Fluid Dynamics

CAx Tutorial: Pressure Along a Streamline

Basic Tutorial #3

Deryl O. Snyder
C. Greg Jensen

Brigham Young University
Provo, UT 84602

Special thanks to:

PACE, Fluent, UGS Solutions, Altair Engineering;

and to the following students who assisted in the creation of the Fluid Dynamics tutorials:

Leslie Tanner, Cole Yarrington, Curtis Rands, Curtis Memory, and Stephen McQuay.

Pressure Along a Streamline

Introduction

In this tutorial, Gambit® will be used to create and mesh the geometry for the problem. Once this is complete, FLUENT will be used to solve for the flow field within the domain and calculate the pressure distribution along a streamline from the domain inlet to the stagnation point on the surface of the cylinder.

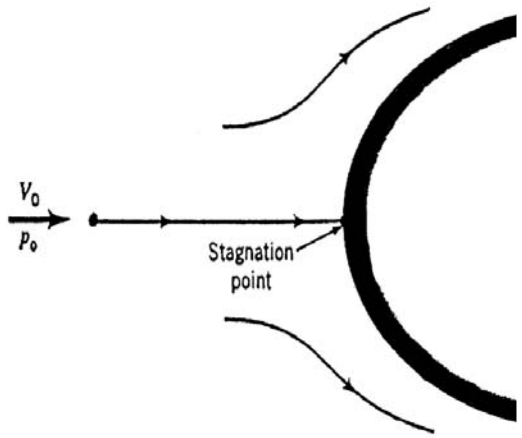
This 2D tutorial will provide more experience with 2D flows and also introduce you to geometry creation FLUENT.

The methods expressed in these tutorials represent just one approach to modeling, defining and solving 2D problems. Our goal is the education of students in the use of CAx tools for modeling, constraining and solving fluids application problems. Other techniques and methods will be used and introduced in subsequent tutorials.

An incompressible, inviscid fluid flows steadily past a cylinder. The fluid velocity along the streamline that ends at the stagnation point is found to be:

$$V_x = V_\infty\left(1-\left(\frac{a}{x}\right)^2\right)$$

where *a* is the radius of the cylinder and Vo is the upstream velocity. Determine and plot the velocity and pressure gradients along the streamline.



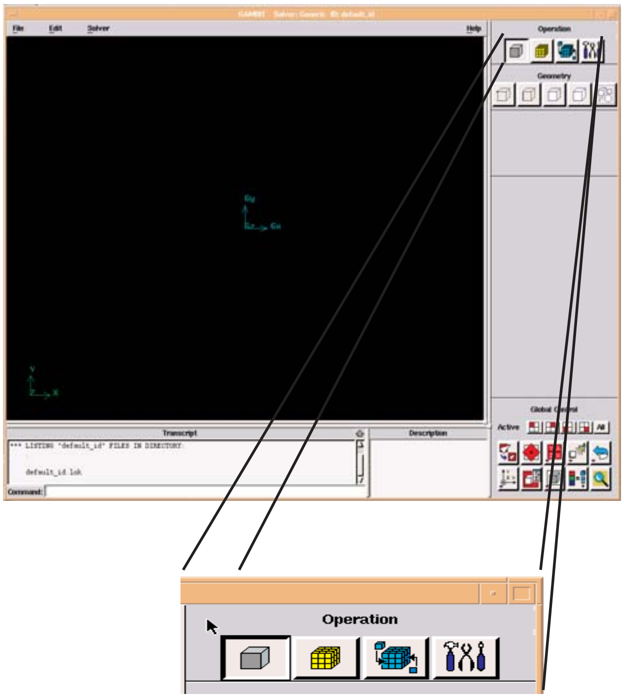
Pressure Along a Streamline

Creating Geometry

Begin the problem by creating geometry in **Gambit**.

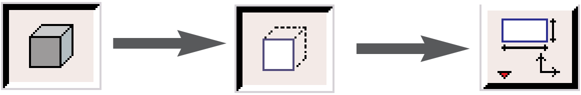
The **Gambit** standard display should be open.

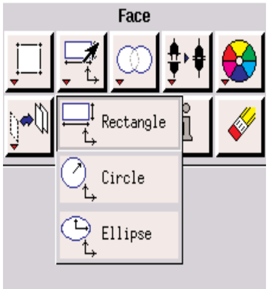
Meshes are generated in **Gambit** by following left to right the menu icons located in the top right of the display window.



