# *Tutorial: Thermal Model of Head Lamp using DO Radiation Model*

## Introduction

This tutorial illustrates the set up and solution of flow and thermal model of an automotive head lamp. The discrete ordinates (DO) radiation model is used to model the radiation. The shell conduction capability of ANSYS FLUENT can be used to model conduction in thin sheets, such as reflector, housing, and shield without having to mesh. Using the shell conduction, the solver will grow one-layer of prism to model conduction in the planar direction. In this tutorial, planar conduction will be applied only at coating.

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 the flow equation.
- Initialize and obtain solution.
- Postprocess the resulting data.

## Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from ANSYS FLUENT 13.0 Tutorial Guide, 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

An automotive headlamp is shown in Figure 1. Forty watts of electric power is dissipated inside the filament. Some of this heat is transfered from the filament by radiation and some by natural convection. Some of the radiation emitted by the filament is transmitted through the bulb, while some is reflected and some is absorbed. The bulb of the headlamp is made of glass whereas the lens, housing, and the reflector are made of polycarbonate.

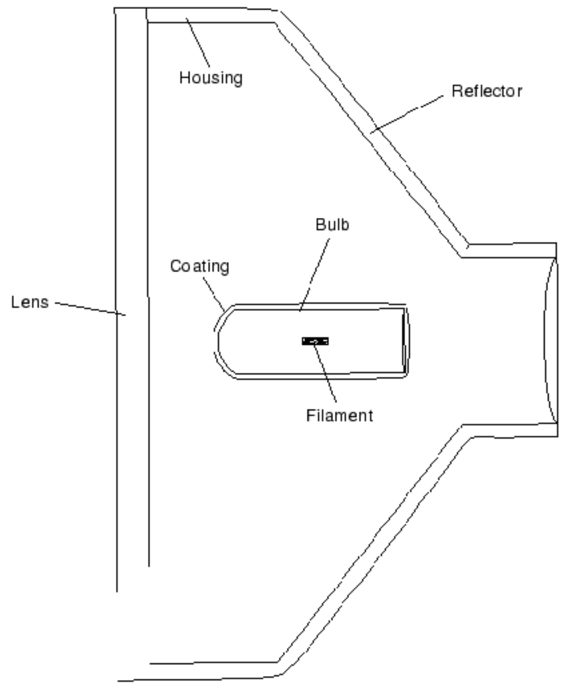



Figure 1: Schematic of the Headlamp

## Preparation

1. Copy the mesh file, (`head-lamp.msh`) to the working directory.
2. Use FLUENT Launcher to start the 3D version of ANSYS FLUENT.

   *For more information about* FLUENT Launcher *see Section* 1.1.2 Starting ANSYS FLUENT Using FLUENT Launcher *in the* ANSYS FLUENT *13.0 User's Guide.*
3. Enable Double-Precision in the Options list.

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

### Step 1: Mesh

1. Read the mesh file `head-lamp.msh.gz`.

   File ⟶ Read ⟶ Mesh...

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

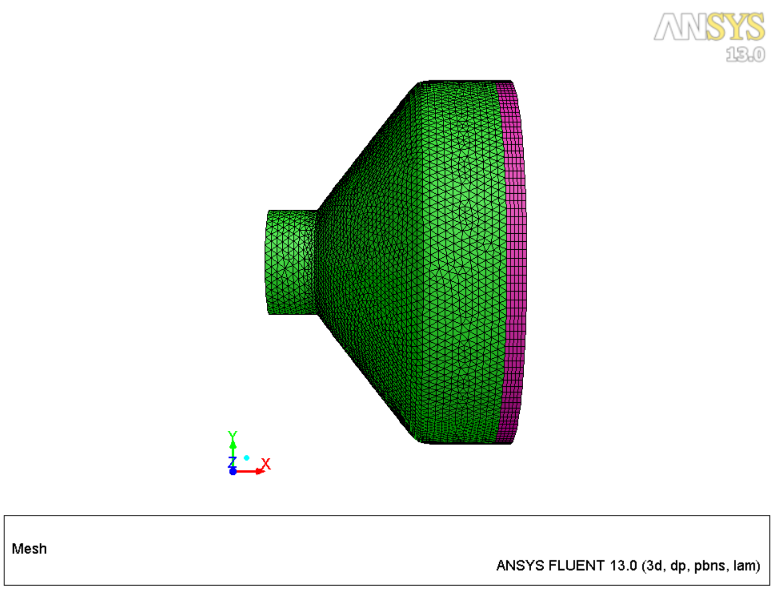



Figure 2: Mesh

### Step 2: General

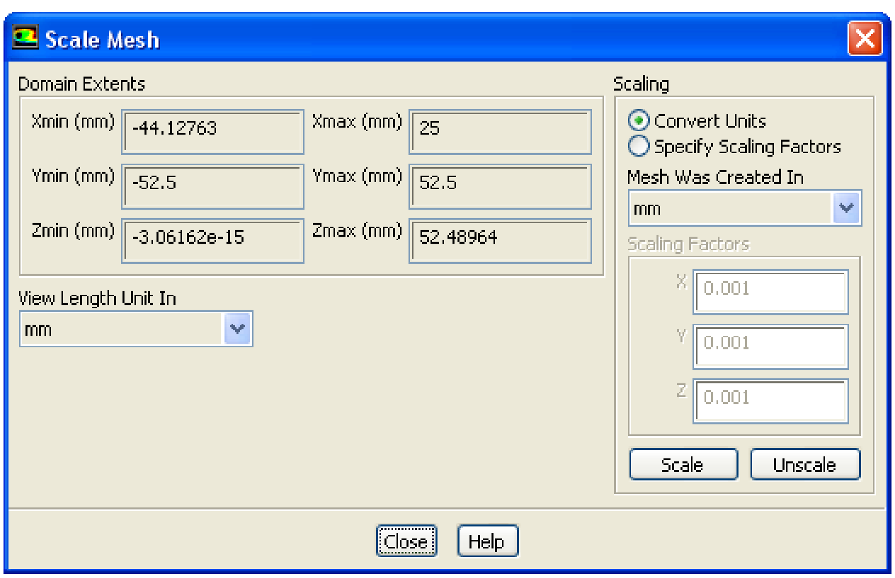
1. Check the mesh.

   General ⟶ Check

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

2. Scale the mesh.

   General ⟶ Scale...



(a) Select mm from the Mesh Was Created In drop-down list.

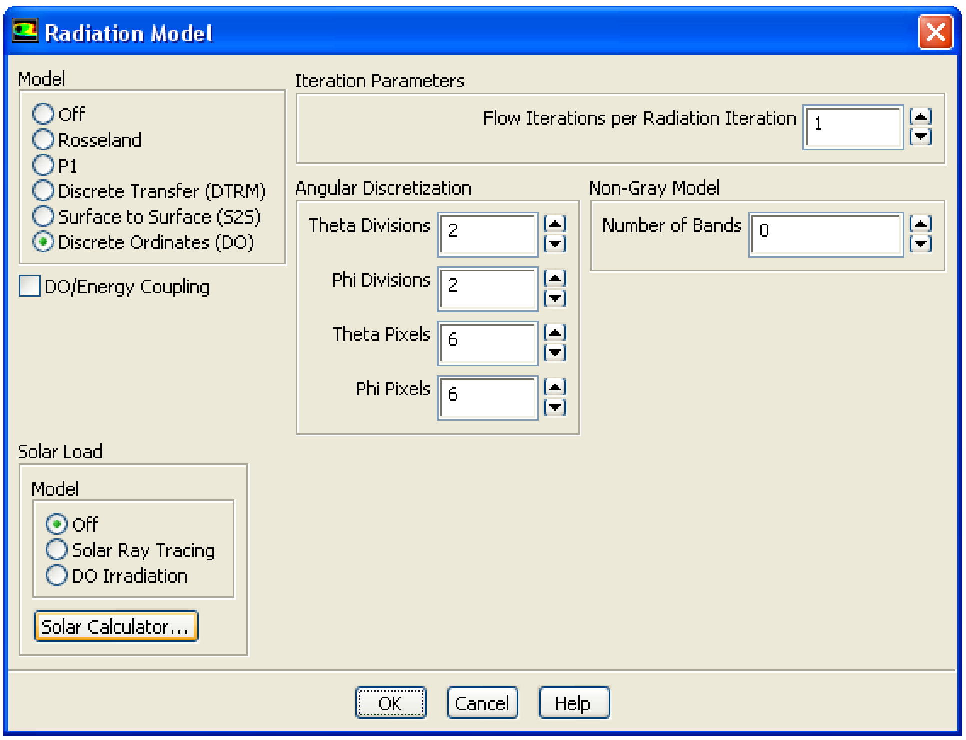
(b) Click Scale.

