure 7.5 Contours of Wall Temperature



3. Display contours of static temperature.

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



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.

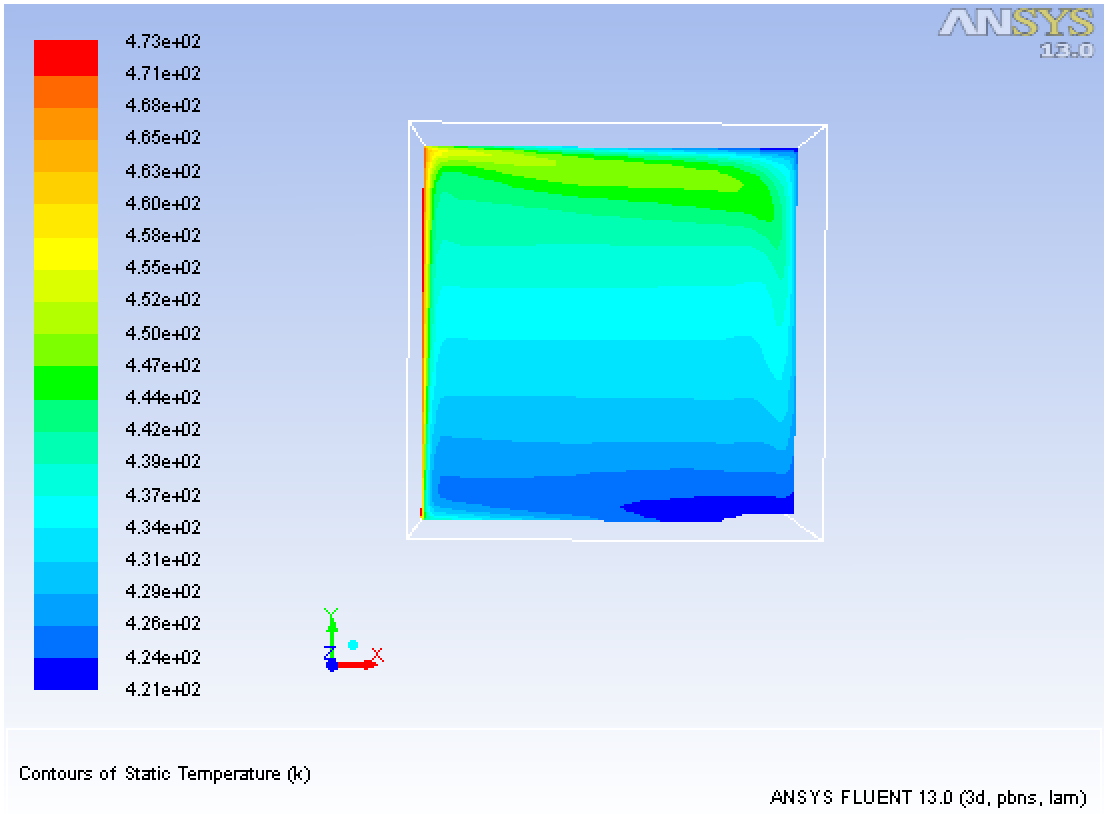
e. Enable **Node Values**.

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 7.6* (p. 301).

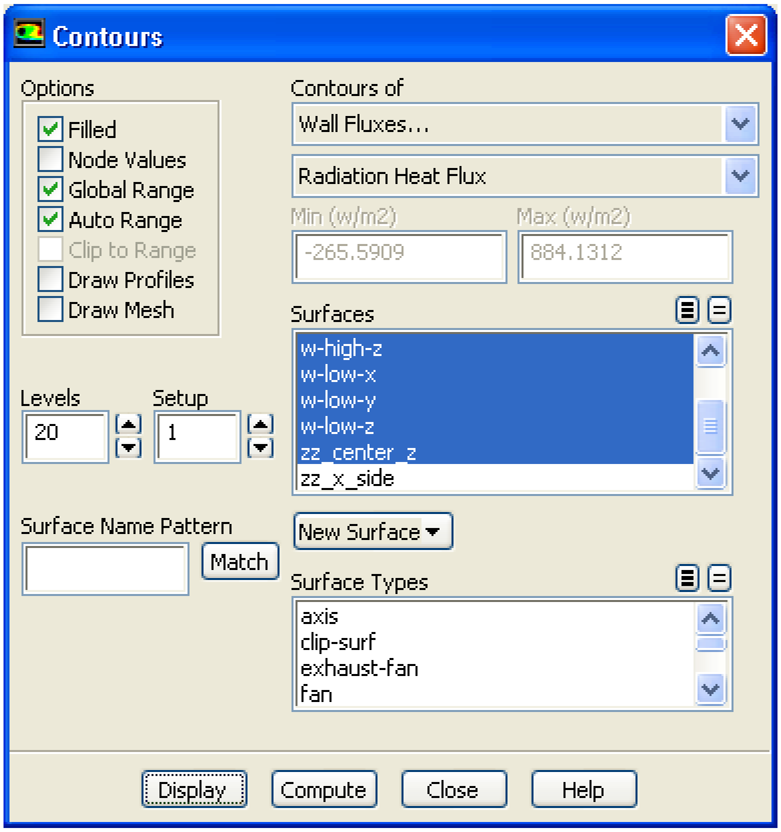
Figure 7.6 Contours of Static Temperature



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

4. Display contours of radiation heat flux.

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.

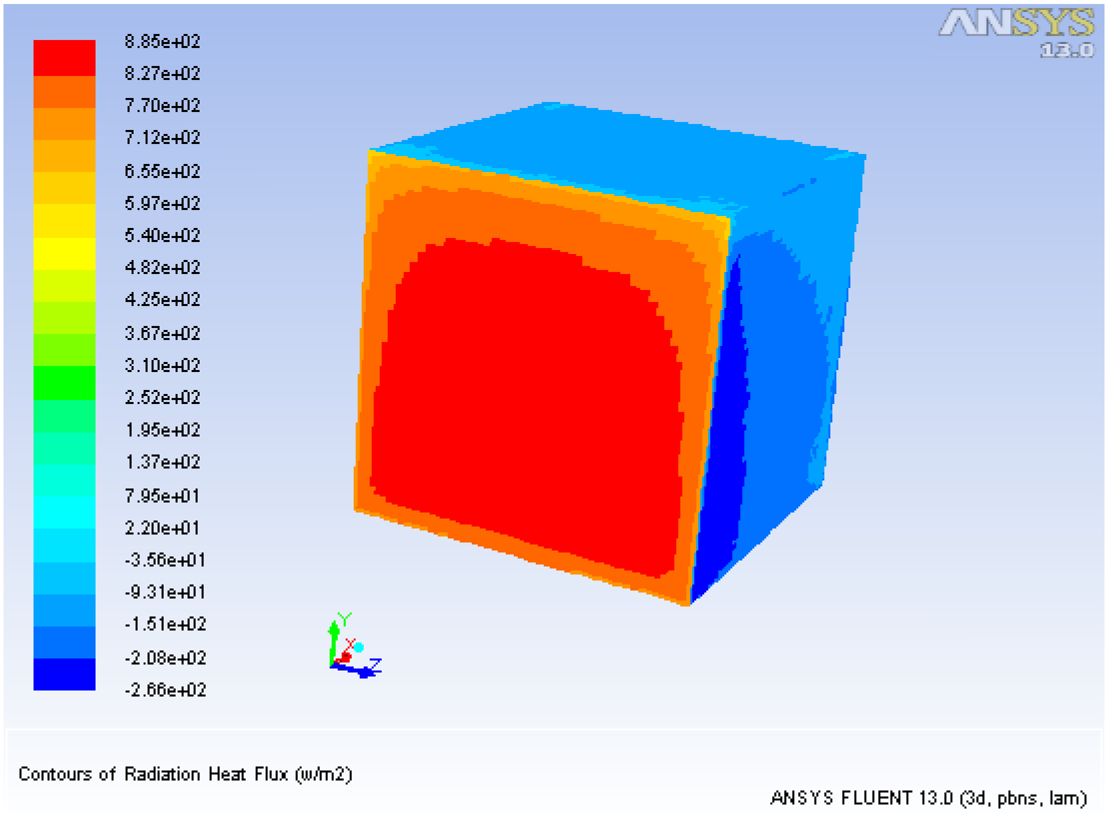
d. Select all surfaces except **default-interior** and **zz_x_side**.

e. Click **Display** and rotate the view as shown in *Figure 7.7* (p. 303).

f. Close the **Contours** dialog box.

