Chapter 8: Using the Discrete Ordinates Radiation Model

This tutorial is divided into the following sections:

8.1. Introduction

This tutorial illustrates the set up and solution of flow and thermal modelling of a headlamp. The discrete ordinates (DO) radiation model will be used to model the radiation.

This tutorial demonstrates how to do the following:

- Read an existing mesh file into ANSYS FLUENT.
- Set up the DO radiation model.
- Set up material properties and boundary conditions.
- Solve for the energy and flow equations.
- Initialize and obtain a solution.
- Postprocess the resulting data.
- Understand the effect of pixels and divisions on temperature predictions and solver speed.

8.2. Prerequisites

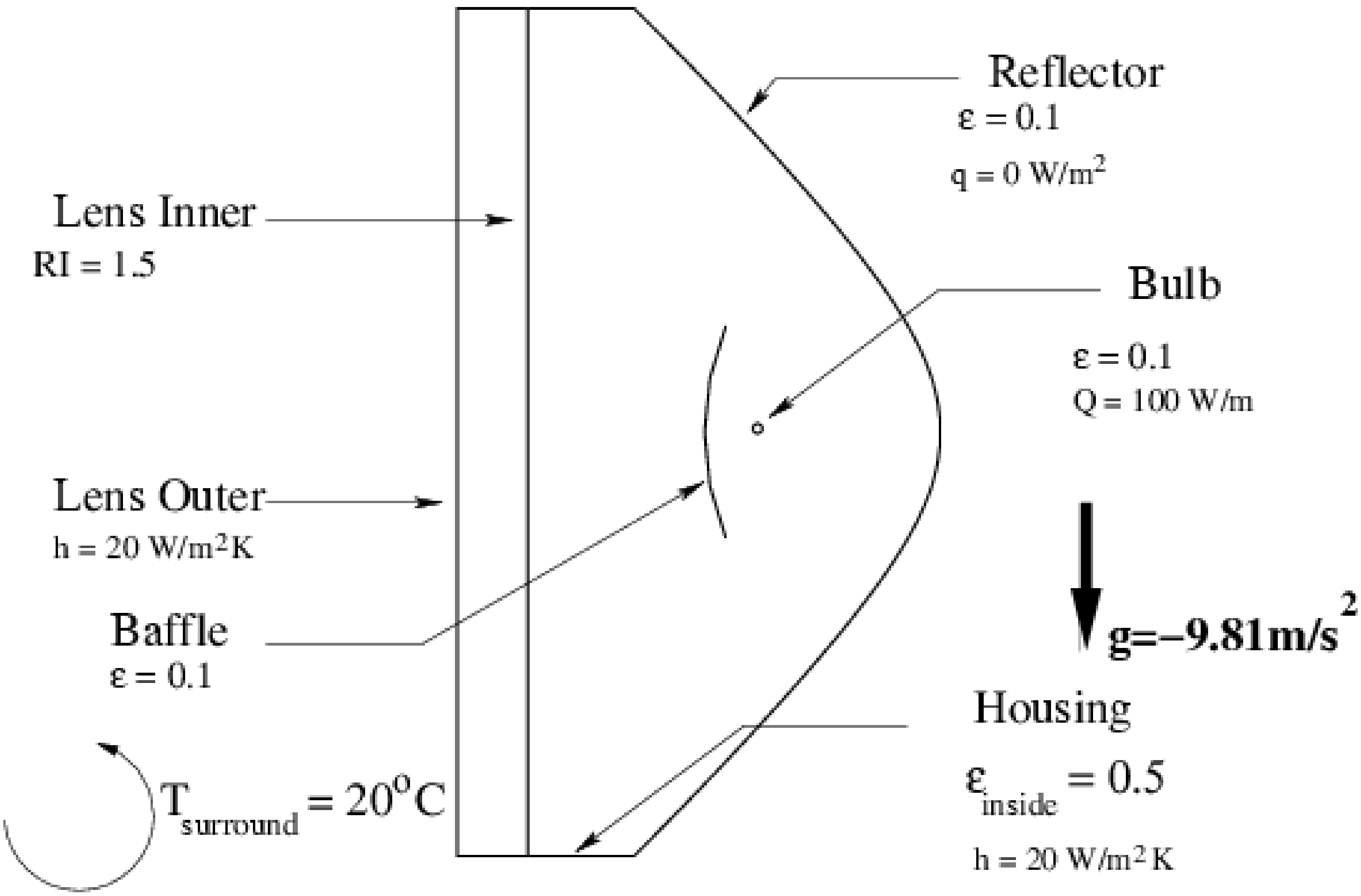
This tutorial is written with the assumption that you have completed *Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 111), and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not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

8.3. Problem Description

The problem to be considered is illustrated in *Figure 8.1* (p. 324), showing a simple two-dimensional section of a headlamp construction. The key components to be included are the bulb, reflector, baffle, lens, and housing. For simplicity, the heat output will only be considered from the bulb surface rather than the filament of the bulb. The radiant load from the bulb will cover all thermal radiation - this includes visible (light) as well as infra-red radiation.

The ambient conditions to be considered are quiescent air at 20 C. Heat exchange between the lamp and the surroundings will occur by conduction, convection and radiation. The rear reflector is assumed to be well insulated and heat losses will be ignored. The purpose of the baffle is to shield the lens from direct radiation. Both the reflector and baffle are made from polished metal having a low emissivity and mirror-like finish; their combined effect should distribute the light and heat from the bulb across the lens. The lens is made from glass and has a refractive index of 1.5.

Figure 8.1 Schematic of the Problem



8.4. Setup and Solution

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

8.4.1. Preparation

1. Download `do_rad.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 `do_rad.zip`.

   The mesh file `do.msh.gz` can be found in the `do_rad` folder created after unzipping the file.

3. Use FLUENT Launcher to start the **2D** 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 in the mesh, it will be displayed in the embedded graphics window.

8.4.2. Step 1: Mesh

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

   File → **Read** → **Mesh...**

   As the mesh file is read, ANSYS FLUENT will report the progress in the console.

8.4.3. Step 2: General Settings

General

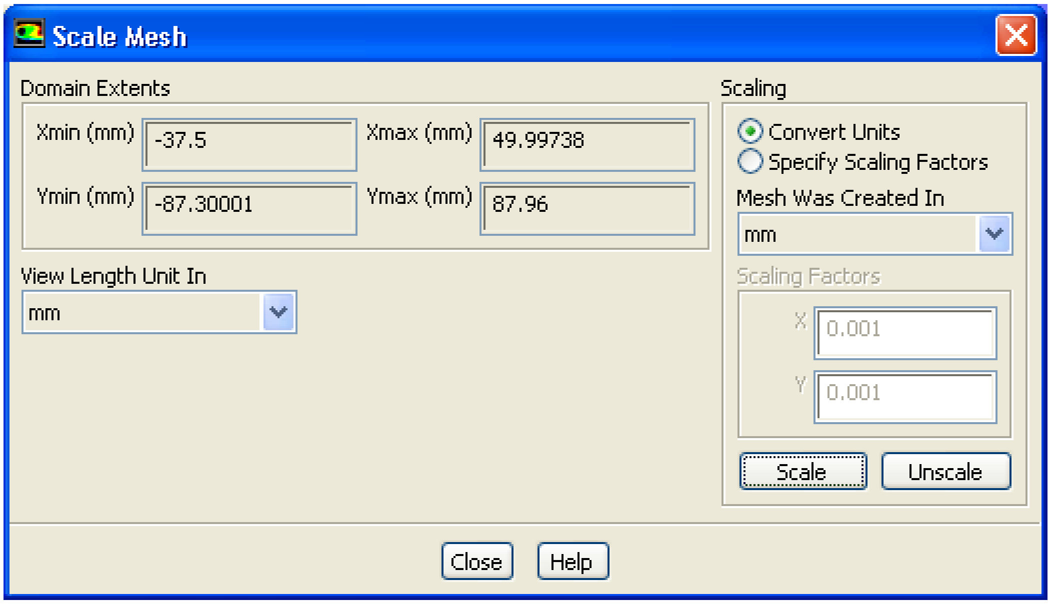
1. Check the mesh.

   General → **Check**

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

2. Scale the mesh.

   General → **Scale...**



a. Select **mm** from the **View Length Unit In** drop-down list.

   *The **Domain Extents** will be reported in **mm**.*

b. Select **mm** from the **Mesh Was Created In** drop-down list.

c. Click **Scale** and close the **Scale Mesh** dialog box.

3. Check the mesh.

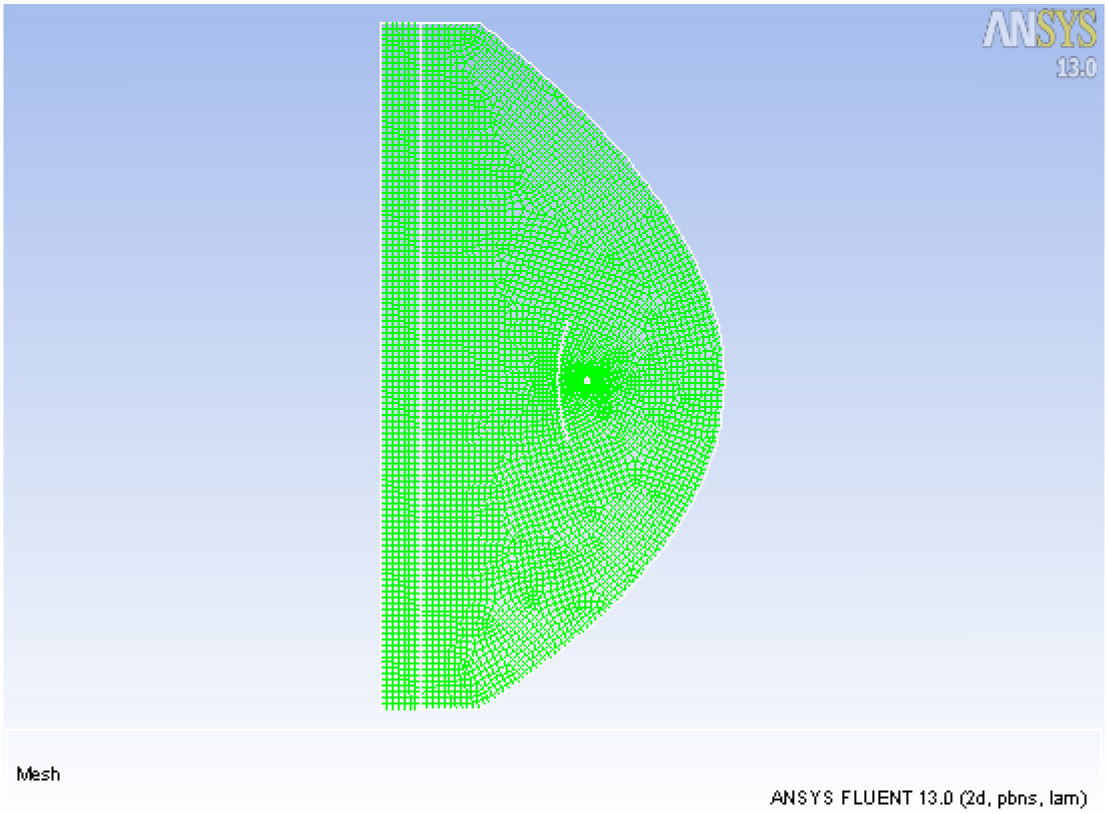
   General → **Check**

   Note

   It is good practice to check the mesh after manipulating it (scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap).

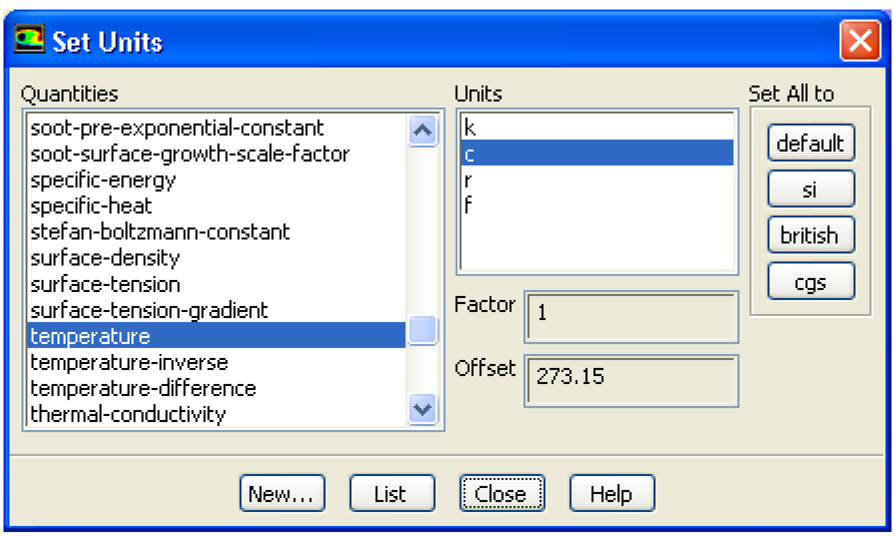
4. Examine the mesh.

Figure 8.2 Graphics Display of Mesh



5. Change the unit of temperature to centigrade.

   General → **Units...**



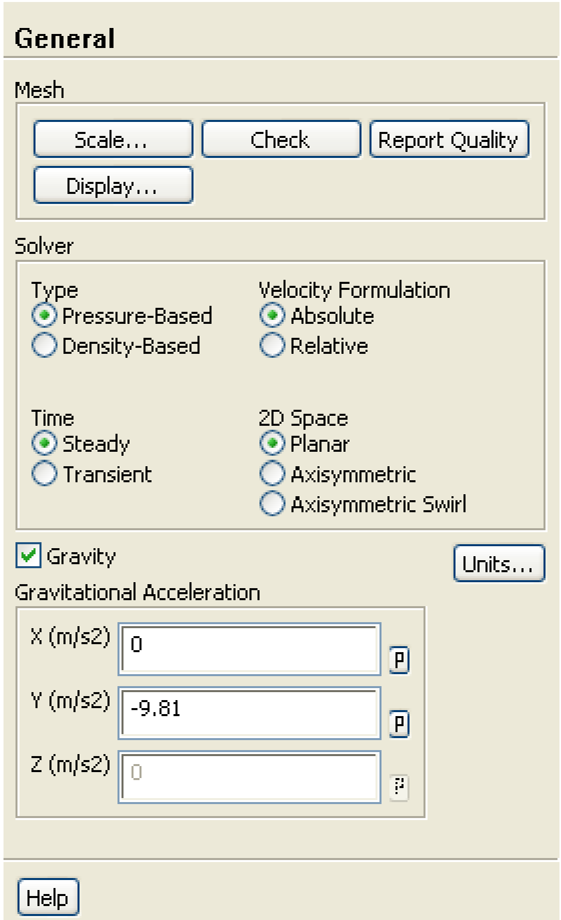
   a. Select **temperature** from the **Quantities** selection list.

   b. Select **c** from the **Units** selection list.

   c. Close the **Set Units** dialog box.

6. Retain the default solver settings.

General



7. Enable **Gravity**.

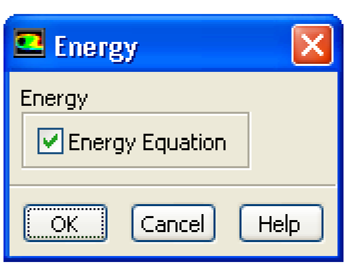
   a. Enter `-9.81` m/s^2 for **Gravitational Acceleration** in the **Y** direction.

8.4.4. Step 3: Models

Models

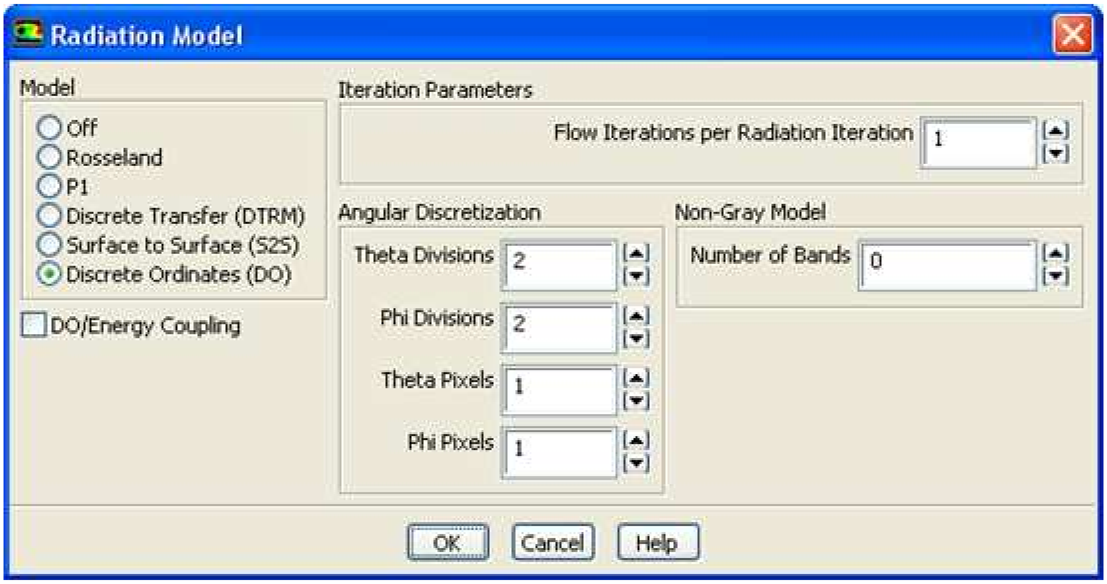
1. Enable the energy equation.

