
Tutorial 2.

Flow in an Intake Manifold

Introduction

The purpose of this tutorial is to model turbulent flow in a simple intake manifold geometry. An intake manifold is a system of passages which carry the fuel mixture from the carburetor to the intake valves of the engine. It is used to distribute the fuel into the cylinder ports. This tutorial studies the flow distribution in such type of manifold.

In this tutorial you will learn how to:

- Setup a turbulent flow model.
- Set the turbulent boundary conditions for the inlet and outlet.
- Monitor the convergence using point monitors.
- Inject inert particle in the domain.
- Postprocess the flow variables and display the particle tracks.

Prerequisites

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

Problem Description

Figure 2.1 shows the schematic of geometry to be considered in this tutorial. The air is flowing at a velocity of 15 m/sec. The dimensions of the inlet and outlet boundaries are as follows:

Boundary	Length (m)
Inlet	0.07841
Outlet 1	0.01826
Outlet 2	0.03008
Outlet 3	0.03415
Outlet 4	0.02657

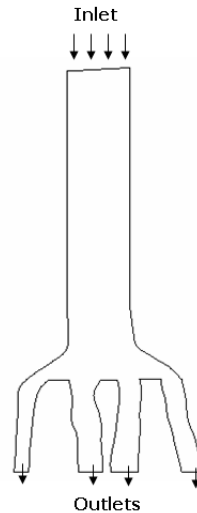


Figure 2.1: Problem Schematic

Preparation

1. Copy the mesh file, `intake.msh` to your working directory.
2. Start the 2D solver of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the grid file, `intake.msh`.

File → **Read** → Case...

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

2. Check the grid.

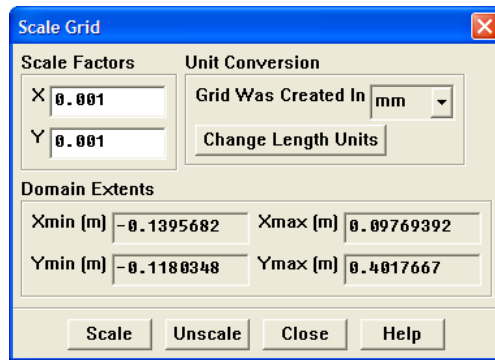
Grid → Check

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 they do not, then the grid has to be scaled with proper units. In this case, the geometry was created in mm and needs to be scaled.

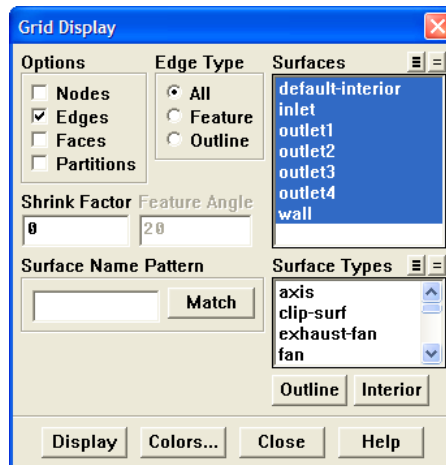


- (a) Select mm in the Grid Was Created In drop-down list.
- (b) Click Scale.

Check if the domain extents are modified or not.

4. Display the grid (Figure 2.2).

Display → Grid...



- (a) Click Display to display the grid (Figure 2.2).

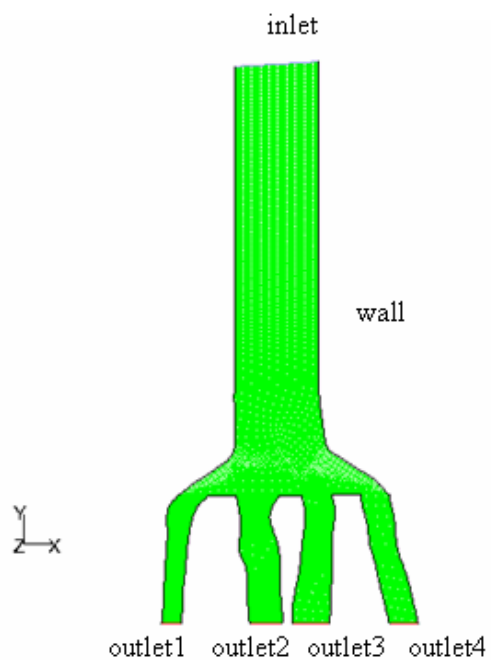


Figure 2.2: Grid Display

Step 2: Models

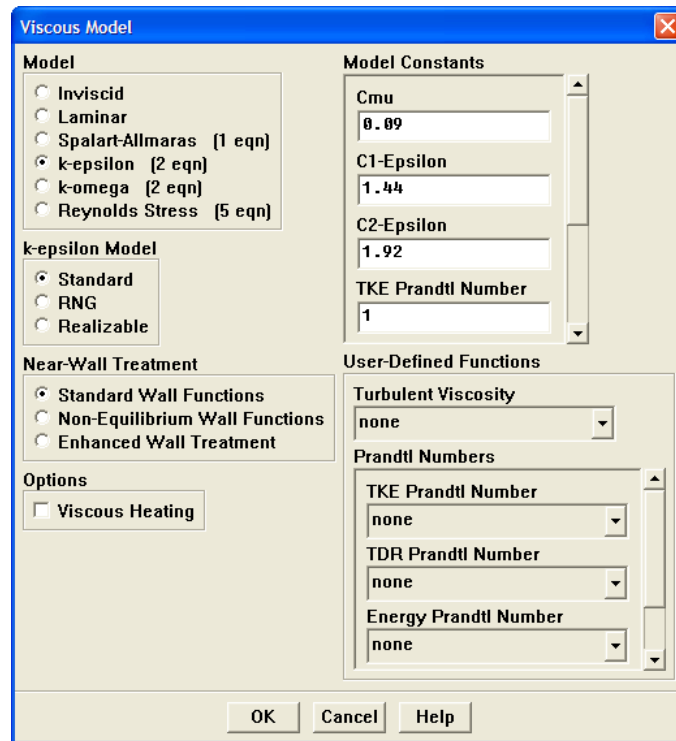
1. Retain the default solver settings.

