Step by Step Gueide for Boundary Layer on a Flat Plate

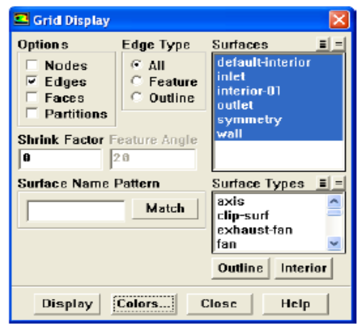
I. Preparation

1. Download the file "Flow-Over-FlatPlate.msh" from Kalam and save it to your working folder
2. Open workbench and drag "Fluid Flow (Fluent)" toolbox to the "Project Schematic" window
3. Right click on Mesh >Import mesh file>Browse and select the saved mesh file.
4. Click the Setup to start the 2D double precision solver of FLUENT.

II. General

1. Display the grid (See the Figure). Make sure all the surfaces are highlighted.

   Display > mesh...



2. Click Close and close the Grid Display panel.
3. Click the Report Quality panel. Make sure that the orthogonal quality is close to 1.

III. Modeling

1. Since problem to be solved is laminar, retain the default solver settings.

   Click Models on the left menu then Viscous > Laminar...

IV. Defining Materials

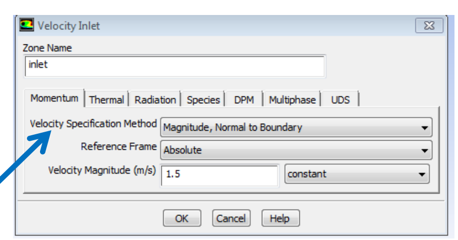
1. retain the default properties of air

   Materials...

V. Boundary Conditions

Click Boundary Conditions on the left menu

1. Set the boundary condition for velocity inlet (inlet).
   a. Select inlet from the Zone selection list.
   b. Select Velocity-inlet from the Type selection list
   c. Click the Edit... button to open the Velocity-inlet panel.



d. Enter 1.5 m/s for Velocity Magnitude.

e. 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.

b. Select pressure-outlet from the Type selection list.

c. Click the Edit... button to open the Pressure Outlet panel.

d. Make sure that the Gauge Pressure value is 0.

3. Retain the default boundary conditions for the plate (wall).

4. Set the boundary condition for top.

a. Select top from the Zone selection list.

b. Select symmetry from the Type selection list.

VI. Reference Values

Click Reference Values on the left menu

Select inlet under Compute from

VII. Solution

1. Set the solution methods.

Click on Method on the left menu

a. Select second order upwind for momentum from the Spatial Discretization drop-down lists.

b. Retain the remaining

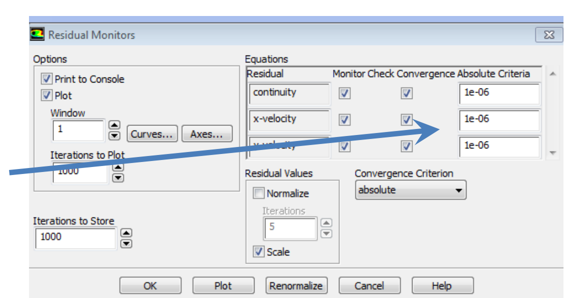
2. Set the convergence criteria.

Click Monitors on the left menu

a. Double Click Residuals-print, plot from the "Residuals, statistics and force monitors" drop-down list.

b. Enable Plot in the Options group box.

c. 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.

d. Click OK to close the Residual Monitors panel.

3. Initialize the flow.

   Click Initialization on the left menu

   a. Select Standard Initialization from the Initialization Methods list
   b. 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. Make sure that all values are correct.

   c. Click Initialize.

      For faster convergence, set all the velocities to zero.

4. Save the case file (filename.cas.gz).

   File > Write > Case...

   Retain the default Write Binary Files option so that you can write a binary file.