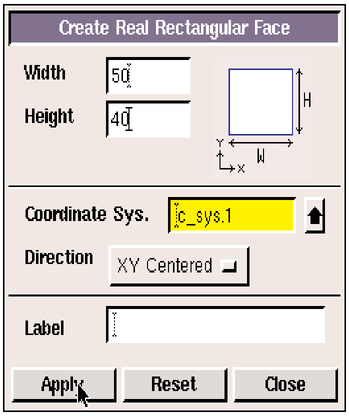
First create a rectangle as the computational domain.
Operation > Geometry > Face
*Note: Icons with a red arrow have a pull down menu. To activate the pull down menu select the icon with **MB3**.*

Select the **Create Face** icon with **MB3**, and make sure the rectangle option is selected





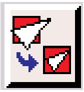
An additional working window should appear named **Create Real Rectangular Face**.



Enter the dimensions for a rectangle of **width = 50**, and **height = 40**.

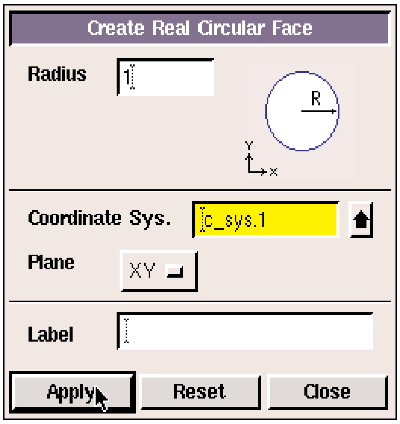
Select **Apply**.

Note: To fit the geometry to the screen in Gambit, simply select the top left icon from the menu on the bottom right of the display window.

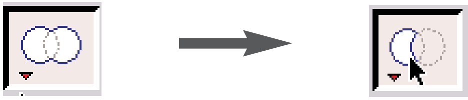


Creating Geometry

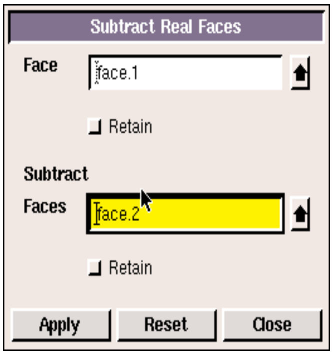
Now create the cylinder in the center of the rectangle by selecting the same **Create Face** icon wtih **MB3** and activating the **Create Real Circle Face**. Make a circle of **radius = 1** select **Apply** and then **Close**.



Since there is no flow inside of the cylinder, the circle face must be subtracted from the rectangle face to create the desired domain. Click on the **Join Faces** button with **MB3** and select **Subtract Real Faces**.



Pick the rectangle with **Shift + MB1** as the first face and the circle (**Shift + MB1**) as the face to be subtracted. Since only the newly created face is required for the solution, leave the two **Retain** boxes unchecked and select **Apply**.



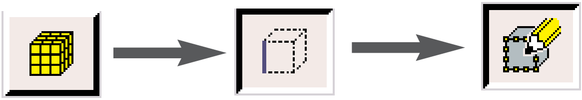
*If problems are encountered in creating the geometry, the geometry can be loaded from the file **"Stagnation_geometry.dbs"**.*

Pressure Along a Streamline

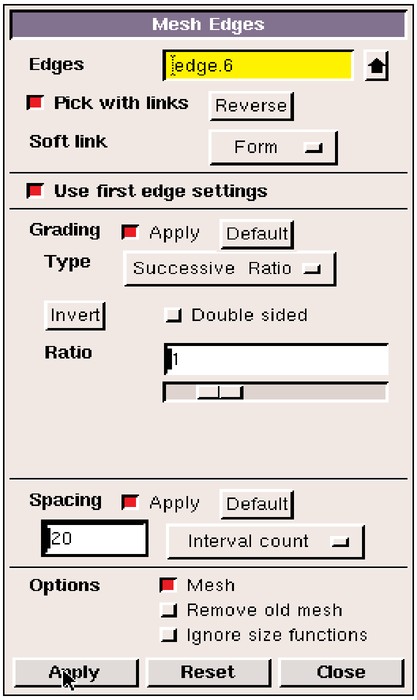
Meshing Geometry

In order to create a mesh for this 2d domain, the edges of the geometry must first be meshed. In this case the only edge mesh required is that of the cylinder.

