Fluid #3: Analyzing Flow in a System of Pipes USING FLOTRAN (Carnegie Mellon)

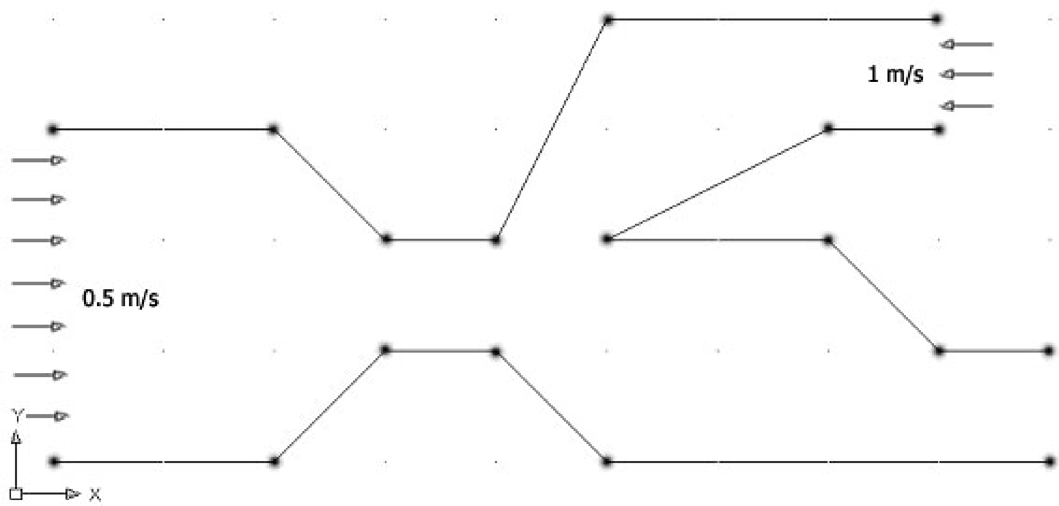
Introduction: In this example you will model a system of pipes filled with water.

Physical Problem: Compute and plot the velocity distribution in the pipe system shown in the figure.

Problem Description:

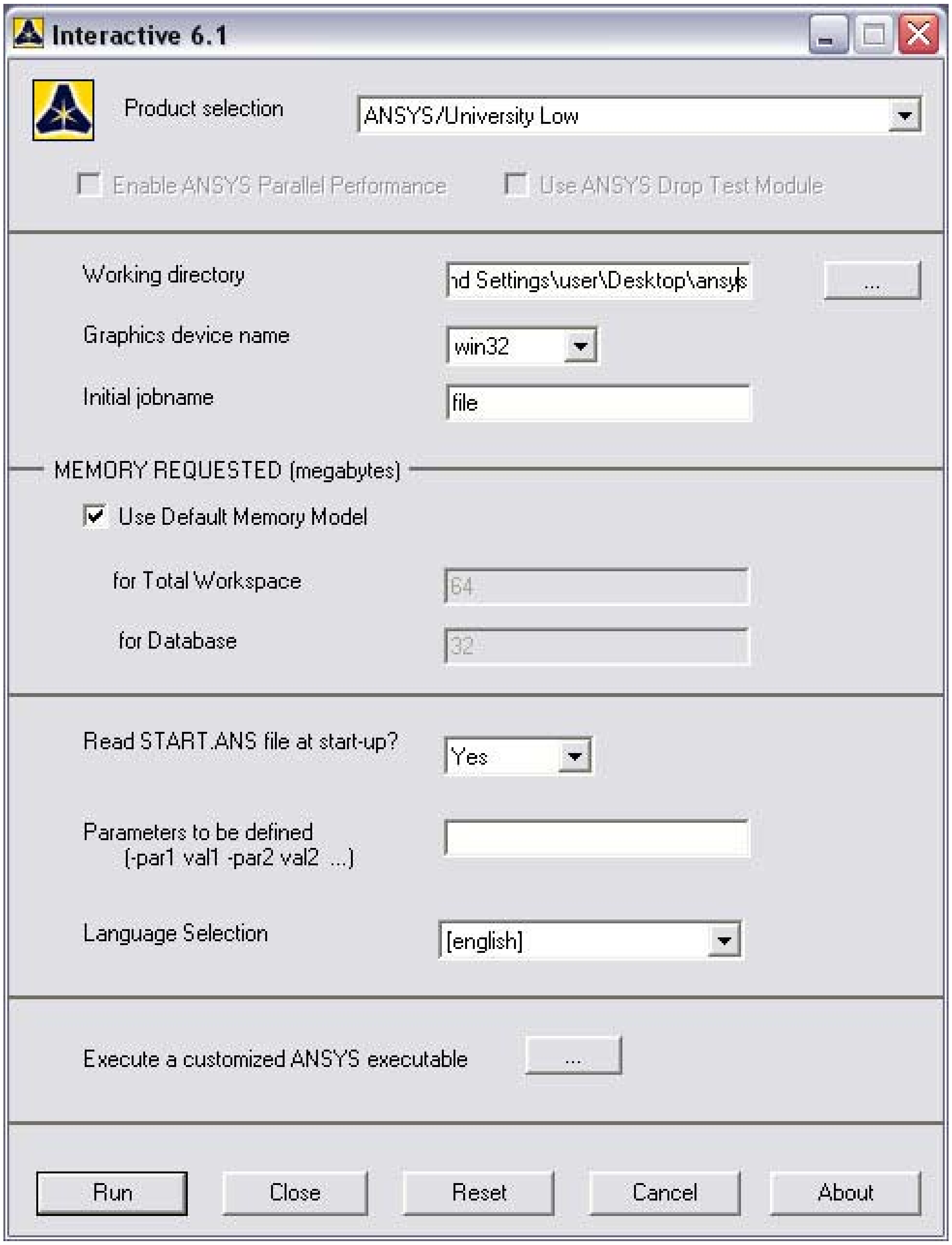
- The shape of the pipe grid is shown in the figure. (Each point is spaced evenly at 0.33m)