(c) Close the Scale dialog box.

**Step 2: Models**

1. Enable the Discrete Ordinates (DO) radiation model.

   Models ⟶ Radiation ⟶ Edit...




(a) Enter 1 for Flow Iterations per Radiation Iteration in the Iteration Parameters group box.

(b) Retain the default value of 2 for Theta Divisions and Phi Divisions in the Angular Discretization group box.

*For radiation through a semi-transparent medium, a minimum number of* 3 *pixels is recommended for* Theta Divisions *and* Phi Divisions.

(c) Enter 6 for Theta Pixels and Phi Pixels.

**Note:** *Calculation of radiation through a semi-transparent medium can be CPU intensive.*

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

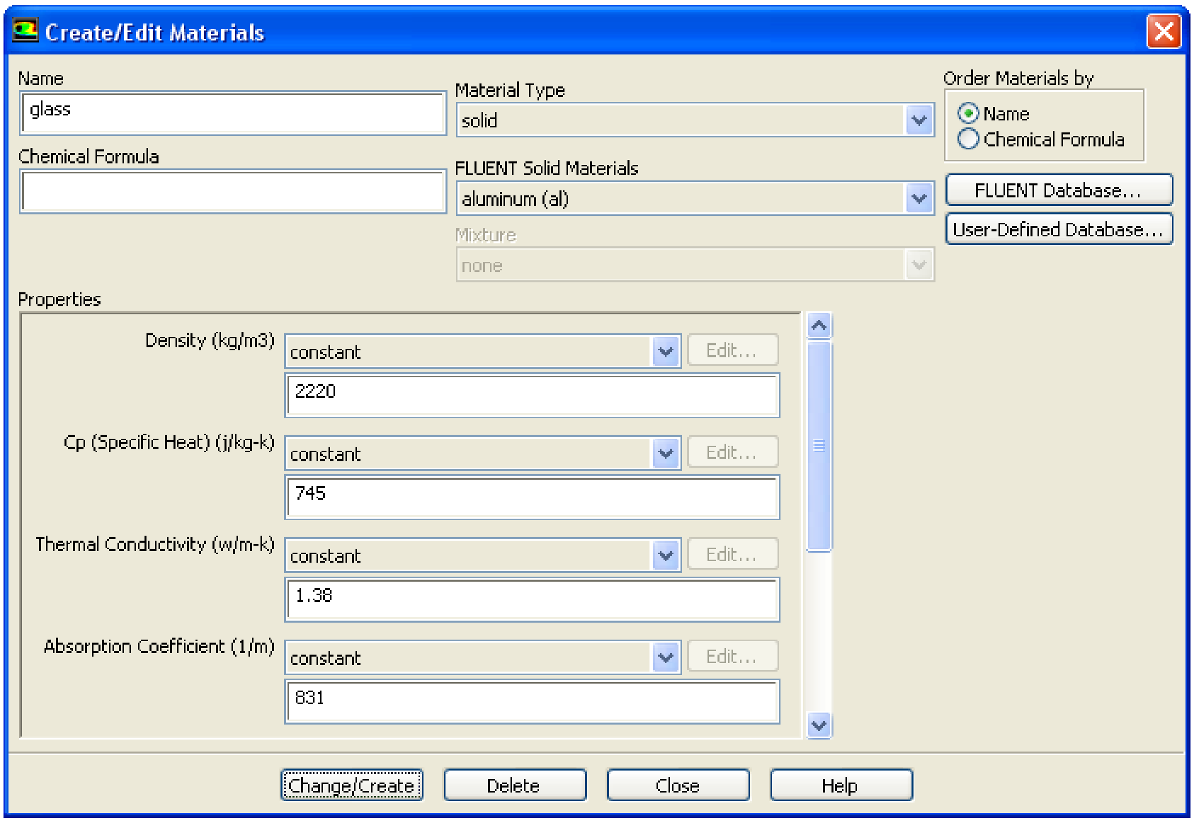
*An* Information *dialog box will appear informing that material properties have changed. Click* OK.

**Note:** *Enabling the radiation model automatically enables the energy equation.*

### Step 3: Materials

1. Create a new solid glass.

   Materials ⟶ Solid ⟶ Create/Edit...




   (a) Enter `glass` for Name and delete the entry for Chemical Formula.

   (b) Enter 2220 kg/$m^3$ for Density and 745 J/kg-K for Cp.

   (c) Enter `1.38` W/m-K and `831` 1/m for Thermal Conductivity and Absorption Coefficient respectively.

   (d) Retain the default value of `0` for Scattering Coefficient.

   *There are no particles in the glass that would scatter the radiation in different directions.*

   (e) Enter `1.5` for Refractive Index.

   (f) Click Change/Create.

   *A* Question *dialog box will appear asking whether to overwrite aluminum. Click* No.

   If you overwrite aluminum, it will be removed from the material list.

2. Similarly create a new solid polycarbonate with the properties shown in the table:

| Property | Value |
|---|---|
| Density | 1200 |
| Cp | 1250 |
| Thermal Conductivity | 0.3 |
| Absorption Coefficient | 930 |
| Scattering Coefficient | 0 |
| Refractive Index | 1.57 |

3. Create a new solid `coating` with the properties shown in the table:

| Property | Value |
|---|---|
| Density | 2000 |
| Cp | 400 |
| Thermal Conductivity | 0.5 |
| Absorption Coefficient | 0 |
| Scattering Coefficient | 0 |
| Refractive Index | 1 |

4. Create the solid material socket with the properties shown in the table:

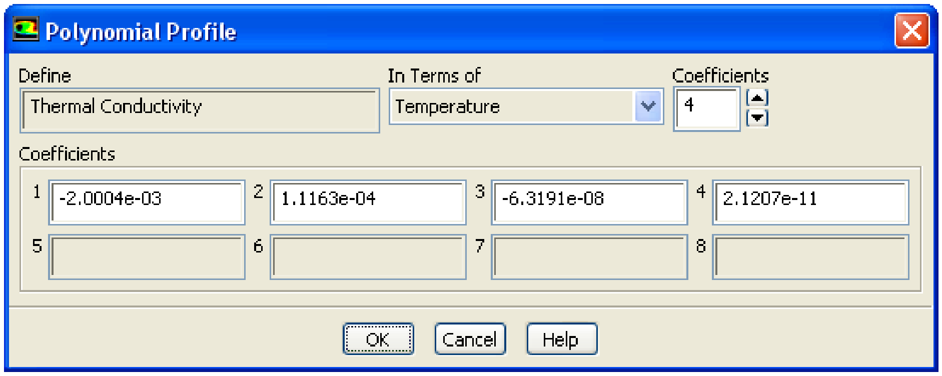
| Property | Value |
|---|---|
| Density | 2719 |
| Cp | 871 |
| Thermal Conductivity | 0.7 |
| Absorption Coefficient | 0 |
| Scattering Coefficient | 0 |
| Refractive Index | 1 |

5. Modify the properties of air.

   Materials ⟶ air ⟶ Create/Edit...

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

   (b) Select polynomial from the Thermal Conductivity drop-down list to open the Polynomial Profile dialog box.




**Note:** *The temperature in the head lamp varies from* 2800 *K to* 350 *K. Therefore, the thermal conductivity will vary considerably.*

*Thermal conductivity was curve fitted to the nth-order polynomial using property data at atmospheric pressure condition. The least-squares approximation method was used to determine the coefficients.*

i. Increase the number of Coefficients to 4.

ii. Enter the values for the coefficients as `-2.0004e-03`, `1.1163e-04`, `-6.3191e-08`, and `2.1301e-11` respectively.

iii. Click OK to close the Polynomial Profile dialog box.

(c) Retain the default values for other parameters.