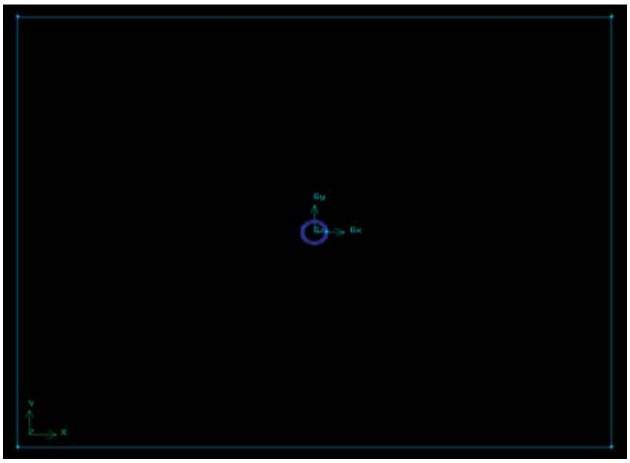
Operation > Mesh > Edge > Mesh Edges



Select the circle edge. In the **Spacing** section change the pull down menu to **Interval Count** and enter **30**. Click **Apply**.



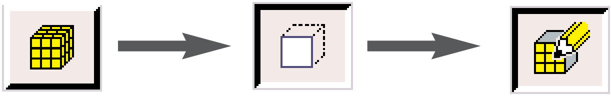
Your domain should resemble the one shown to the right.



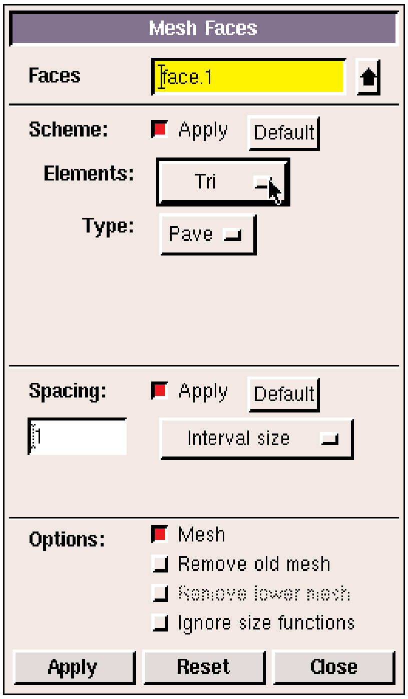
Meshing Geometry

Now create the interior mesh.

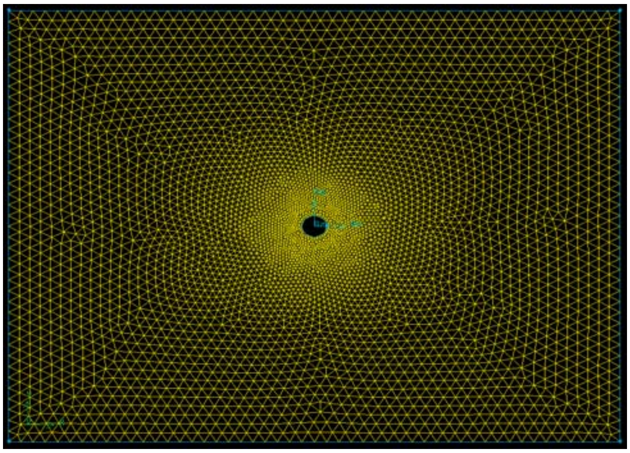
Operation > Mesh > Face > Mesh Faces



Select the entire face by clicking on any one of the edges with **Shift+MB1**. In the **Elements** pull down menu change the element type to **Tri** and click **Apply**.



The mesh should resemble the one shown to the right.



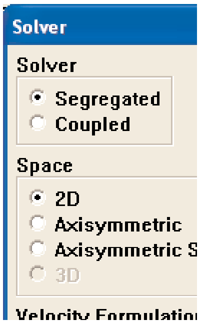
Pressure Along a Streamline

Boundary Conditions

The next task is to specify the bondary conditions.

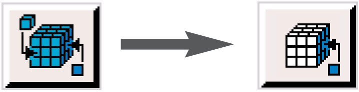
The first step is to specify the solver that will be used.