Define → Models → Solver...

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

2. Enable standard $k - \epsilon$ turbulence model.

Define → Models → Viscous...



- (a) Under Model, enable k-epsilon (2 eqn).
- (b) Click OK to accept the settings and close the panel.

The Reynolds number can be calculated at the inlet using a built-in calculator in FLUENT.

$$Re = \frac{U \times D_i \times \rho}{\mu} \quad (2.1)$$

where,

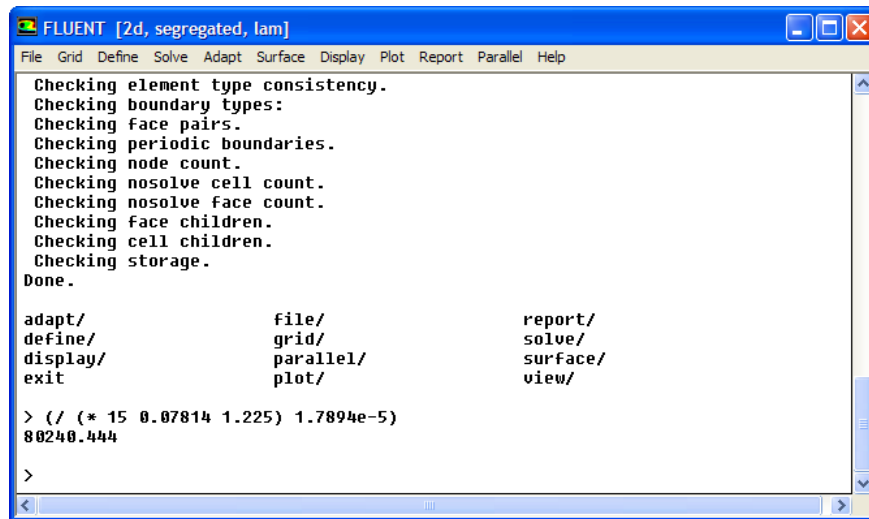
D_i = The inlet diameter (0.07814m).

U = The velocity at inlet (15 m/s).

This expression can be evaluated in the FLUENT console window by entering the above equation using the corresponding values in the console as shown below:

```
(/ (* 15 0.07814 1.225) 1.7894e-5)
```

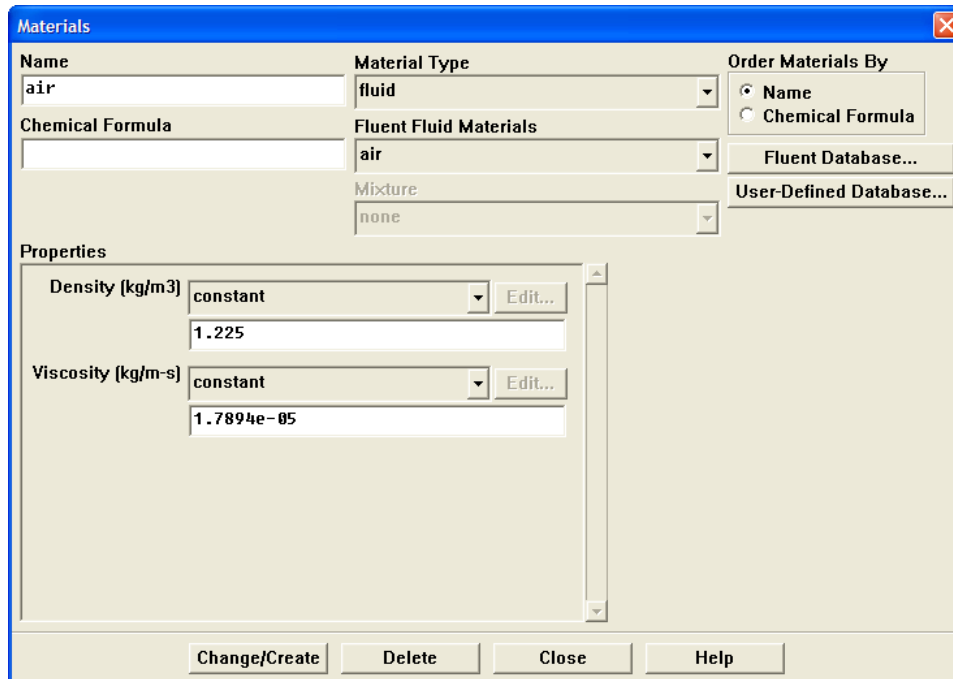
Press Enter key. The value of Re will be reported as 80240.444 showing that the flow is turbulent.



Step 3: Materials

1. Keep default material properties.

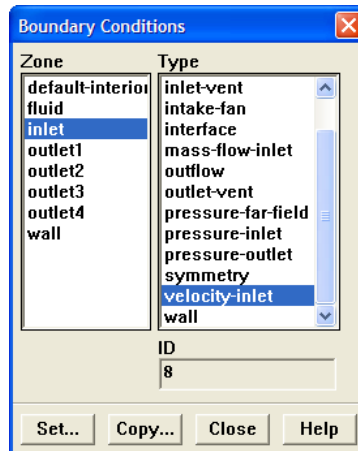
Define → Materials...



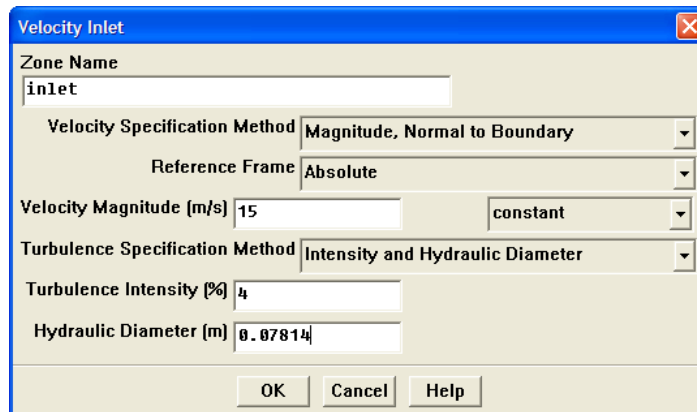
Step 4: Boundary Conditions

Define → Boundary Conditions...