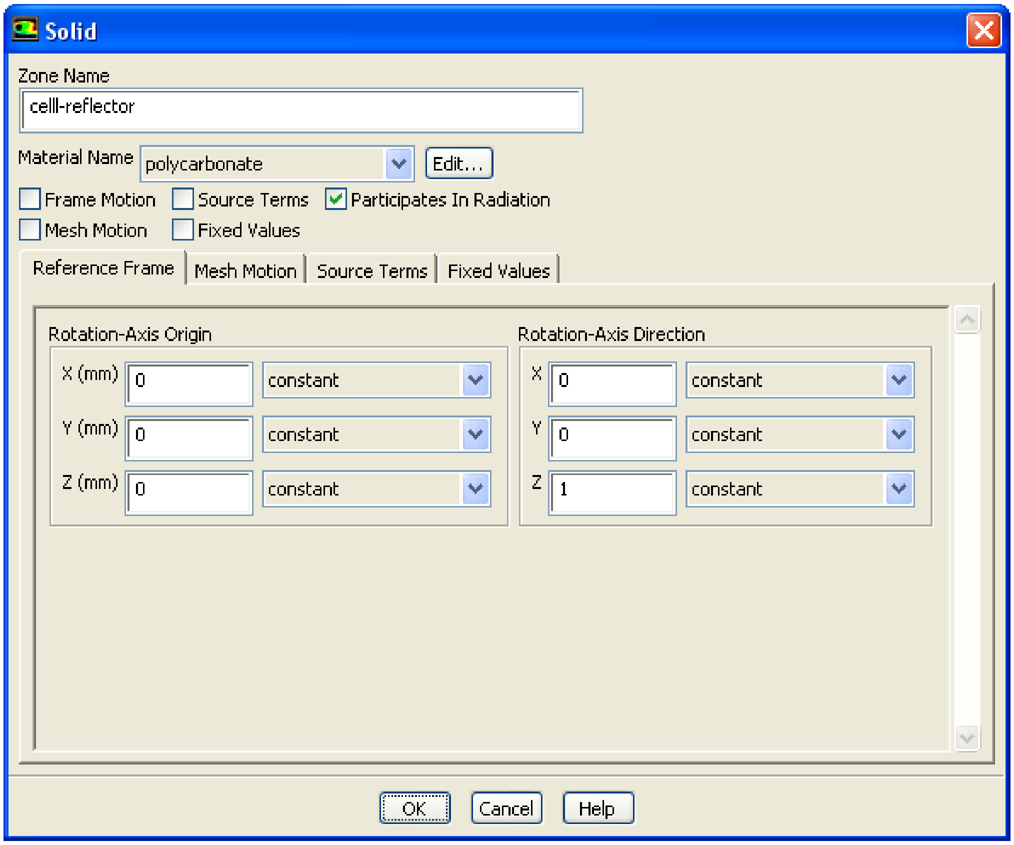
- *The* Absorption Coefficient *(a) for air is negligible at such high operating temperatures. The value of optical thickness (a×L) is much smaller than 1 (where L is some characteristic length*
- *For* Scattering Coefficient*, there are no particles in air to scatter the radiation in different directions, so the scattering coefficient is set to zero (assuming zero humidity*

(d) Click Change/Create and close the Create/Edit Materials dialog box.

## Step 4: Cell Zone Conditions

1. Specify continuum condition for the reflector.

   Cell Zone Conditions ⟶ celll-reflector ⟶ Edit...




(a) Select polycarbonate from the Material Name drop-down list.

(b) Enable Participates In Radiation and click OK to close the Solid dialog box.

2. Specify continuum condition for the thickness of the bulb.

   Cell Zone Conditions ⟶ cells-bulb ⟶ Edit...

   (a) Select glass from the Material Name drop-down list.

   (b) Enable Participates In Radiation and click OK to close the Wall dialog box.

   *This will consider transmission and absorption of radiation inside the bulb thickness.*

3. Specify continuum condition for air inside the housing.

   Cell Zone Conditions ⟶ cells-housing-air ⟶ Edit...

   (a) Ensure that Participates In Radiation is enabled.

   (b) Click OK to close the Fluid dialog box.

   *This will consider surface-to-surface radiation from the bulb to the other components.*

4. Specify continuum condition for the lens.

   Cell Zone Conditions ⟶ cells-lens ⟶ Edit...

(a) Select polycarbonate from the Material Name drop-down list.

(b) Enable Participates In Radiation and click OK to close the Wall dialog box.

*This will calculate for transmission and absorption inside the bulb thickness.*

**Step 5: Operating Conditions**

Cell Zone Conditions ⟶ Operating Conditions...

1. Enable Gravity.

   *The* Operating Conditions *dialog box will expand to show the related inputs.*

2. Enter `-9.81` for Y (m/s2) in Gravitational Acceleration group box.

3. Click OK to close the Operating Conditions dialog box.

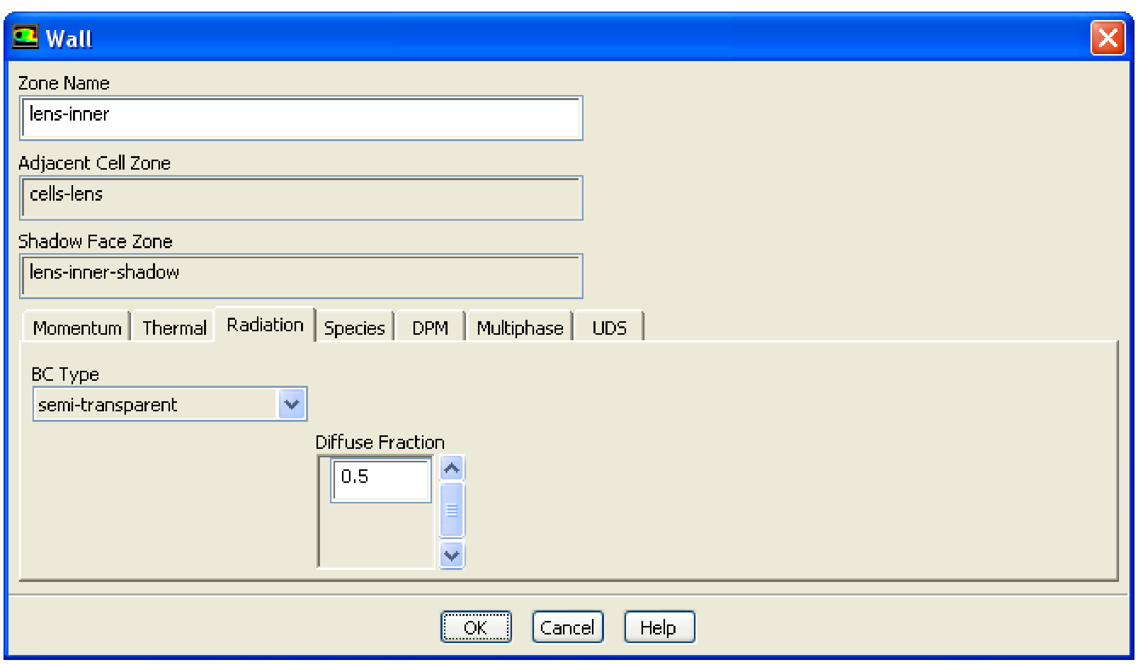
   *Since density is defined as a function of temperature (incompressible ideal gas), the reference density in the body-force term in the y-direction momentum equation needs to be specified. If* Variable-Density Parameters *is not specified, at each flow iteration, the average air density through out the domain will be used for the operating density.*

**Step 6: Boundary Conditions**

1. Specify the inner surface of the lens as semi-transparent.

   Boundary Conditions ⟶ lens-inner ⟶ Edit...

   *This is an internal wall with cells on both sides, so there is also a shadow zone corresponding to it. The shadow is facing the fluid zone.*



(a) Click the Radiation tab and select semi-transparent from the BC Type drop-down list.

**Note:** *The net amount of the incoming radiation is computed. Of this, the Diffuse Fraction is reflected and transmitted diffusely. The reflectivity and transmissivity of a semi-transparent boundary can be computed for each incoming direction by integrating over the entire incoming solid angle using Snell's law. The remainder is then treated specularly.*

*For a clean surface, the diffuse fraction is* `0`*. However, for rough glass, some fraction would be diffused.*

i. Enter `0.05` for Diffuse Fraction.