- Objective:
  - To plot the velocity profile within the pipe.
  - To graph the variation of velocity out the bottom pipe.
- You are required to hand in print outs for the above.
- Figure:



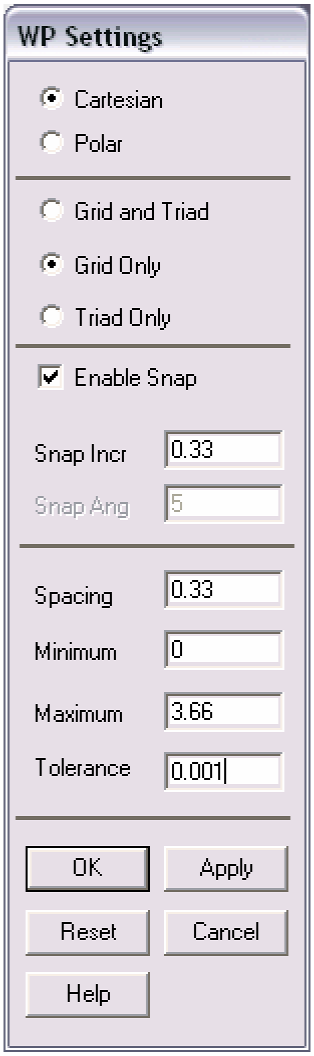
STARTING ANSYS

- Click on ANSYS in the programs menu.
- Select **Interactive**.
- The following menu that comes up. Enter the working directory. All your files will be stored in this directory. Also enter 64 for Total Workspace and 32 for Database.
- Click on Run.

