

---

## Tutorial 3. Flow and Heat Transfer over a Flat Plate

---

### Introduction

The purpose of this tutorial is to illustrate the setup and solution of the 2D laminar fluid flow over a flat plate. This type of flow is of interest to study the convective heat transfer and the development of thermal boundary layer. Heat transfer is highly dependent on the Reynolds number (based on the plate length). Also observe how the Nusselt number varies with Reynolds number.

This tutorial demonstrates how to do the following:

- Read an existing mesh file in FLUENT.
- Check the grid for dimensions and quality.
- Study convective heat transfer.
- Specify solver settings and perform iterations.
- Create isosurfaces and points.
- Perform postprocessing.
- Compare the Nusselt number calculations with the literature.

### Prerequisites

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

### Problem Description

In this tutorial, we consider a flat plate with a length  $L = 1$  m (Figure 3.1). The flow of air is at a velocity of 1.4607 m/sec, such that the Reynolds number based on plate length is  $1e+05$ . The plate is maintained at a temperature of 400 K.

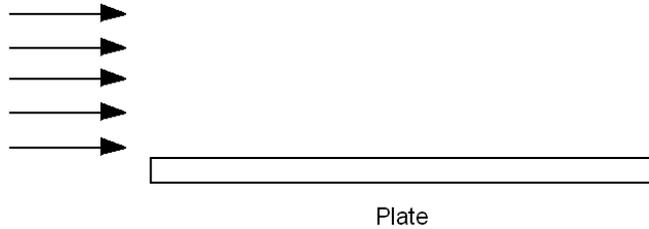


Figure 3.1: Problem Schematic

### Preparation

1. Copy the following files to your working folder:
  - plate.msh
  - point-x.jou
  - iso-x.jou
2. Start the 2D double precision (2ddp) solver of FLUENT.

### Setup and Solution

#### Step 1: Grid

1. Read the grid file plate.msh.

→  → Case...

FLUENT *will read the mesh file and report the progress in the console.*

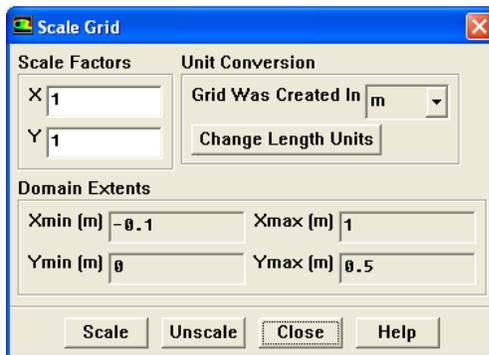
2. Check the grid.

→ Check

*This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.*

3. Check the scale of the grid.

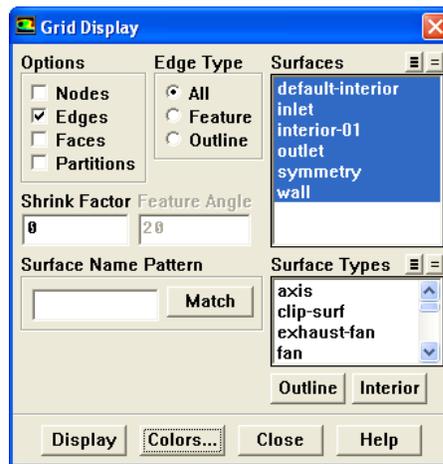
Grid → Scale...



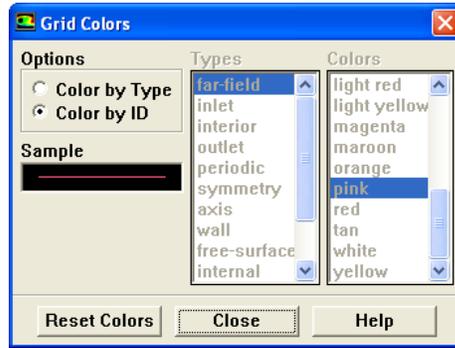
*Check the domain extents to see if they correspond to the actual physical dimensions. If not, the grid has to be scaled with proper units.*

4. Display the grid (Figure 3.2).

Display → Grid...



(a) Click the Colors... button to open the Grid Colors panel.



- i. Select Color by ID in the Options group box.
- ii. Close the Grid Colors panel.

(b) Click Display and close the Grid Display panel.

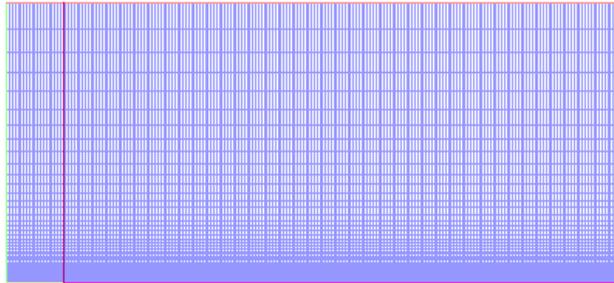


Figure 3.2: Grid Display

*The grid adjacent to the walls is finer compared to that in the central region. The purpose of such fine mesh is to capture sharp gradients near the walls correctly. To avoid a sudden start to the boundary layer, a symmetry boundary of length  $0.1 \times L$  is included before the plate.*

## Step 2: Models

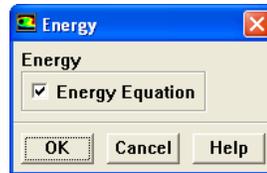
1. Retain the default solver settings.

Define → Models → Solver...

*The problem is to be solved in steady state with 2D laminar conditions.*

2. Enable Energy Equation.

Define → Models → Energy...