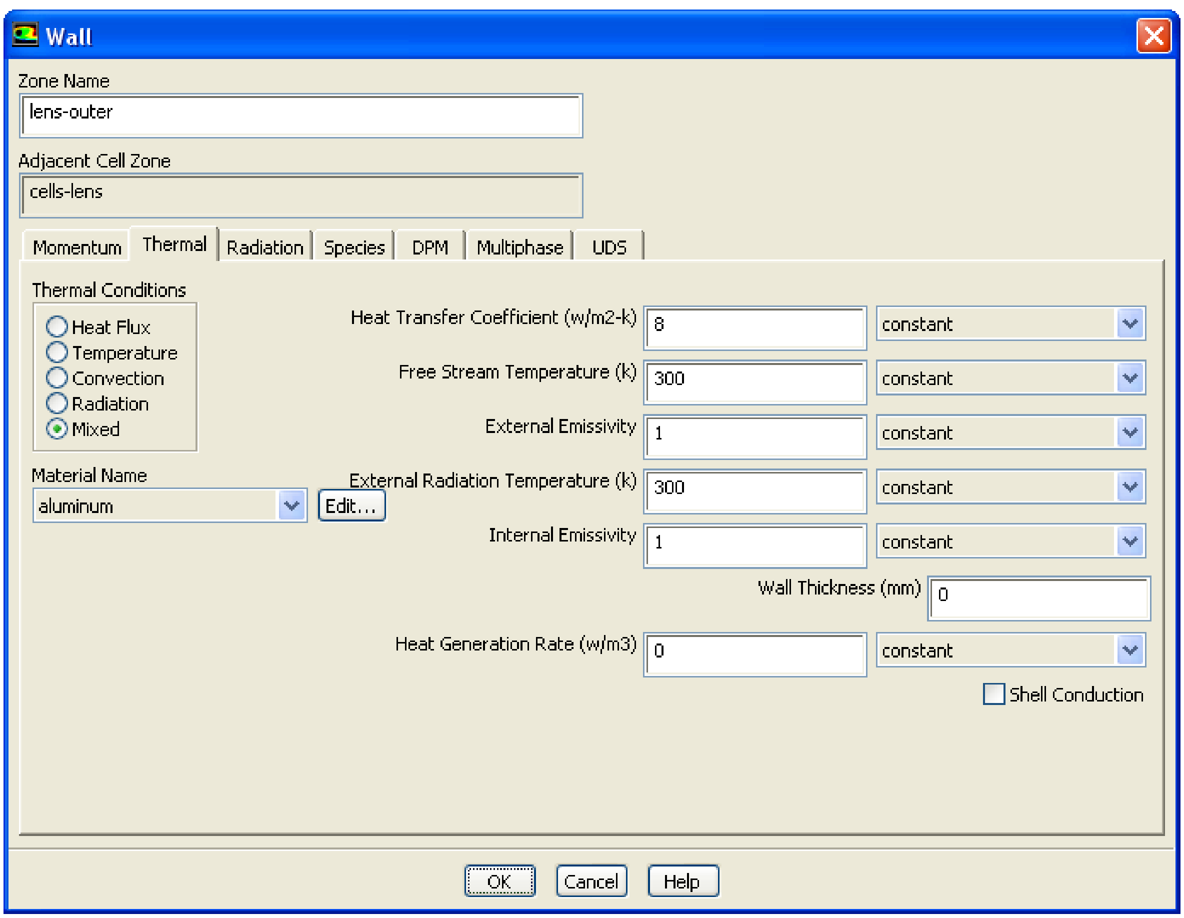
ii. Click OK to close the Wall dialog box.

2. For lens-inner-shadow enter `0.5` for Diffuse Fraction.

Boundary Conditions ⟶ lens-inner-shadow ⟶ Edit...

3. Specify the outer surface of the lens as semi-transparent.

Boundary Conditions ⟶ lens-outer ⟶ Edit...




(a) Click the Thermal tab.

i. Select Mixed in the Thermal Conditions group box.

ii. Enter 8 W/m$^2$-K for Heat Transfer Coefficient.

*The outer surface of the lens, which is not modeled, is exposed to ambient air. Assuming that the car is in stationary position, the cooling mechanism on the outer surface of the lens is therefore by natural convection and radiation. The convective heat transfer to the outside can be approximated using flat plate Nusselt number correlations. Based on average known temperature, the convective heat transfer is determined to be 8 W/m$^2$K.*

iii. Retain the default values for other parameters.

*Since the incoming radiation ($Q_{in}$, rad) is diffuse, you can specify* Mixed *at semi-transparent boundary wall, and then provide* External Emissivity*=1 ($E_e$=1) and* External Radiation Temperature *($T_e$). You will also specify convection conditions such as heat transfer coefficients and reference temperature for convection. If the wall is semi-transparent,* ANSYS FLUENT *will use the following relation for the incoming radiation:*

$$Q_{in}, rad = E_e \times Stefan{-}Bolzmann\_Constant \times T_e^4$$

*The outgoing radiation is calculated from inside. But if it is a diffuse wall (not semi-transparent), where the outgoing radiation is not calculated, then* ANSYS FLUENT *uses the following relation:*

$$Q_{net} = E_e \times Boltzmann\_Constant \times (T_{wall}^4 \text{-} T_e^{4)}$$

*to calculate net radiation exchange to outside.*

*If the incoming radiation is directional, say a very close-by heatsource, then you need not use* Mixed*, but can specify the beam direction, beam width, and irradiation.*

iv. Retain the default values for the other parameters.

**Note:** *Extreme temperatures are experienced in the headlamp when the car is stationary.*

(b) Click the Radiation tab and select semi-transparent from the BC Type drop-down list.

(c) Click OK to close the Wall dialog box.

4. Specify the outer surface of the bulb as semi-transparent.

*Semi-transparent BC type applied on the surface allows the radiation to be transmitted through this surface. It also accounts for reflection. Internal emissivity will not be used at semi-transparent walls, instead, emission and absorption is now a volumetric phenomenon that is accounted for using the absorption coefficient of the glass.*

Boundary Conditions ⟶ bulb-outer ⟶ Edit...

*This is an internal wall with cells on both sides so there is a corresponding shadow zone. The shadow faces the* cells-bulb *zone.*

(a) Click the Radiation tab.

(b) Select semi-transparent from the BC Type drop-down list.

(c) Enter 0.05 for Diffuse Fraction.

*Since most bulbs have a fine surface finish, and assuming that this bulb has a clean surface, you will enter a value of* 0.05*.*

(d) Click OK to close the Wall dialog box.

5. Enter 0.05 for Diffuse Fraction under Radiation tab for bulb-outer-shawdow.

Boundary Conditions ⟶ bulb-outer-shawdow ⟶ Edit...

6. Specify the inner surface of the bulb as semi-transparent.

*This is an internal wall, with cells on both sides, so there is a corresponding shadow zone. The shadow faces the* cells-bulb *zone.*

Boundary Conditions ⟶ bulb-inner ⟶ Edit...

(a) Click the Radiation tab.

(b) Select semi-transparent from the BC Type drop-down list.

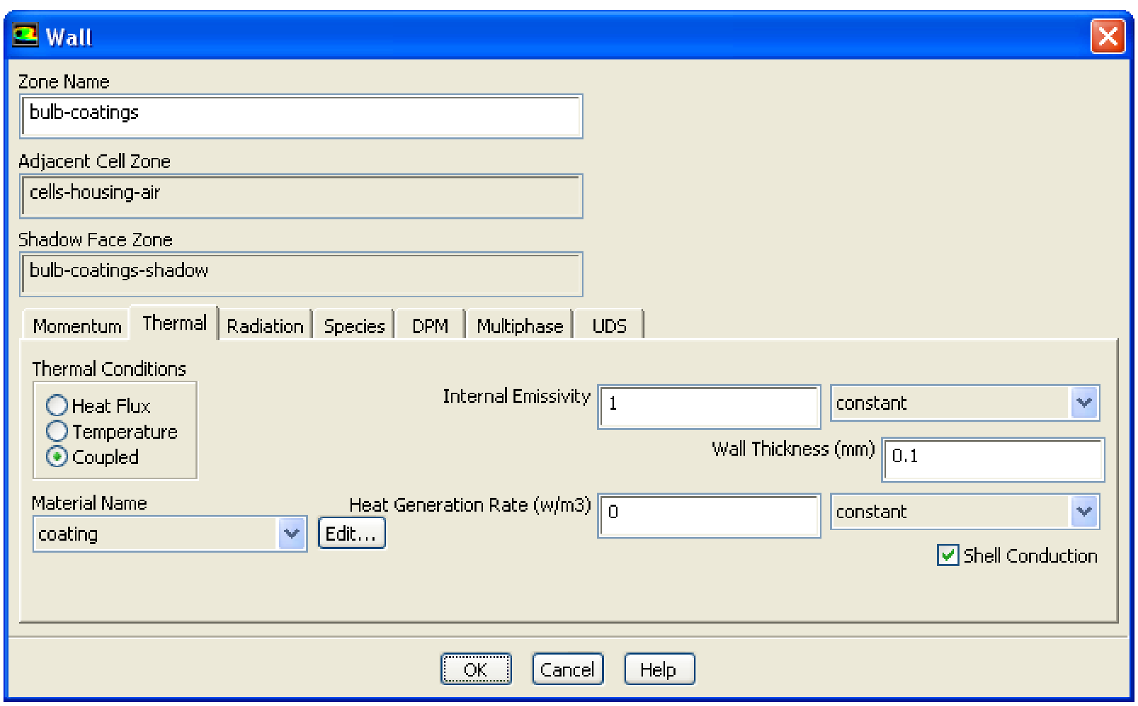
(c) Enter 0.05 for Diffuse Fraction.

7. Enter 0.05 for Diffuse Fraction under Radiation tab for bulb-inner-shawdow.

Boundary Conditions ⟶ bulb-inner-shawdow ⟶ Edit...

8. Specify the boundary condition for the coating on the outer surface of the bulb.

Boundary Conditions ⟶ bulb-coatings ⟶ Edit...