   Models → **Energy** → **Edit...**



2. Enable the **DO** radiation model.

Models → **Radiation** → **Edit...**



a. Select **Discrete Ordinates (DO)** in the **Model** list.

   *The **Radiation Model** dialog box expands to show the related inputs.*

b. Set the **Flow Iterations per Radiation Iteration** to 1.

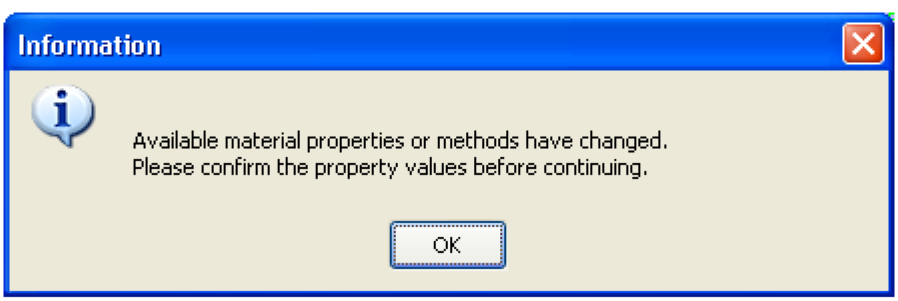
   As radiation will be the dominant mode of heat transfer, it is beneficial to reduce the interval between calculations. For this small 2D case we will reduce it to 1.

c. Retain the default settings for **Angular Discretization**.

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

   *An **Information** dialog box will appear, informing that material properties have changed.*

e. Click **OK** in the **Information** dialog box.

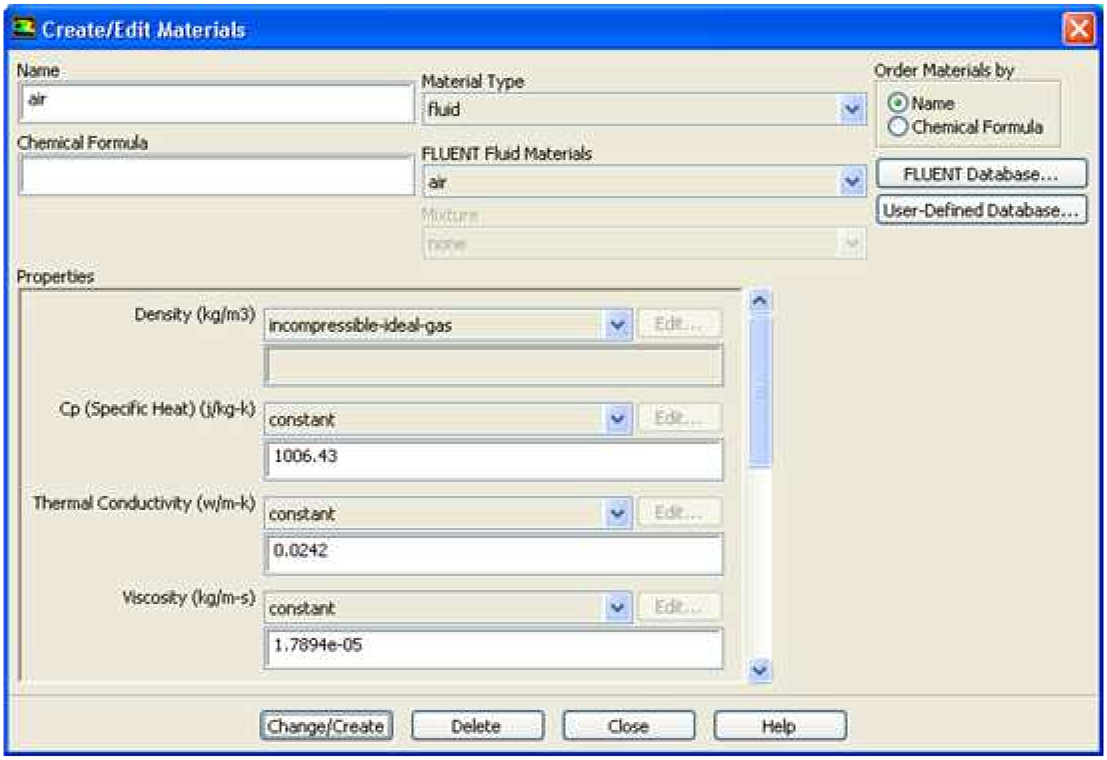


8.4.5. Step 4: Materials

Materials

1. Set the properties for **air**.

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



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

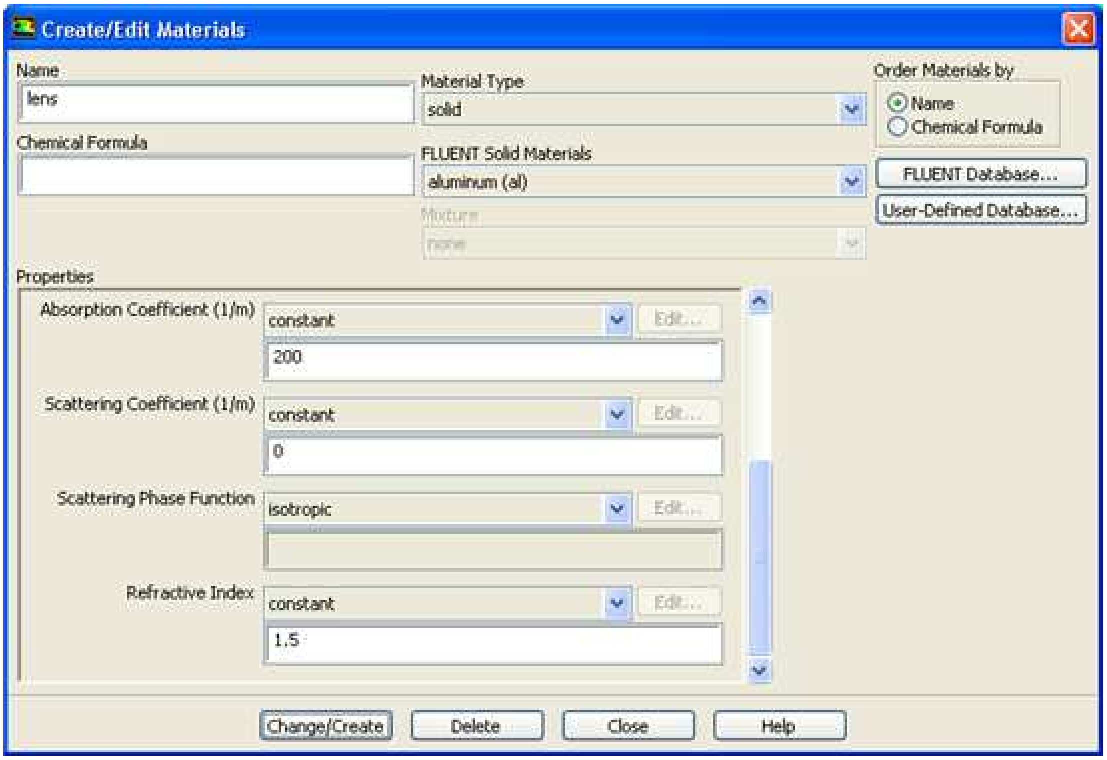
Since pressure variations are insignificant compared to temperature variation, we choose incompressible-ideal-gas law for density.

b. Retain the default settings for all other parameters.

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

2. Create a new material, **lens**.

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



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

b. Enter `2200` Kg/m^3 for **Density**.

c. Enter `830` J/Kg-K for **Cp (Specific Heat)**.

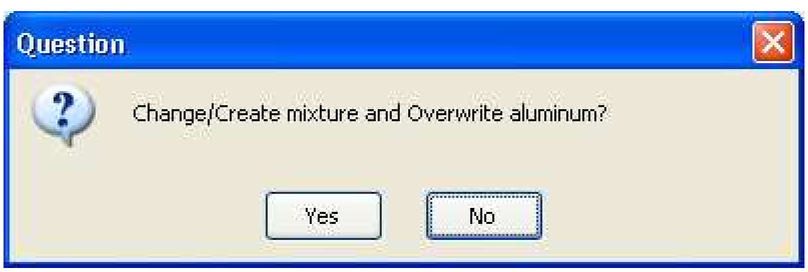
d. Enter `1.5` W/m-K for **Thermal Conductivity**.

e. Enter `200` 1/m for **Absorption Coefficient**.

f. Enter `1.5` for **Refractive Index**.

g. Click **Change/Create**.

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



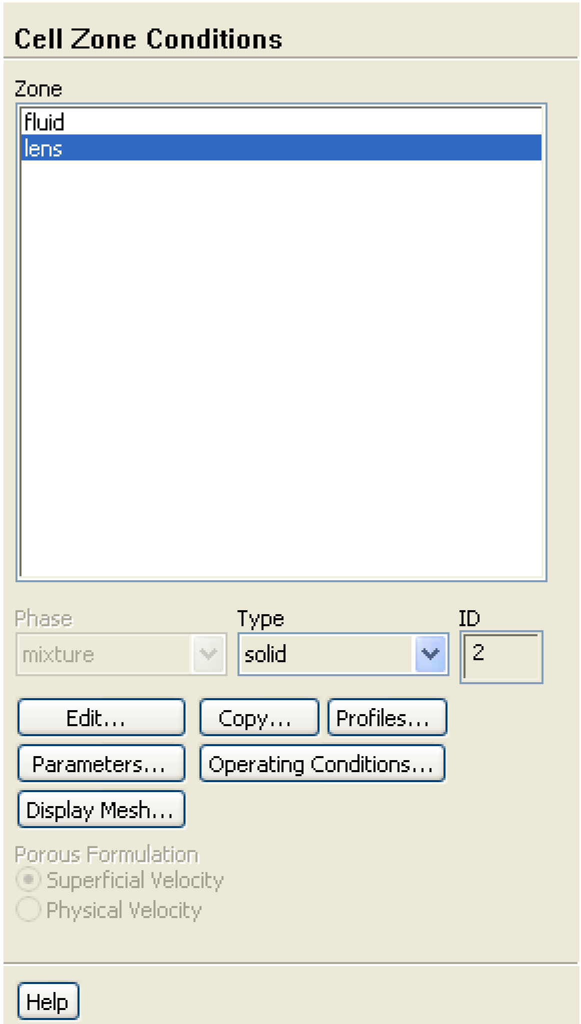
h. Click **No** in the **Question** dialog box to retain **aluminum** and add the new material (**lens**) to the materials list.

   *The **Create/Edit Materials** dialog box will be updated to show the new material, **lens**, in the **ANSYS FLUENT Solid Materials** drop-down list.*

i. Close the **Create/Edit Materials** dialog box.

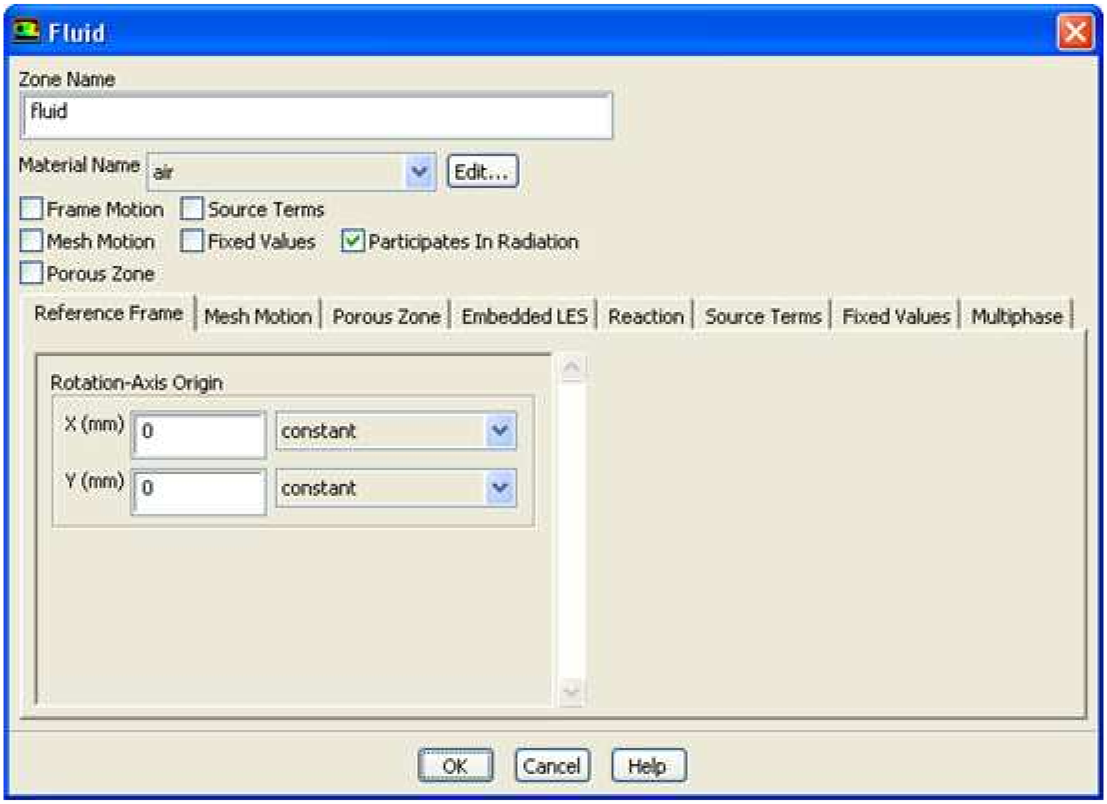
8.4.6. Step 5: Cell Zone Conditions

Cell Zone Conditions



1. Ensure that **air** is selected for fluid.

   Cell Zone Conditions → **fluid** → **Edit...**

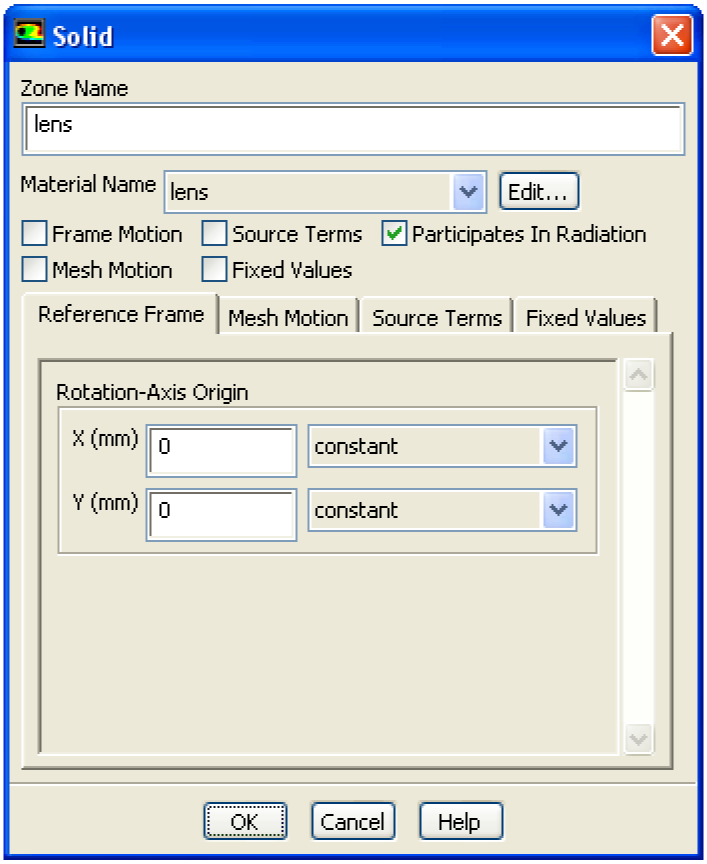


a. Retain the default selection of **air** from the **Material Name** drop-down list.

b. Click **OK** to close the **Fluid** dialog box.

2. Set the cell zone conditions for the **lens**.

Cell Zone Conditions → **lens** → **Edit...**



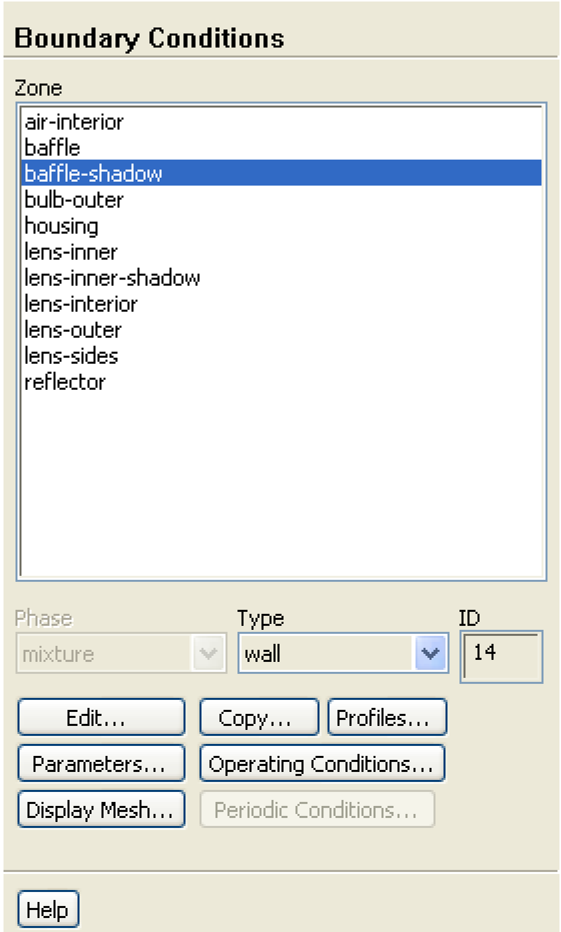
a. Select **lens** from the **Material Name** drop-down list.

b. Enable **Participates In Radiation**.