## Step 3: Materials

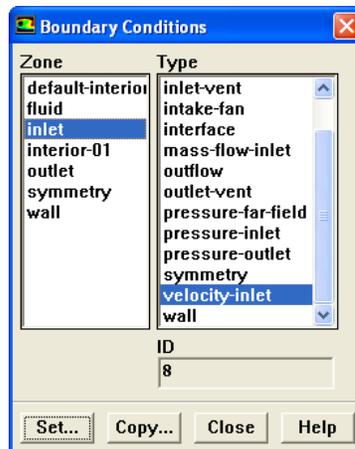
Define → Materials...

*Retain the default properties of air.*

## Step 4: Boundary Conditions

Define → Boundary Conditions...

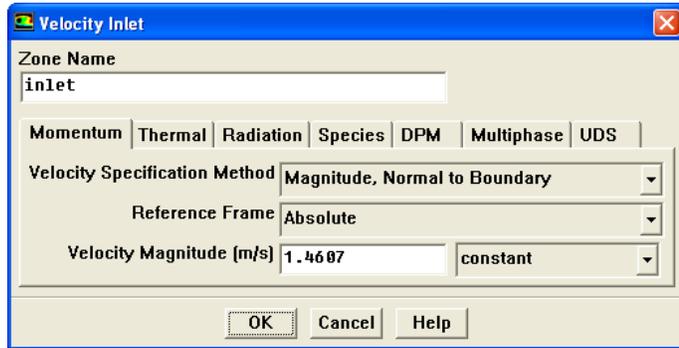
1. Set the boundary condition for velocity inlet (inlet).



- (a) Select inlet from the Zone selection list.

*This Type will be reported as velocity-inlet.*

- (b) Click the Set... button to open the Velocity Inlet panel.



- (c) Enter 1.4607 m/s for Velocity Magnitude.

*The flow Reynolds number is  $1e+5$  and the velocity can be calculated using the expression:*

$$Re = \left[ \frac{UH\rho}{\mu} \right]$$

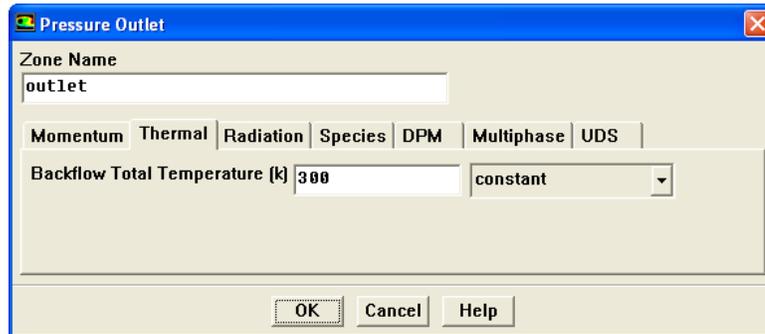
- (d) Click OK to close the Velocity Inlet panel.

2. Retain the default boundary conditions for pressure outlet (outlet).

- (a) Select outlet from the Zone selection list.

*The Type will be reported as pressure-outlet.*

- (b) Click the Set... button to open the Pressure Outlet panel.



- (c) Ensure that Backflow Total Temperature is set to 300 K.

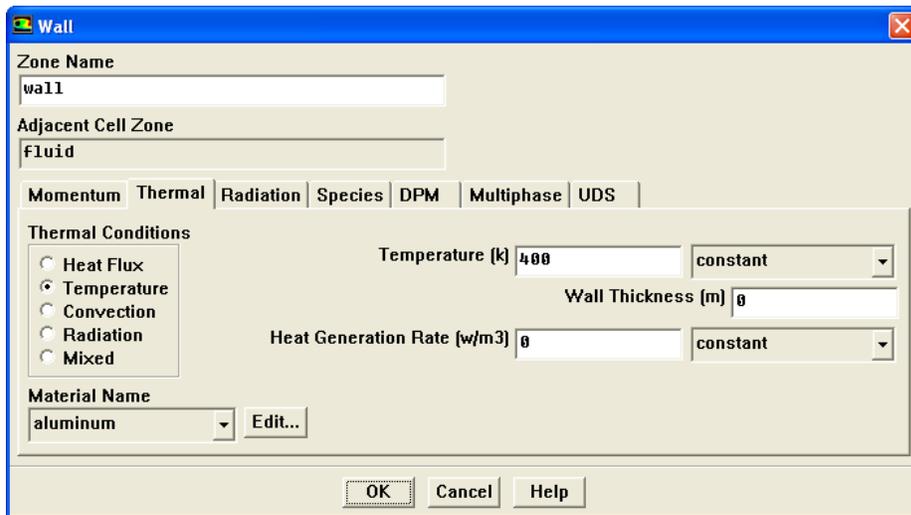
- (d) Click OK to close the Pressure Outlet panel.

3. Set the boundary condition for wall.

- (a) Select wall from the Zone selection list.

*The Type will be reported as wall.*

(b) Click the Set... button to open the Wall panel.