**Note:** *Planar conduction will be used to model conduction along the planar direction. Black coating (usually ceramic) is used on the bulb tip to shield the reflector and lens components from the high intensity radiation arising from the filament.*

(a) Click Thermal tab.

i. Select coating from the Material Name drop-down list.

ii. Enter 0.1 mm for Wall Thickness.

*The coating on the outer surface of the bulb has a thickness of about* 0.1 *mm.*

iii. Enable Shell Conduction.

iv. Retain the default values for the other parameters and click OK to close the Wall dialog box.

*???A message showing that new conduction zones have been created will be displayed in the* ANSYS FLUENT *console.*

v. Type the following scheme command at the TUI:

```
(rpsetvar 'temperature/shell-secondary-gradient? #f)
```

*To achieve robustness, this command ignores the secondary gradient for highly skewed shell conduction cells.*

9. Specify the boundary condition for the outer surface of the reflector.

Boundary Conditions ⟶ reflector-outer ⟶ Edit...

(a) Select Mixed from the Thermal Conditions group box.

*The fluid over the outer cylindrical wall of the housing is not modeled. But the outer surface is cooled by natural convection as well. Also, there is a radiation exchange between the outer surface and the ambient.*

(b) Enter a value of 7 W/m$^2$-K for Heat Transfer Coefficient (h).

(c) Retain the default value of 300 K for Free Stream Temperature (T$_{\mathrm{ref}}$).

ANSYS FLUENT *uses Newton's law of cooling,* $q = h(T_s - T_{ref})$*, to determine heat loss due to convection.* $h$ *may be determined using Churchill and Chu's correlation for natural convection over a cylinder.*

(d) Enter a value of 0.95 for External Emissivity (e).

(e) Retain the default value of 300 K for External Radiation Temperature (T$_e$).

ANSYS FLUENT *uses* $q = Boltzmann\_Constant \times e \times (T_s^4 - T_e^4)$ *to determine the net radiation exchange to the ambient.* $T_s$ *is the calculated temperature at the outer surface of the reflector.*

(f) Retain the default values for the other parameters and click OK to close the Wall dialog box.

*The outer surface of the reflector has a black coating and the inner surface is coated with a highly reflective material. Inspite of that, radiation exchange between the inner and outer surface takes place.*

10. Specify the boundary condition for the inner surface of the reflector.

   Boundary Conditions ⟶ reflector-inner ⟶ Edit...

   (a) Enter 0.2 for Internal Emissivity in the Thermal tab.

   (b) Enter a value of 0.3 for Diffuse Fraction in the Radiation tab.

   *If the diffuse fraction is* 0*, all the incoming radiation is reflected specularly (like a clean mirror), where the incident angle is equal to the reflected angle. In reality, the reflectors are not 100% reflectors and are dusty. The reflected portion of the incoming radiation is given by the following equation:*

   $$Q_{\text{reflected}} = (1 - df) \times Q_{\text{incoming}} + df \times (1 - e) \times Q_{\text{incoming}}$$

   *where* df *is diffuse fraction, and* e *is internal emissivity.*

   *The first term on the right hand side is the part reflected specularly and the second term is the part reflected diffusely. The portion of the incoming radiation that is absorbed is* $e \times df \times Q_{\text{incoming}}$ *and the portion emitted is* $df \times e \times n^2 \times \sigma \times T^4$*, where* n *is the refractive index of the fluid. As seen in the above equations, there will be some absorption if emissivity* e*, defined in the* Thermal *tab, is not zero and if the* Diffuse Fraction *is greater than zero.*

   (c) Click OK to close the Wall dialog box.

11. Specify the boundary condition for the reflector-inner-shadow surface.

   Boundary Conditions ⟶ reflector-inner-shawdow ⟶ Edit...

   (a) Enter 0.2 for Internal Emissivity in the Thermal tab.

   (b) Enter a value of 0.3 for Diffuse Fraction in the Radiation tab.

   (c) Click OK to close the Wall dialog box.

item Specify the heat flux at the filament.

Boundary Conditions ⟶ filament ⟶ Edit...

*The electric power dissipated as heat from the filament is 40 W. The area of the filament is* $6.9413e^{-6}$ $m^2$*. So the heat flux is* $40/6.9413e^{-6}$*, which is equal to* 5760000 $W/m^2$*.*

   (a) Enter 5760000 $W/m^2$ for Heat Flux in the Thermal tab.

   (b) Retain the default values for the other parameters and click OK to close the Wall dialog box.

### Step 6: Solution

*The solution process will be performed in a series of steps. First, the energy and the radiation equation will be decoupled from the flow. Then the energy and radiation equation will be solved without the flow equation. When the temperatures on the components develop sufficiently, the energy and the flow equation will be solved.*

*The flow and the energy will be converged. Then, the energy and the radiation equation will be iterated to convergence. This process will be repeated until there is no significant change in the solution monitors or the residuals.*

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

   Solution Methods

2. Set the solution parameters.

   Solution Controls

   (a) Enter the following values for the Under-Relaxation Factors.

   | | |
   |---|---|
   | Pressure | 0.3 |
   | Density | 0.8 |
   | Body Forces | 0.8 |
   | Momentum | 0.6 |
   | Energy | 0.8 |
   | Discrete Ordinates | 0.8 |

   (b) Click Equations, deselect Flow and retain the selection of Energy and Discrete Ordinates from the Equations selection list.

   **Note:** *Due to the crude initial guess, starting the calculation with reduced under-relaxation factors for* Energy *and* DO *keeps the residuals stable.*

3. Display surfaces of symmetry type.

   Graphics and Animations ⟶ Mesh ⟶ Set Up...

   (a) Deselect all the surfaces from the Surfaces selection list.

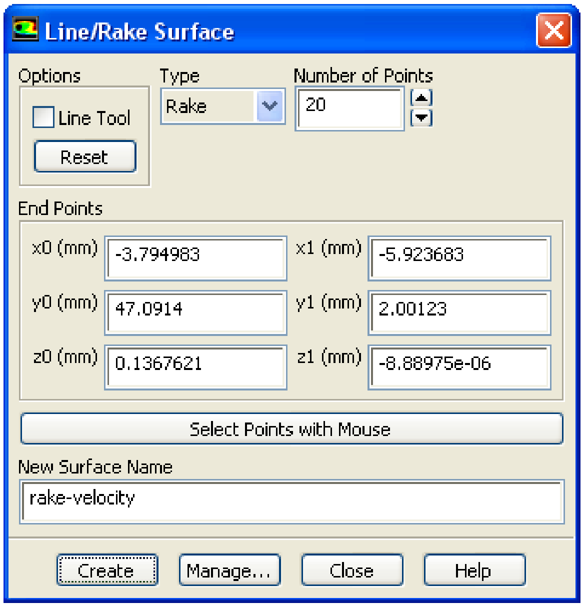
   (b) Select Outline in the Edge Type group box.

   (c) Select symmetry from the Surface Types selection list.

   (d) Click Display.

4. Create a rake to monitor velocity on a symmetry plane (Figure 4).

   Surface ⟶ Line/Rake...




(a) Select Rake from the Type drop-down list.

(b) Increase the Number of Points to 20.

(c) Click the Select Points with Mouse button.

*The* Working *dialog box will be displayed asking you to click on two locations in the graphics window with the mouse button.*

(d) Right-click on the graphics window to define two end points of the rake as shown in the Figure 3.

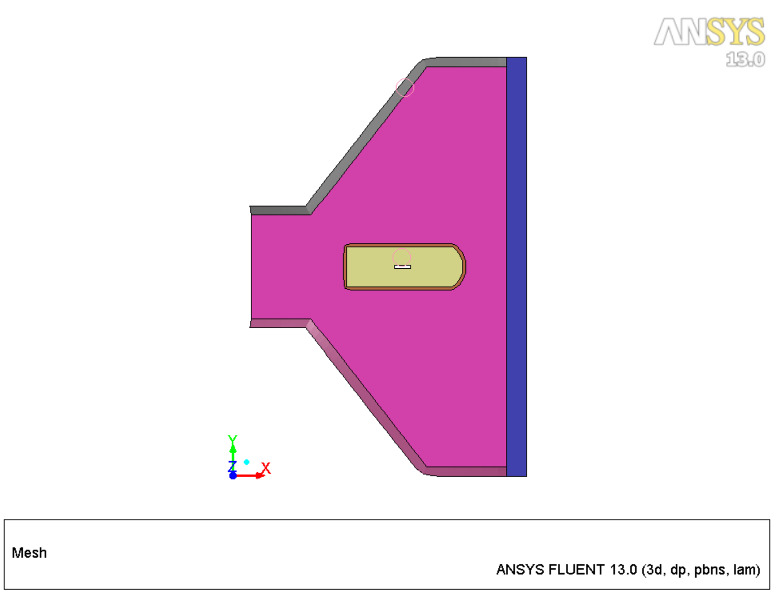



Figure 3: Points for Rake Creation

(e) Enter rake-velocity for New Surface Name.