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



- (a) Under Zone, select inlet.
This Type will be reported as velocity-inlet.
- (b) Click Set....
The Velocity Inlet panel opens.



- i. Specify a value of 15 for Velocity Magnitude (m/s).
- ii. Select Intensity and Hydraulic Diameter as Turbulence Specification Method.
- iii. Specify a value of 4 for Turbulent Intensity and a value of 0.07814 for Hydraulic Diameter.

Turbulent Intensity can be calculated using the relation:

$$T.I = 0.16 \times Re^{-1/8}$$

The length of inlet can be used as Hydraulic Diameter.

iv. Click OK to close Velocity Inlet panel.

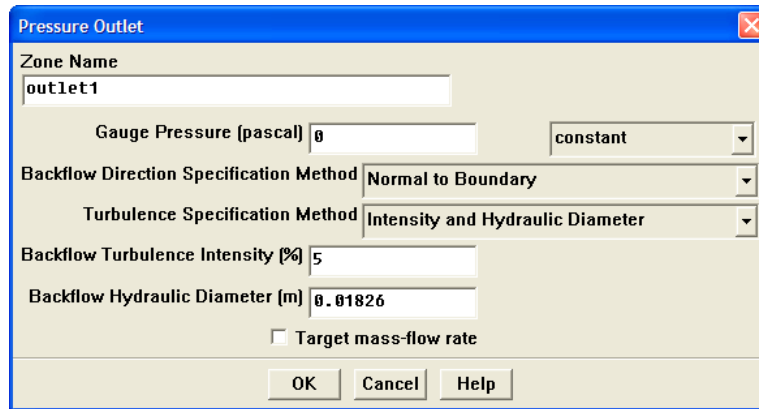
2. Set the boundary condition for outlet (outlet1).

(a) Under Zone, select outlet1.

This Type will be reported as pressure-outlet.

(b) Click Set....

The Pressure Outlet panel opens.



i. Keep default value of Gauge Pressure.

ii. Select Intensity and Hydraulic Diameter as Turbulence Specification Method.

iii. Set Backflow Turbulent Intensity to 5.

This will only be used in case of reverse flow from this boundary.

iv. Set Hydraulic Diameter to 0.01826.

This value corresponds to the length of outlet1 boundary.

3. Specify similar boundary conditions for the remaining outlets. The only change is to specify the Hydraulic Diameter for each outlet corresponding to the dimensions mentioned in problem description.

Step 5: Solution

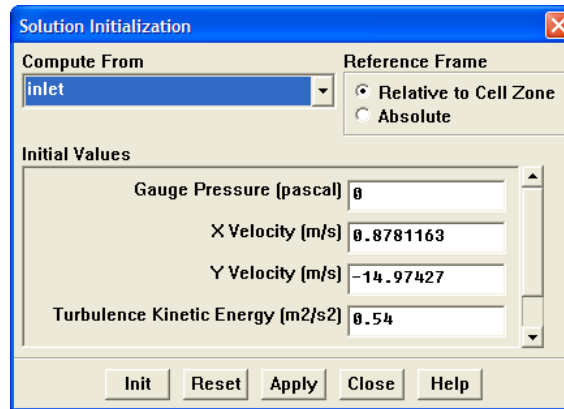
1. Set the solution controls.

Solve → Controls → Solution...

- (a) Keep default parameters in the Solution Controls panel.

2. Initialize the flow.

Solve → Initialize → Initialize...



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

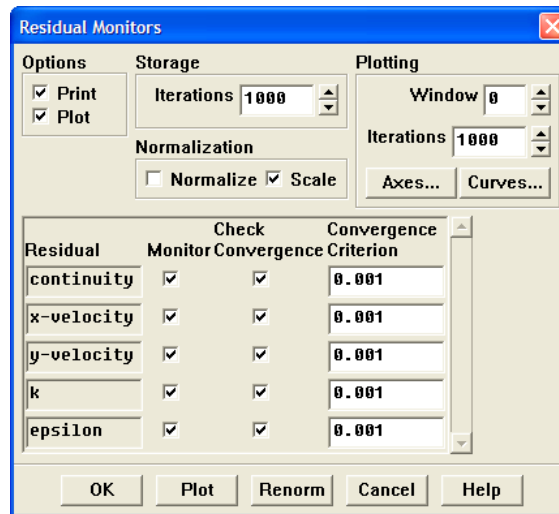
The values of all the variables based on the boundary conditions at inlet will get updated.

- (b) Click Init and close the panel.

The flow will get initialized with appropriate values of velocity and turbulence parameters.

3. Enable the plotting of residuals during the calculation.

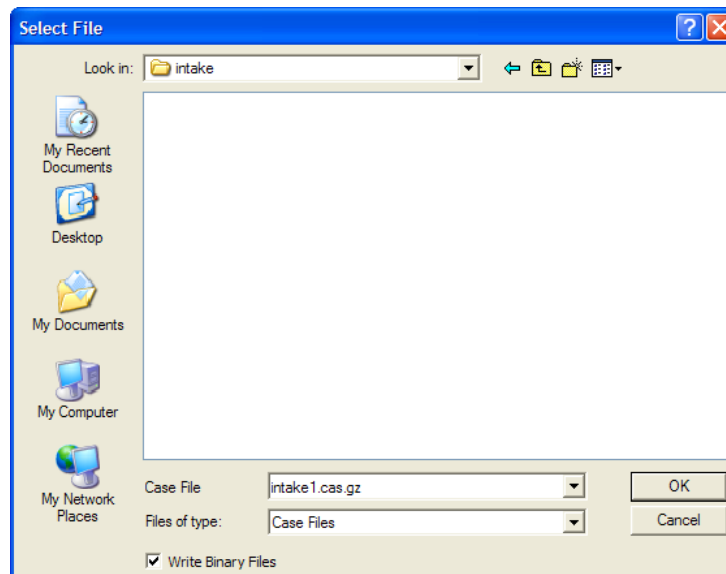
Solve → Monitors → Residuals...