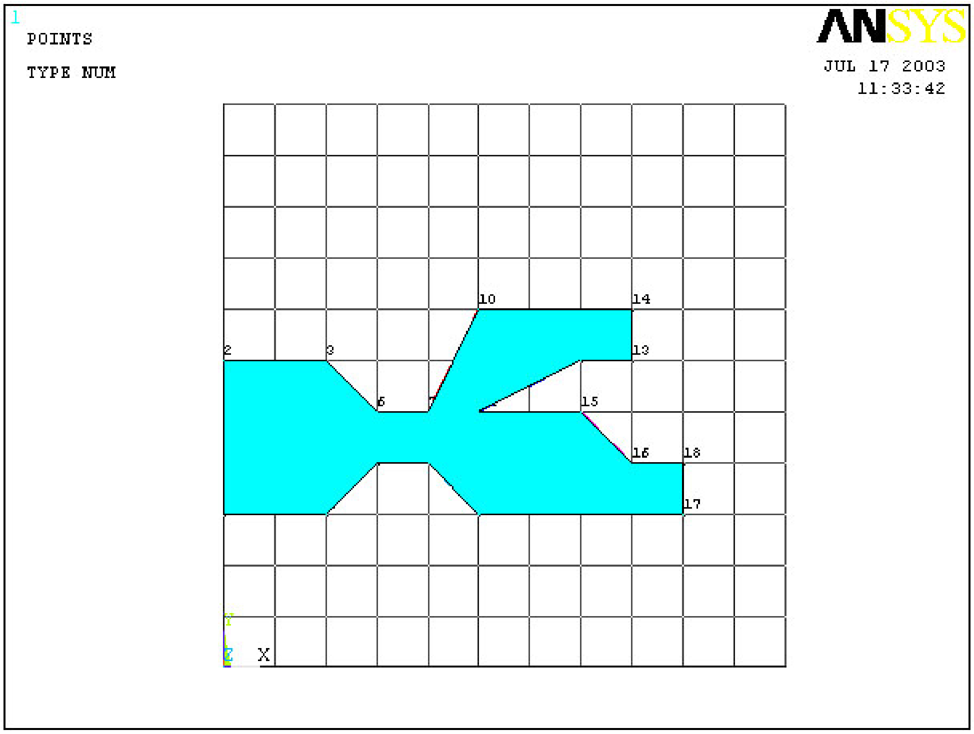


MODELING THE STRUCTURE

- Go to the ANSYS Utility Menu
- Click **Workplane>WP Settings**
- The following window comes up



- Check the **Cartesian and Grid Only** buttons
- Enter the **values** shown in the figure above.
- Go to the ANSYS Main Menu
- In this problem we will model the pipe grid and then apply fluid flow to it.
- Click **Preprocessor>-Modeling->** and create the pipe grid as shown below.
- Hint: You can use key points and then create the area

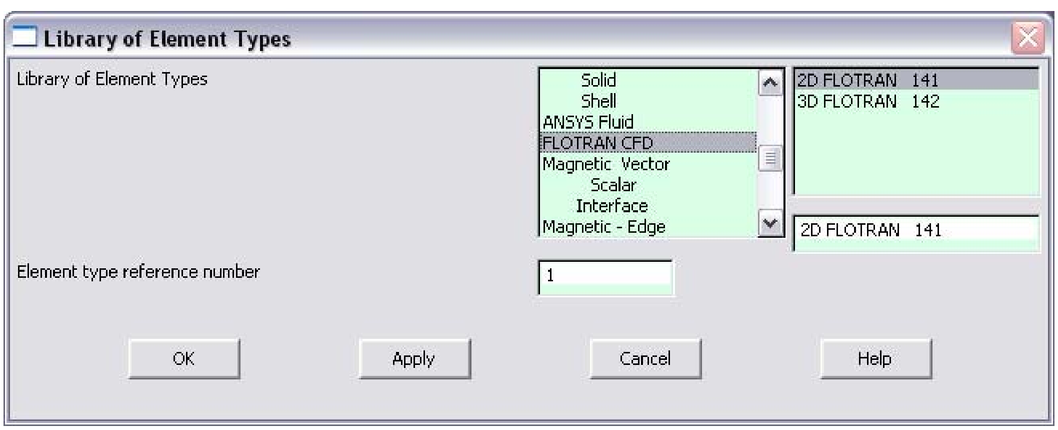


The modeling of the problem is done.

ELEMENT PROPERTIES

SELECTING ELEMENT TYPE:

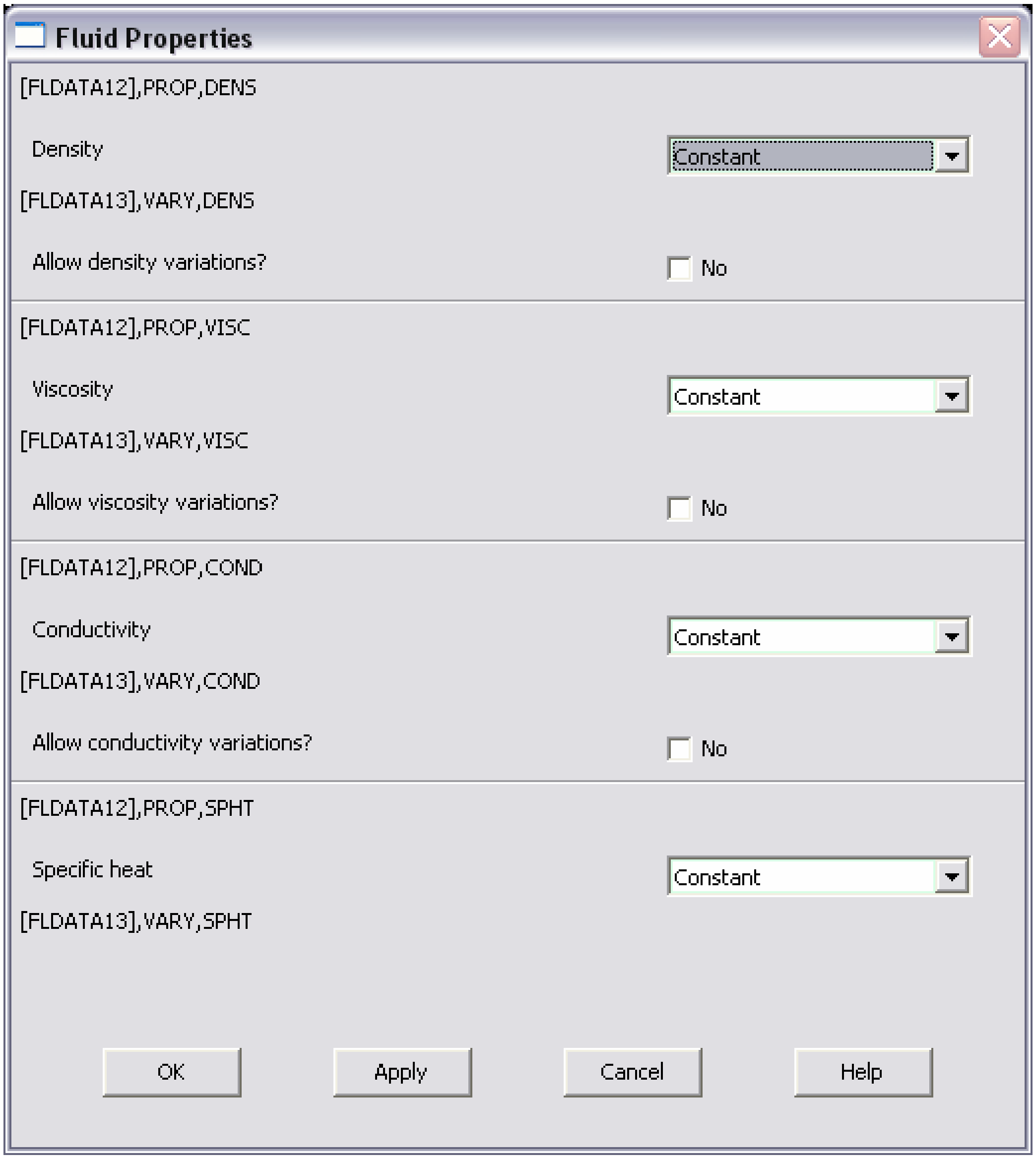
- Click **Preprocessor>Element Type>Add/Edit/Delete**... In the 'Element Types' window that opens click on **Add**... The following window opens:

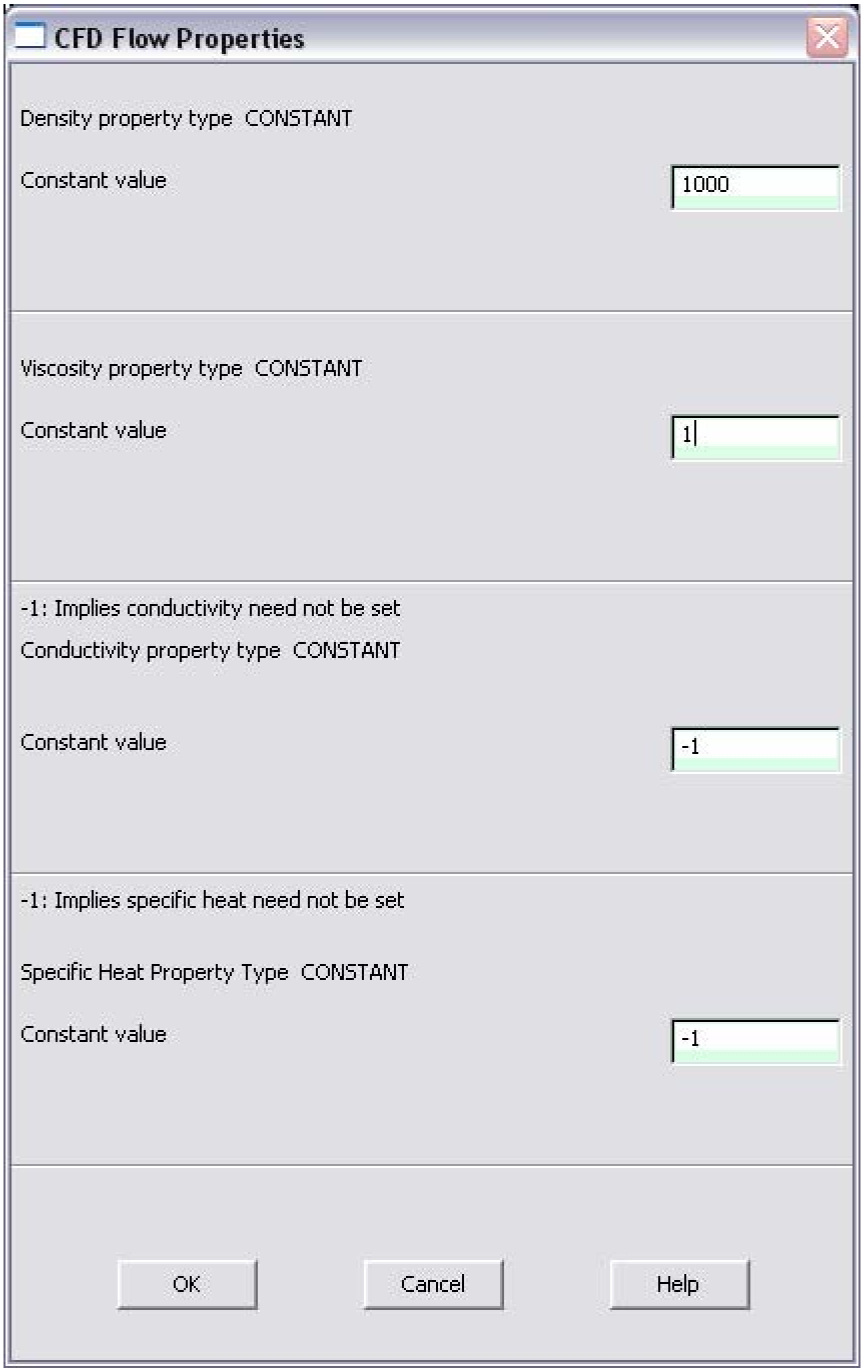


- Type **1** in the Element type reference number.
- Click on **Flotran CFD** and select **2D Flotran 141**. Click **OK**. Close the 'Element types' window.
- So now we have selected Element type 1 to be a Flotran element. The component will now be modeled using the principles of fluid dynamics. This finishes the selection of element type.

DEFINE THE FLUID PROPERTIES:

- Go to **Preprocessor>Flotran Set Up>Fluid Properties.**
- On the box, shown below, make sure the first two input fields read **Constant,** and then click on **OK**. Another box will appear. Fill in the values as shown below, then click **OK**.