Solver > Fluent 5/6



Next, bring up the **Specify Boundary Types** window.

Operation > Zones > Specify Boundary Types



From the pull down menu under **Entity**, select **Edges**.

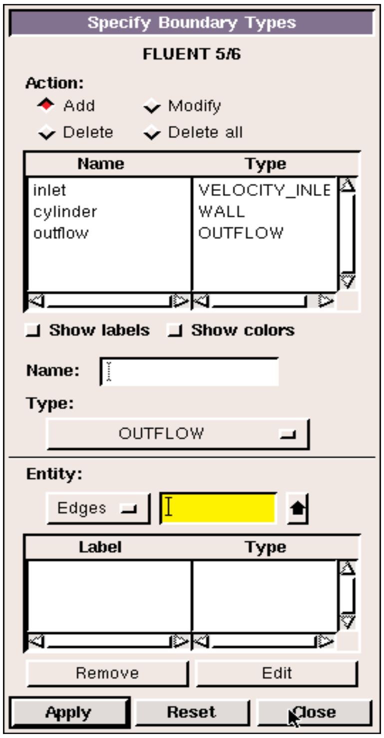
Select the **top, left, and bottom edges** of the rectangle.

Type the label **inlet** next to **Name**.

Select **Velocity Inlet** from the **Type** pull down menu

Select **Apply**. The boundary condition should appear in the list at the top of the **Boundary Types** window.

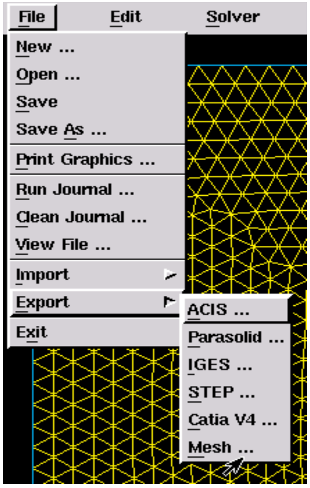
Now create the last two boundary conditions in the same way. The far right edge of the rectangle as an **outflow** and the cylinder as a **wall**. The list should resemble the image to the right.



Exporting the Mesh

Now export the mesh that will be imported into Fluent

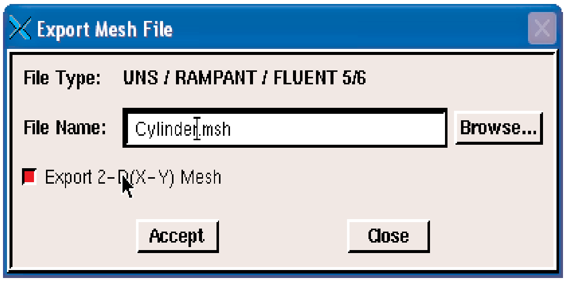
File > Export > Mesh



Make sure the **Export 2d (X-Y mesh)** box is selected.

Select **Browse...** to choose the saving directory for the mesh file.

Save the mesh as **Cylinder.msh**.



Also save the Gambit file.

File > Save

Save the file as **Cylinder**

*If problems are encountered in meshing the geometry, the mesh can be loaded from the file **"Stagnation_mesh.msh"**.*

Pressure Along a Streamline

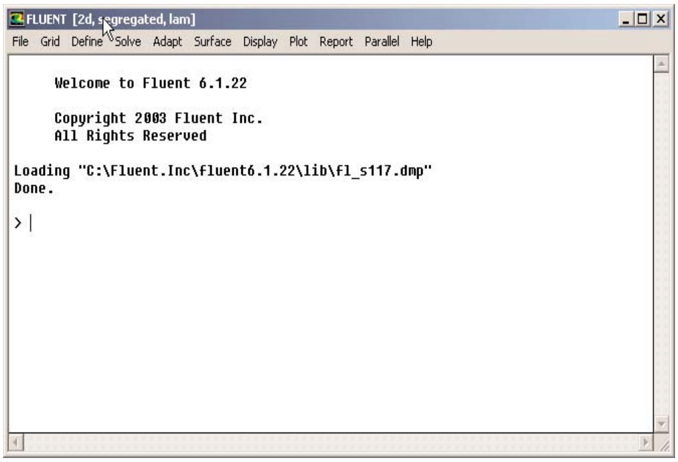
Starting in Fluent

Open **FLUENT** from the Desktop or Start menu.