- (a) Under **Options**, enable **Plot**.
 - (b) Click **OK** to accept the settings and close the panel.
4. Save the case file (`intake1.cas.gz`).

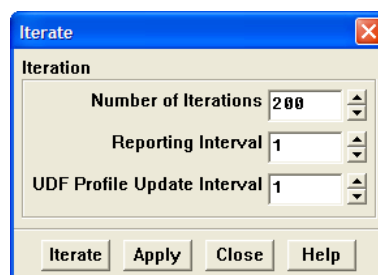
File → **Write** → **Case...**

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



5. Start the calculation by requesting 200 iterations.

Solve → **Iterate...**



- (a) Set **Number of Iterations** to 200.
- (b) Click **Iterate**.

The solution converges in about 82 iterations with default convergence criteria. The residuals plot is shown in Figure 2.3.

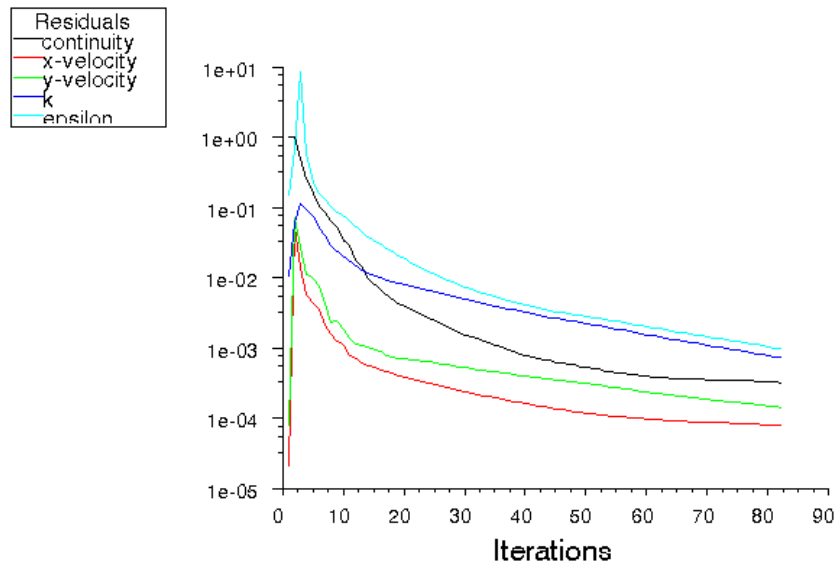
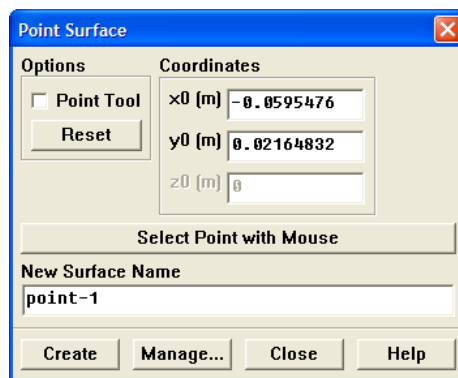


Figure 2.3: Scaled Residuals

6. Create point surfaces.

To check whether the flow variables have reached a constant value or not, monitor the average values for some of the relevant parameters. In this case, track velocity magnitude, pressure, and turbulence kinetic energy over three points in the flow domain.

Surface → Point...



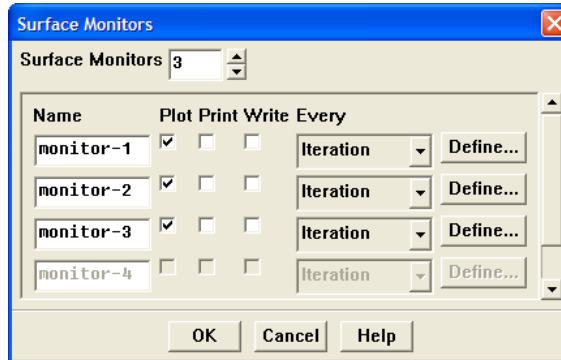
- Under Coordinates, specify a value of -0.0595476 for x_0 and a value of 0.02164832 for y_0 .
- Set the New Surface Name as `point-1`.
- Click Create.

(d) Similarly, create two more monitor points using following values:

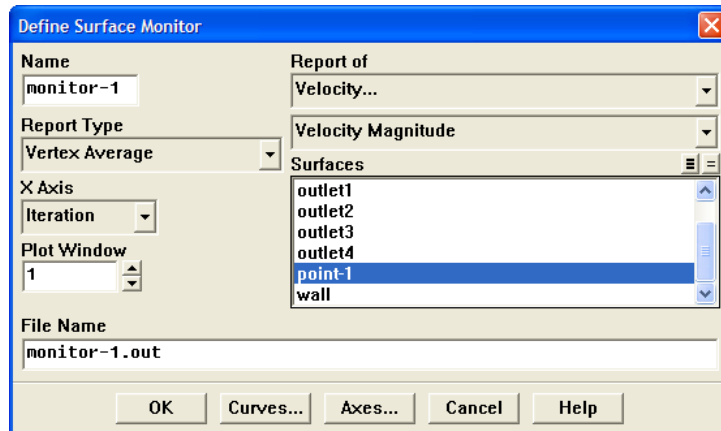
Point	x0	y0
point-2	0.001064202	0.02015189
point-3	0.03839816	0.01478293

7. Monitor vertex average velocity magnitude at the first point surface.

Solve → Monitors → Surface...