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

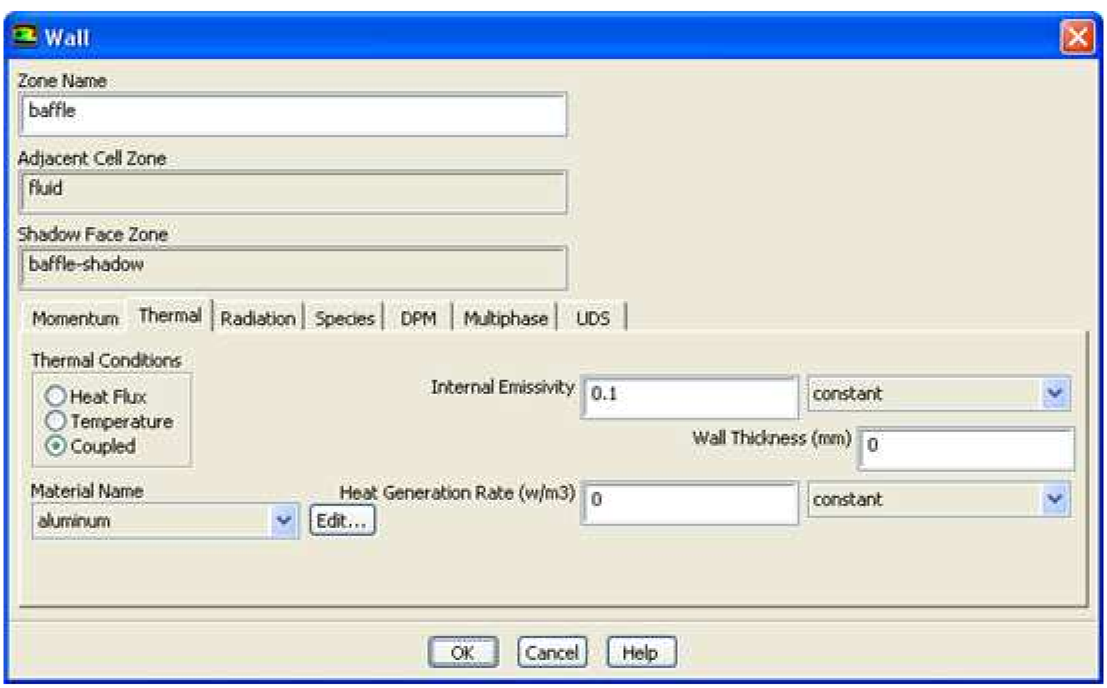
8.4.7. Step 6: Boundary Conditions

Boundary Conditions



1. Set the boundary conditions for the **baffle**.

   Boundary Conditions → **baffle** → **Edit...**



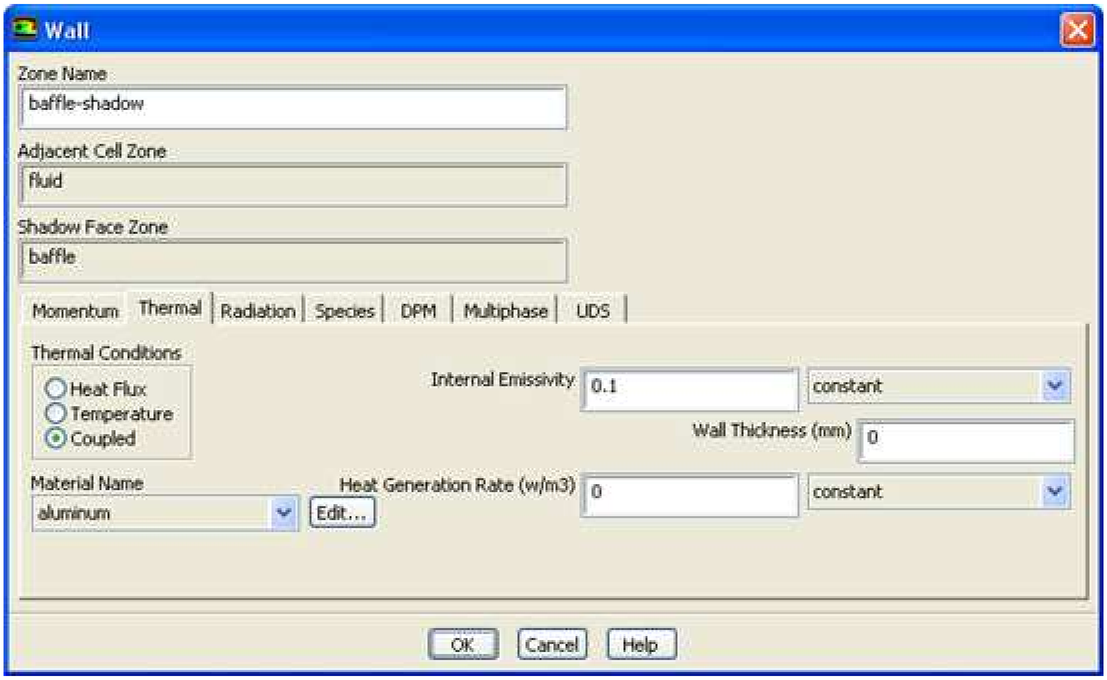
a. Click the **Thermal** tab and enter `0.1` for **Internal Emissivity**.

b. Click the **Radiation** tab and enter `0` for **Diffuse Fraction**.

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

2. Set the boundary conditions for the **baffle-shadow**.

Boundary Conditions → **baffle-shadow** → **Edit...**



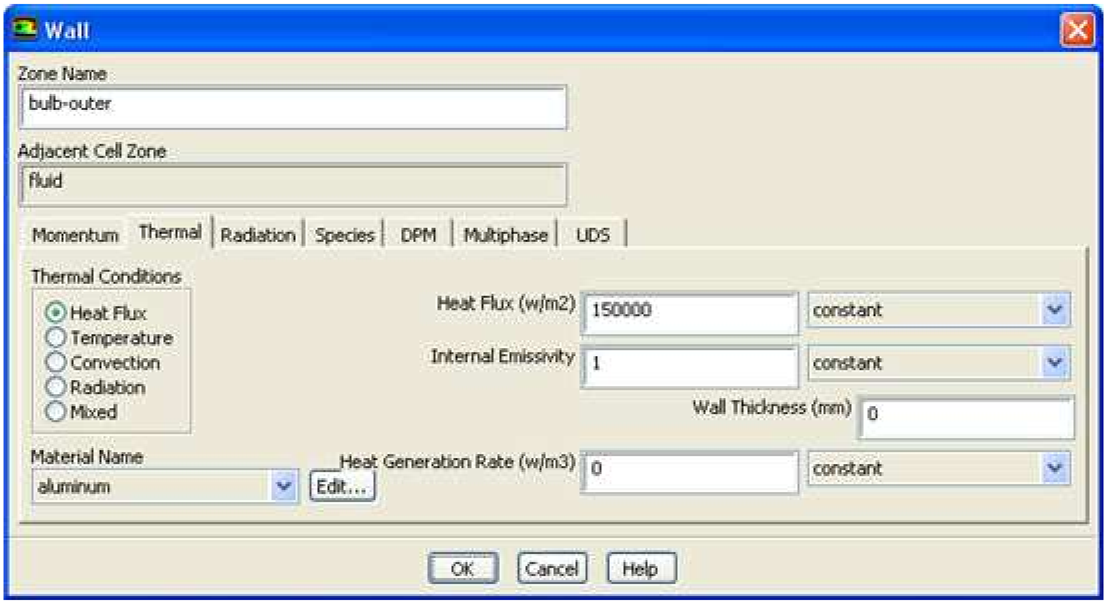
a. Click the **Thermal** tab and enter `0.1` for **Internal Emissivity**.

b. Click the **Radiation** tab and enter `0` for **Diffuse Fraction**.

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

3. Set the boundary conditions for the **bulb-outer**.

Boundary Conditions → **bulb-outer** → **Edit...**



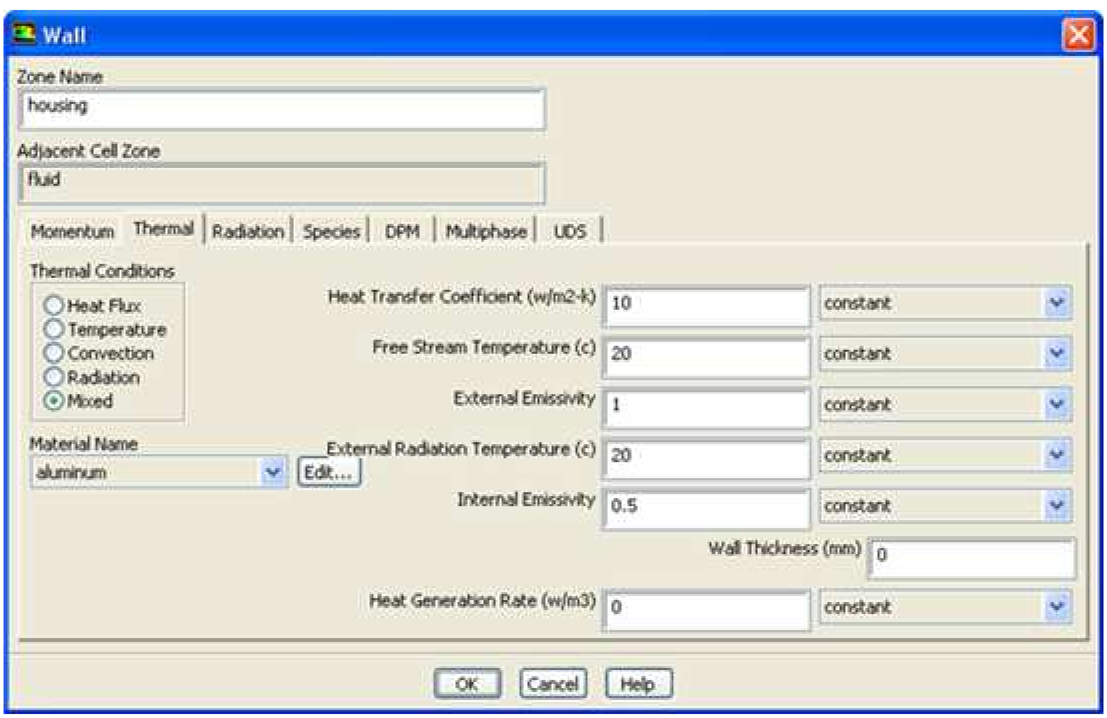
a. Click the **Thermal** tab and enter `150000` W/m^2 for **Heat Flux**.

b. Retain the value of `1` for **Internal Emissivity**.

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

4. Set the boundary conditions for the **housing**.

Boundary Conditions → **housing** → **Edit...**



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

b. Enter `10` W/m^2-K for **Heat Transfer Coefficient**.

c. Enter `20` C for **Free Stream Temperature**.

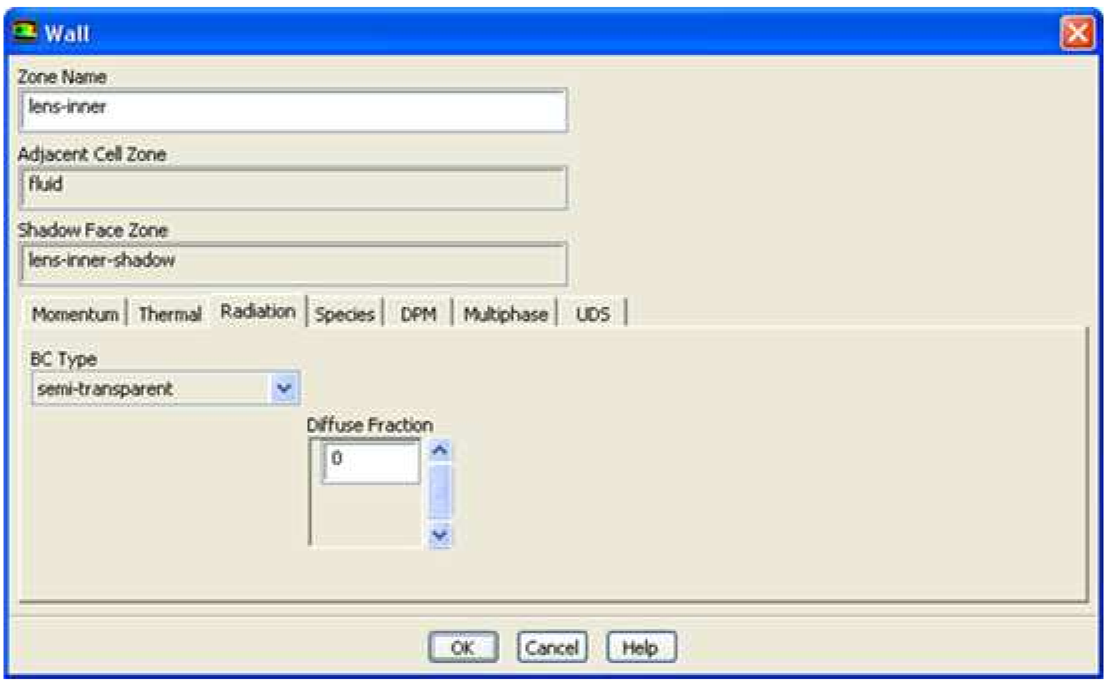
   d. Retain the value of 1for **External Emissivity**.
   e. Enter 20 C for **External Radiation Temperature**.
   f. Enter 0.5 for **Internal Emissivity**.
   g. Click **OK** to close the **Wall** dialog box.

5. Set the boundary conditions for the **lens-inner**.

   Boundary Conditions → **lens-inner** → **Edit...**

   The inner and outer surface of the lens will be set to semi-transparent conditions. This allows radiation to be transmitted through the wall between the two adjacent participating cell zones. It also calculates the effects of reflection and refraction at the interface. These effects occur because of the change in refractive index (set through the material properties) and are a function of the incident angle of the radiation and the surface finish. In this case, the lens is assumed to have a very smooth surface so the diffuse fraction will be set to 0.

   On the internal walls (wall/wall-shadows) it is important to note the adjacent cell zone: this is the zone the surface points into and may influence the settings on diffuse fraction (these can be different on both sides of the wall).



   a. Click the **Radiation** tab.
   b. Select **semi-transparent** from the **BC Type** drop-down list.
   c. Enter 0 for **Diffuse Fraction**.
   d. Click **OK** to close the **Wall** dialog box.

6. Set the boundary conditions for the **lens-inner-shadow**.

   Boundary Conditions → **lens-inner-shadow** → **Edit...**

   a. Click the **Radiation** tab.
   b. Retain the default selection of **semi-transparent** from the **BC Type** drop-down list.

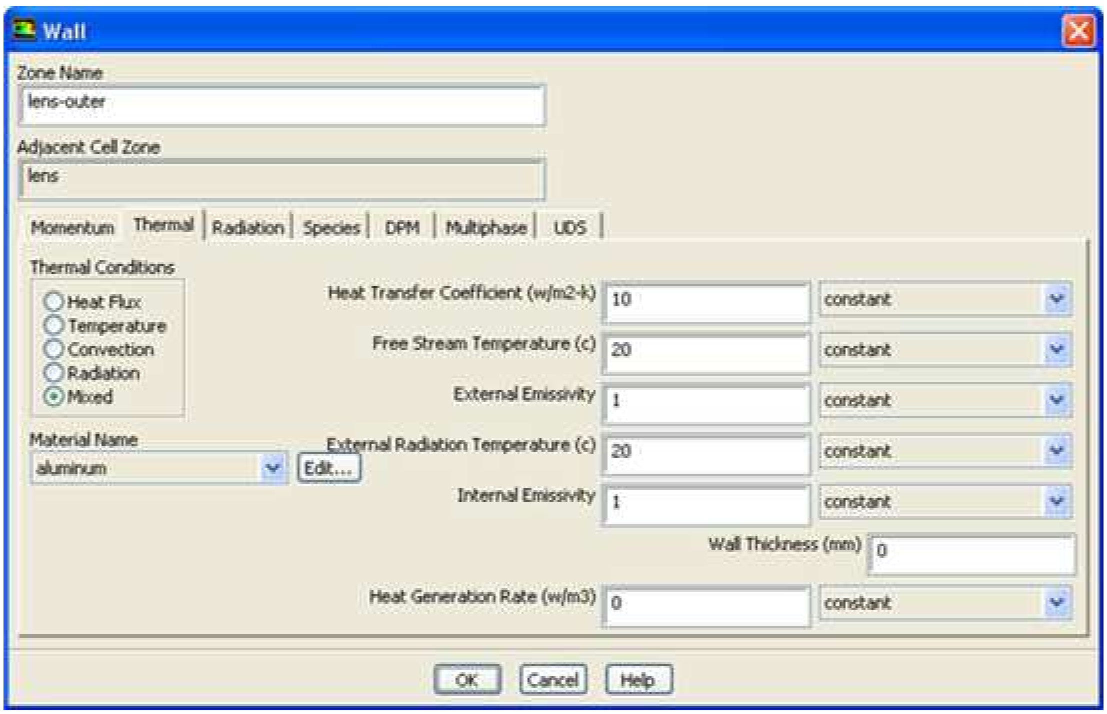
c. Enter `0` for **Diffuse Fraction**.