(f) Click Create and close the dialog box.

5. Display the rake (Figure 4).

   Graphics and Animations ⟶ Mesh ⟶ Set Up...

   (a) Retain the previous settings and select rake-velocity from the Surfaces selection list.

   (b) Click Display.

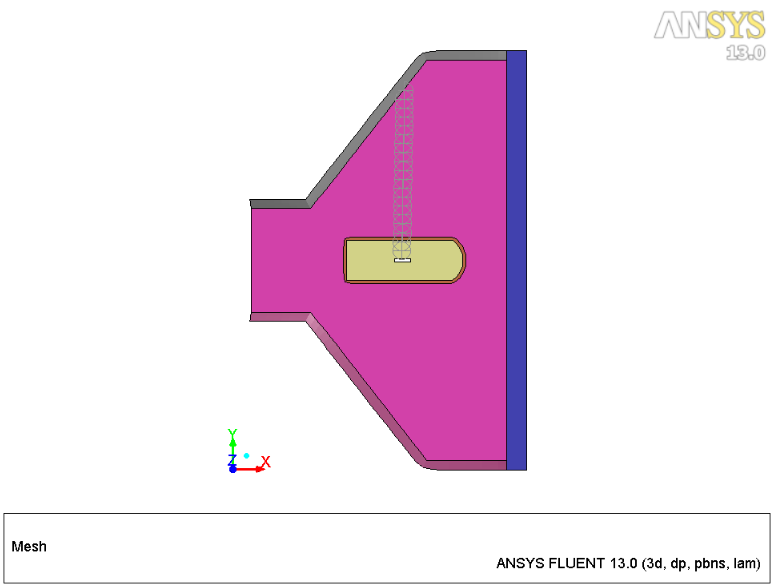



Figure 4: Mesh with Rake

6. Enable the plotting of velocity at rake-velocity during the solution process.

   Monitors (Surface Monitors) ⟶ Create...

   (a) Enable Plot, Print, and Write for the monitors.

   (b) Enter `head-lamp-v.out` for File Name.

   **Note:** *If you are using a UNIX/LINUX machine, click on the letter R on the keyboard, to quickly select the zone name starting with the letter r. But first, you need to select one of the zones for it to work. Then, click the second time to select the second zone starting with the same letter r.*

   (c) Select Area-Weighted Average from the Report Type drop-down list.

   (d) Select Velocity... and Velocity Magnitude from the Report of drop-down lists.

   (e) Select rake-velocity from the Surfaces selection list.

   (f) Click OK to close the Define Surface Monitor dialog box.

7. Enable the plotting of maximum temperature at reflector-inner.

   Monitors (Surface Monitors) ⟶ Create...

(a) Enable Plot, Print, and Write for the monitors.

(b) Enter `head-lamp-t.out` for File Name.

(c) Select Temperature... and Static Temperature from the Report of drop-down lists.

(d) Select Facet Maximum from the Report Type drop-down list.

(e) Select reflector-inner from the Surfaces selection list.

(f) Click OK to close the Surface Monitor dialog box.

8. Initialize the solution.

   Solution Initialization

   *The values for velocities and pressure are set to zero.*

   (a) Ensure that the value for initial temperature is set to 300 K.

   (b) Click Initialize.

9. Patch cells-bulb-inside with a high temperature (500 K).

   Solution Initialization ⟶ Patch...

   (a) Select Temperature from the Variable list.

   (b) Select cells-bulb-inside from the Zones To Patch list.

   (c) Enter 500 K for Value.

   (d) Click Patch and close the Patch dialog box.

10. Save the case file (`auto-hlamp.cas.gz`).

    File ⟶ Write ⟶ Case...

11. Start the calculation by requesting 20 iterations.

    Run Calculation ⟶ Calculate

12. Adjust solution parameters.

    Solution Controls

    (a) Increase the under-relaxation factors for Energy and Discrete Ordinates to 1.

    *Since the solution is now stabilized, you can increase the under-relaxation factors for* Energy *and* Discrete Ordinates *to* `1.0`*. This will speed-up the iteration process significantly for the* Discrete Ordinates *equation.*

13. Continue the calculation of Energy and Discrete Ordinates by requesting another 500 iterations (Figures 5, 6, and 7).

    Run Calculation ⟶ Calculate

    *In some cases, with an under-relaxation factor of* `1.0` *for* Energy *and* Discrete Ordinates*, the residuals may become "flat" after about 80 to 100 iterations. Reducing them back to* `0.8` *will force it to converge further.*

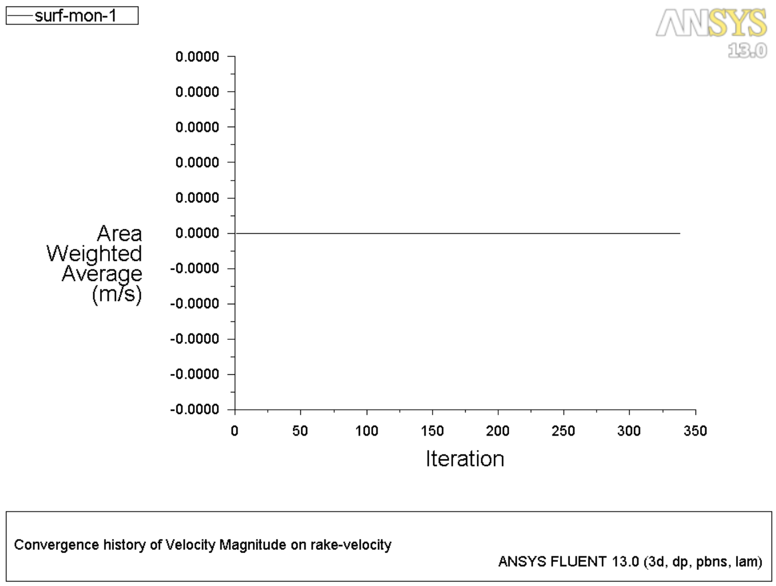



Figure 5: Convergence History of Velocity Magnitude on rake-velocity

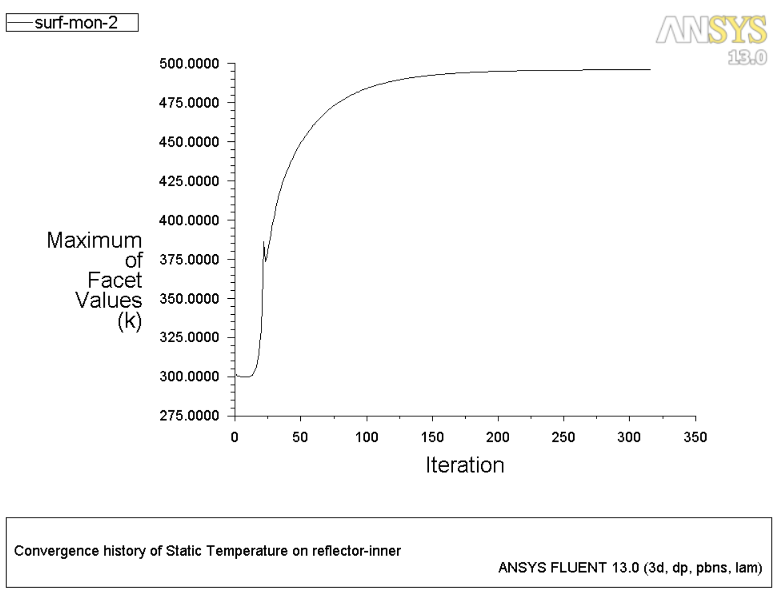



Figure 6: Convergence History of Static Temperature on reflector-inner

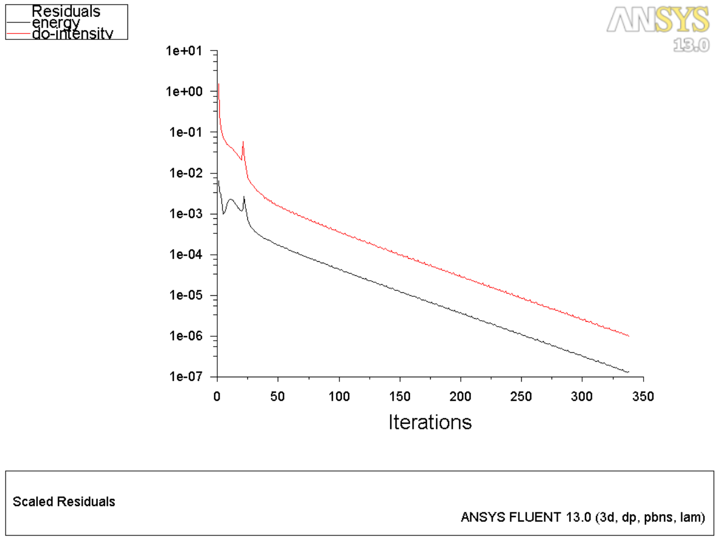



Figure 7: Scaled Residuals

14. Calculate flow and energy equation.

    Solution Controls ⟶ Equations...