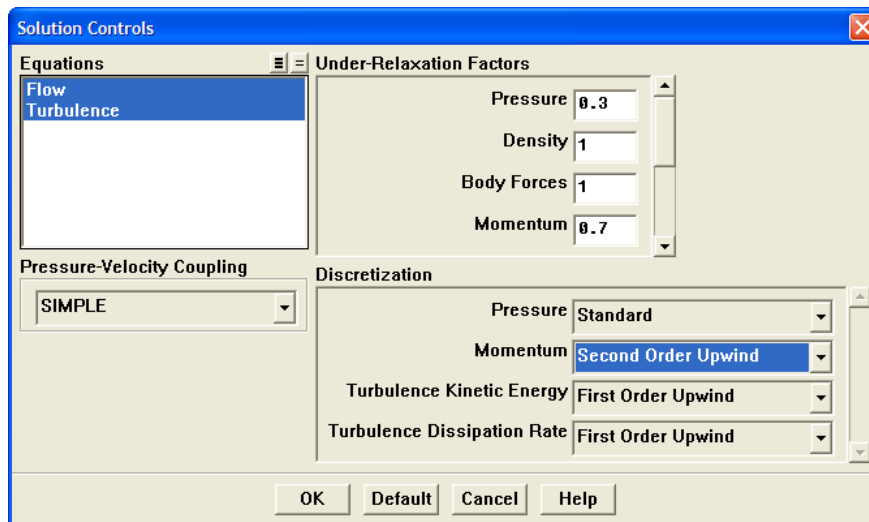
- (a) Increase the number of Surface Monitors to 3.
- (b) Enable Plot for all three monitors.
- (c) Click on Define... next to monitor-1.



- i. Select Velocity... and Velocity Magnitude in the Report of drop-down list.
 - ii. Under Surfaces, select point-1.
 - iii. Select Vertex Average in the Report Type drop-down list.
 - iv. Click OK to accept the settings.
- (d) Similarly, set the monitors for static pressure and turbulent kinetic energy at point-2 and point-3 respectively.

8. Set second order upwind scheme for momentum.

Solve → Controls → Solution...



- (a) Under Discretization, select Second Order Upwind for Momentum.
- (b) Click OK.

9. Disable the convergence criteria for all the equations.

For a better convergence, the iterations need to be performed till all the monitors flatten out. Therefore, disable the convergence criteria.

Solve → Monitors → Residuals...

- (a) Disable the Check Convergence option for all equations.
- (b) Click OK.

10. Iterate the solution till all the monitors reach a constant value, as shown in Figures 2.4 and 2.5.

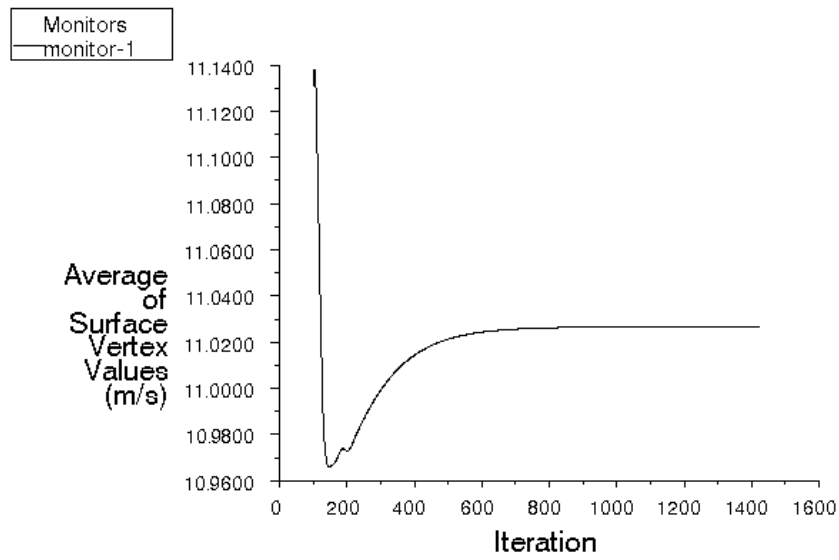


Figure 2.4: Convergence History of Velocity Magnitude on point-1

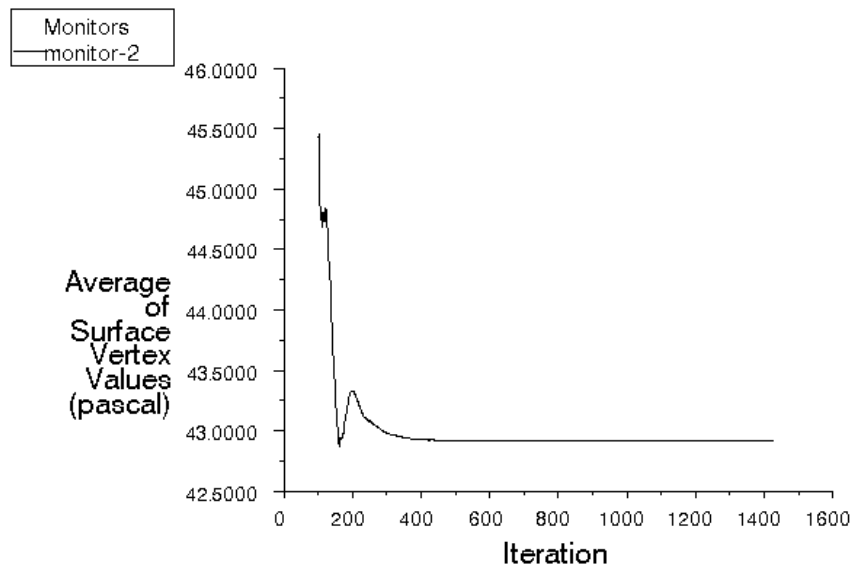
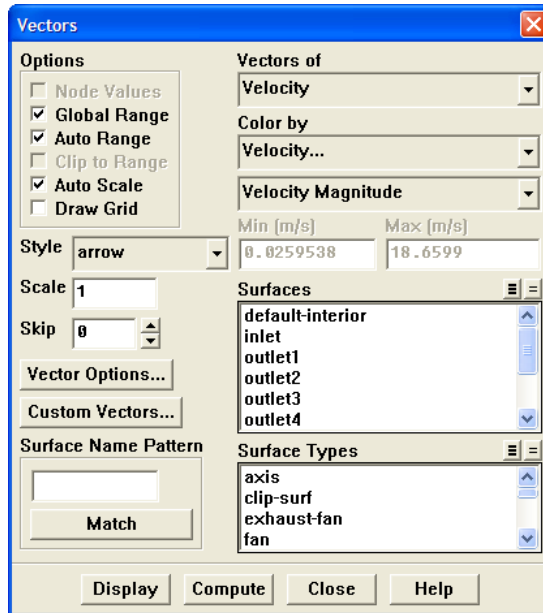