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

7. Set the boundary conditions for the **lens-outer**.

Boundary Conditions → **lens-outer** → **Edit...**

The surface of the lamp cools mainly by natural convection to the surroundings. As the outer lens is transparent it must also lose radiation to the surroundings, while the surroundings will supply a small source of background radiation associated with the temperature. For the lens, a semi-transparent condition is used on the outside wall. A mixed thermal condition provides the source of background radiation as well as calculating the convective cooling on the outer lens wall. For a semi-transparent wall, the source of background radiation is added directly to the DO radiation rather than to the energy equation - an external emissivity of 1 is used, in keeping with the assumption of a small object in a large enclosure. As the background radiation is supplied from the thermal conditions, there is no need to supply this as a source of irradiation under the ***Radiation*** *tab for the wall boundary condition. The only other setting required here is the surface finish of the outer surface of the lens - the diffuse fraction should be set to 0 as the lens is assumed to be smooth.*



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

b. Enter `10` W/m^2-K for **Heat Transfer Coefficient**.

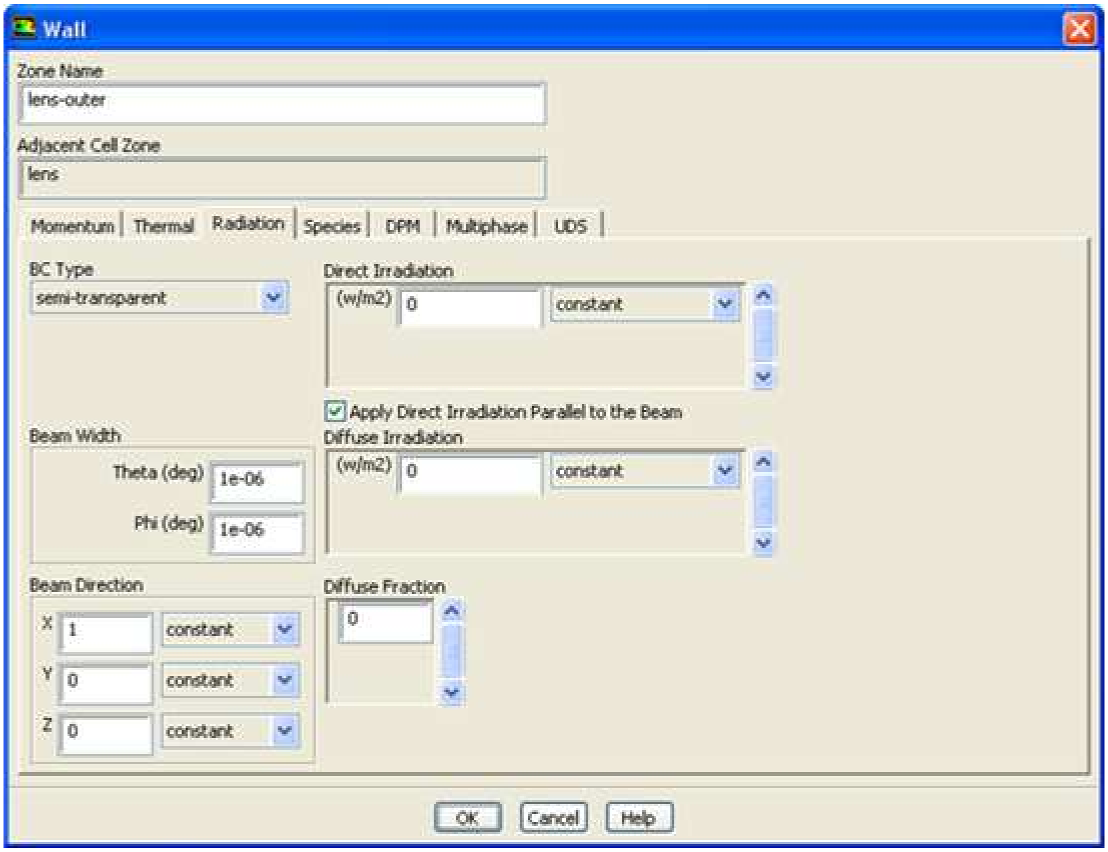
c. Enter `20` C for **Free Stream Temperature**.

d. Retain the value of `1`for **External Emissivity**.

For a semi-transparent wall the internal emissivity has no effect as there is no absorption or emission on the surface. So the set value is irrelevant.

e. Enter `20` C for **External Radiation Temperature**.

f. Click the **Radiation** tab.



g. Select **semi-transparent** from the **BC Type** drop-down list.

h. Enter `0` for **Diffuse Fraction**.

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

8. Set the boundary conditions for the **reflector**.

Boundary Conditions → **reflector** → **Edit...**

Like the baffles, the reflector is made of highly polished aluminum, giving it highly reflective surface property. About 90% of incident radiation reflects from this surface. Only 10% gets absorbed. Based on Kirchhoff's law, we can assume emissivity equals absorptivity. Therefore, we apply internal emissivity = 0.1. We also assume a clean reflector (diffuse fraction = 0).

a. Click the **Thermal** tab and enter `0.1` for **Internal Emissivity**.

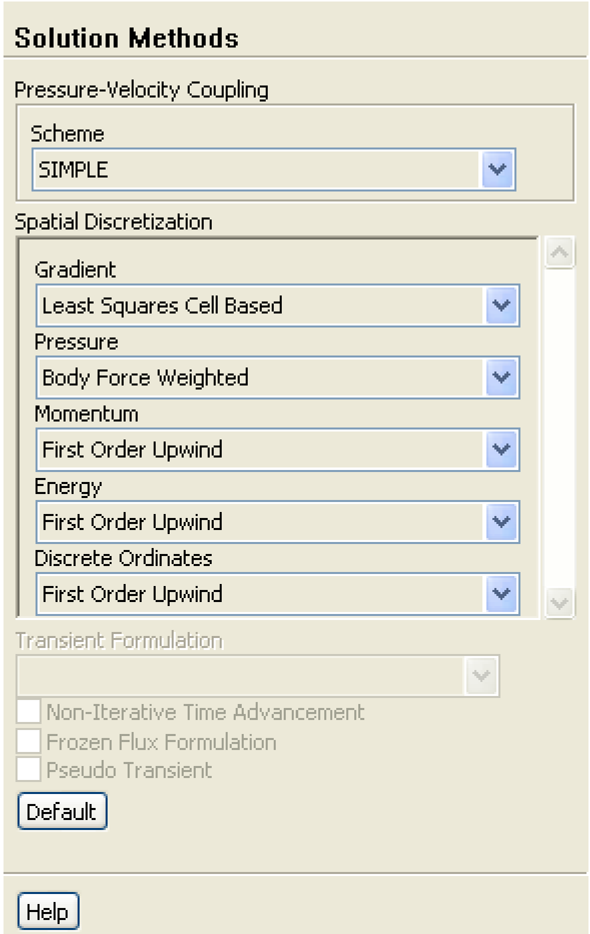
b. Click the **Radiation** tab and enter `0` for **Diffuse Fraction**.

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

8.4.8. Step 7: Solution

1. Set the solution parameters.

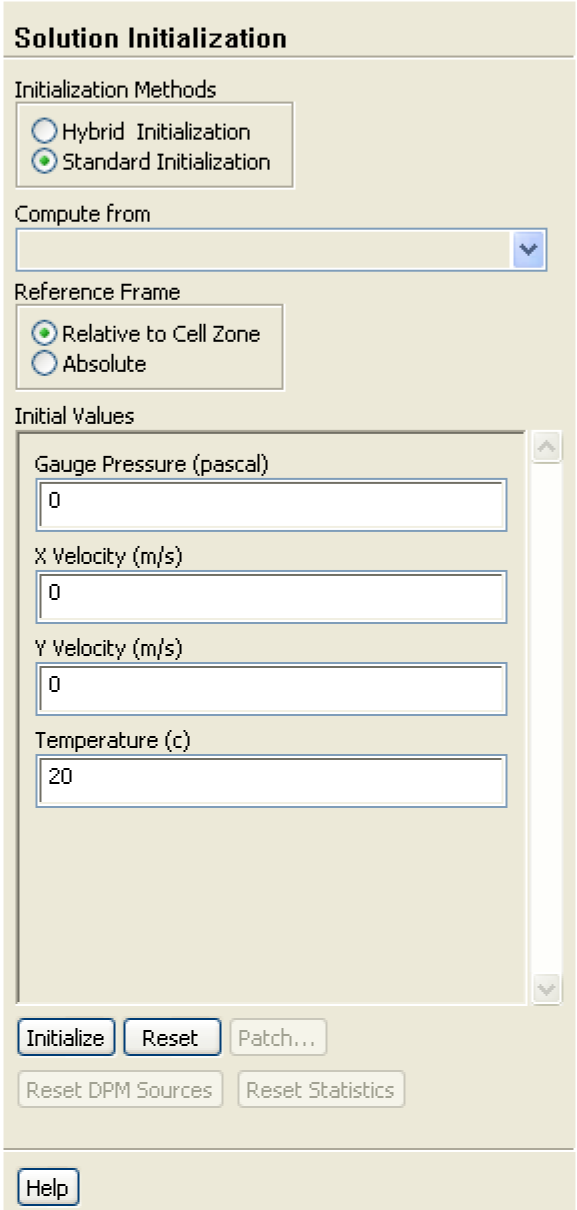
Solution Methods



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

2. Initialize the solution.

Solution Initialization



a. Retain the selection of **Standard Initialization** from the **Initialization Methods** group box.

b. Enter `20` C for **Temperature**.

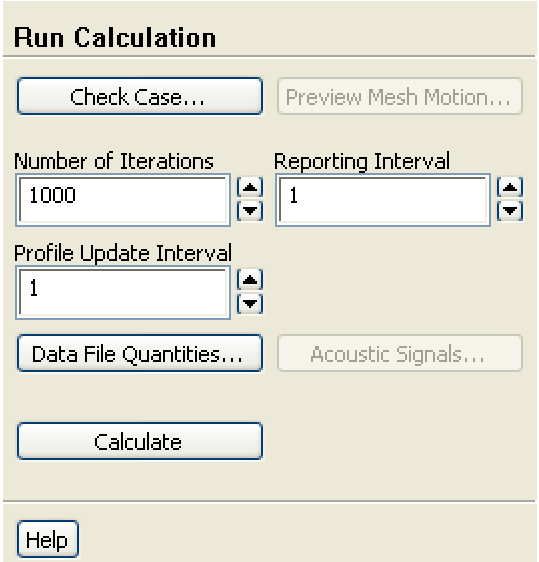
c. Click **Initialize**.

3. Save the case file (`do.cas.gz`)

   File → **Write** → **Case...**

4. Start the calculation by requesting 1000 iterations.

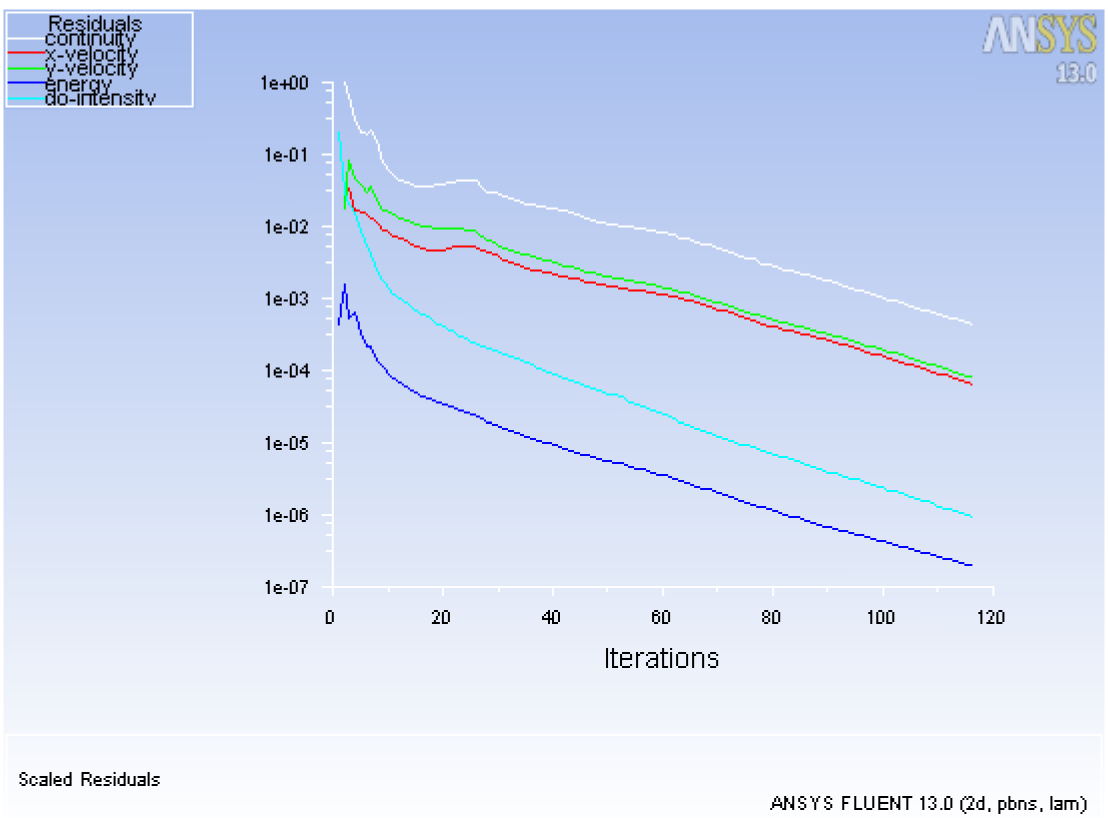
   Run Calculation



a. Enter `1000` for **Number of Iterations**.

b. Click **Calculate**.

Figure 8.3 Residuals



The solution will converge in approximately 120 iterations.

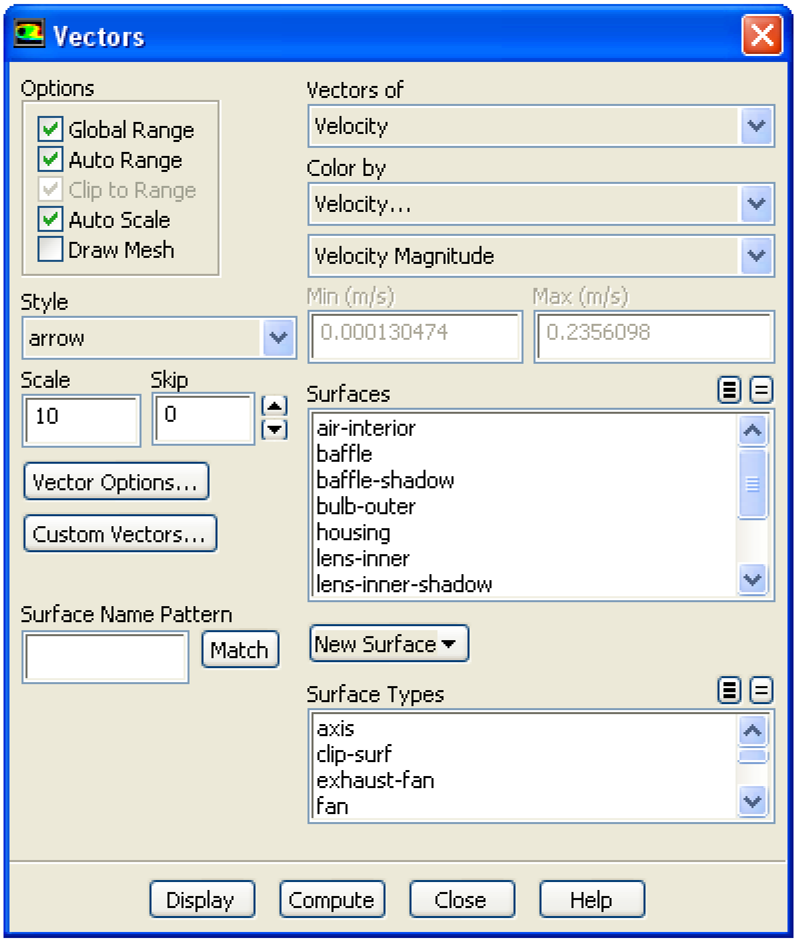
5. Save the case and data files (`do.cas.gz` and `do.dat.gz`).

File → **Write** → **Case & Data...**

8.4.9. Step 8: Postprocessing

1. Display velocity vectors.

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



   a. Enter `10` for **Scale**.

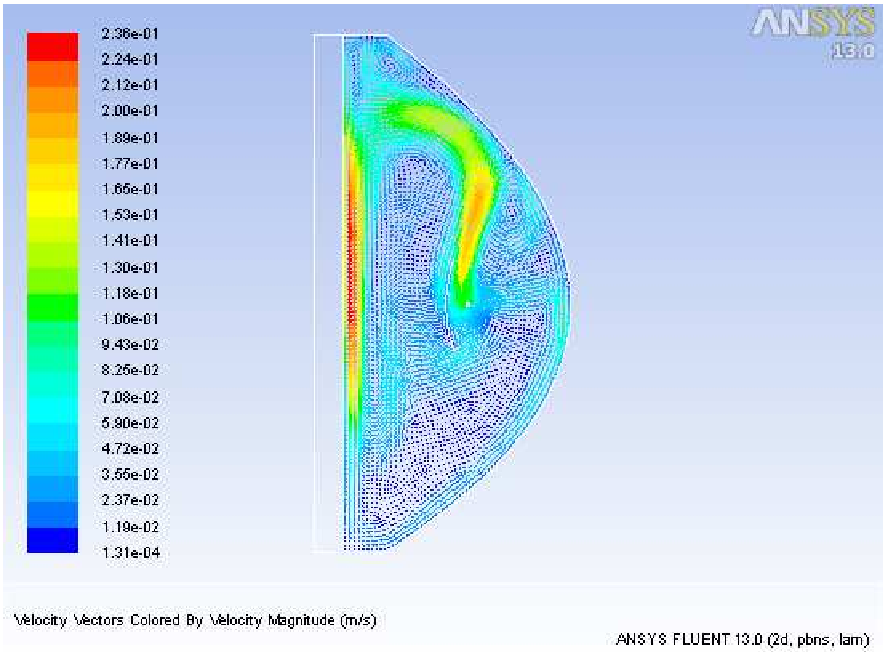
   b. Retain the default selection of **Velocity** from the **Vectors of** drop-down list.

   c. Retain the default selection of **Velocity...** and **Velocity Magnitude** from the **Color by** drop-down list.

   d. Click **Display** (*Figure 8.4* (p. 345)).