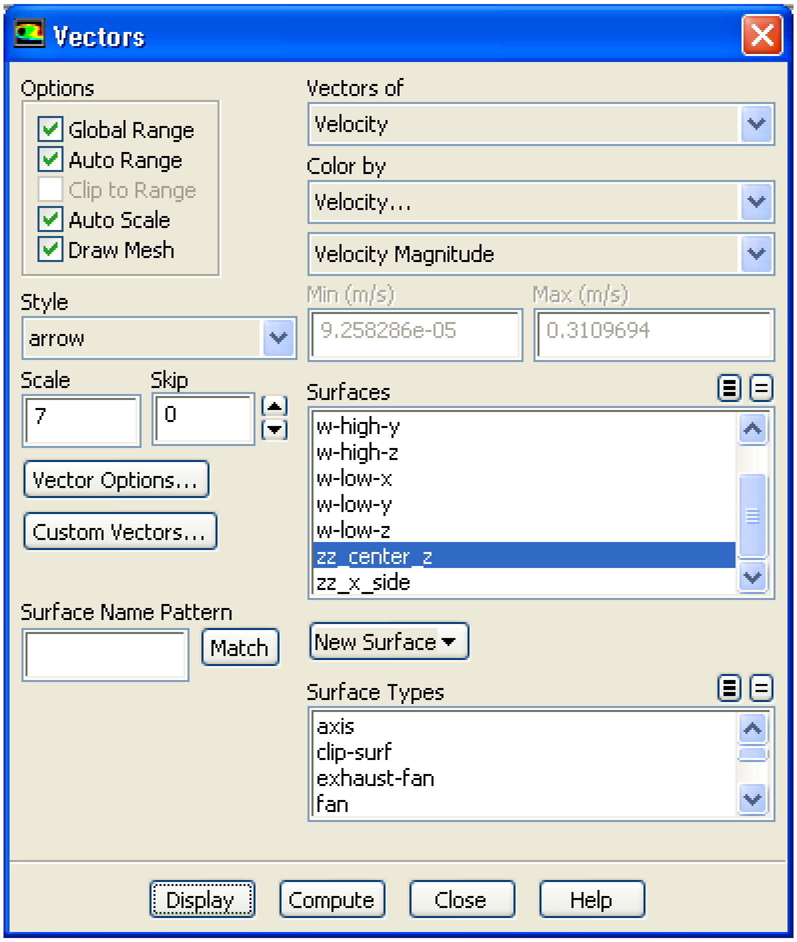
*Figure 7.7 (p. 303)shows the radiating wall (**w-low-x**) with positive heat flux and all other walls with negative heat flux.*

Figure 7.7 Contours of Radiation Heat Flux



5. Display vectors of velocity magnitude.

Graphics and Animations → **Vectors** → **Set Up...**



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.

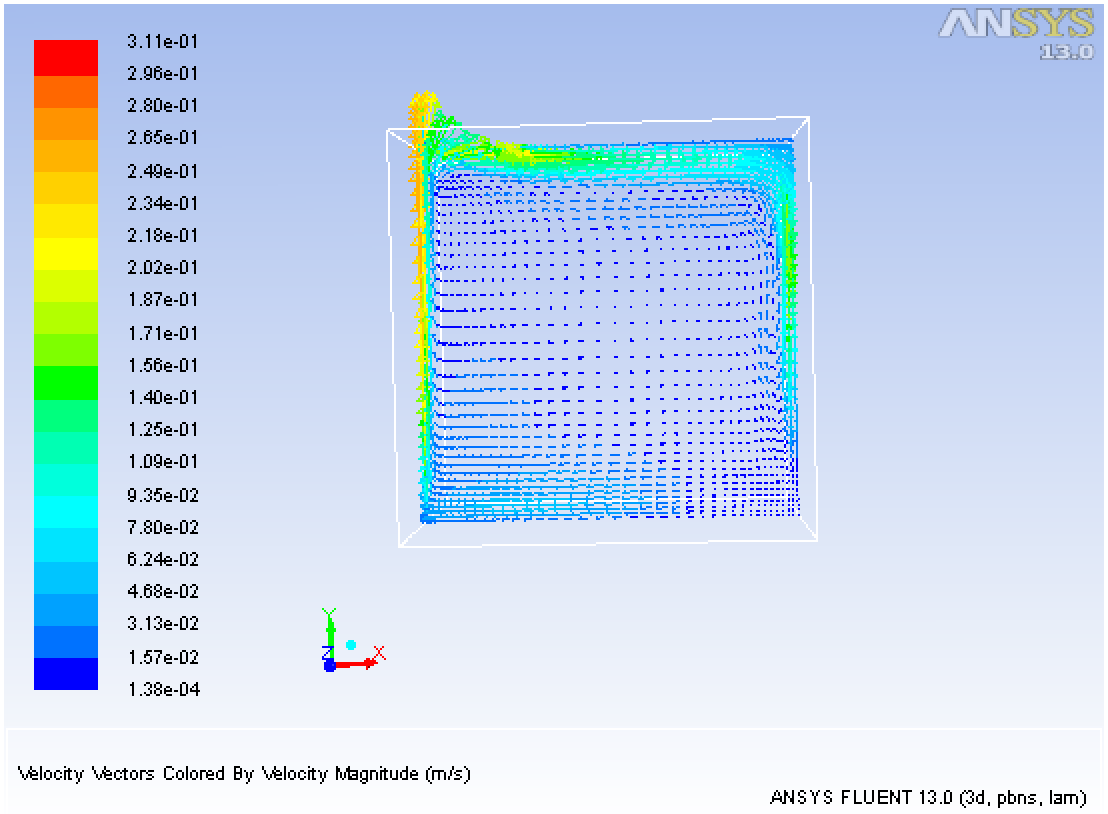
   ii. Click **Display** and close the **Mesh Display** dialog box.

e. Enter 7 for **Scale**.

f. Click **Display** and rotate the view as shown in *Figure 7.8* (p. 305).

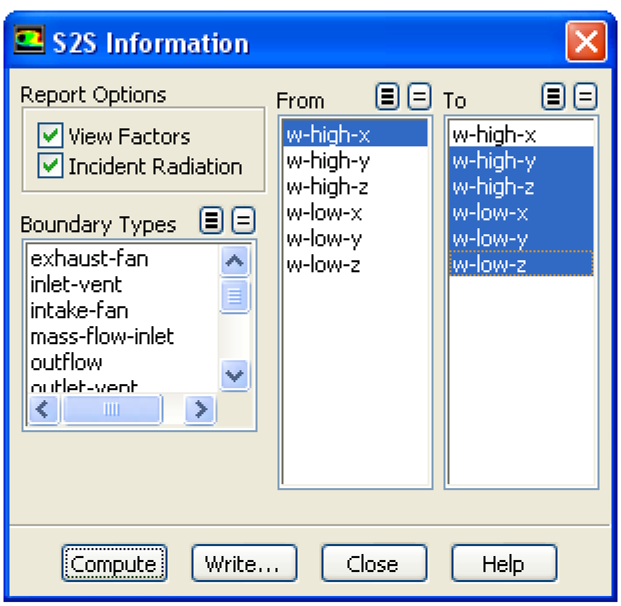
g. Close the **Vectors** dialog box.

Figure 7.8 Vectors of Velocity Magnitude



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.

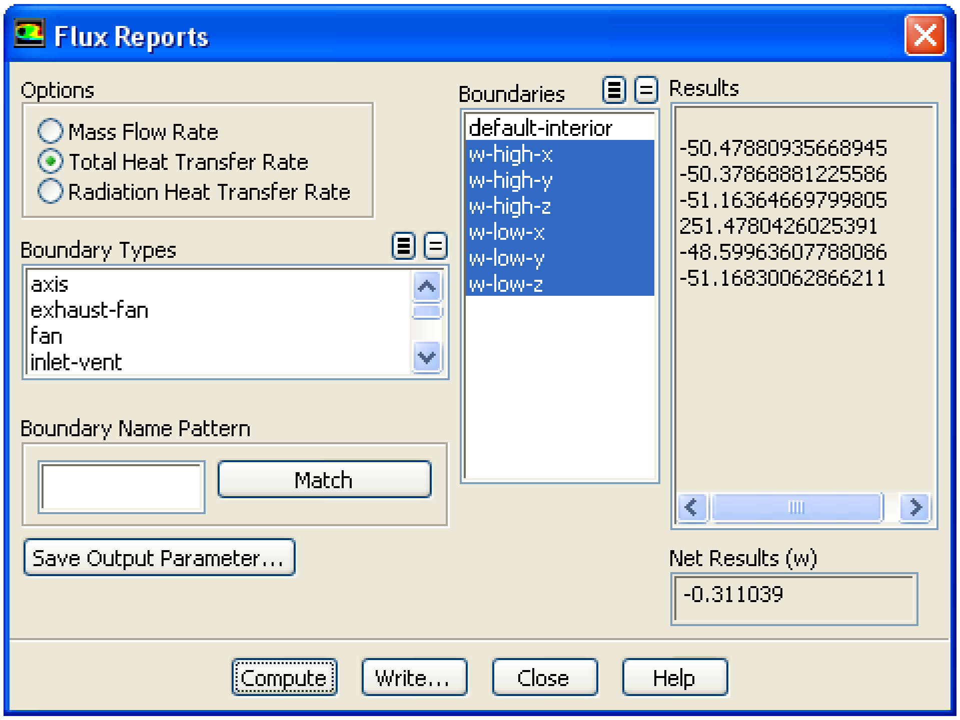
d. Select all zones except **w-high-x** from the **To** selection list.

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.

Reports → **Fluxes** → **Set Up...**



a. Select **Total Heat Transfer Rate** in the **Options** group box.