Figure 2.5: Convergence History of Static Pressure on point-2

Step 6: Postprocessing

1. Display velocity vectors (Figure 2.6).

Display → Vectors...



- (a) Select Velocity in the Vectors of drop-down list.
- (b) Select Velocity... and Velocity Magnitude in the Color by drop-down list.
- (c) Click Display (Figure 2.6).

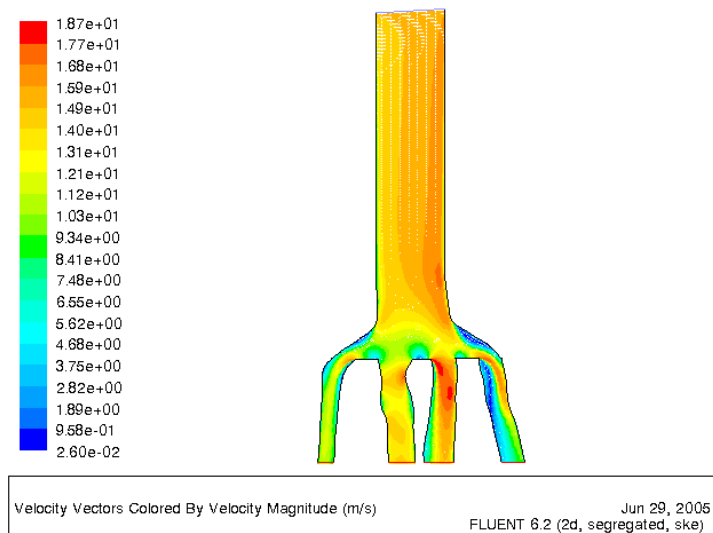
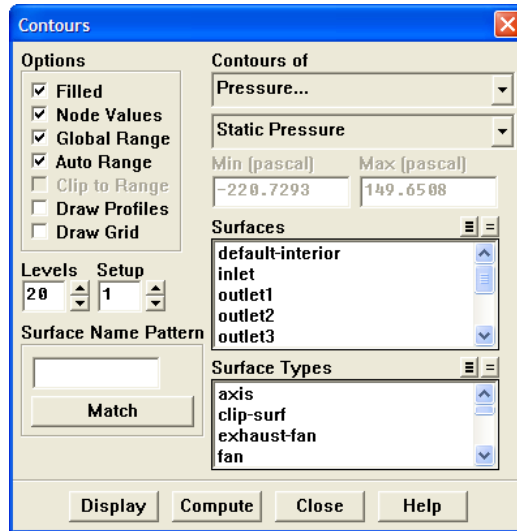


Figure 2.6: Velocity Vectors

2. Display the pressure contours (Figure 2.7).

Display → Contours...



- (a) Under Contours of, retain the default options, Pressure... and Static Pressure.
- (b) Under Options, enable Filled.
- (c) Click Display.

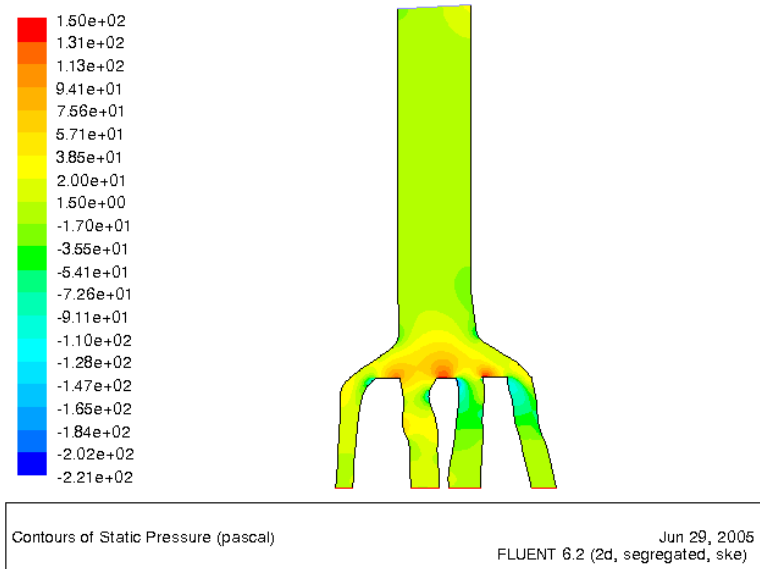
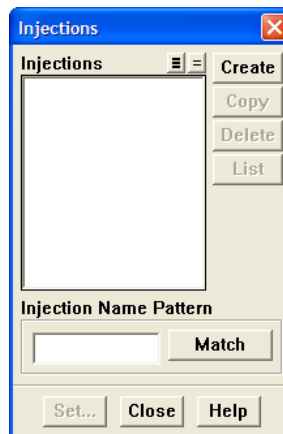