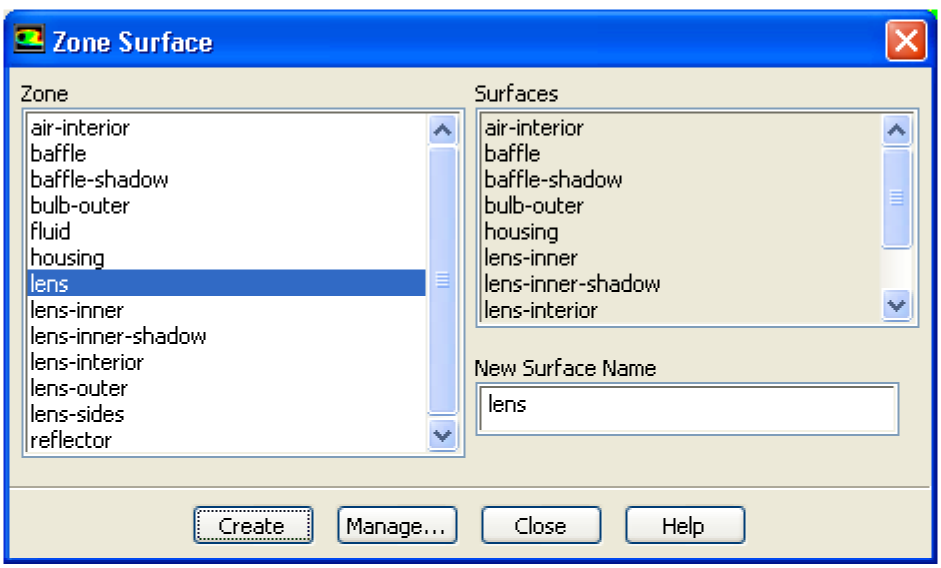
   e. Close the **Vectors** dialog box.

Figure 8.4 Vectors of Velocity Magnitude



2. Create the new surface, **lens**.

   Surface → **Zone...**

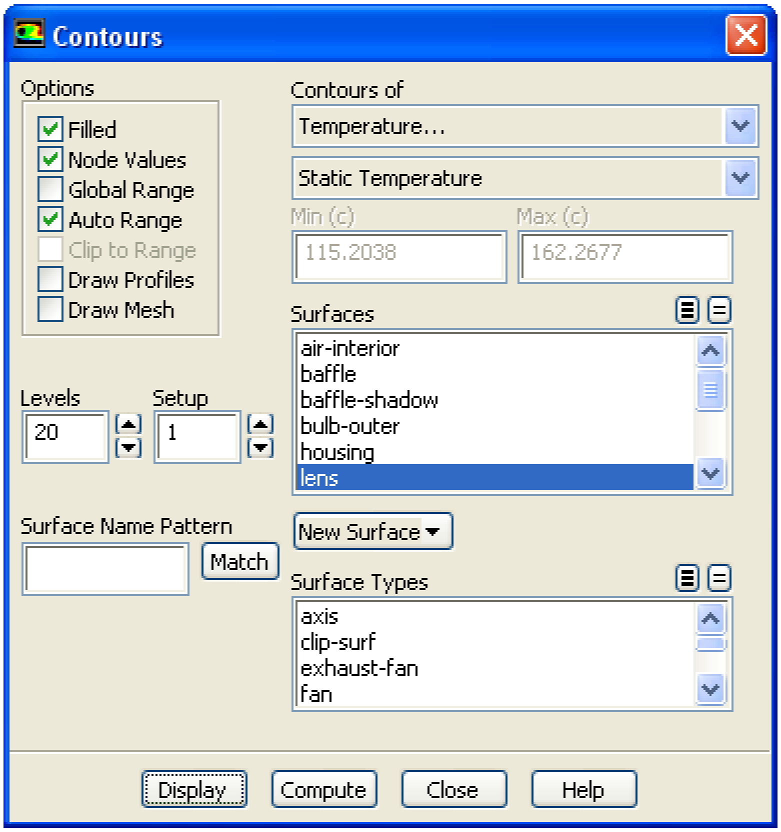


   a. Select **lens** from the **Zone** selection list.

   b. Click **Create** and close the **Zone Surface** dialog box.

3. Display contours of static temperature.

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



a. Enable **Filled** in the **Options** group box.

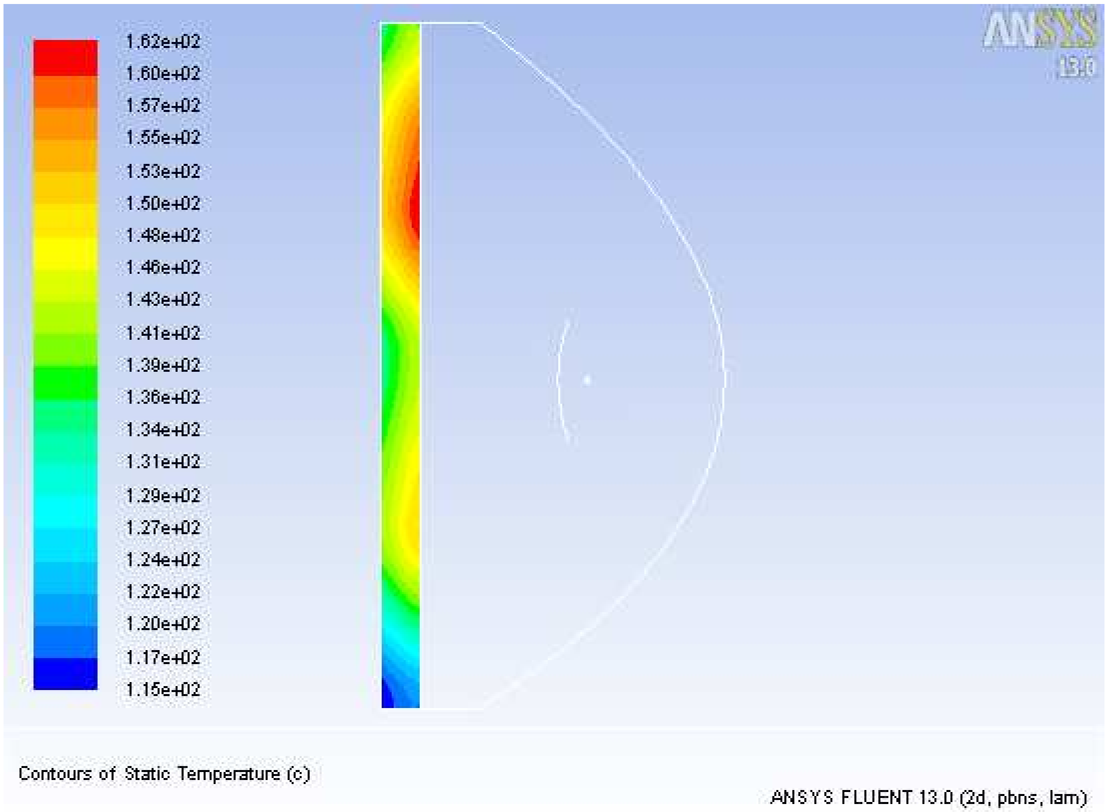
b. Disable **Global Range** in the **Options** group box.

c. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

d. Select **lens** from the **Surfaces** selection list.

e. Click **Display** (*Figure 8.5* (p. 347)).

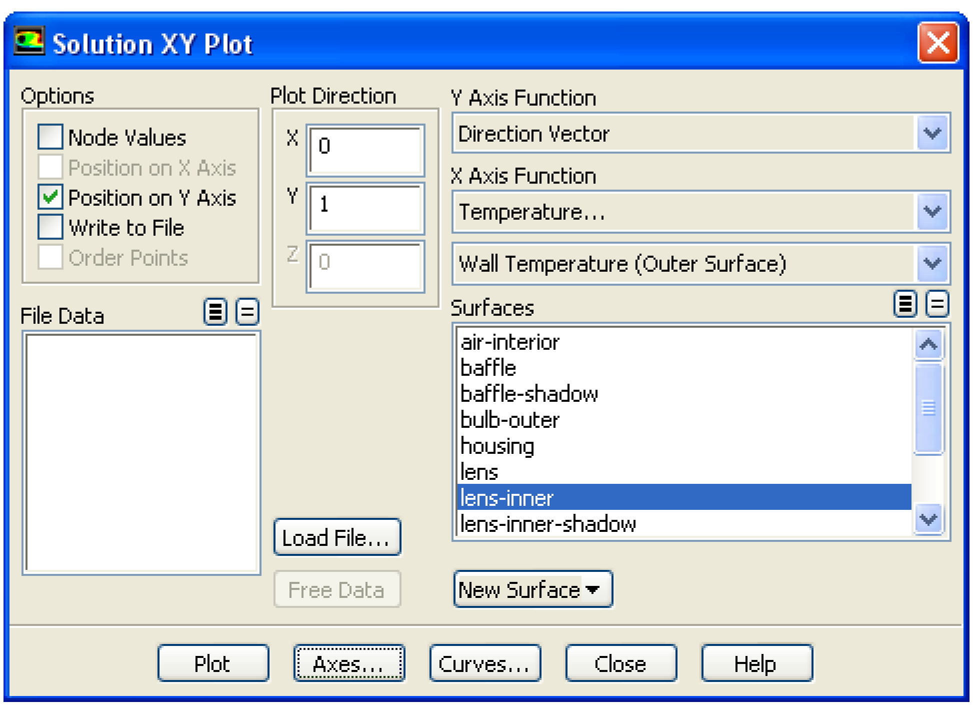
Figure 8.5 Contours of Static Temperature



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

4. Display temperature profile for the **lens-inner**.

Plots → **XY Plot** → **Set Up...**



a. Disable both **Node Values** and **Position on X Axis** in the **Options** group box.

b. Enable **Position on Y Axis**.

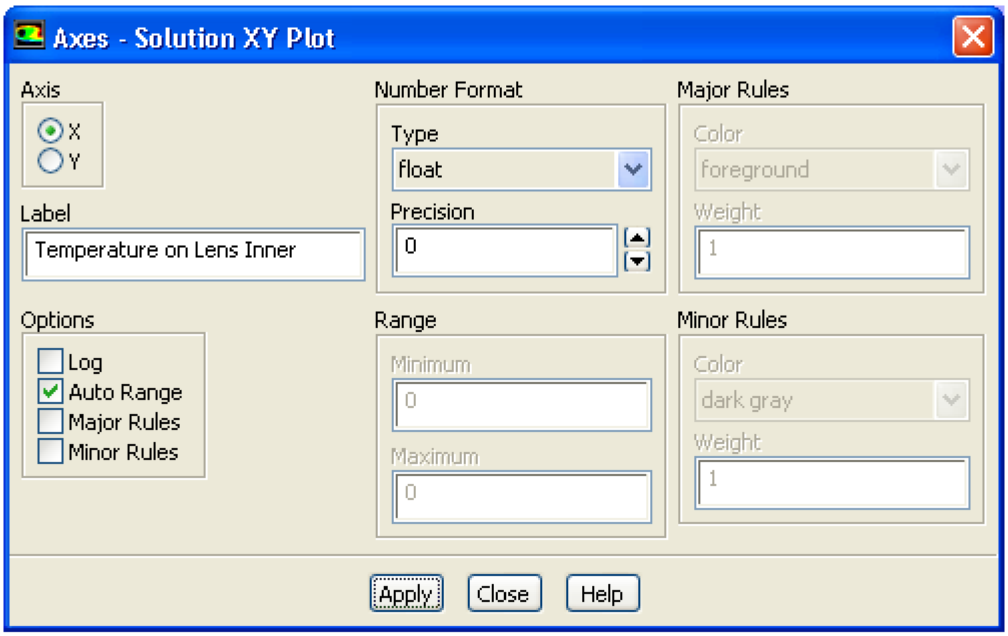
c. Enter `0` and `1` for **X** and **Y** in the **Plot Direction** group box.

d. Retain the default selection of **Direction Vector** from the **Y Axis Function** drop-down list.

e. Select **Temperature...** and **Wall Temperature (Outer Surface)** from the **X Axis Function** drop-down lists.

f. Select **lens-inner** from the **Surfaces** selection list.

g. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box.



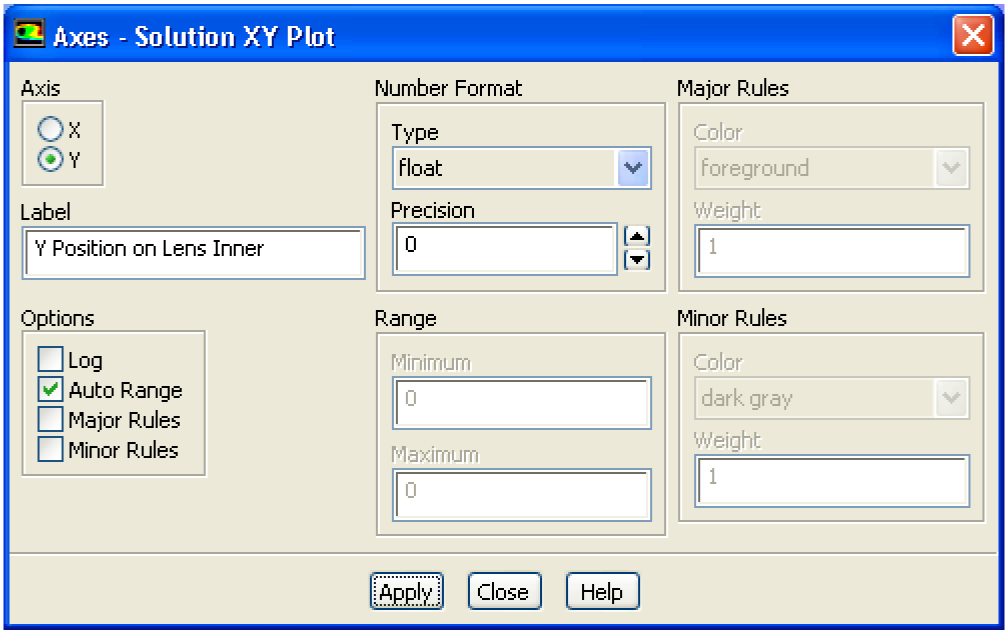
i. Ensure that **X** is selected in the **Axis** list.

ii. Enter `Temperature on Lens Inner` for **Label**.

iii. Select **float** from the **Type** drop-down list in the **Number Format** group box.

iv. Set **Precision** to `0`.

v. Click **Apply**.



vi. Select **Y** in the **Axis** list.

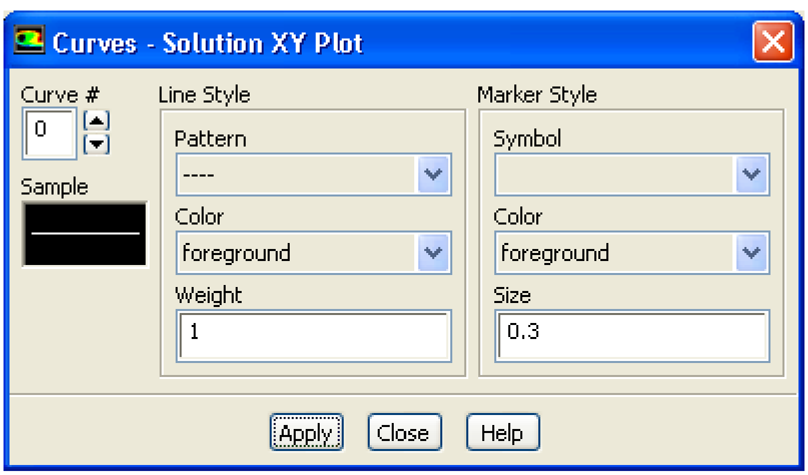
vii. Enter `Y Position on Lens Inner` for **Label**.

viii. Select **float** from the **Type** drop-down list in the **Number Format** group box.

ix. Set **Precision** to `0`.

x. Click **Apply** and close the **Axes - Solution XY Plot** dialog box.

h. Click the **Curves...** button to open the **Curves - Solution XY Plot** dialog box.