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

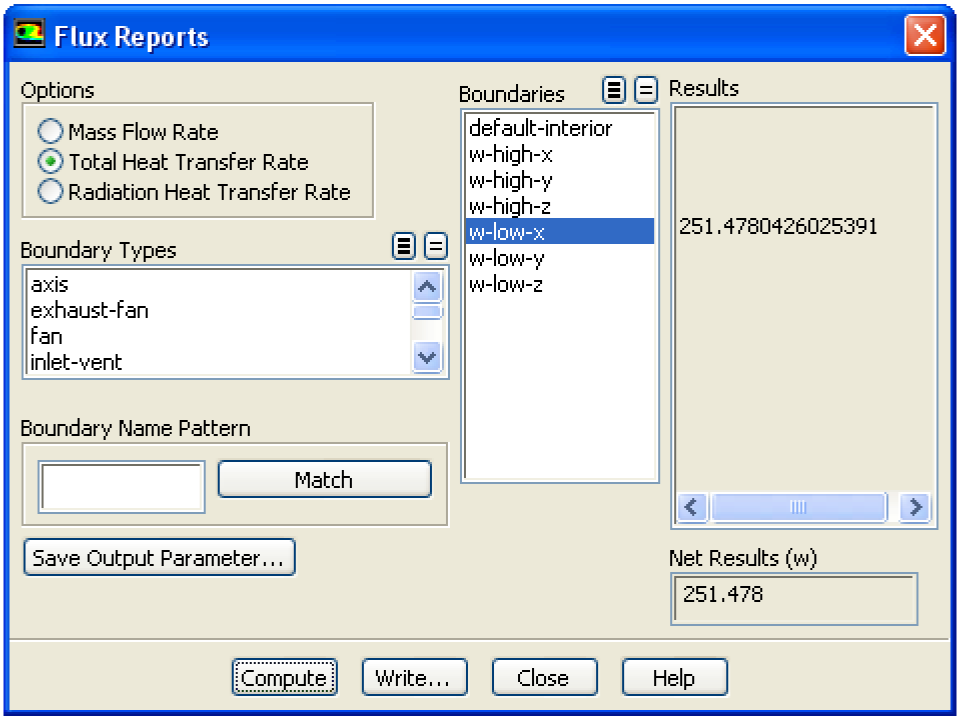
c. Click **Compute**.

Note

The energy imbalance is approximately 0.12%.

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

Reports → **Fluxes** → **Set Up...**



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.

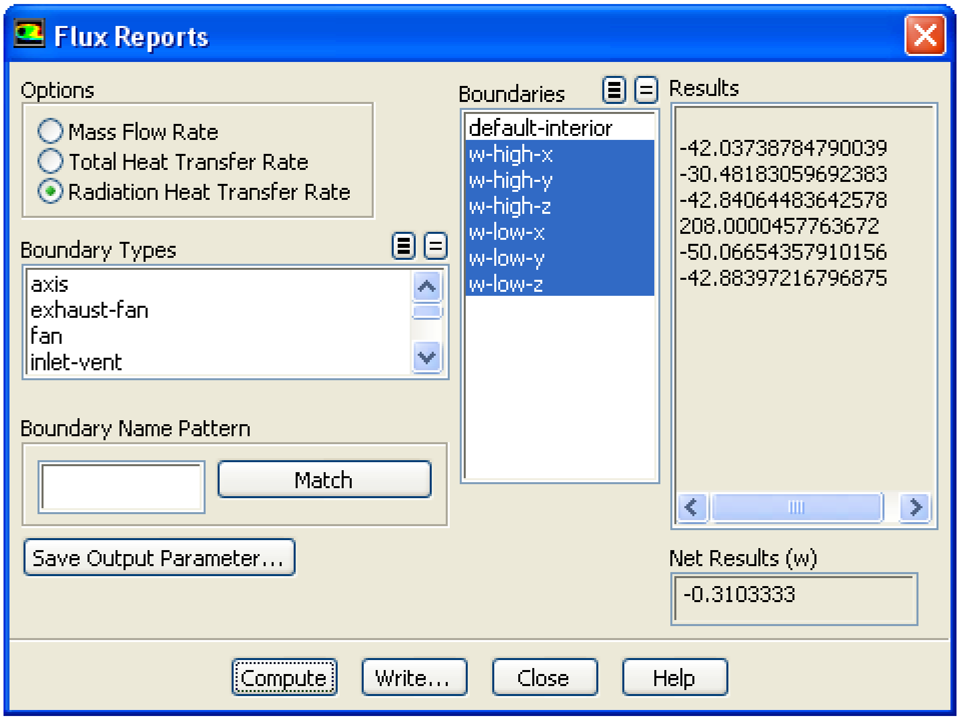
c. Click **Compute**.

Note

The net heat load is approximately 251.48 W

9. Compute the radiation heat transfer rate..

Reports → **Fluxes** → **Set Up...**



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

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

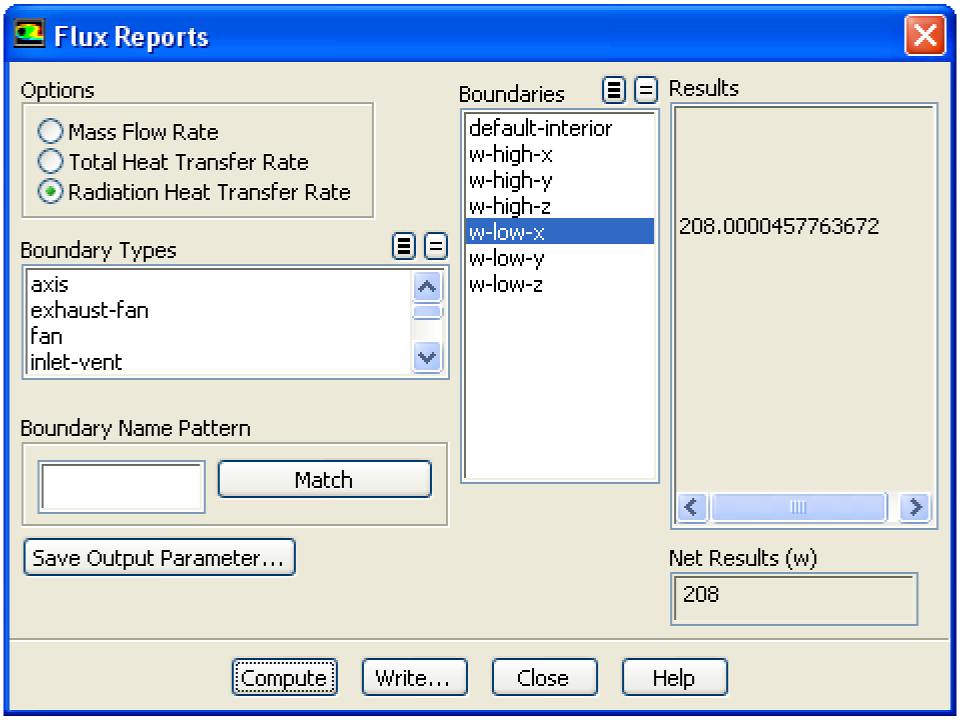
c. Click **Compute**.

Note

The net heat load is approximately -0.31 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.

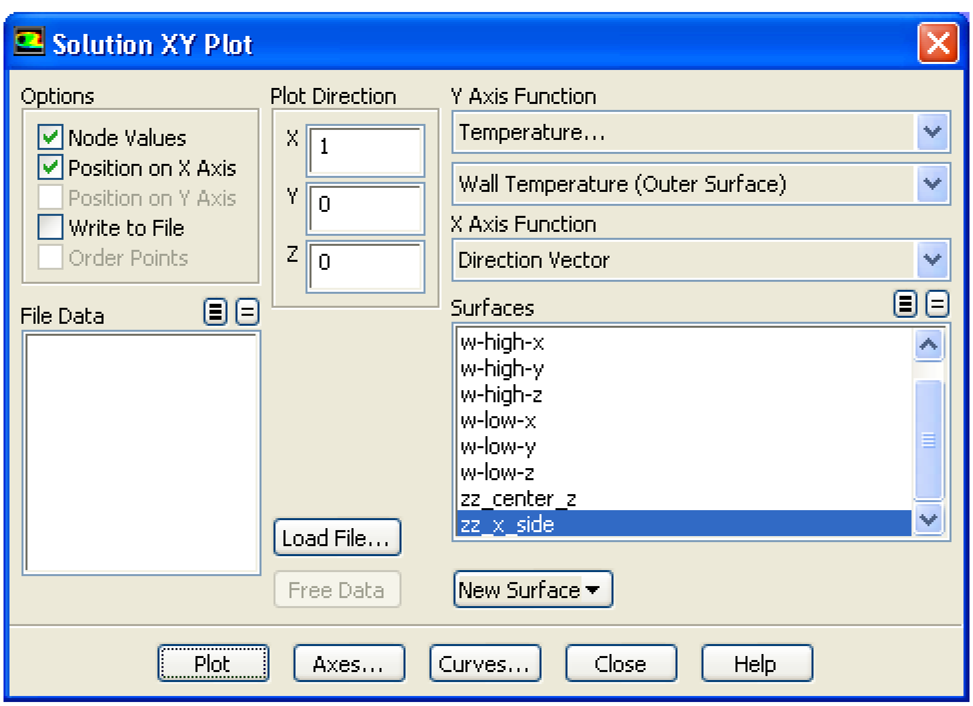
b. Deselect all boundary zones and select **w-low-x** from the **Boundaries** selection list.

c. Click **Compute** and close the **Flux Reports** dialog box.