    (a) Deselect Discrete Ordinates, and select Flow and Energy from the Equations selection list.

    (b) Click OK to close the Equations dialog box.

15. Continue the calculation by requesting another 1000 iterations (Figures 8, 9, and 10).

    Run Calculation ⟶ Calculate

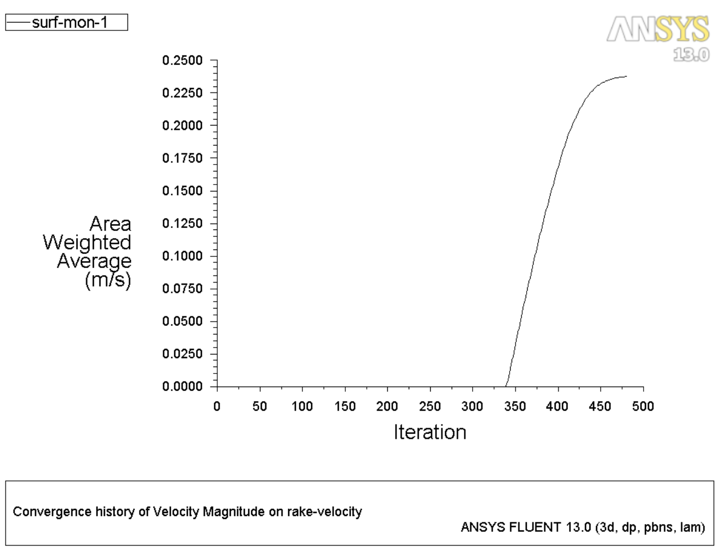



Figure 8: Convergence History of Velocity Magnitude on rake velocity

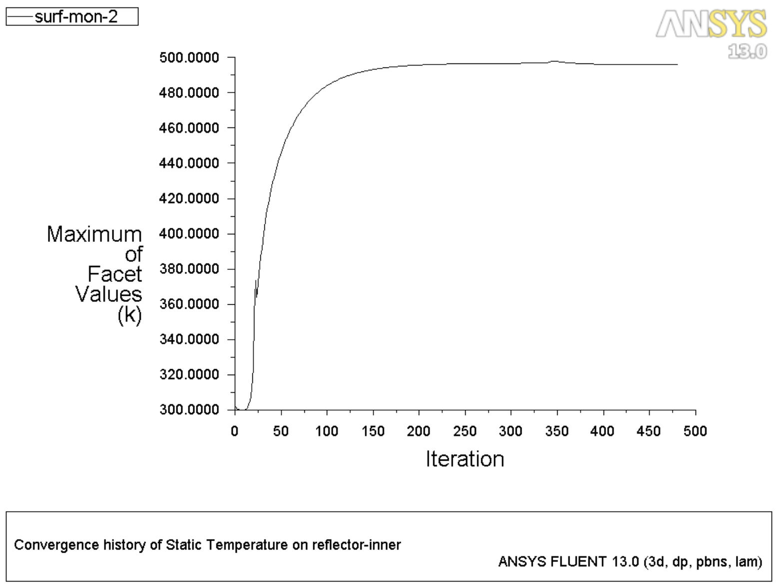



Figure 9: Convergence history of Static Temperature on reflector-inner

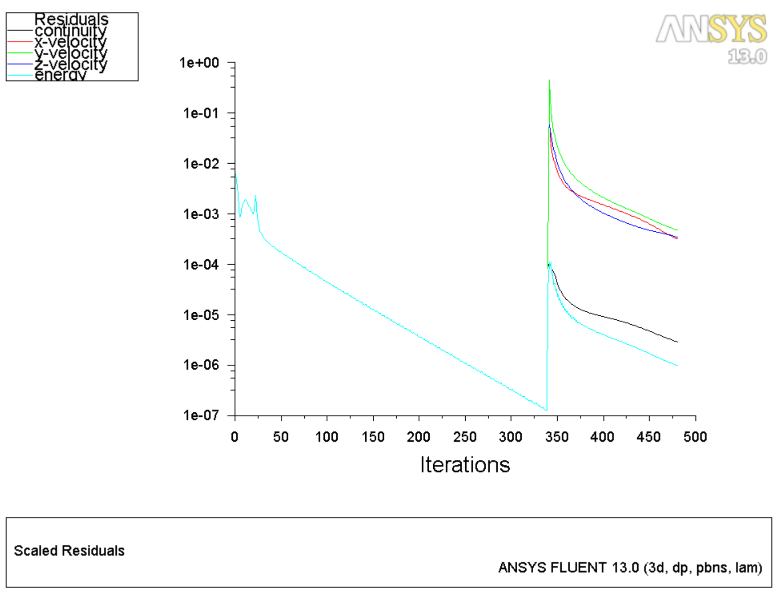



Figure 10: Scaled Residuals

16. Solve for all the flow equations, DO and energy.

    Solution Controls ⟶ Equations...

    (a) Select all from the Equations selection list.

    (b) Ensure that the under-relaxation factor for Energy and Discrete Ordinates is 1.

    (c) Click OK to close the Equations dialog box.

17. Continue the calculation by requesting another 500 iterations (Figures 11, 12, and 13.

    *You can stop the calculations once you find that the monitors are stable,*

    Run Calculation ⟶ Calculate

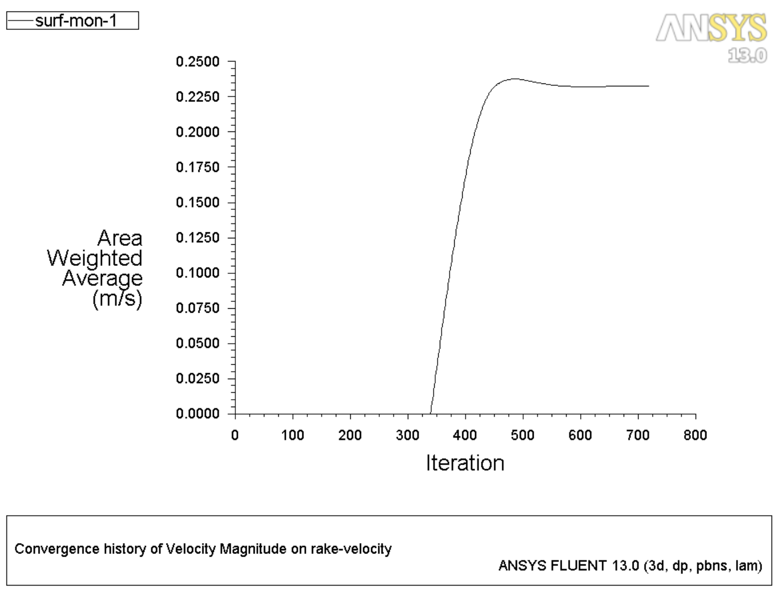



Figure 11: Convergence History of Velocity Magnitude on rake velocity

18. Save the case and data file (auto-hlamp.dat.gz).

    File ⟶ Write ⟶ Data....