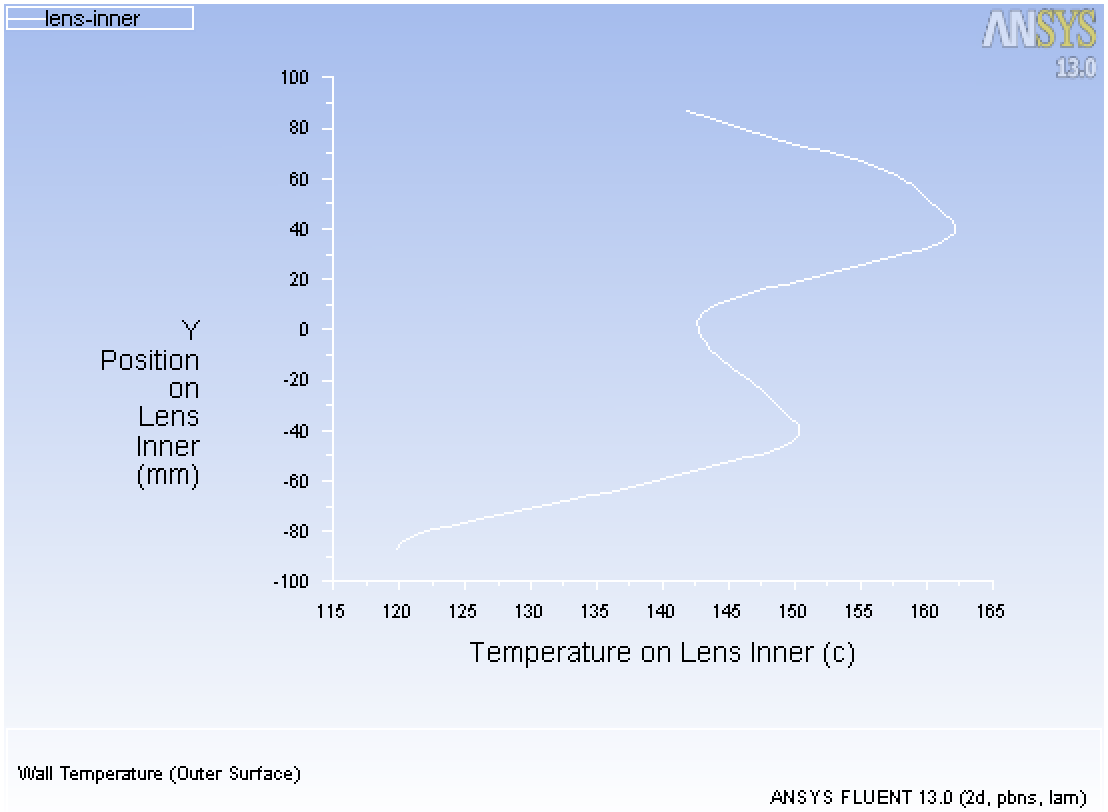
i. Select the line pattern as shown in the **Curves - Solution XY Plot** dialog box.

ii. Select the symbol pattern as shown in the **Curves - Solution XY Plot** dialog box.

iii. Click **Apply** and close the **Curves - Solution XY Plot** dialog box.

i. Click **Plot** (*Figure 8.6* (p. 350)).

Figure 8.6 Temperature Profile forlens-inner



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

   i. Enter `do_2x2_1x1.xy` for **XY File** and close the **Select File** dialog box.

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

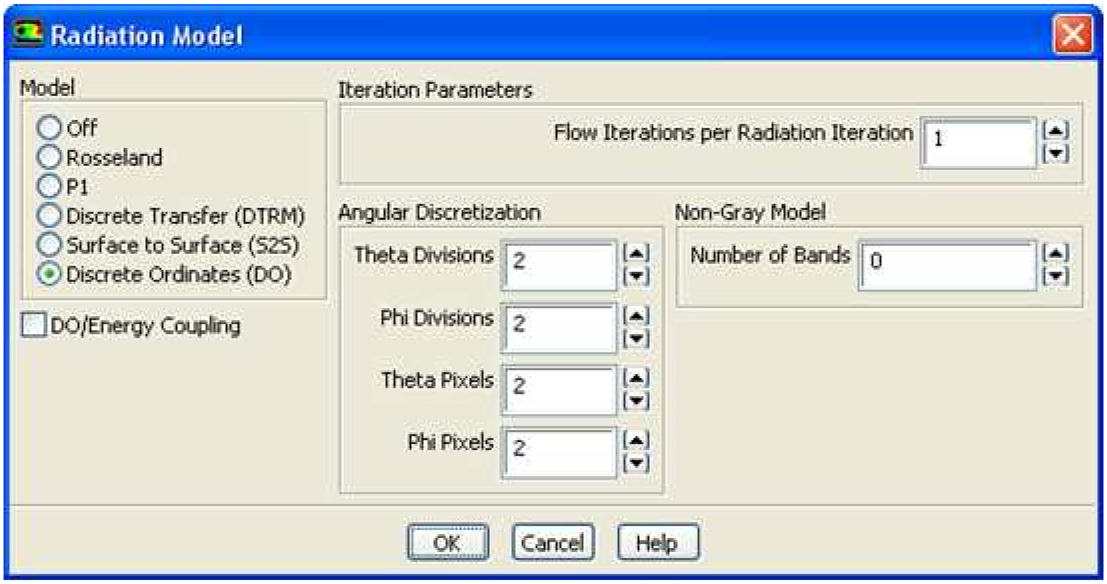
*The key in this plot is changed to **1x1** instead of **lens-inner**.*

8.4.10. Step 9: Iterate for Higher Pixels

1. Increase pixelation for accuracy.

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

   For semi-transparent and reflective surfaces, increasing accuracy by increasing pixilation is more efficient than increasing theta and phi divisions.



a. Set both **Theta Pixels** and **Phi Pixels** to 2.

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

2. Request 1000 more iterations.

Run Calculation

The solution will converge in approximately 100 additional iterations.

3. Save the case and data files (`do_2x2_2x2_pix.cas.gz` and `do_2x2_2x2_pix.dat.gz`).

File → **Write** → **Case & Data...**

4. Display temperature profile for the **lens-inner**.

Plots → **XY Plot** → **Set Up...**

a. Disable **Write to File**.

b. Retain the default settings and plot the temperature profile.

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

   i. Enter `do_2x2_2x2_pix.xy` for **XY File** and close the **Select File** dialog box.

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

   i. Select **do_2x2_1x1.xy** and click **OK** to close the **Select File** dialog box.

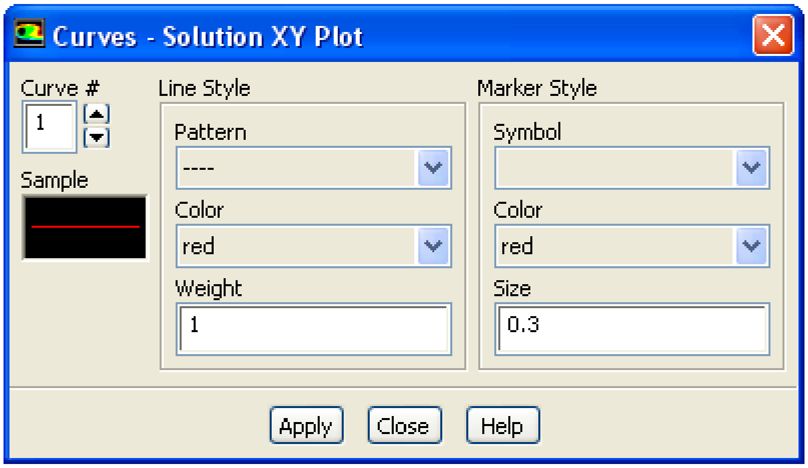
e. Click the **Curves...** button to open **Curves - Solution XY Plot** dialog box.

   i. Set **Curve #** to 1.

   ii. Select the line pattern as shown in the **Curves - Solution XY Plot** dialog box.

   iii. Select the symbol pattern as shown in the **Curves - Solution XY Plot** dialog box.

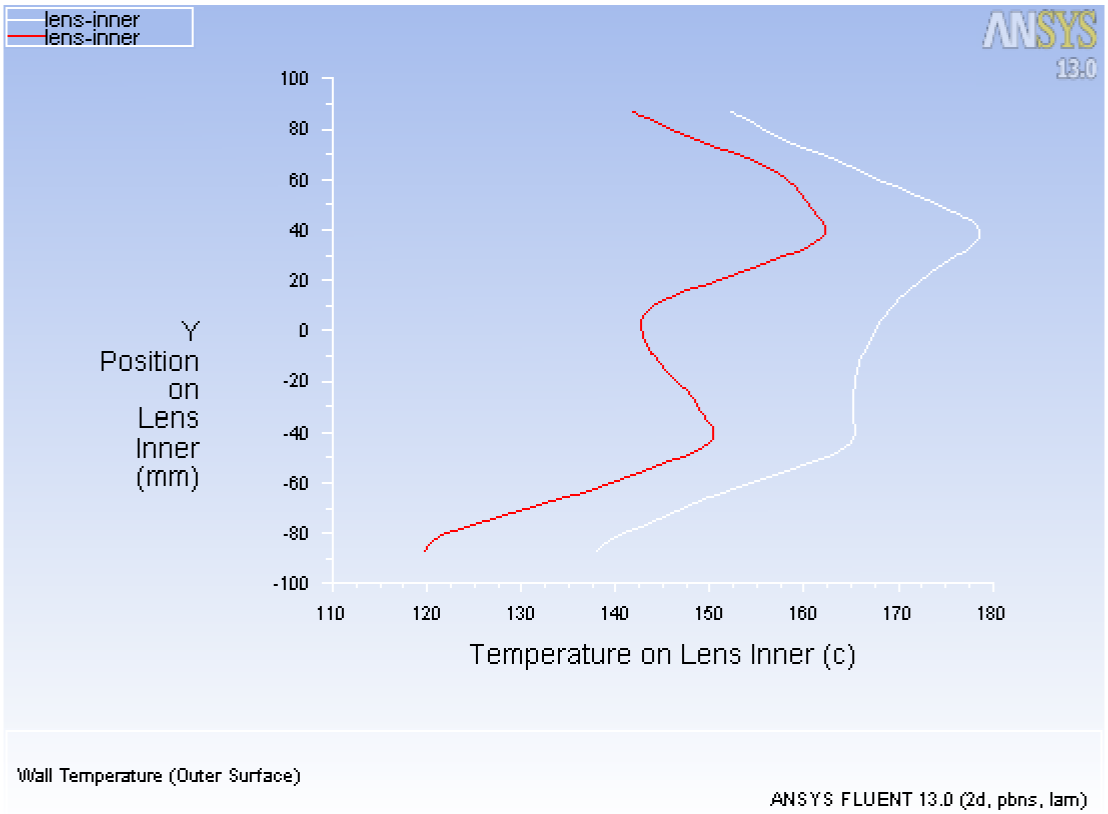
   iv. Click **Apply** and close the **Curves - Solution XY Plot** dialog box.



f. Disable **Write to File**.

g. Click **Plot** (*Figure 8.7* (p. 352)).

Figure 8.7 Temperature Profile forlens-inner



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

5. Increase both **Theta Pixels** and **Phi Pixels** to 3 and continue iterations.

Models → **Radiation** → **Edit...**

6. Click the **Calculate** button.

Run Calculation

The solution will converge in approximately 100 additional iterations.

7. Save the case and data files (`do_2x2_3x3_pix.cas.gz` and `do_2x2_3x3_pix.dat.gz`).

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

8. Display temperature profile for the **lens-inner**.

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

   a. Make sure **Write to File** is disabled.
   b. Ensure that all files are deselected from the **File Data** selection list.
   c. Ensure that **lens-inner** is selected from the **Surfaces** selection list.
   d. Click **Plot**.
   e. Click **Write to File** and save the file as `do_2x2_3x3_pix.xy`.

9. Repeat the procedure for 10 **Theta Pixels** and **Phi Pixels** and save the case and data files (`do_2x2_10x10_pix.cas.gz` and `do_2x2_10x10_pix.dat.gz`).

   a. Save the file as `do_2x2_10x10_pix.xy`.

10. Read in all the files and plot them.

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

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

       i. Select all the xy files and close the **Select File** dialog box.

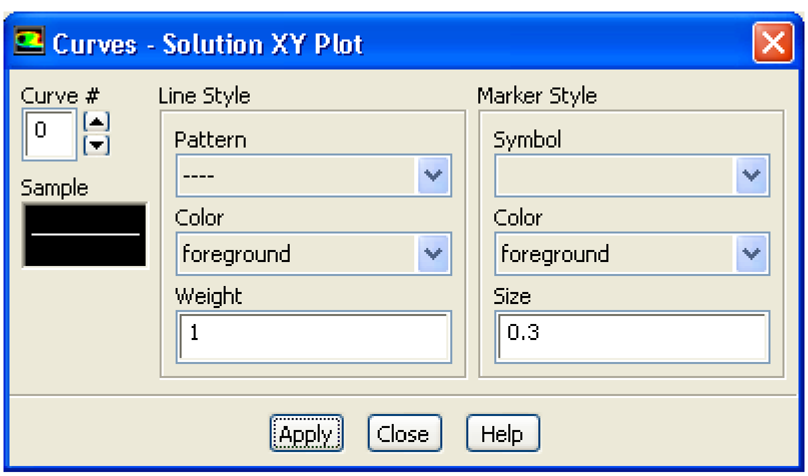
       Note

       Selected files will be listed in the **XY File(s)** selection list.

       *Make sure you deselect **lens-inner** from the **Surfaces** list so that there is no duplicated plot.*

    b. Click the **Curves...** button to open **Curves - Solution XY Plot** dialog box.

       *Make sure you deselect **lens-inner** from the **Surfaces** list so that there is no duplicated plot.*



       i. Select the line pattern as shown in the **Curves - Solution XY Plot** dialog box.

ii. Select the symbol pattern as shown in the **Curves - Solution XY Plot** dialog box.

iii. Click **Apply** to save the settings for curve zero.

iv. Set **Curve #** to 1.

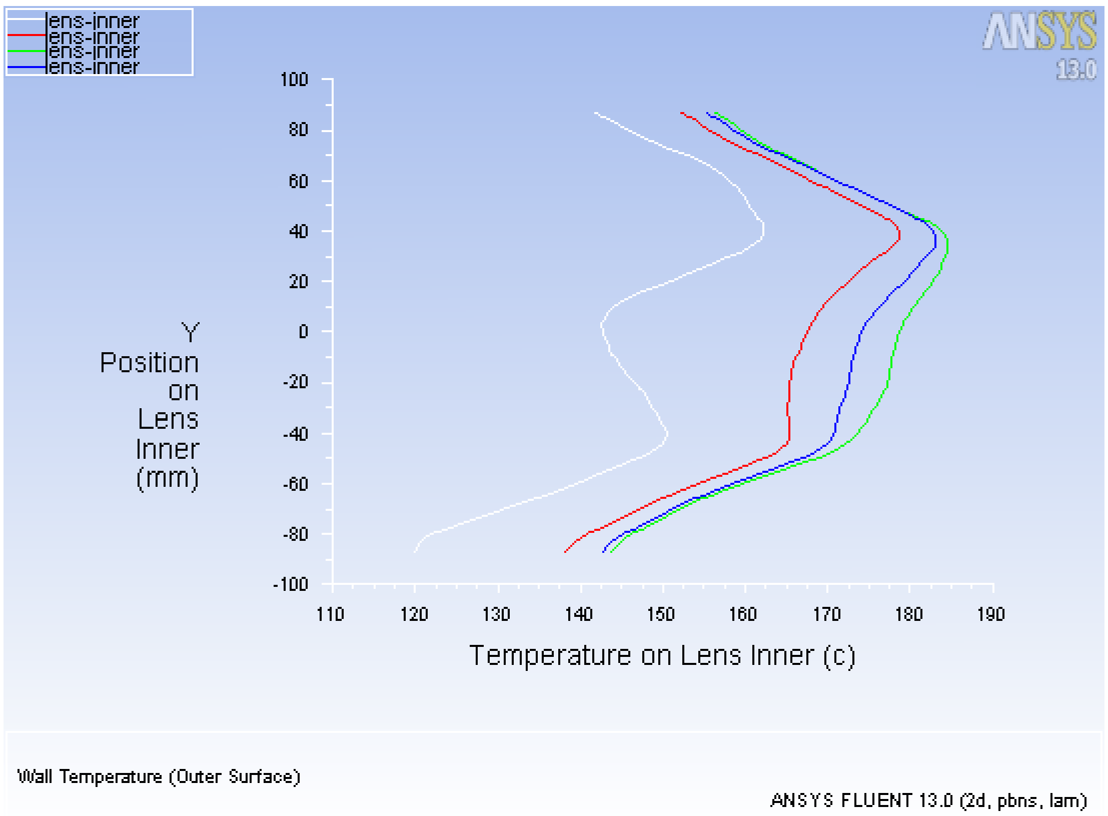
v. Follow the above instructions for curves 2, 3, and 4.

vi. Click **Apply** and close the **Curves - Solution XY Plot** dialog box.

c. Click **Plot** (*Figure 8.8* (p. 354)).