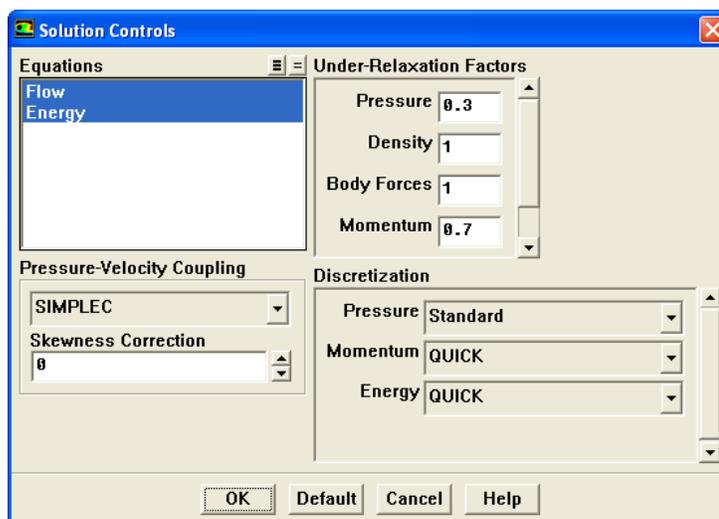


- i. Click the Thermal tab and select Temperature from the Thermal Conditions list.
- ii. Enter 400 K for Temperature.
- iii. Click OK to close the Wall panel.

4. Close the Boundary Conditions panel.

### Step 5: Solution

1. Set the solution controls.



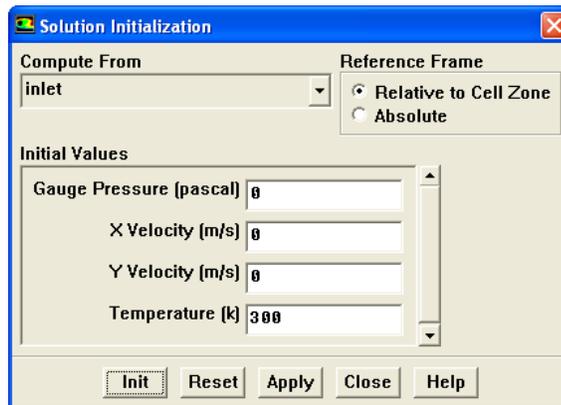
- (a) Select SIMPLEC from the Pressure-Velocity Coupling drop-down list.

*SIMPLEC is a better option for uncomplicated problems, where convergence depends on pressure-velocity coupling. In SIMPLEC, the pressure-correction under-relaxation factor is generally set to 1.0, which helps speed up convergence. Higher order schemes will give more accurate results.*

- (b) Select QUICK from the Momentum and Energy drop-down lists.  
(c) Click OK to close the Solution Controls panel.

item Initialize the flow.

Solve → Initialize → Initialize...



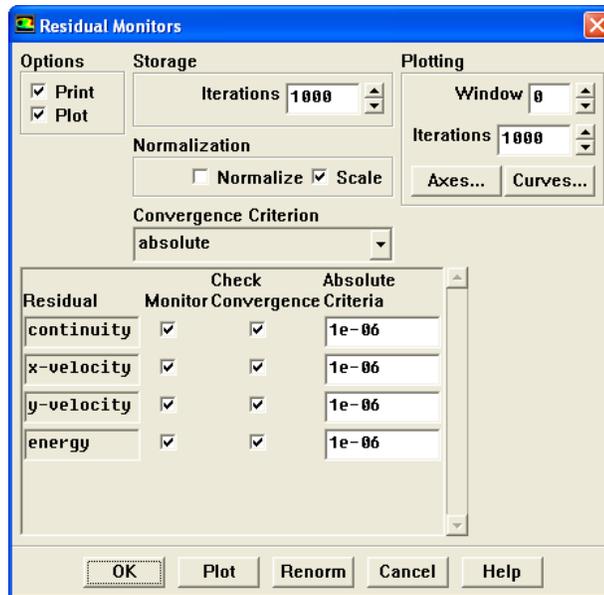
- (a) Select inlet from the Compute From drop-down list.

*It will update values of all the variables based on the boundary conditions at the inlet.*

- (b) Enter 0 for X Velocity.  
(c) Click Init and close the Solution Initialization panel.  
*For faster convergence, set all the velocities to zero.*

2. Enable the plotting of residuals during the calculation.

Solve → Monitors → Residuals...



- (a) Enable Plot in the Options group box.
- (b) Enter 1e-06 for Absolute Criteria for all the equations.

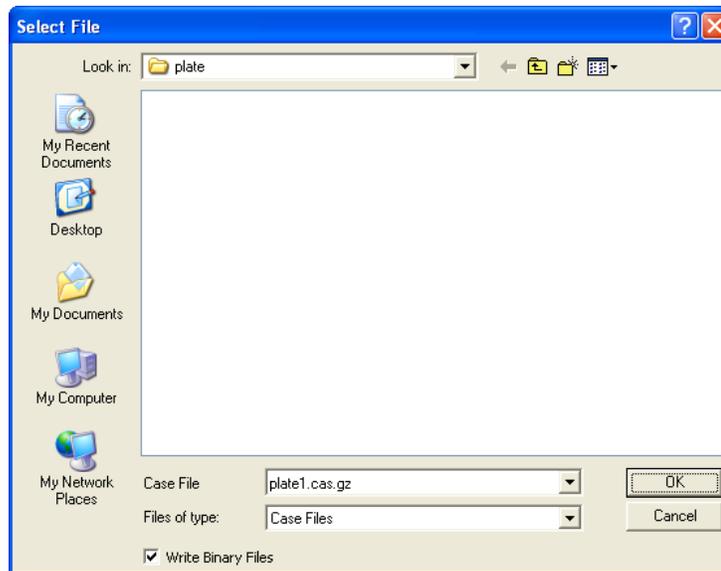
*Higher order discretization schemes and tighter convergence criteria are desirable for accurate resolution of boundary thermal boundary layer.*

- (c) Click OK to close the Residual Monitors panel.

3. Save the case file (plate1.cas.gz).

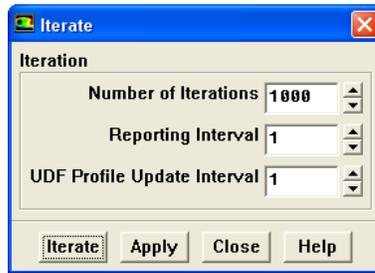
**File** → **Write** → Case...