Select **2D**

Select **Run**

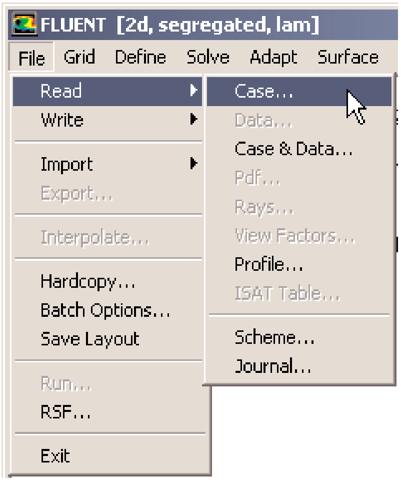
The following window should appear.



This is the **FLUENT** user interface. Most tasks are completed using the menus across the top. The menus are designed to guide you through the analysis in an orderly fashion, going from top to bottom through each menu, and left to right across the menu bar.

Now import the mesh into Fluent.

File > Read > Case



A browse window should appear.

Locate **Cylinder.msh** and select **OK**.

FLUENT will read in the geometry and mesh you created. If there are problems reading the mesh, return to the beginning of the tutorial and make sure you follow the steps carefully. If there are no problems the command window should state "**done**".

Now check the grid for errors.

Grid > Check

Any errors will be listed, otherwise the command window should again state "**done**".

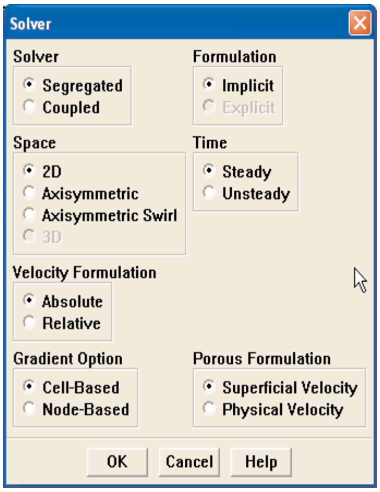
Pressure Along a Streamline

Defining the Problem

Default settings for the Fluent Solver will be sufficient for this problem.

Define > Models > Solver

Confirm that the Solver options are the same as shown on the left.



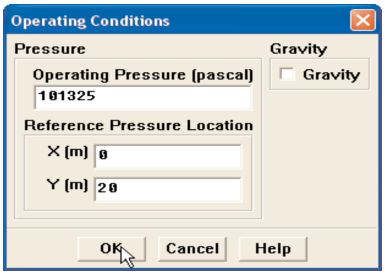
Now change the viscosity model to an inviscid flow.

Define > Models > Viscous

Select **Inviscid** and then **Ok**

Since the cylinder is centered at (0,0) in the domain, the reference pressure for the solution must be moved into the domain.

Define > Operating Conditions...



Move the reference pressure location to the top wall by setting **X** to **0 m** and **Y** to **20 m**.

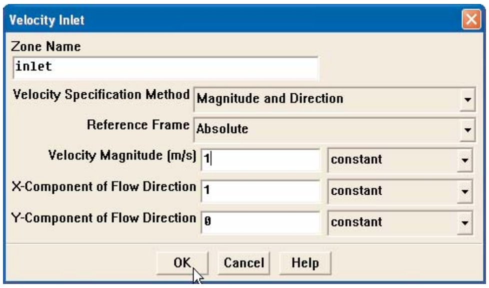
Now set the desired inlet velocity in the **Boundary Conditions** menu.

Define > Boundary Conditions...

Select the **inlet** from the column on the left and then click on **Set...**

Defining the Problem

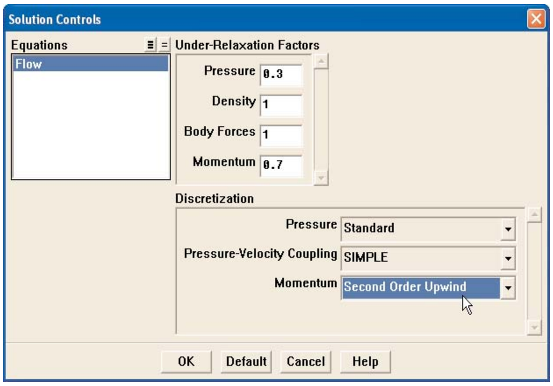
Change **Velocity Specification Method** to **Magnitude and Direction** and set **Velocity Magnitude** to **1**. Ensure that the **X-Component of Flow Direction** is set to **1** and that the **Y Component** is **0**. Click **OK** and exit from the Boundary Conditions window.