Figure 2.7: Contours of Static Pressure

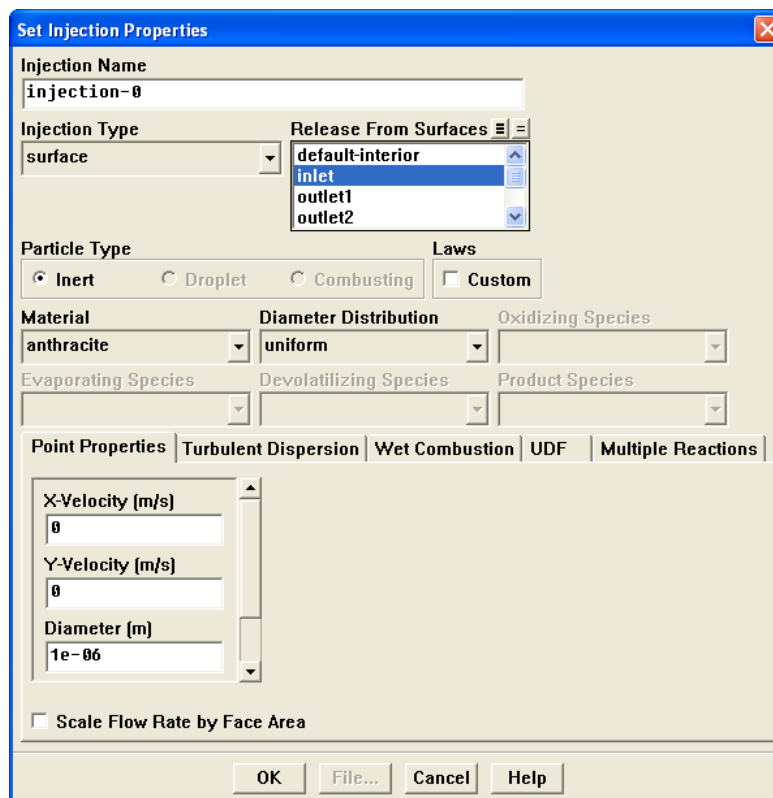
Right-click on a point in the domain to display the value of the corresponding contour in the console window.

3. Define injections.

Define → Injections...



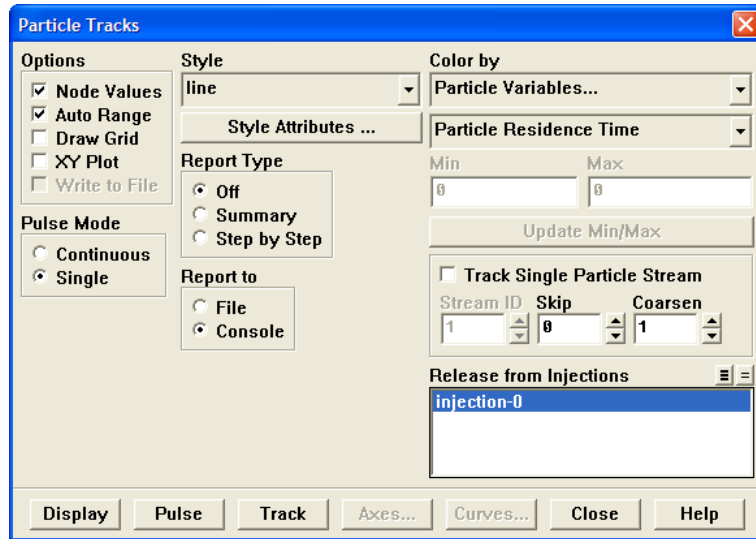
(a) In the Injections panel, click Create.



- (b) Inject inert particles from the inlet.
- i. Select Surface in the Injection Type drop-down list.
 - ii. Under Release From Surfaces, select inlet.
 - iii. Click OK to accept the settings.

4. Display the particle tracks (Figure 2.8).

Display → Particle Tracks...



- (a) Select injection-0 under Release from Injections.
- (b) Click Display.

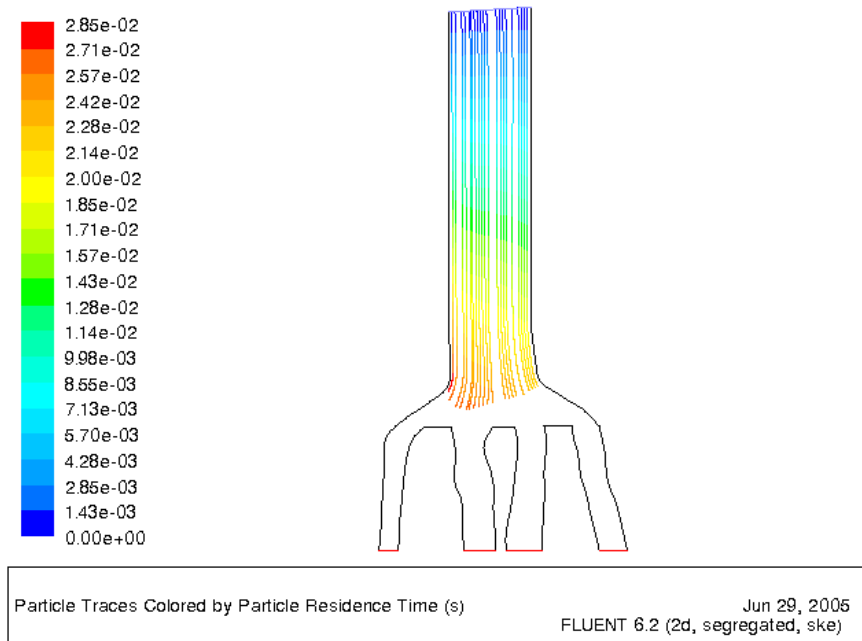
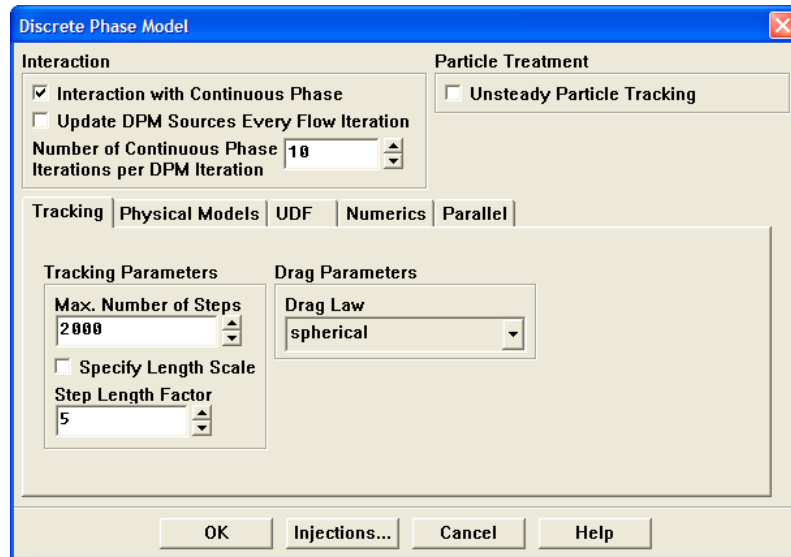