*Retain the default Write Binary Files option so that you can write a binary file. The .gz extension will save compressed files on both, Windows and LINUX/UNIX platforms.*



4. Start the calculation by requesting 1000 iterations.

Solve → Iterate...



(a) Set Number of Iterations to 1000.

(b) Click Iterate.

*The solution converges in about 380 iterations, with specified convergence criteria. The residuals plot is shown in Figure 3.3.*

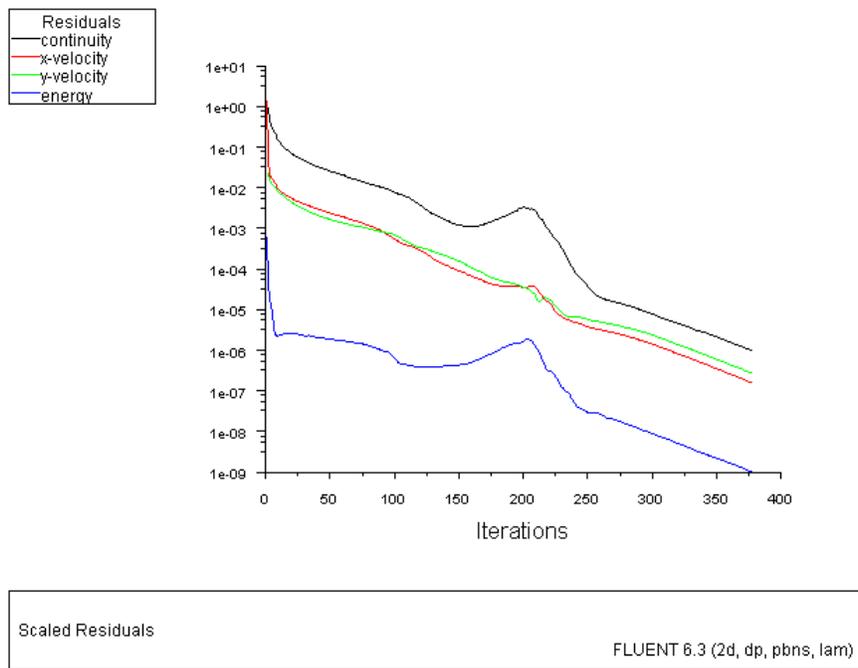


Figure 3.3: Scaled Residuals

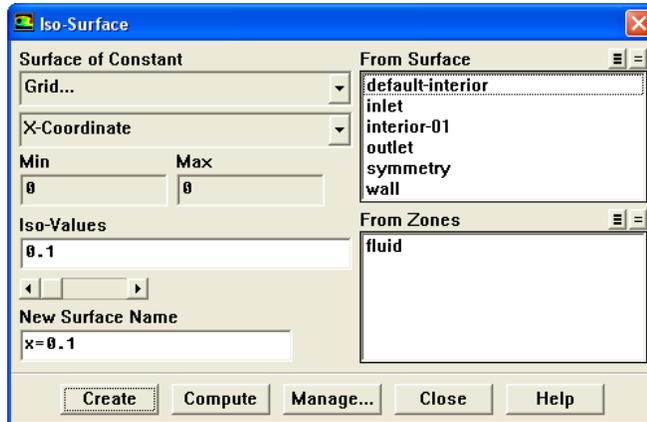
(c) Close the Iterate panel.

5. Create iso-surfaces.

To create the plot of Nusselt number and skin friction coefficient along the length of plate, you will have to create some isosurfaces along the length of the plate. Create 10 equidistant lines with spacing of 0.1m. Bulk temperature of fluid will be calculated using these isosurfaces.

(a) Create the iso-surface using the Iso-Surface panel.

Surface → Iso-Surface...



- i. Select Grid... and X-Coordinate from the Surface of Constant drop-down lists.
- ii. Enter 0.1 for Iso-Values.
- iii. Enter x=0.1 for New Surface Name.
- iv. Click Create and close the Iso-Surface panel.

(b) Create isosurface using TUI commands.

To create the rest of the iso-surfaces, the process can be automated using the use of Text User Interface (TUI) commands. For almost all the graphical commands, a corresponding TUI command is available on FLUENT's console window.

The TUI command such as

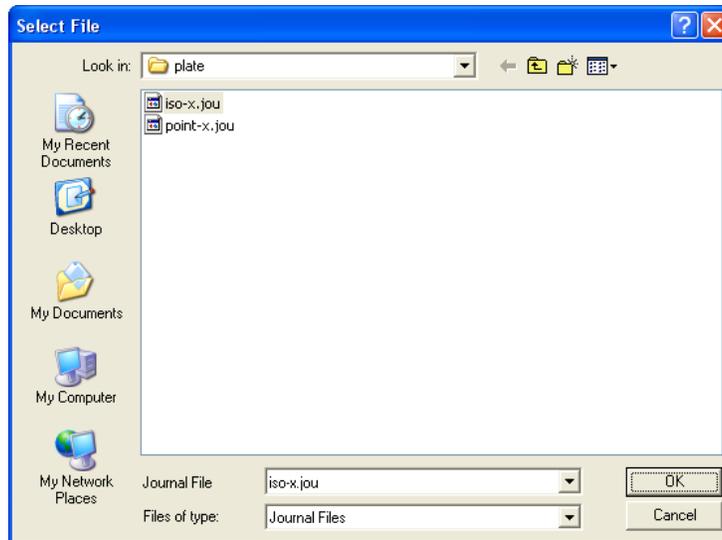
`surface iso-surface x-coordinate x=0.2 , , 0.2 ,`

creates an isosurface at  $x=0.2$ .