The net heat load is approximately 208 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.

Plots → **XY Plot** → **Set Up...**



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 7.9* (p. 310)).

e. Enable **Write to File** and click the **Write...** button to open the **Select File** dialog box.

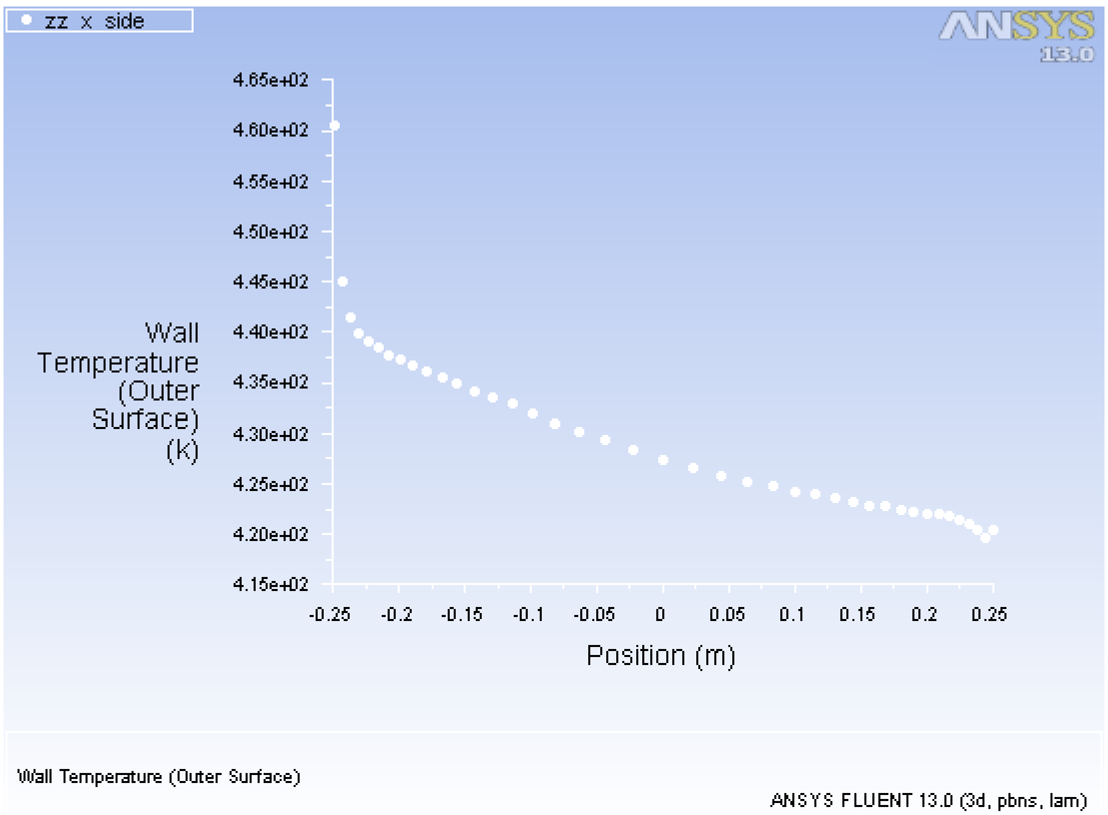
   i. Enter `tp_1.xy` for **XY File**.

   ii. Click **OK** in the **Select File** dialog box.

f. Disable the **Write to File** option.

g. Close the **Solution XY Plot** dialog box.

Figure 7.9 Temperature Profile Along Outer Surface



12. Save the case and data files (`rad_b_1.cas.gz` and `rad_b_1.dat.gz`).

    File → **Write** → **Case & Data...**

7.4.9. 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 on the **Settings...** button to open the **View Factors and Clustering** dialog box.

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

ii. Click **Apply to All Walls** and close the **View Factors and Clustering** dialog box.

b. Click **Compute/Write/Read...** 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 605 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 7.10 Contours of Wall Temperature (Outer Surface): 1 Face per Surface Cluster

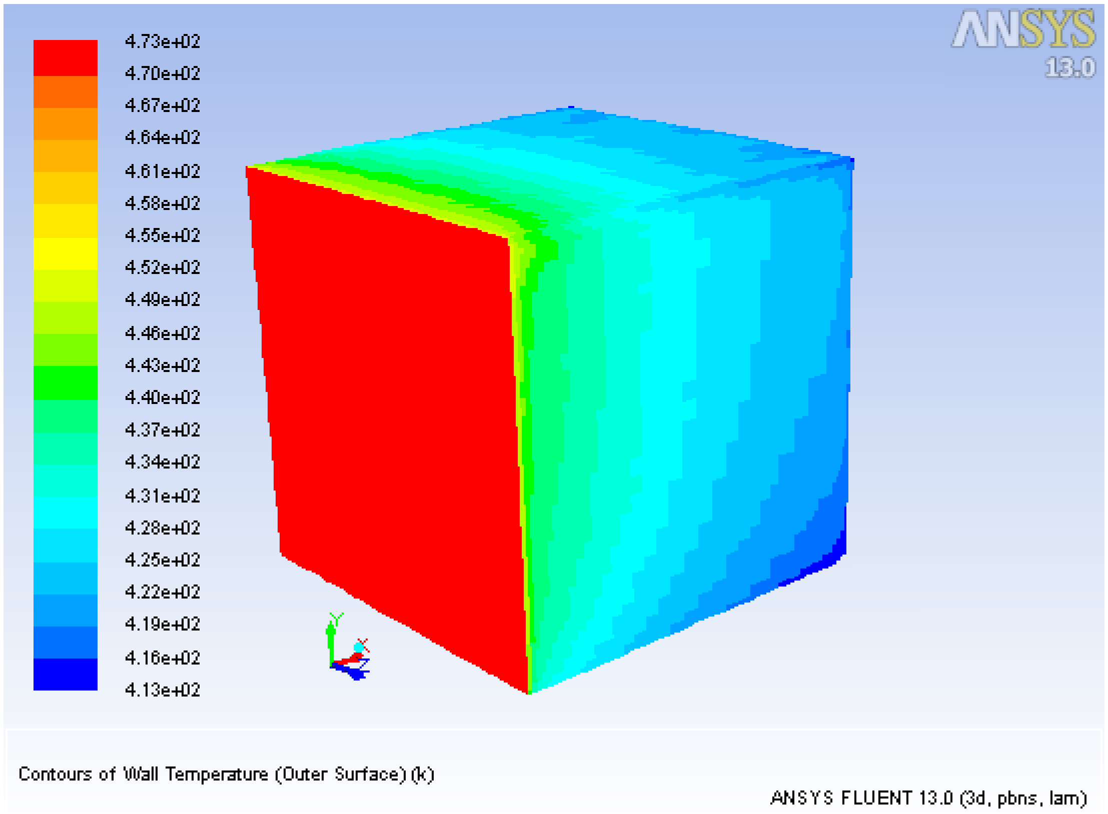


Figure 7.11 Contours of Wall Temperature (Outer Surface): 10 Faces per Surface Cluster

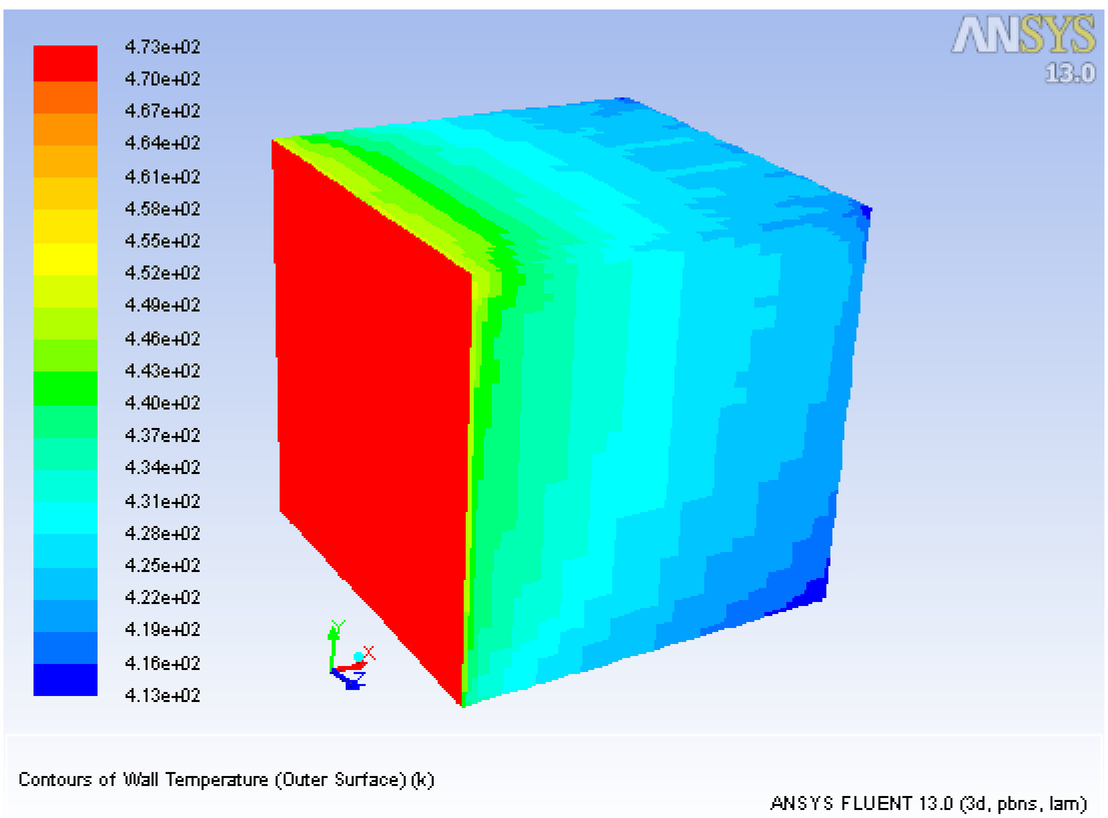


Figure 7.12 Contours of Wall Temperature (Outer Surface): 100 Faces per Surface Cluster