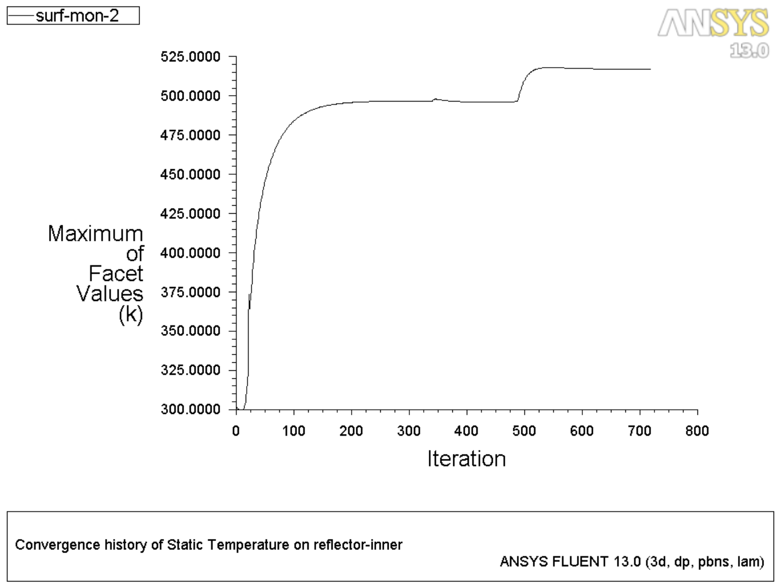



Figure 12: Convergence History of Static Temperature on reflector-inner

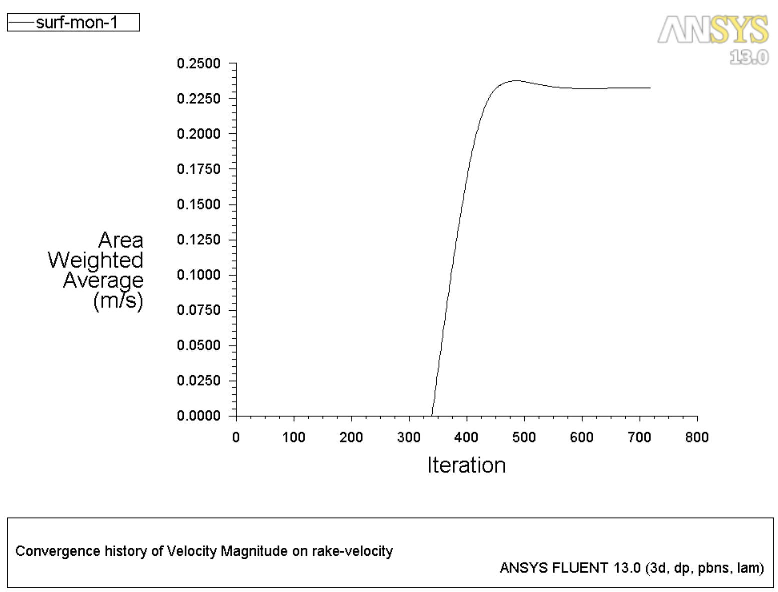



Figure 13: Scaled Residuals

### Step 7: Postprocessing

1. Display velocity vectors.

   Graphics and Animations ⟶ Vectors ⟶ Set Up...

   (a) Deselect all the surfaces from the Surfaces selection list.

   (b) Select symmetry from the Surface Types selection list.

   (c) Click Display (Figure 14).

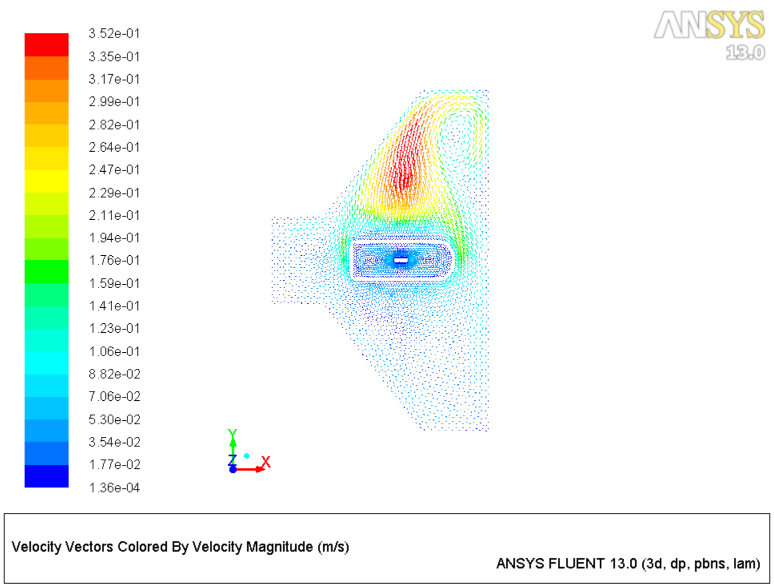



Figure 14: Velocity Vectors

2. Display contours of static temperature.

   Graphics and Animations ⟶ Contours ⟶ Set Up...

   (a) Disable Global Range from the Options group box.

   (b) Select Temperature... and Static Temperature from the Contours of drop-down lists.

   (c) Select housing-inner, lens-inner, and socket-inner from the Surfaces selection list.

   (d) Click Display.

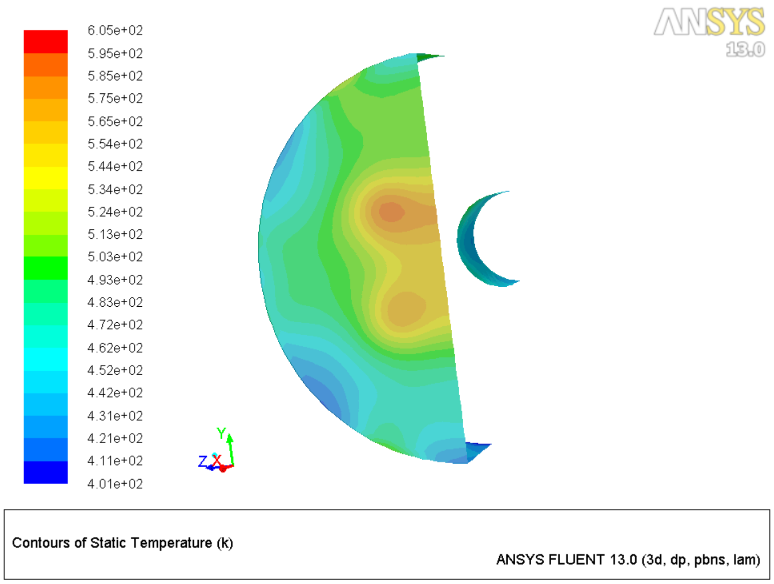



Figure 15: Contours of Static Temperature

*The contours of static temperature is shown in Figure 15. You can see two hot spots at the housing in the contour plot.*

3. Display contours of incident radiation.

   Graphics and Animations ⟶ Contours ⟶ Set Up...

   (a) Disable Global Range from the Options group box.

   (b) Select Wall Fluxes... and Surface Incident Radiation from the Contours of drop-down lists.

   (c) Select housing-inner, lens-inner, and socket-inner from the Surfaces selection list.

   (d) Click Display.

   **Note:** *The contours of surface incident radiation is shown in Figure 16. You can see two hot spots at the housing in the contour plot. The radiation in this sample tutorial is highly localized—it is coming from a small source (a high temperature and a relatively small filament). These hot spots can be removed if a higher number of theta and phi divisions are used.*

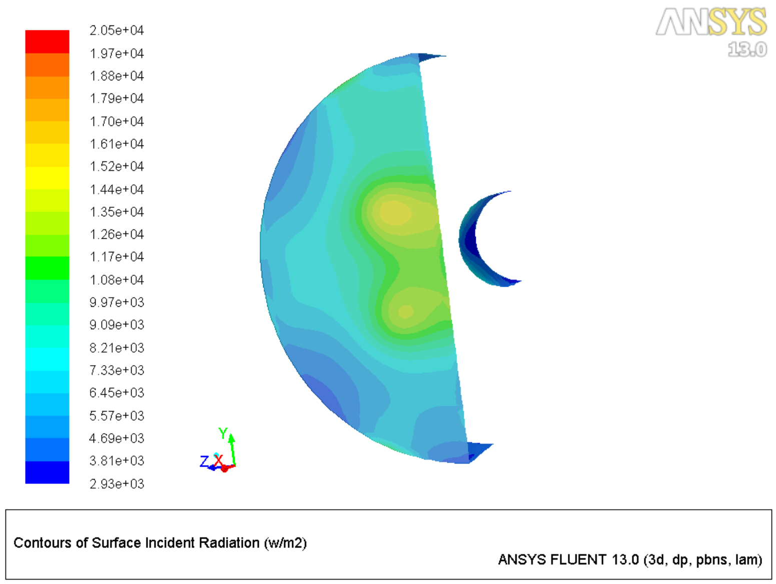



Figure 16: Contours of Incident Radiation

*Similarly, you can also display the contour plots of reflected, absorbed, and transmitted portion of the incident radiation. This can be done only on the semi-transparent walls.*

## Summary :

1. In this tutorial, the spectral distribution of absorption coefficient of the glass and the lens is not considered, and is assumed to be constant (gray model). A wave-length-weighted average of absorption coefficient is used:

$$a = sum(a \times wave\_length)/sum(wave\_length)$$

**Note:** *If more accurate results are required, then the non-gray* DO *model of* ANSYS FLUENT *can be used.*

2. When the mesh is coarse, and there is a high temperature gradient near the filament, it is possible that the energy equation may have difficulty in convergence. One solution would be to make the bulb region finer.

3. In cases with localized heat sources, if more accurate results are required, higher angular discretization should be used. In this example, the radiation is highly localized at the filament which may require up to 4×4 division. An angular discretization of 2×2 is used in this tutorial. A sensitivity study of angular discretization can also be performed. You may start with 2×2, then continue with 4×4, and so forth until there is no considerable change in maximum temperature on the major components.

4. Condensation is an issue in automotive headlamps. It can be modeled using user defined functions (UDF).