A file `iso-x.jou` contains the commands to create all the nine iso-surfaces at different  $x$ -locations.

- i. Read the iso-x.jou file.

File → Read → Journal...



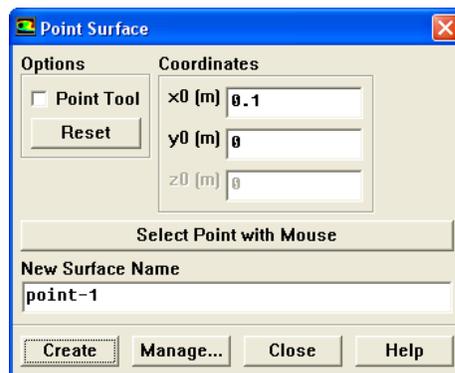
- ii. Select file `iso-x.jou` and click OK.

*All the commands will be displayed in the console and iso-surfaces will be created.*

6. Create points on the plate corresponding to the locations of iso-surfaces.

**Surface** → Point...

*The point surfaces will be used to calculate the heat flux and wall shear stress at different locations. As there are 10 iso-surfaces, you need to create 10 point surfaces.*



- (a) Enter 0.1 m for  $x_0$ .
- (b) Enter 0 m for  $y_0$ .
- (c) Enter `point-1` for New Surface Name.
- (d) Click Create and close the Point Surface panel.

The TUI command such as

```
surface point-surface point-2 0.2 0
```

creates point surface at  $x=0.2$  and  $y=0$ .

Read the file *point-x.jou* containing the commands to create all the nine point surfaces.

- (a) Read the *point-x.jou* file.

File → Read → Journal...

- i. Select file *point-x.jou* and click OK.

*All the commands will be displayed in console and point surfaces will be created.*

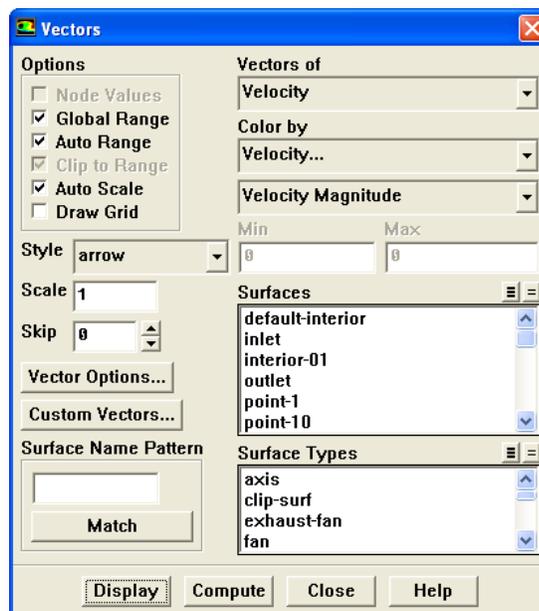
7. Save the case and data files (*plate2.cas.gz* and *plate2.dat.gz*).

File → Write → Case & Data...

## Step 6: Postprocessing

1. Display velocity vectors (Figure 3.4).

Display → Vectors...



- (a) Select Velocity from the Vectors of drop-down list.  
 (b) Select Velocity and Velocity Magnitude from the Color by drop-down lists and click Display.  
 (c) Close the Vectors panel.

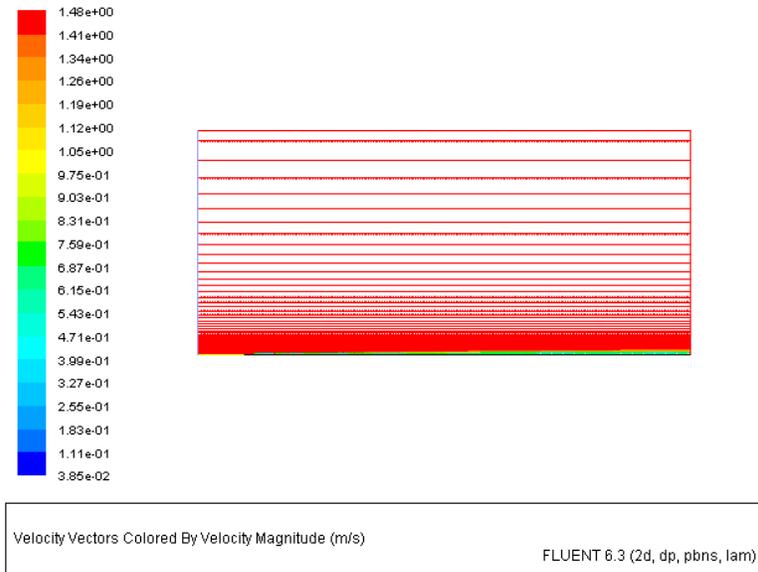
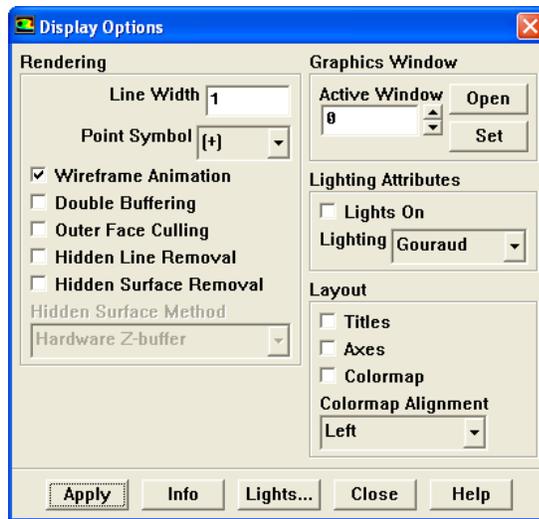


Figure 3.4: Velocity Vectors

2. Disable the captions.

Display → Options...