This case will be solved with a second order discretization, so change the solution controls for momentum to second order.

Solve > Controls > Solution

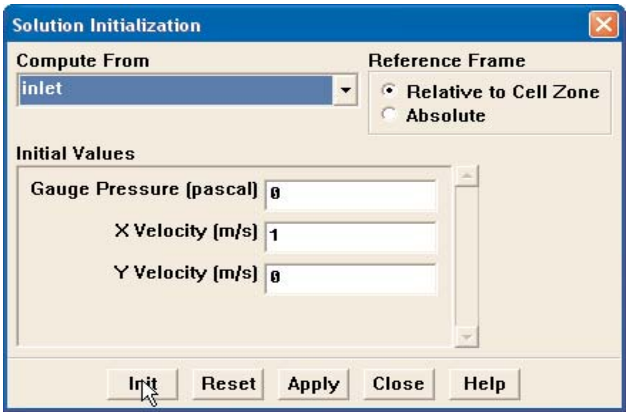
Change the drop down menu for **Momentum** to **Second Order Upwind** and click **Ok**.



The problem can now be initialized.

Solve > Initialize > Initialize...

Select **inlet** in the drop-down menu labeled **Compute From** and verify that the **Initial Values** match the inlet conditions:



Gague Pressure: 0

X-Velocity: 1

Y-Velocity: 0

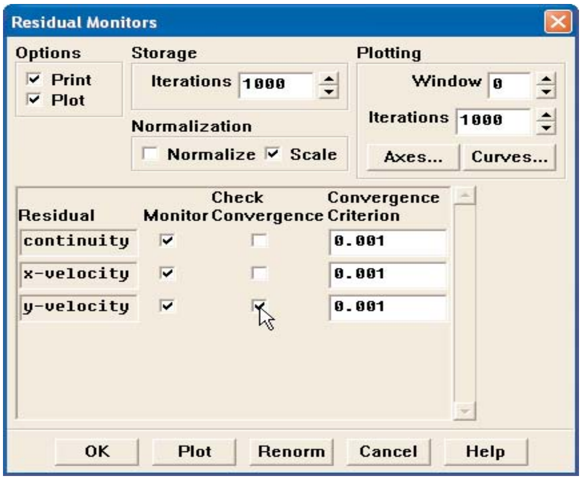
Click on **Init** to initialize the computational domain and then close the window.

Pressure Along a Streamline

Solving the Problem

The problem is now almost ready to be solved.

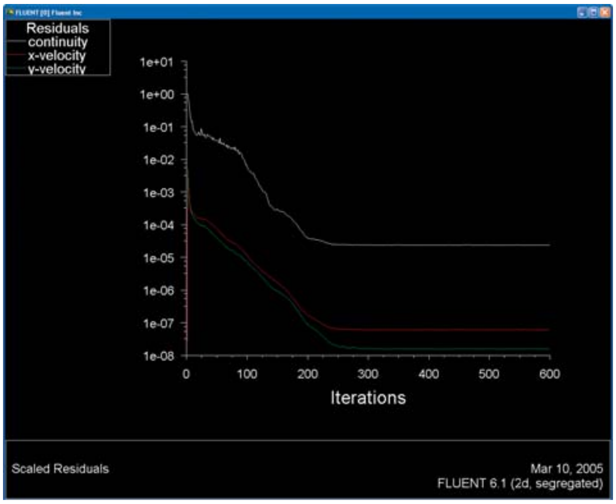
Solve > Monitors > Residual...



Make sure **Plot** is checked in the upper left corner and uncheck all of the boxes in the **Check Convergence** column. Fluent can automatically stop iterating once residuals have reached a certain value. For this case the solution will be monitored manually. Click **OK**.

Now begin iterating on the solution.

Solve > Iterate...



Begin with 600 iterations. Convergence will depend on the computing platform being used. The residuals should taper off after 400 iterations or so. The solution has stopped changing when the residuals become constant. A good indication of a converged solution is that the residuals have dropped at least 4 orders of magnitude.