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

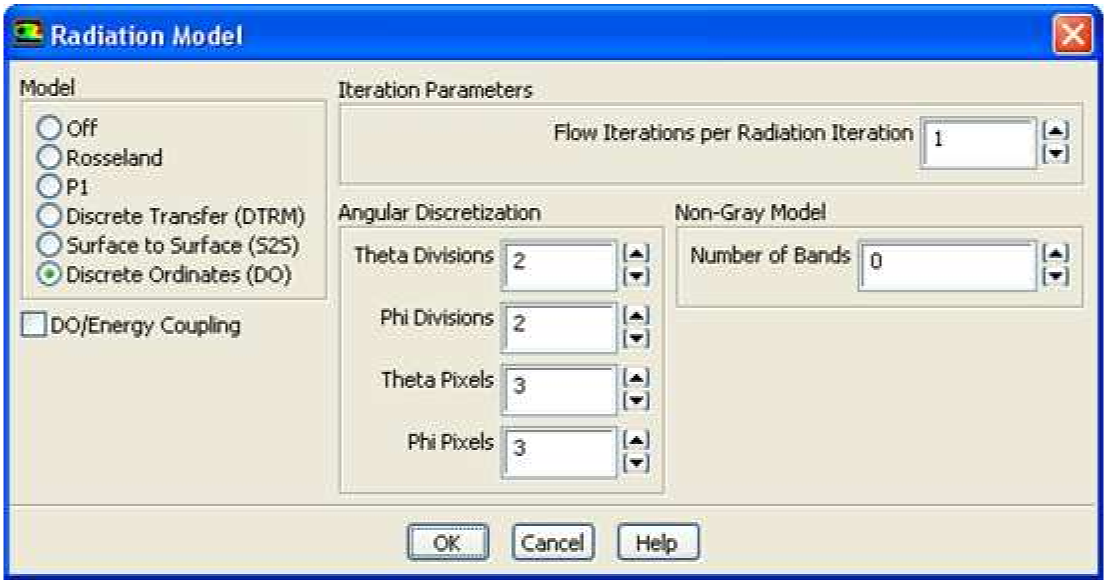
Figure 8.8 Temperature Profile



8.4.11. Step 10: Iterate for Higher Divisions

1. Retain the default division as a base for comparison.

Models → **Radiation** → **Edit...**



a. Retain both **Theta Divisions** and **Phi Divisions** as 2.

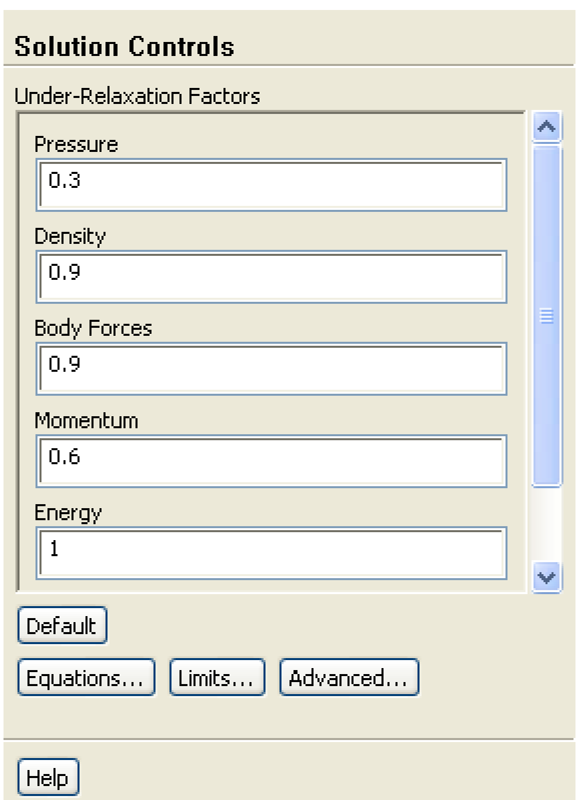
b. Enter a value of 3 for **Theta Pixels** and **Phi Pixels.**

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

This creates a baseline giving better solution efficiency.

2. Set the under-relaxation factors.

Solution Controls



a. Enter `0.9` for **Density**.

b. Enter `0.9` for **Body Forces**.

c. Enter `0.6` for **Momentum**.

3. Request 1000 more iterations.

Run Calculation

The solution will converge in approximately 80 iterations.

4. Save the case and data files (`do_2x2_3x3_div.cas.gz` and `do_2x2_3x3_div.dat.gz`).

File → **Write** → **Case & Data...**

5. Display temperature profiles for the **lens-inner**.

Plots → **XY Plot** → **Set Up...**

a. Select all the files from the **File Data** selection list.

b. Click **Free Data** to remove the files from the list.

c. Retain the settings for **Y axis Function** and **X axis Function**.

d. Select **lens-inner** from the **Surfaces** selection list.

e. Click **Plot**.

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

i. Enter `do_2x2_3x3_div.xy` for **XY File** and close the **Select File** dialog box.

6. Repeat the procedure for 3 **Theta Divisions** and **Phi Divisions**.

a. Save the file as `do_3x3_3x3_div.xy`.

7. Save the case and data files (`do_3x3_3x3_div.cas.gz` and `do_3x3_3x3_div.dat.gz`).

File → **Write** → **Case & Data...**

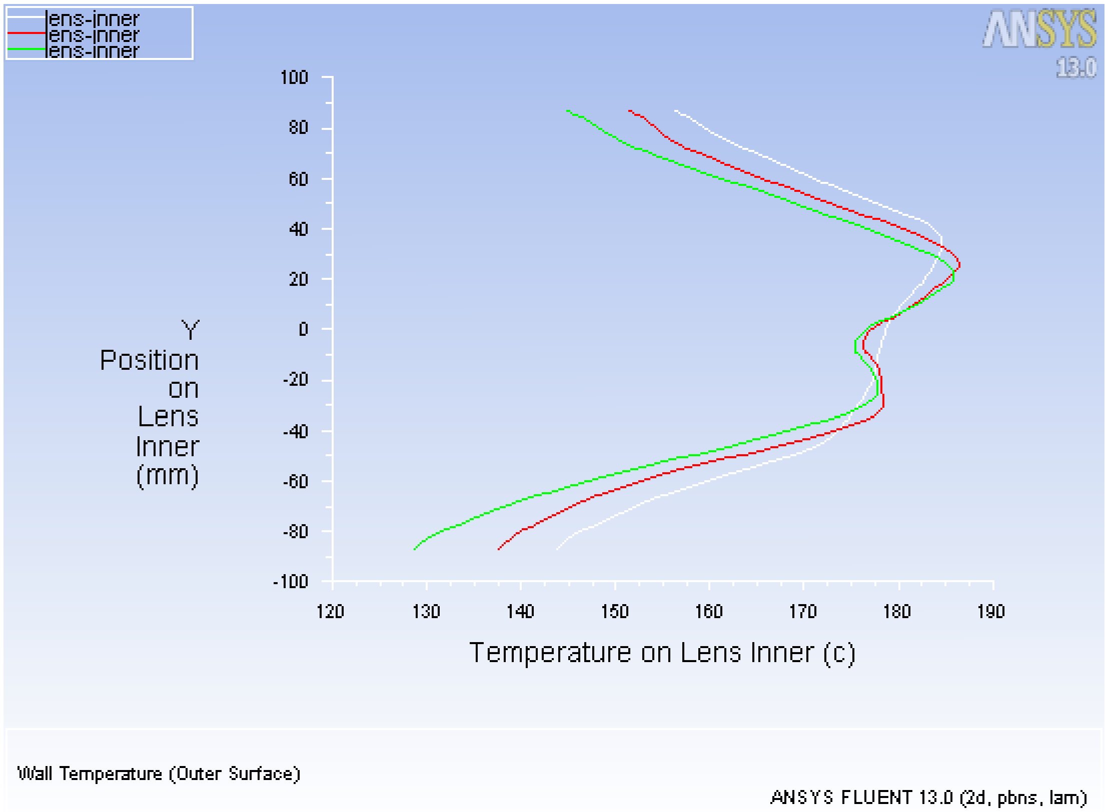
8. Repeat the procedure for 5 **Theta Divisions** and **Phi Divisions**.

a. Save the file as `do_5x5_3x3_div.xy`.

9. Read in all the files for **Theta Divisions** and **Phi Divisions** of 2, 3, and 5 and display temperature profiles.

*Make sure you deselect **lens-inner** from the **Surfaces** list so that no plots are duplicated.*

Figure 8.9 Temperature Profiles for Various Theta Divisions

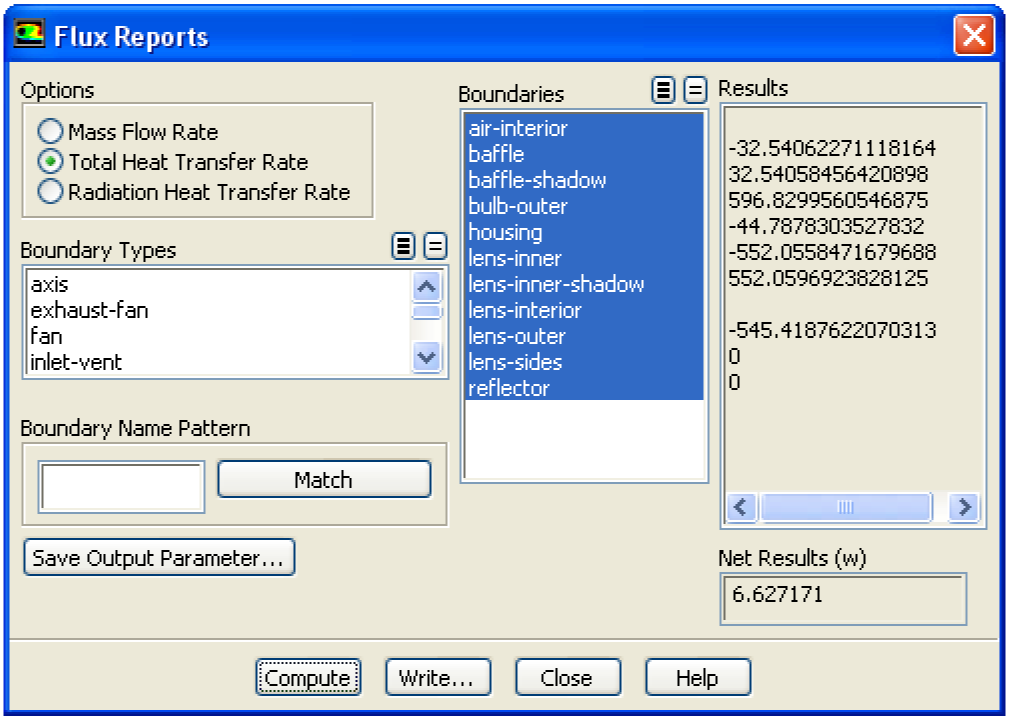


10. Save the case and data files (`do_5x5_3x3_div.cas.gz` and `do_5x5_3x3_div.dat.gz`).

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

11. Compute the total heat transfer rate.

    Reports → **Fluxes** → **Set Up...**



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

b. Select all zones from the **Boundaries** selection list.

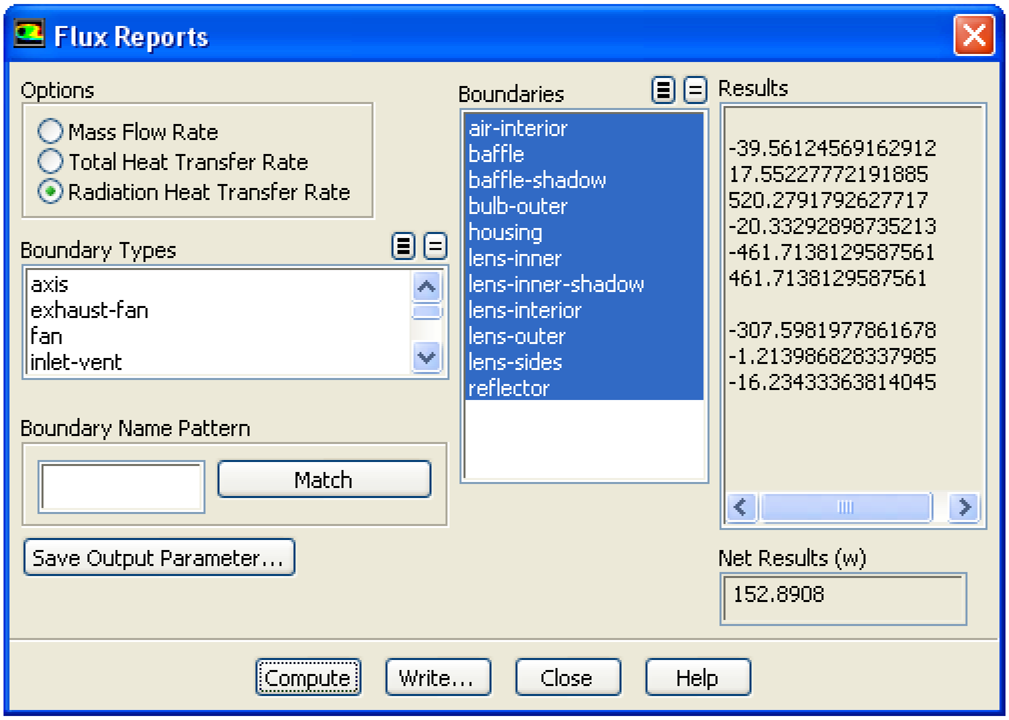
c. Click **Compute**.

Note

The net heat load is 6.627 W, which equates to an imbalance of approximately 1.1% when compared against the heat load of the bulb.

12. Compute the radiation heat transfer rate.

Reports → **Fluxes** → **Set Up...**



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

b. Retain the selection of all boundary zones from the **Boundaries** selection list.

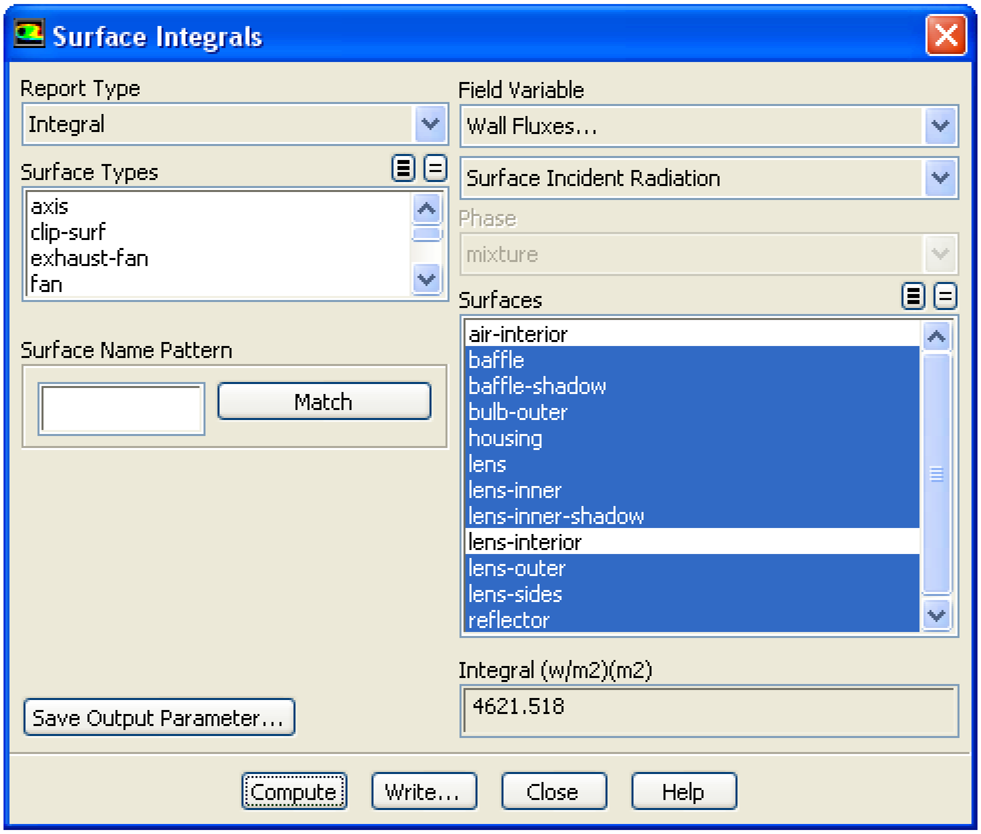
c. Click **Compute** and close the **Flux Reports** dialog box.

Note

The net heat load is 152.92.

13. Compute the radiation heat transfer rate incident on the surfaces.

Reports → **Surface Integrals** → **Set Up...**



a. Select **Integral** from the **Report Type** drop-down list.

b. Select **Wall Fluxes...** and **Surface Incident Radiation** from the **Field Variable** drop-down lists.

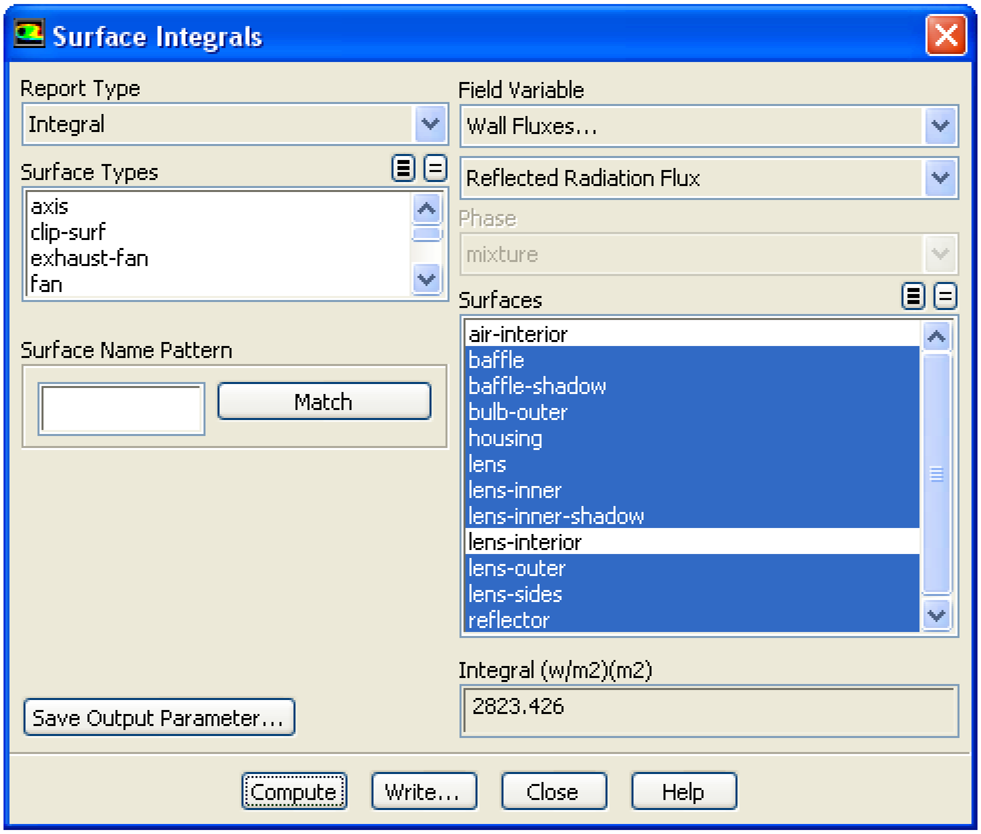
c. Select all surfaces except **air-interior** and **lens-interior** from the **Surfaces** selection list.

d. Click **Compute**.