(a) Disable Titles and Colormap in the Layout group box.

(b) Click Apply and close the Display Options panel.

*Now the captions in the graphics window are no longer visible.*

(c) Zoom the view near plate to view the velocity profile as shown in Figure 3.5.

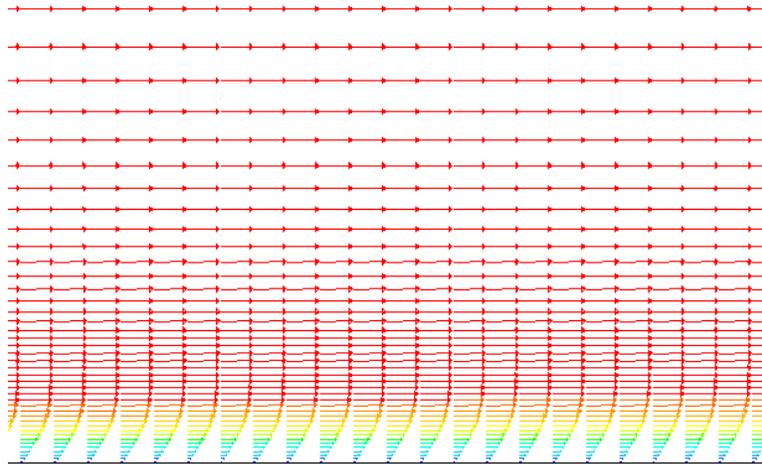
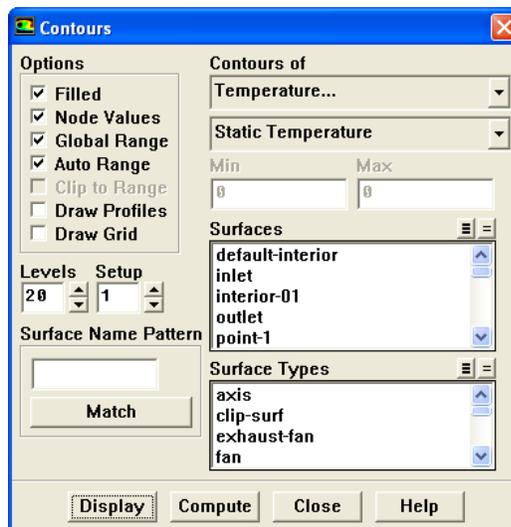


Figure 3.5: Magnified View of Velocity Vectors

3. Display the temperature contours (Figure 3.6).

Display → Contours...



- (a) Select Temperature... and Static Temperature from the Contours of drop-down lists.
- (b) Enable Filled from the Options list.
- (c) Click Display.

*Zoom in to get the complete temperature distribution of temperature along the length of plate (Figure 3.6).*

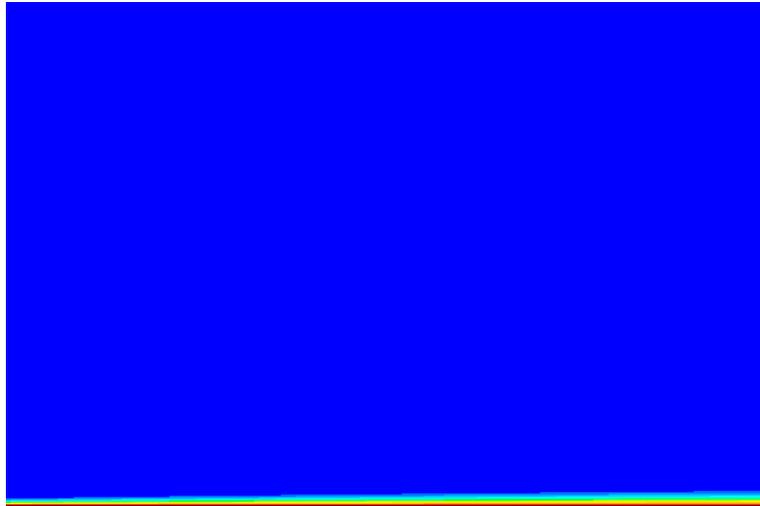


Figure 3.6: Contours of Static Temperature

- (d) Close the Contours panel.
4. Calculate the skin friction coefficients using correlation as follows:

$$C_{f,x} = \frac{0.664}{\sqrt{Re_x}} \quad (3.1)$$

Where,  $Re_x$  (for laminar flows) is calculated based on the location of the point on the plate. The location of **point-1** is 0.1 m.

Therefore, the skin friction based on correlation = 6.64e-3.

Similarly, calculate skin friction coefficient based on the correlation for remaining points.

5. Calculate the skin friction coefficients using FLUENT parameters as follows:

$$C_f = \frac{\tau_w}{1/2\rho v_b^2} \quad (3.2)$$

where,

- $\tau_w$  = Wall shear stress at the point on the plate.
- $\rho$  = Density of fluid.
- $v_b$  = Bulk velocity at the location on the plate.