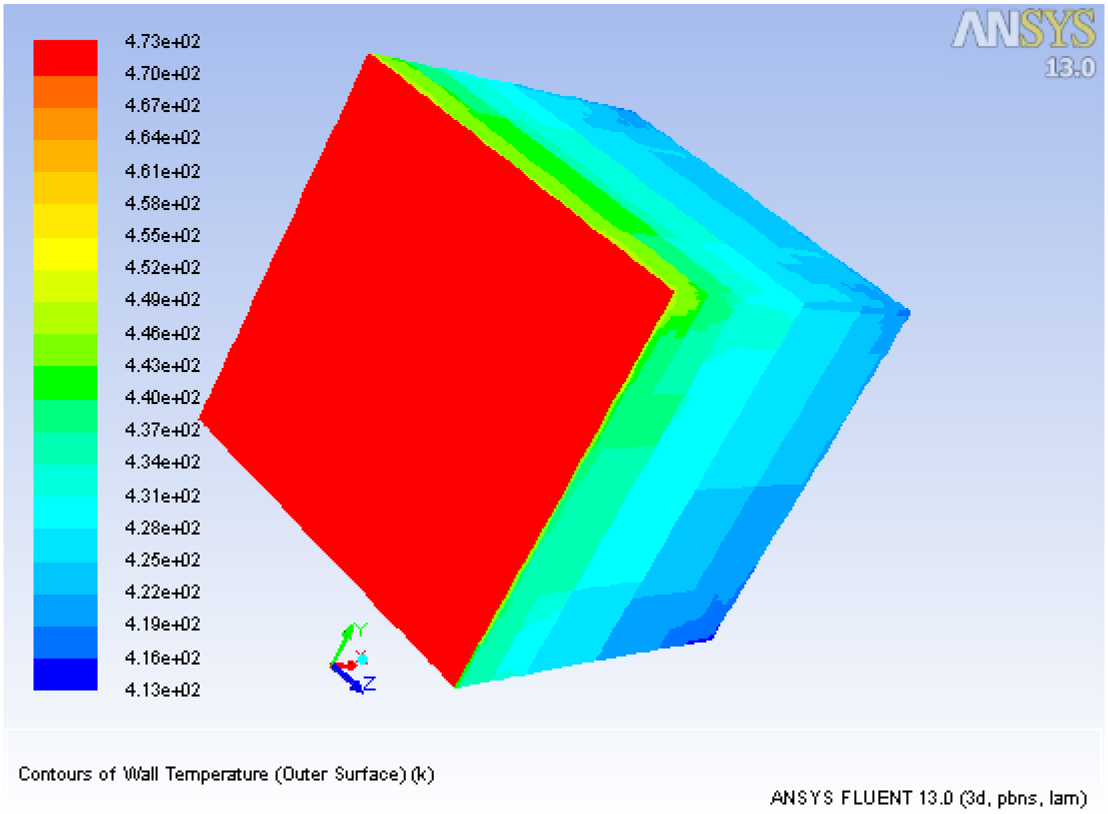


Figure 7.13 Contours of Wall Temperature (Outer Surface): 400 Faces per Surface Cluster

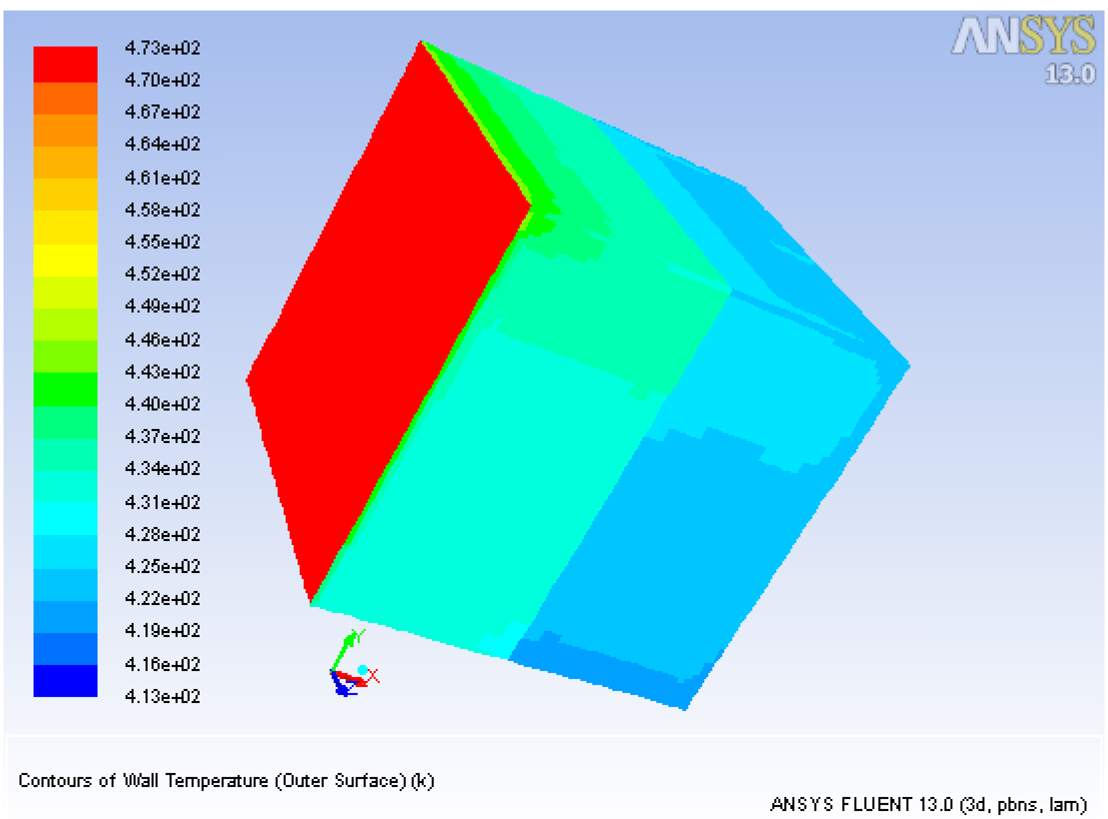


Figure 7.14 Contours of Wall Temperature (Outer Surface): 800 Faces per Surface Cluster