The incident load on lens-inner is slightly less than that on the reflector. This is because some radiation has been absorbed by the housing. However the incident load on the lens-outer is notably lower due to the amount of radiation which has been absorbed in the solid lens.

14. Compute the reflected radiation flux.

Reports → **Surface Integrals** → **Set Up...**



a. Retain the selection of **Integral** from the **Report Type** drop-down list.

b. Select **Wall Fluxes...** and **Reflected Radiation Flux** from the **Field Variable** drop-down lists.

c. Select all surfaces except **air-interior** and **lens-interior** from the **Surfaces** selection list.

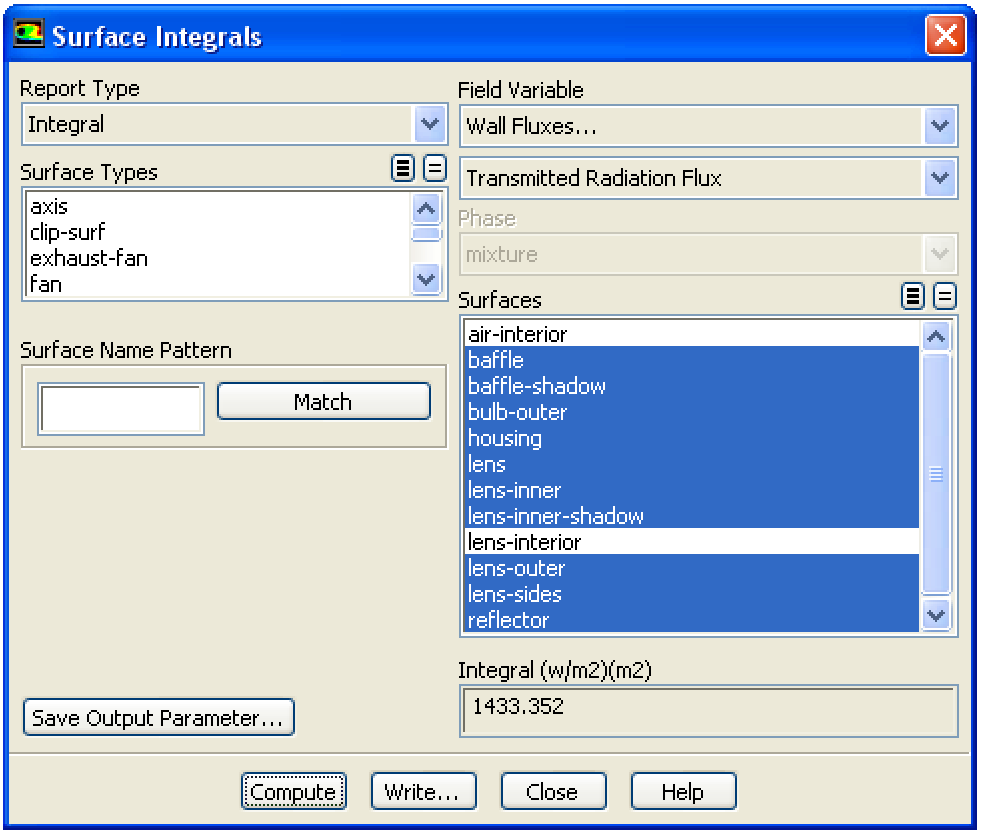
d. Click **Compute**.

*Reflected radiation flux values are printed in the console for all the zones. The zone **baffle** is facing the filament and its shadow (**baffle-shadow**) is facing the lens. There is much more reflection on the filament side than on the lens side, as expected.*

***lens-inner** is facing the fluid and **lens-inner-shadow** is facing the lens. Due to different refractive indexes and non-zero absorption coefficient on the lens, there is some reflection at the interface. Reflection on **lens-inner-shadow** is the reflected energy of the incident radiation from the lens side. Reflection on **lens-inner** is the reflected energy of the incident radiation from the fluid side.*

15. Compute the transmitted radiation flux.

Reports → **Surface Integrals** → **Set Up...**



a. Retain the selection of **Integral** from the **Report Type** drop-down list.

b. Select **Wall Fluxes...** and **Transmitted Radiation Flux** from the **Field Variable** drop-down lists.

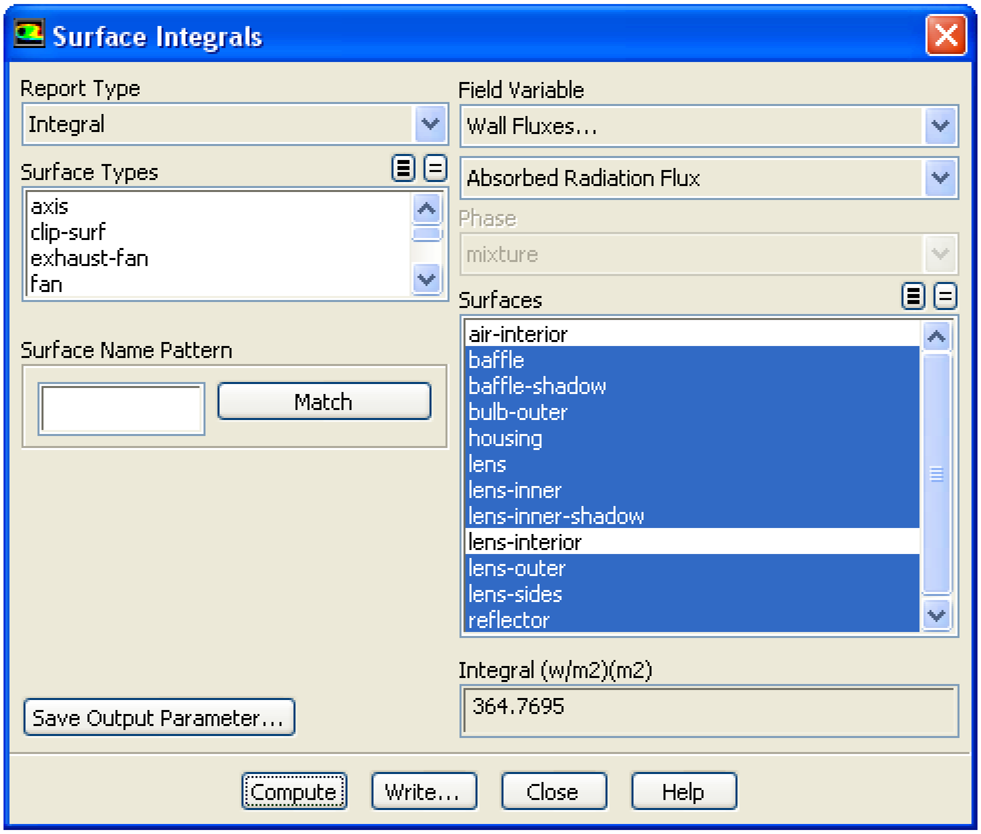
c. Ensure that all surfaces are selected except **air-interior** and **lens-interior** from the **Surfaces** selection list.

d. Click **Compute**.

Transmitted radiation flux values are printed in the console for all the zones. All surfaces are opaque except lens. Zero transmission for all surfaces indicate that they are opaque.

16. Compute the absorbed radiation flux.

Reports → **Surface Integrals** → **Set Up...**



a. Retain the selection of **Integral** from the **Report Type** drop-down list.

b. Select **Wall Fluxes...** and **Absorbed Radiation Flux** from the **Field Variable** drop-down lists.

c. Ensure that all surfaces are selected except **air-interior** and **lens-interior** from the **Surfaces** selection list.

d. Click **Compute**.

e. Close the **Surface Integrals** dialog box.

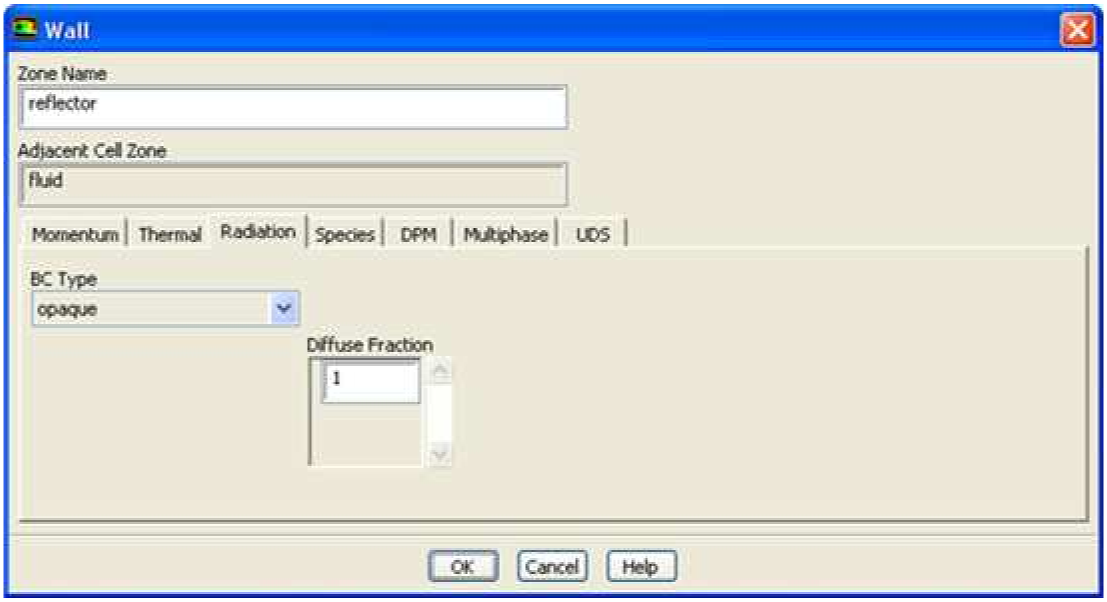
Absorption will only occur on opaque surface with a non-zero internal emissivity adjacent to participating cell zones. Note that absorption will not occur on a semi-transparent wall (irrespective of the setting for internal emissivity). In semi-transparent media, absorption and emission will only occur as a volumetric effect in the participating media with non-zero absorption coefficients.

8.4.12. Step 11: Make the Reflector Completely Diffuse

1. Read in the case and data files (`do_3x3_3x3_div.cas.gz` and `do_3x3_3x3_div.dat.gz`).

2. Increase the diffuse fraction for **reflector**.

   Boundary Conditions → **reflector** → **Edit...**



a. Click the **Radiation** tab and enter 1 for **Diffuse Fraction**.

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

3. Request another 1000 iterations.

Run Calculation

The solution will converge in approximately 80 additional iterations.

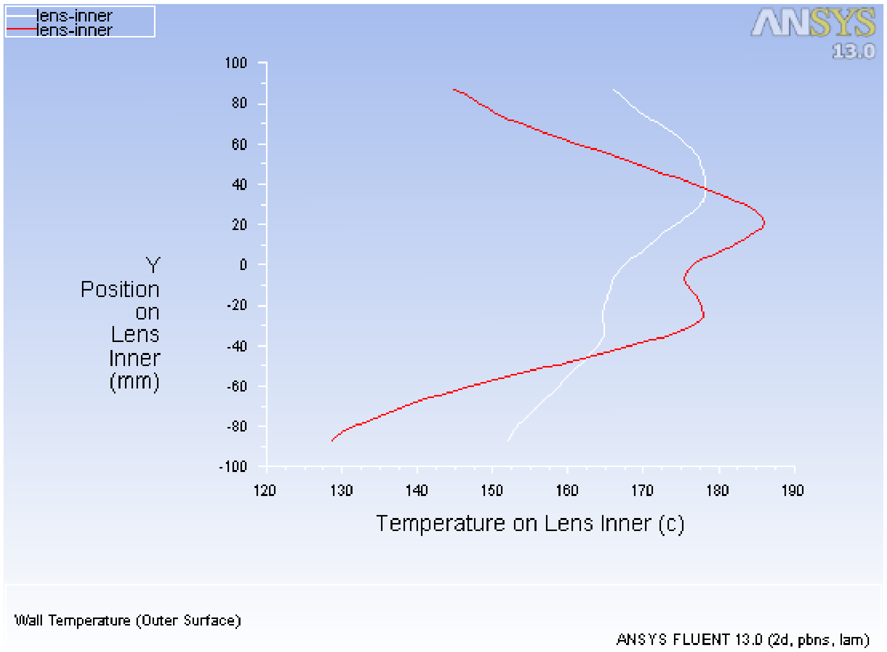
4. Plot the temperature profiles after increasing the diffuse fraction for the **reflector**.

Plots → **XY Plot** →**Set Up...**

a. Save the file as `do_3x3_3x3_div_df=1.xy`.

b. Save the case and data files as `do_3x3_3x3_div_df1.cas.gz` and `do_3x3_3x3_div_df1.dat.gz`.

Figure 8.10 Temperature Profile for Higher Diffuse Fraction



8.4.13. Step 12: Change the Boundary Type of Baffle

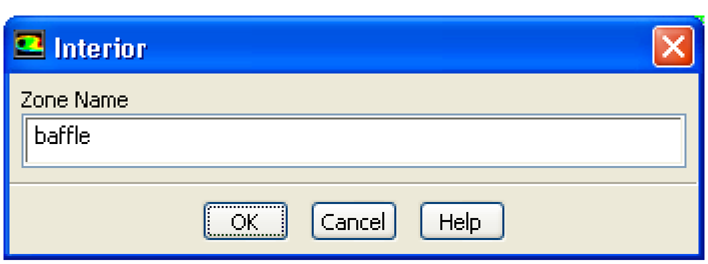
1. Read in the case and data files (`do_3x3_3x3_div.cas.gz` and `do_3x3_3x3_div.dat.gz`).
2. Change the boundary type of **baffle** to **interior**.

Boundary Conditions → **baffle**

a. Select **interior** from the **Type** drop-down list.

*A **Question** dialog box will open, asking if you want to change **Type** of **baffle** to **interior**.*

b. Click **Yes** in the **Question** dialog box.



c. Click **OK** in the **Interior** dialog box.

3. Request another 1000 iterations.

Run Calculation

The solution will converge in approximately 160 additional iterations.

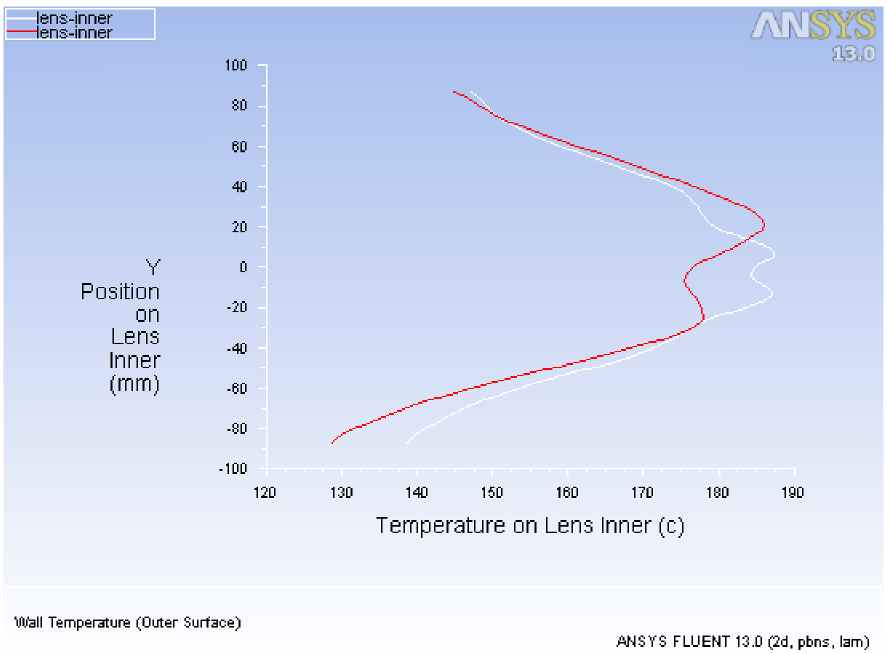
4. Plot the temperature profile for **baffle interior**.

Plots → **XY Plot** → **Set Up...**

a. Save the file as `do_3x3_3x3_div_baf_int.xy`.

b. Save the case and data files as `do_3x3_3x3_div_int.cas.gz` and `do_3x3_3x3_div_int.dat.gz`.

Figure 8.11 Temperature Profile of baffle interior



8.5. Summary

This tutorial demonstrated the modeling of radiation using discrete ordinates (DO) radiation model in ANSYS FLUENT. In this tutorial, you learned the use of angular discretization and pixelation available in discrete ordinates radiation model and solved for different values of **Pixels** and **Divisions**. You studied the change in behavior for higher absorption coefficient. Changes in internal emissivity, refractive index, and diffuse fraction are illustrated with the temperature profile plots.

8.6. 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).