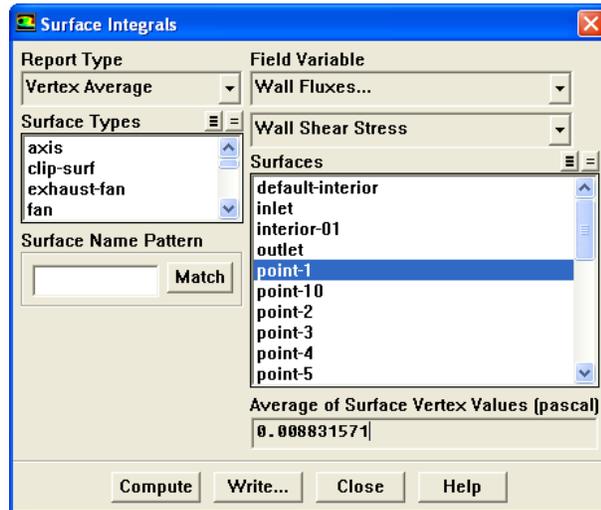
*The values of  $\rho$  and  $v_b$  are constant as set in the Reference Values panel. The skin friction coefficient does not take into account the variation of bulk velocity and*

density at each location. Hence, it is not used as the postprocessing variable in FLUENT.

But in literature, bulk velocity is being evaluated at each location to calculate the skin friction coefficient. Therefore, in this step you will calculate  $\tau_w$  and  $v_b$  at each point location.

6. Obtain the values of wall shear stress ( $\tau_w$ ) and bulk velocity ( $v_b$ ) to calculate skin friction coefficient at point-1.
  - (a) Obtain the value of shear stress at point-1.

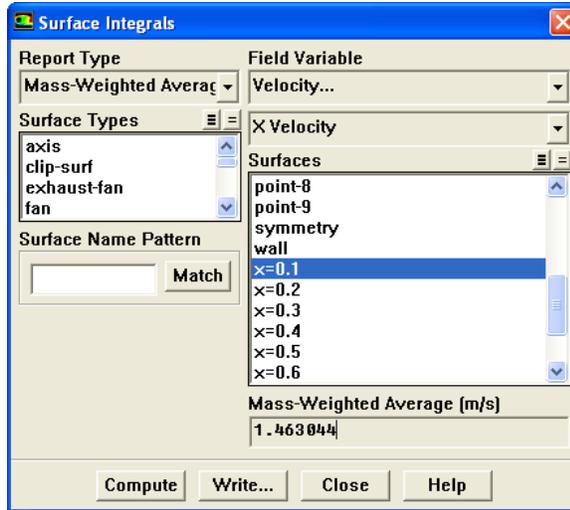
Report → Surface Integrals...



- i. Select Vertex Average from the Report Type drop-down list.
- ii. Select Wall Fluxes... and Wall Shear Stress drop-down lists.
- iii. Select point-1 from the Surfaces selection list.
- iv. Click Compute.

*The value of shear stress will be updated for Average of Surface Vertex Values as 0.00883157 pascal.*

- (b) Obtain the bulk velocity of flow at point-1.



- i. Select Mass-Weighted Average from the Report Type drop-down list.
- ii. Select Velocity... and X Velocity drop-down lists.
- iii. Select x=0.1 from the Surfaces selection list .
- iv. Click Compute and close the Surface Integrals panel.

*The value of bulk velocity will be updated for Mass-Weighted Average as 1.463044 m/s.*

*Density of fluid (air) is 1.225 Kg/m<sup>3</sup>.*

- (c) Using these values of shear stress, bulk velocity, and density of air in Equation 3.2, calculate skin friction coefficient at point-1 (6.736e-3).
- (d) Similarly, calculate skin friction coefficients at remaining locations. See following table for details.

Location of point (m)	Skin friction (FLUENT)	Skin friction (correlation)
0.1	6.736e-3	6.640e-3
0.2	4.774e-3	4.695e-3
0.3	3.899e-3	3.830e-3
0.4	3.375e-3	3.320e-3
0.5	3.016e-3	2.969e-3
0.6	2.750e-3	2.710e-3
0.7	2.540e-3	2.510e-3
0.8	2.372e-3	2.347e-3
0.9	2.228e-3	2.213e-3
1	2.106e-3	2.099e-3

*The values obtained from FLUENT results are close to the values obtained from the correlation. Using this data, generate a plot to compare the skin friction coefficients calculated using correlation and simulation results.*

7. Calculate the Nusselt number using correlation as follows:

$$Nu = 0.332 \times \sqrt{Re} \times (Pr)^3 \quad (3.3)$$

where,  $Re$  is calculated based on the location of the point on the plate.  $Pr$  (Prandtl number) for air is 0.744176. The location of **point-1** is 0.1 m. Nusselt number based on correlation is 30.08.

Similarly, calculate skin friction coefficient based on the correlation for remaining points.

8. Calculate the Nusselt number using FLUENT parameters as follows:

$$Nu = \frac{q_p X_p}{(T - T_b)K} \quad (3.4)$$

where,

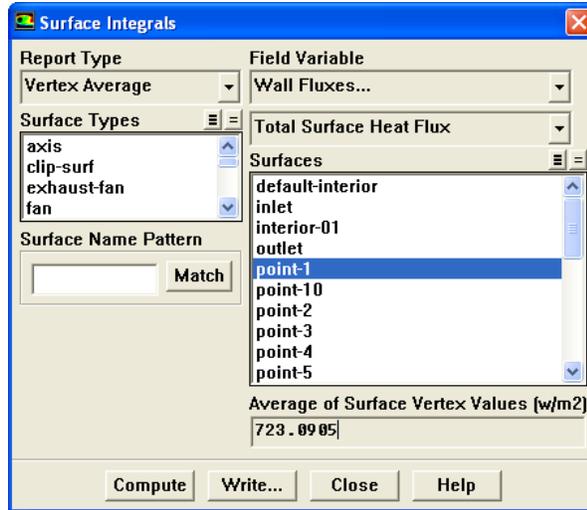
- $q_p$  = Heat flux at point  $p$ .
- $X_p$  = Location of point  $p$ , on the plate.
- $T$  = Temperature of plate (400 K in this case).
- $T_b$  = Bulk temperature of fluid at point  $p$ .
- $K$  = Thermal conductivity of fluid.