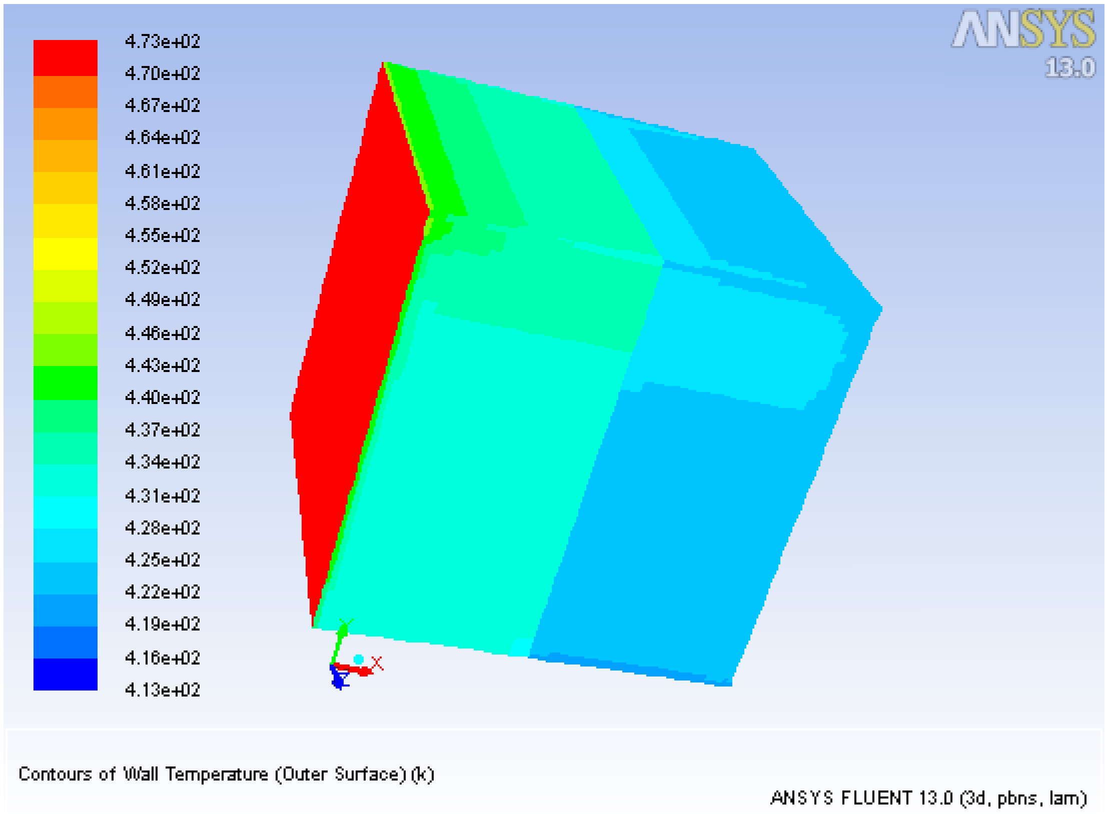
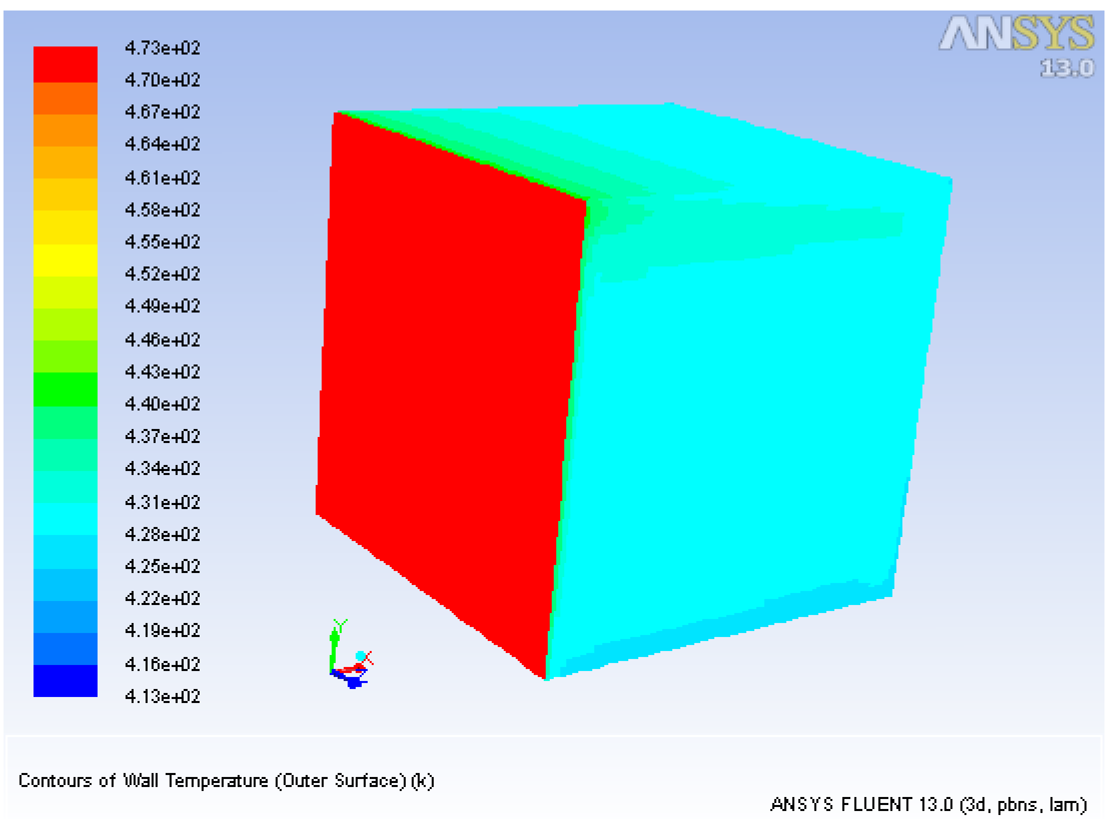
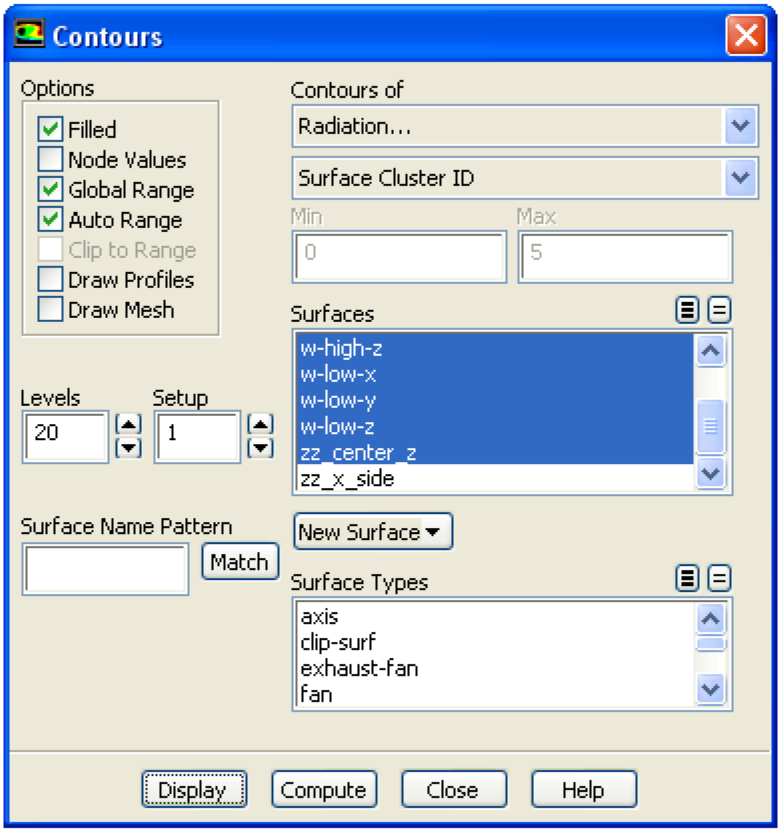


Figure 7.15 Contours of Wall Temperature (Outer Surface): 1600 Faces per Surface Cluster



8. Display contours of surface cluster ID for 1600 faces per surface cluster (*Figure 7.16* (p. 316)).

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



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

   b. Make sure that **Node Values** is disabled.

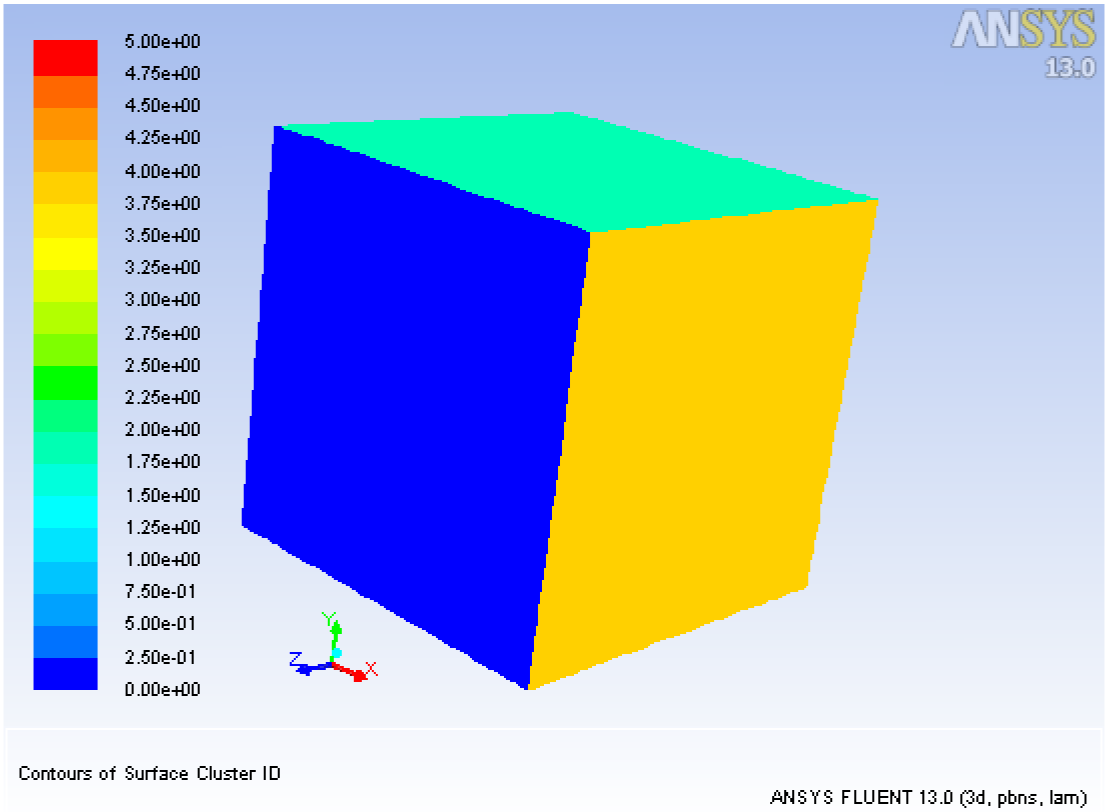
   c. Select **Radiation...** and **Surface Cluster ID** from the **Contours of** drop-down lists.

   d. Select all surfaces except **default-interior** and **zz_x_side**.

   e. Click **Display** and rotate the figure as shown in *Figure 7.16* (p. 316).

   f. Close the **Contours** dialog box.