Pressure Along a Streamline

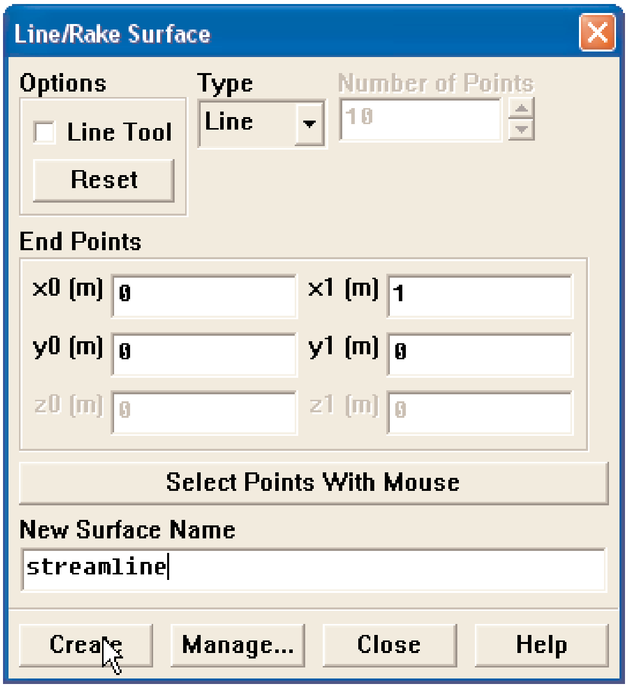
Analyzing the Solution

The problem statement calls for a graph of the pressure and velocity distributions along a streamline from the edge of the domain up to the stagnation point on the cylinder. In order to plot distributions along this streamline, a line must be created in Fluent along the desired path.

Fluent can create lines in the domain based on two user-specified points (the points require both X and Y coordinates) in the domain that are used to create the slope of the line. Fluent automatically takes this slope and draws a line across the entire domain.

Surface > Line/Rake...

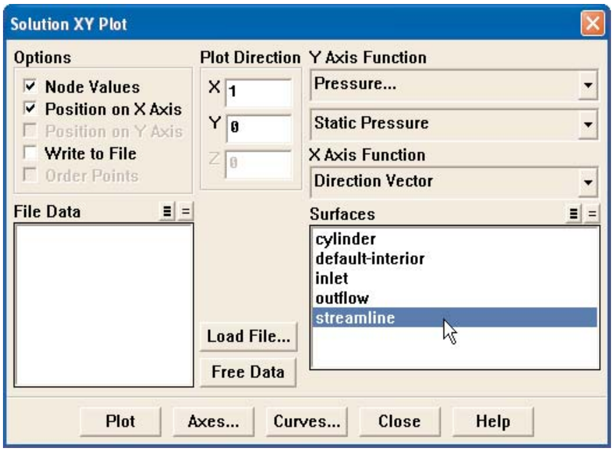
Create a horizontal line along the stagnation streamline by entering the first point **(X0,Y0)** at **(0,0)** and the second point **(X1,Y1)** at **(1,0)**. Name the line **streamline** under **New Surface Name**. Click **Create** and then **Close**.



Pressure Along a Streamline

Analyzing the Solution

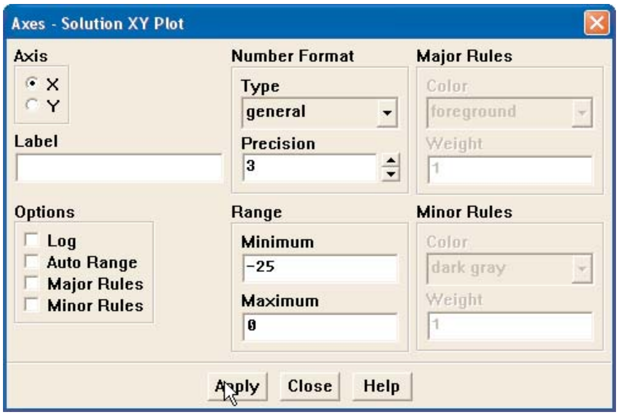
Now that a line has been specified, plots of the pressure and velocity can be created and compared to the analytical solution.



Plot > XY Plot...