Figure 2.8: Particle Tracks

In Figure 2.8, observe that the particle tracks end before reaching the outlet. In each step, particles travel a distance and there is a limit on the total number of steps. By default it is set to 500. As this value is not sufficient to get the complete particle track, you need to set the maximum number of steps such that the particles escape through the outlet. It can be achieved by changing the Max. Number of Steps in the Discrete Phase Model panel.

5. Change the maximum number of steps.

Define → Models → Discrete Phase...



- (a) Set the Max. Number of Steps to 2000.
 - (b) Click OK to accept the settings.
6. Display particle tracks again (Figure 2.9).
The particle track now passes through the outlets.

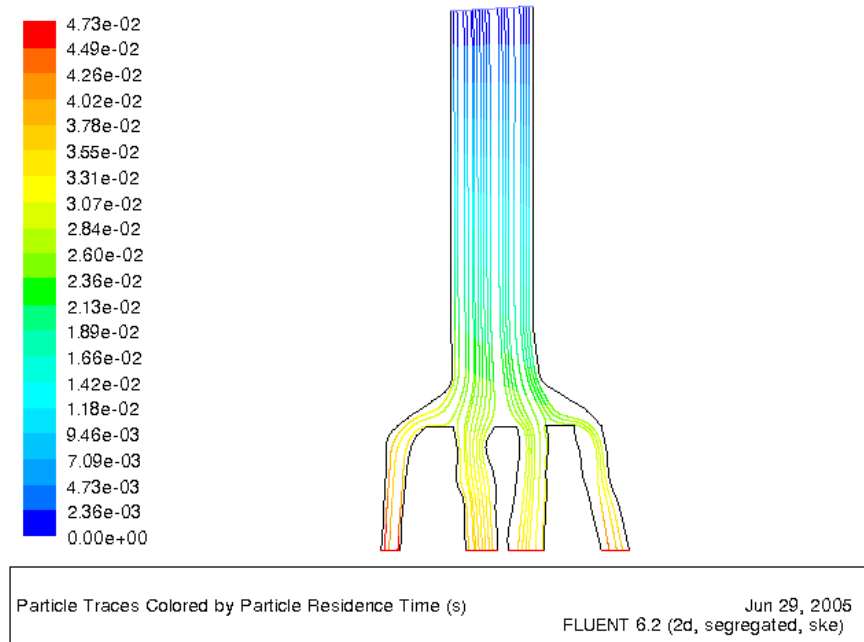
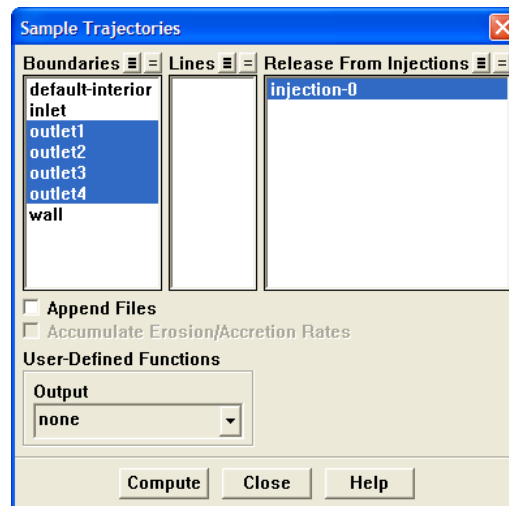


Figure 2.9: Particle Tracks

7. Write particle trajectories to a file.

Report → Discrete Phase → Sample...



- (a) Under Release From Injections, select injection-0.
- (b) Under Boundaries, select all the outlet surfaces.

(c) Click **Compute**.

This will write out four .dpm files containing the information of particles that pass through the corresponding outlet. This information can be used to check the fraction of particles cross the outlet. It should correspond to the fraction of mass flow rate flowing through the particular outlet.

8. Save the case and data files (intake2.cas.gz and intake2.dat.gz).

File → Write → Case & Data...

Summary

In this tutorial you learned how to judge a convergence by employing couple of monitors for scalar variables or integrated quantities of interest. You also learned that the iterations should not be stopped when residuals drop below the default convergence criteria. Particle tracks can be used to visualize the flow path by adjusting the maximum number of steps.

References

1. J.B. Hywood, *Internal Combustion Engine Fundamentals*, McGraw Hill Int. Ed., Ch. 9: pp. 309.

Exercises/Discussions

1. How will you change the set up if same flow is expected from all the outlets?
2. What effect will the turbulence models have on the flow distribution?
3. If the particle diameter is increased, how will they move in the domain? Will they start affecting the main flow in that case?

Links for Further Reading

- http://www.innerauto.com/Automotive_Systems/Fuel_System/Intake_Manifold/
- <http://www.autoswalk.com/fuelinsys.html>
- http://www.edelbrock.com/automotive/man_sbchev.html