Figure 7.16 Contours of Surface Cluster ID—1600 Faces per Surface Cluster (FPSC)



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 7.17* (p. 317)).

Figure 7.17 Contours of Surface Cluster ID—400 FPSC

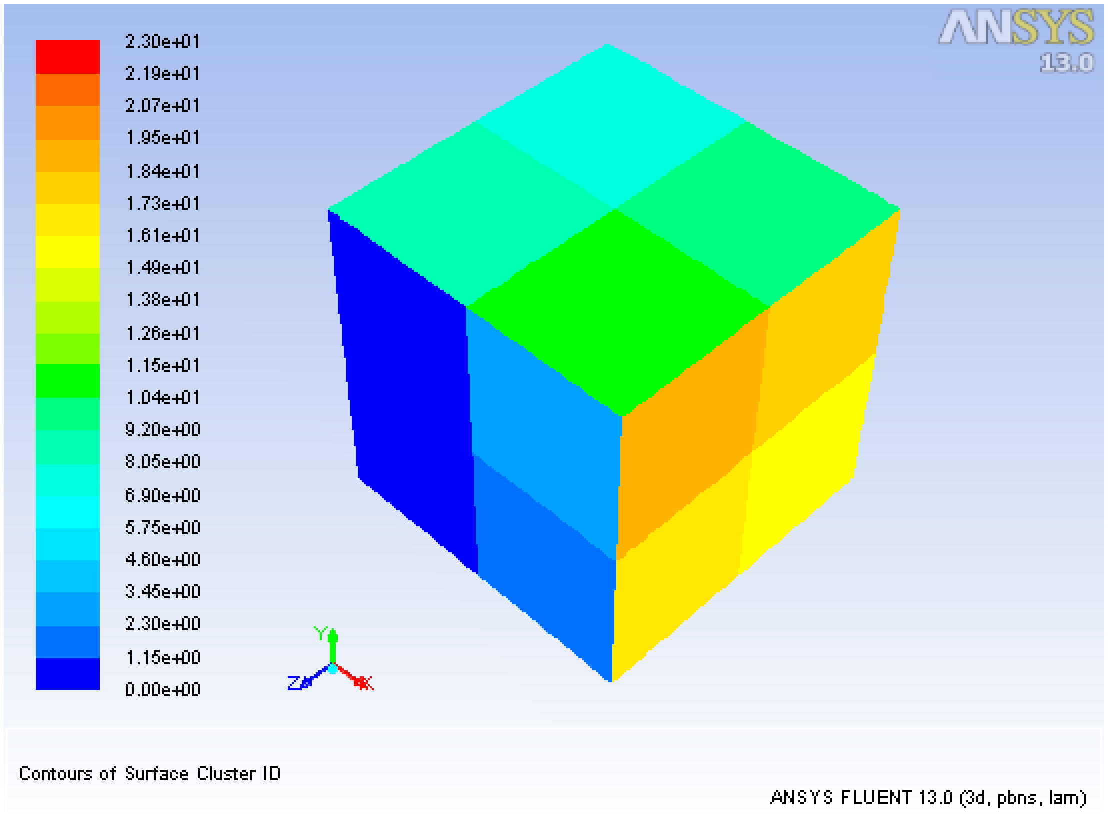


*Figure 7.17 (p. 317) shows contours of **Surface Cluster ID** for 400 FPSC. This case shows better clustering compared to all of the other cases.*

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.

    Plots → **XY Plot** → **Set Up...**

    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 **zz_x_side** is selected from the **Surfaces** selection list.

    e. Click **Plot** (*Figure 7.18* (p. 318)).

    f. Click the **Load File...** button to open the **Select File** dialog box.

       i. Select **tp_1.xy**.

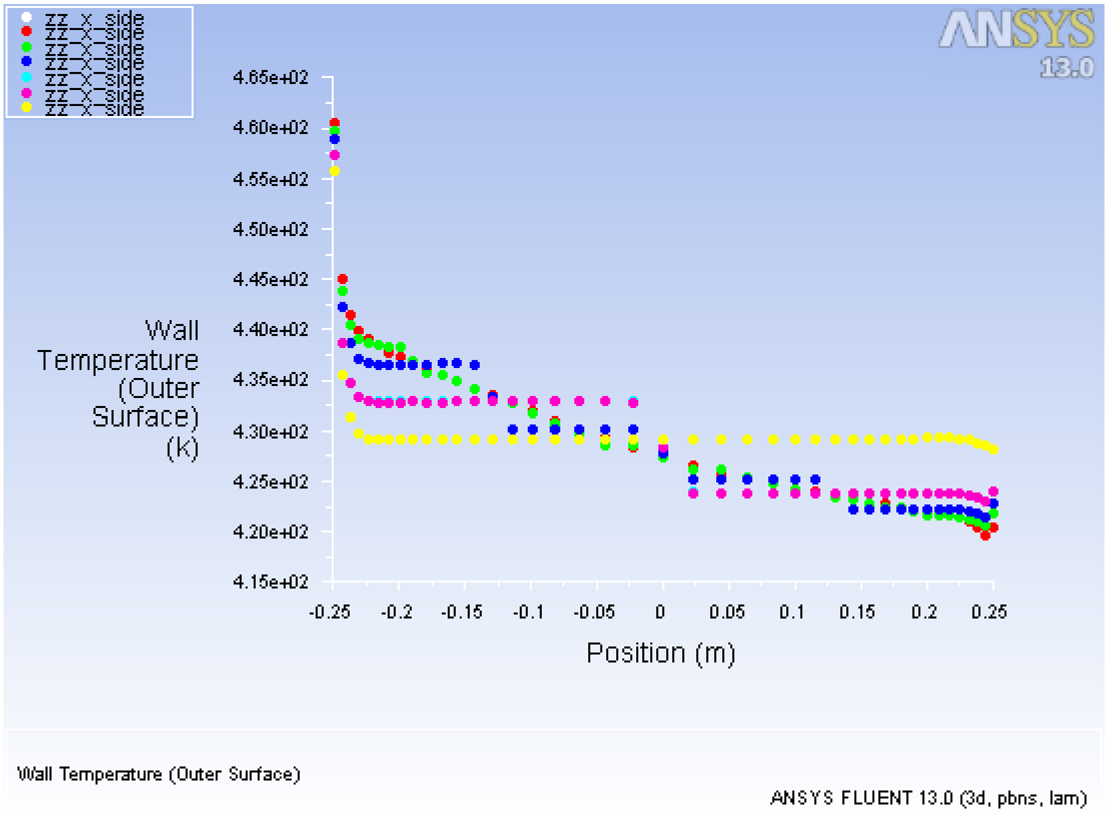
       ii. Click **OK** to close the **Select File** dialog box.

    g. Click **Plot**.

    h. In a similar manner, click the **Load File...** button to read the files **tp_10.xy**, **tp_100.xy**, **tp_800.xy**, and **tp_1600.xy**, and plot the temperature profiles.

    i. Close the **Solution XY Plot** dialog box.

Figure 7.18 A Comparison of Temperature Profiles along the Outer Surface



Note

The legend entries in *Figure 7.18* (p. 318) have been changed for display purposes. You will see similar changes in *Figure 7.19* (p. 322). You do not need to make these changes.

7.4.10. 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.

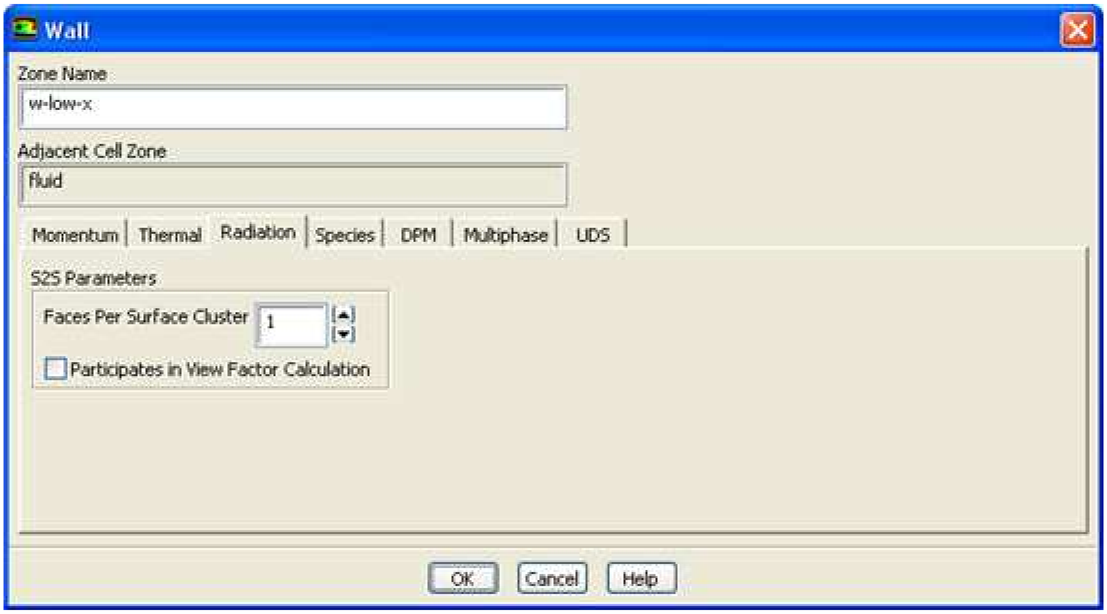
*In the steps that follow, you will specify the radiating wall (**w-low-x**) as a boundary zone that is not participating in the S2S radiation model. Consequently, you will specify the partial enclosure temperature for the wall. The partial enclosure option may not yield accurate results in cases that 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`).