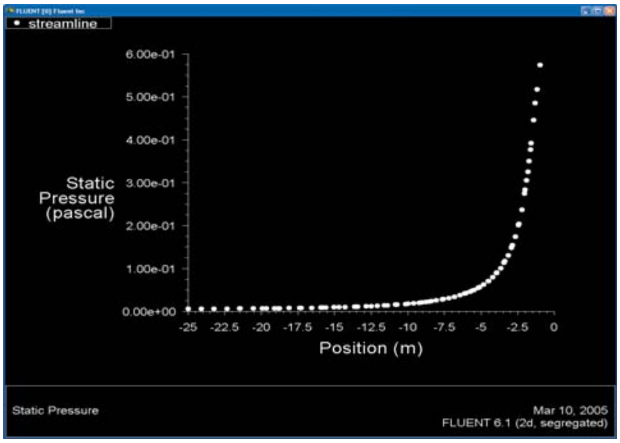
Change the two pull-down menus under the **Y Axis Function** section to **Pressure...** and **Static Pressure** respectively. **X Axis Function** should be set to **Direction Vector**. Select the **streamline** from the list of edges to plot against and click **Plot** at the bottom of the window. A window should appear with the static pressure distribution along the entire streamline. Since nothing downstream of the stagnation point is required, change the X-Axis range to display only the desired section.

Click on **Axes...** in the **XY Plot window**.



Unchecking **Auto Range** under **Options** will highlight the **Range** section in the middle of the window. Change the **Minimum** to **-25** (the far edge of the domain) and then click **Apply** and then **Close**.

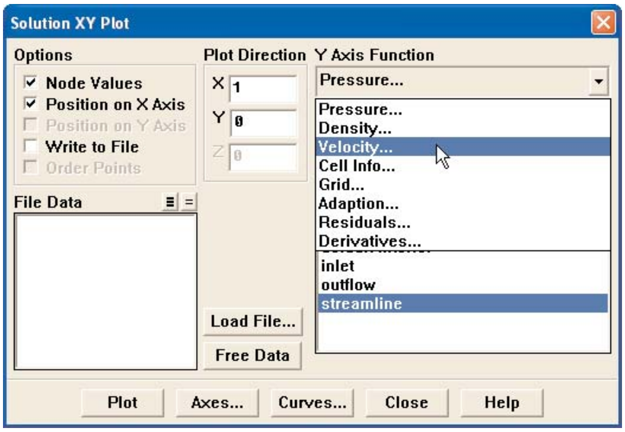
Now replot the graph by selecting **Plot** one more time. The new plot should look simliar to the image shown.



Pressure Along a Streamline

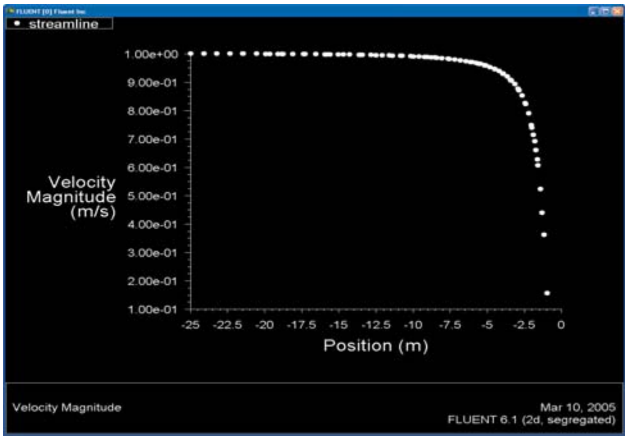
Analyzing the Solution

Now plot the velocity distribution along the same streamline by changing **Y Axis Function** in the plot window to **Velocity...** instead of **Pressure...**



Make sure that the **streamline** is still selected in the **Surfaces** menu. The X-axis settings will remain the same from the pressure plot.

The velocity distribution should be similar to the image to the left.



If problems are encountered in setting up or analyzing this problem in Fluent, the solved problem can be read in as ***Case & Data****.... from the file “****Stagnation _cylinder.cas****”.*

Pressure Along a Streamline

Analytical Solution

Using Bernoulli's equation with the velocity profile given in the problem statement the pressure distribution can be found to be:

$$P_x = P_\infty + \rho V_\infty^2 \left(\frac{a}{x}\right)^2 \left(1 - \frac{1}{2}\left(\frac{a}{x}\right)^2\right)$$

Plotting the analytical velocity and pressure distributions with corresponding Fluent output reveals good agreement as can be seen in the plots to the right.