5. Start the calculation by requesting 1000 iterations.

   Solve > Run Calculation...

   a. Set Number of Iterations to 1000.
   b. Click Calculate.
   c. Close the Calculate panel

VIII. <u>Postprocessing</u>

1. Display pressure and velocity contours and vectors.

   Display > Graphics and Animations...

   a. Double click Contours from the Graphics and Animation drop-down lists
   b. Select Pressure... and Static Pressure from the "Contours of" drop-down lists.
   c. Enable Filled from the Options list.
   d. Click Display.

Zoom in to get the complete temperature distribution of temperature along the length of plate

Similarly, for the velocity contour:

e. Select Velocity... and Velocity magnitude from the "Contours of" drop-down lists.

f. Enable Filled from the Options list.

g. Click Display.

Zoom in to get the complete temperature distribution of temperature along the length of plate

2. Display the velocity vectors.

   a. Double click Vectors from the Graphics and Animation drop-down lists

   b. Select Velocity... and Velocity Magnitude from the "Vector of " drop-down lists.

   c. Enable Filled from the Options list.

   d. Click Display.

   Zoom in to get the complete temperature distribution of temperature along the length of plate

3. Generate lines at x = 0.1 m , x = 0.2 m, x = 0.3 m and at x = 0.4 m ,…. Up to x =1 m

   a. You need to examine the boundary layer profiles and boundary layer thicknesses in detail at all *x* locations along the flat plate. As an example five locations namely at inlet (x=0, x = 0.2 m, 0.4 m, x = 0.6 m, 0.8 m, and outlet (x=1 m) from the leading edge are explained below.

   b. In the main FLUENT menu, click **Surface-Line/Rake**. Type in the desired starting and ending *x* and *y* locations of the vertical line, i.e. a vertical line going from (0.2, 0) to (0.2, 0.5).

   c. The New Surface Name should be assigned at this point. It is suggested that this line be called "*x*=0.2-m-profile" or something descriptive of its intended purpose.

   d. Click **Create** to create the line.

   e. Similarly, create a lines at *x* = 0.4, *x* = 0.6 and *x* = 0.8; suggested label are "*x*=0.4-m-profile", "*x*=0.6-m-profile", and "*x*=0.8-m-profile".

   f. **Close** Line/Rake Surface.

   g. To view these newly created lines, click Display>Mesh. Under "Surfaces", Left Mouse Click (LMC) the default interior called "interior-surface_body" to unselect it. LMC the names of the newly created lines to select them instead.

h. Click **Display**. The lines should be visible at the appropriate locations. If not, create them again more carefully.

i. Close the Mesh Display window.

4. Plot the velocity profiles at inlet (x=0), x = 0.2 m , x = 0.6 m, at x = 0.1 m , and at outlet (x=1 m)

Display > Plots...

a. Double click XY Plots from the Plots drop-down lists

b. Select (highlight) the lines from the list of surfaces on the right side under Results, Plots-XY Plot-Set Up.

c. select Velocity from the X Axis Function,

d. Also select outlet, which is at x = 1 m.

e. Click Plot

f. In the upper left corner of the window, turn off (uncheck) Position on X Axis, and turn on (check) Position on Y Axis. This will make the vertical axis the y position on the plot, as desired.

g. In the upper middle part of that window, set Plot Direction to $X = 0$ and $Y = 1$. This will make the Y- coordinate position appear on the vertical axis, as desired for a standard velocity profile plot.

h. The boundary layer profiles should be there, but may be difficult to see since the vertical axis extends all the way to the upper boundary of the computational domain. The axes limits can be changed as follows:

i. Click Axes...

j. Choose Y.

k. Unselect Auto Range, and select Major Rules. Set Range from “0” to “0.06” m.

l. Apply then close.

Reference

FLUENT 6.1 Tutorial Guide, February 2003, file:///C:/Users/FKM%2311/Downloads/documents.mx_fluent-tutorials-55844f4e977b8.pdf