   File → **Read** → **Case...**

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

Boundary Conditions → **w-low-x** → **Edit...**



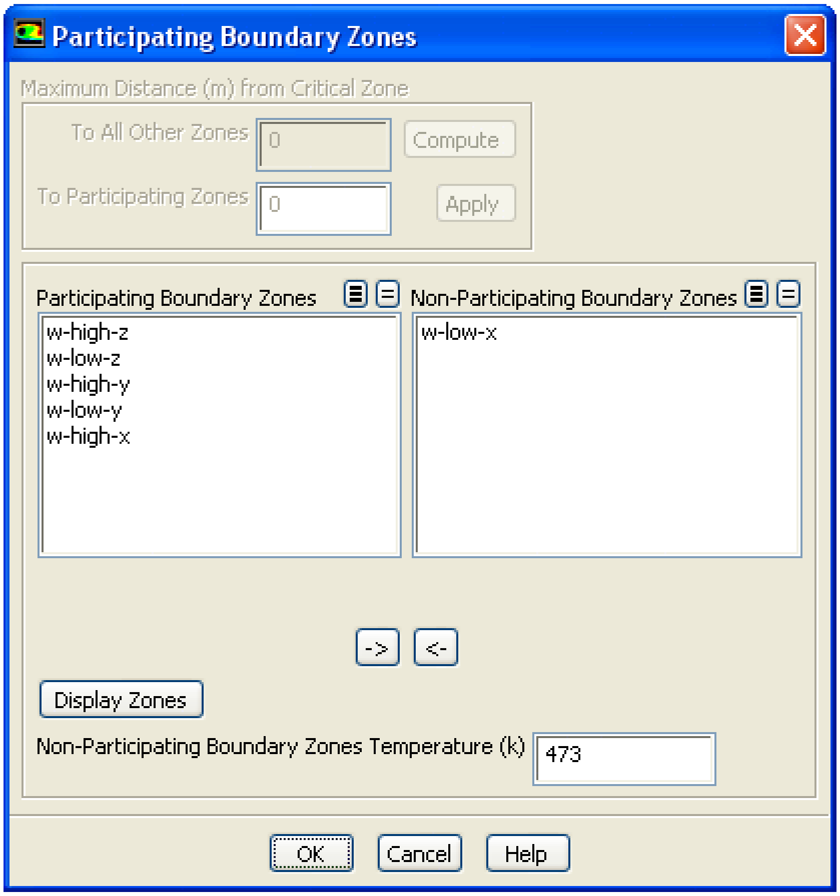
a. Click the **Radiation** tab.

b. Disable **Participates in View Factor Calculation** in the **S2S Parameters** group box.

c. Click **OK** to close the **Wall** dialog box.

3. Compute the view factors for the S2S model.

Models → **Radiation** → **Edit...**



a. Click **Settings...** to open **View Factors and Clustering** dialog box.

b. Click **Select...** to open **Participating Boundary Zones** dialog box.

   i. Enter 473 K for **Non-Participating Boundary Zones Temperature**.

   ii. Click **OK** to close **Participating Boundary Zones** dialog box.

c. Click **OK** to close **View Factors and Clustering** dialog box.

d. Click **Compute/Write/Read...** 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.

e. 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

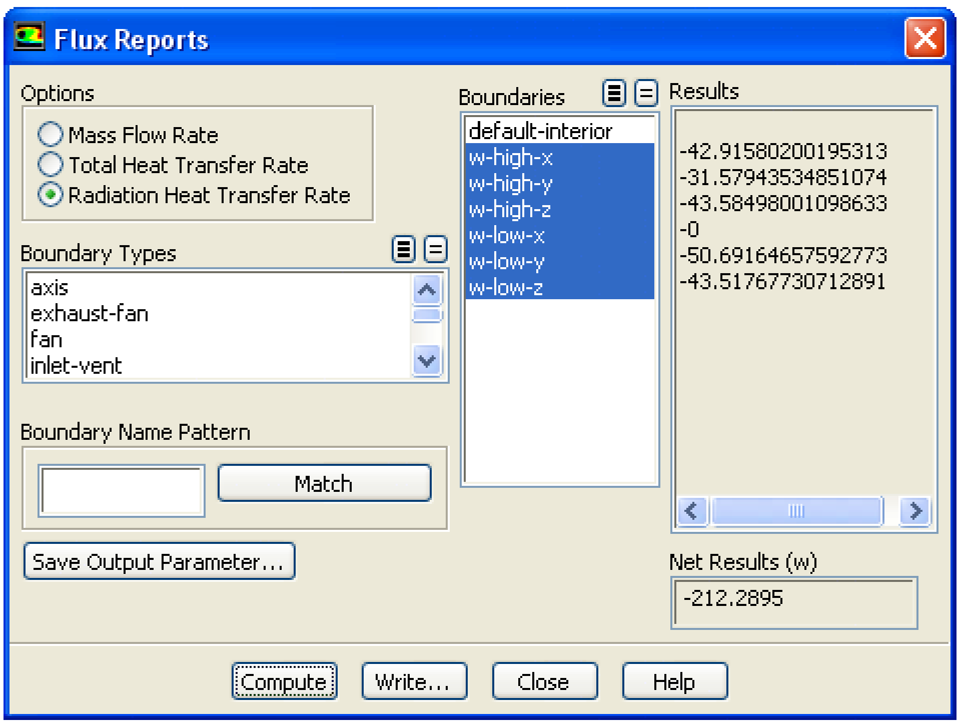
The solution will converge in approximately 625 iterations.

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...**

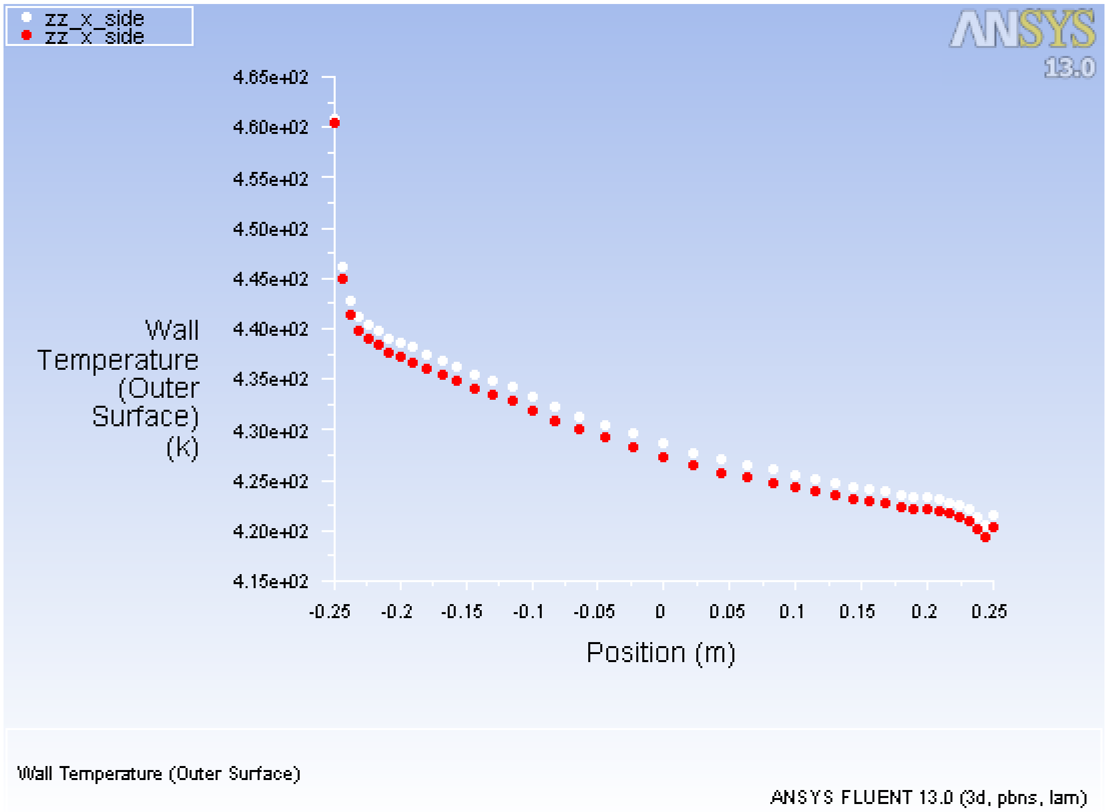
a. Select all of items in the **File Data** selection list and click **Free Data**.

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.

Figure 7.19 Temperature Profile Comparison on Outer Surface



7.5. 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 Modeling Radiation in the User's Guide.

7.6. 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 *Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 111).