*The value of bulk temperature is constant as set in the Reference Values panel. The Nusselt Number does not take into account the variation of bulk temperature at each location. Hence, it is not used as the postprocessing variable in FLUENT.*

*But in literature, heat flux and bulk temperature is being evaluated at each location to calculate the Nusselt number. Therefore, in this step you will calculate  $T_b$  at each point location.*

9. Obtain the values of heat flux ( $q_p$ ) and bulk temperature ( $T_b$ ) to calculate Nusselt number at **point-1**.
- (a) Obtain the value of heat flux at **point-1**.

Report → Surface Integrals...

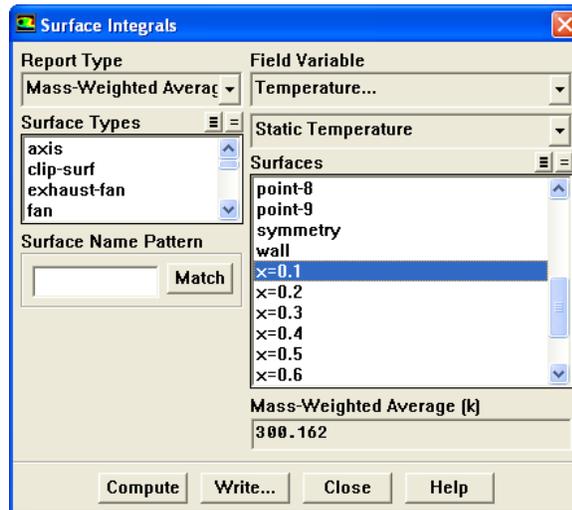


- i. Select Vertex Average from the Report Type drop-down list.
- ii. Select Wall Fluxes... and Total Surface Heat Flux from the Field Variables drop-down lists.
- iii. Select point-1 from the Surfaces selection list.
- iv. Click Compute.

*The value of heat flux will be updated at Area Vertex Average (w/m<sup>2</sup>) as 723.0905.*

*The value of  $X_p$  for point-1 is 0.1 m.*

- (b) Obtain the bulk temperature at point-1.



- i. Select Mass-Weighted Average from the Report Type drop-down list.
- ii. Select Temperature... and Static Temperature from the Field Variable drop-down lists.

- iii. Select  $x=0.1$  from the **Surfaces** selection list.
  - iv. Click **Compute** and close the **Surface Integrals** panel.
- (c) The value of bulk temperature will be updated for **Mass-Weighted Average** as 300.162 K. The value of thermal conductivity for fluid is 0.0242 w/m-k. Using all these values and Equation 3.4, calculate the Nusselt number ( $Nu = 29.928$ ).
- (d) Similarly, calculate Nusselt numbers at remaining locations. See following table for details.

Location of point (m)	Nusselt number (FLUENT)	Nusselt number (correlation)
0.1	29.928	30.086
0.2	42.505	42.548
0.3	52.149	52.110
0.4	60.268	60.172
0.5	67.407	67.274
0.6	73.847	73.695
0.7	79.748	79.600
0.8	85.206	85.096
0.9	90.271	90.258
1	94.974	95.140

*The value obtained from FLUENT results is close to those values expected from the correlation.*

*Follow this procedure to generate values of  $Nu$  for all the points. Using this data generate a plot to compare the Nusselt number using both, correlation and simulation results.*

## Summary

There is a good comparison of both Nusselt number and skin friction coefficient with the values obtained from the correlations. This shows the importance of using higher order discretization schemes and also setting tighter convergence criteria.

- The skin friction comparison shows that the viscous layer has been resolved properly.
- The Nusselt number comparison reveals that thermal boundary layer is correctly predicted.

In order to match the results with standard correlation, bulk velocity and temperature has to be used at corresponding location.

### References

1. Incropera, F.P., and DeWitt, D.P., *Fundamentals of Heat and Mass Transfer*, 4th Ed., Ch. 7.
2. Schlichting H., *Boundary-Layer Theory*, 7th Edition, McGraw-Hill, 1979.

### Exercises/Discussions

1. Run the simulation for lower Reynolds number and compare the results with theoretical values.
2. How does simulation results compare with the theoretical values for turbulent flow? Following correlations can be used for turbulent flows:

$$C_{f,x} = \frac{0.445}{\ln^2(0.06Re_x)}$$
$$Nu = 0.0296 \times Re^{4/5} \times (Pr)^{1/3}$$

Will the existing grid work for any Reynolds number?

3. Observe the distribution of Nusselt number and skin friction coefficient changes as a function of Reynolds number.
4. Set the material properties to water and compare results for laminar and turbulent conditions.
5. Solve a natural convection problem by applying gravity in the flow direction, and setting density as Boussinesq model. Also set the inlet type as pressure inlet, instead of velocity inlet.

### Links for Further Reading

- [http://www.efunda.com/formulae/heat\\_transfer/convection\\_forced/calc\\_lamflow\\_isothermalplate.cfm](http://www.efunda.com/formulae/heat_transfer/convection_forced/calc_lamflow_isothermalplate.cfm)
- <http://www.cartage.org.lb/en/themes/Sciences/Physics/Mechanics/FluidMechanics/RealFluids/BoundaryLayers/BoundaryLayers.htm>