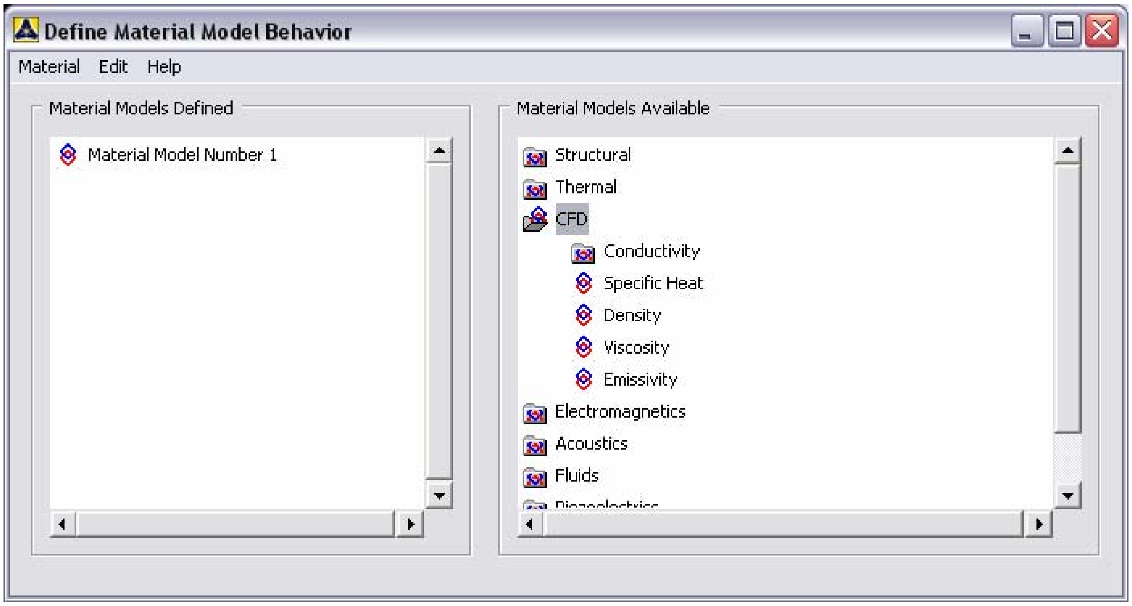




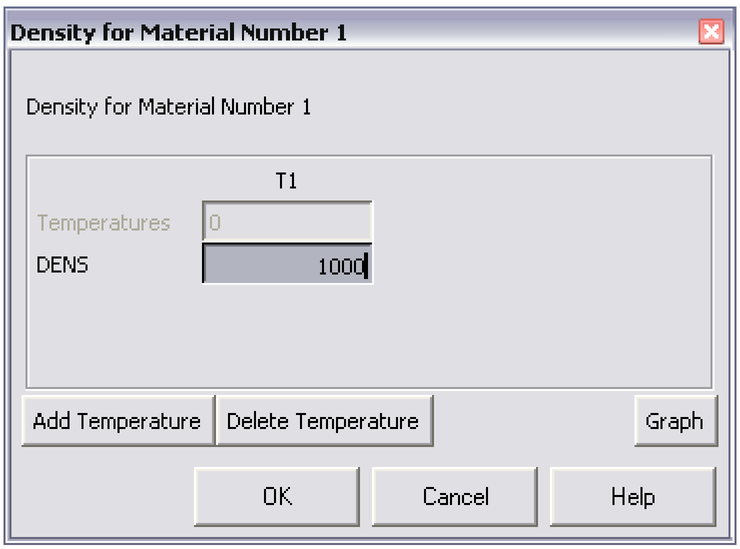
- Now we're ready to define the Material Properties

MATERIAL PROPERTIES

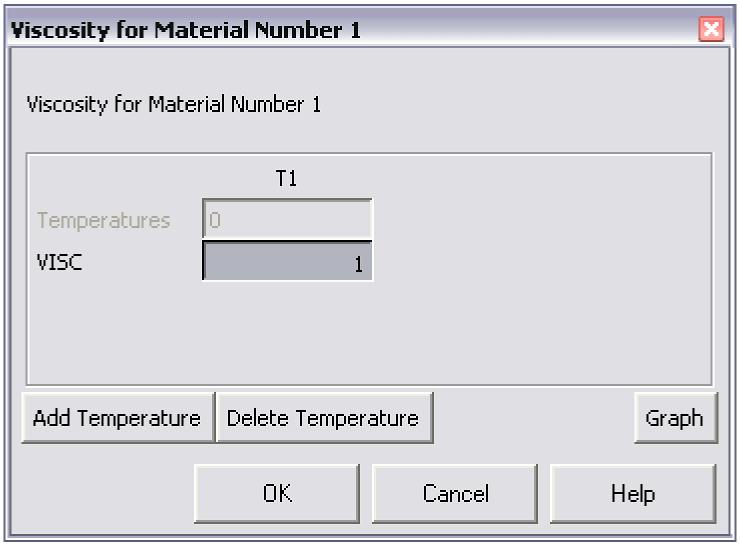
- Go to the ANSYS Main Menu
- Click **Preprocessor>Material Props>Material Models**. The following window will appear



- As displayed, choose **CFD>Density**. The following window appears.



- Fill in 1000 to set the density of Water. Click **OK.**
- Now choose **CFD>Viscosity.** The following window appears:

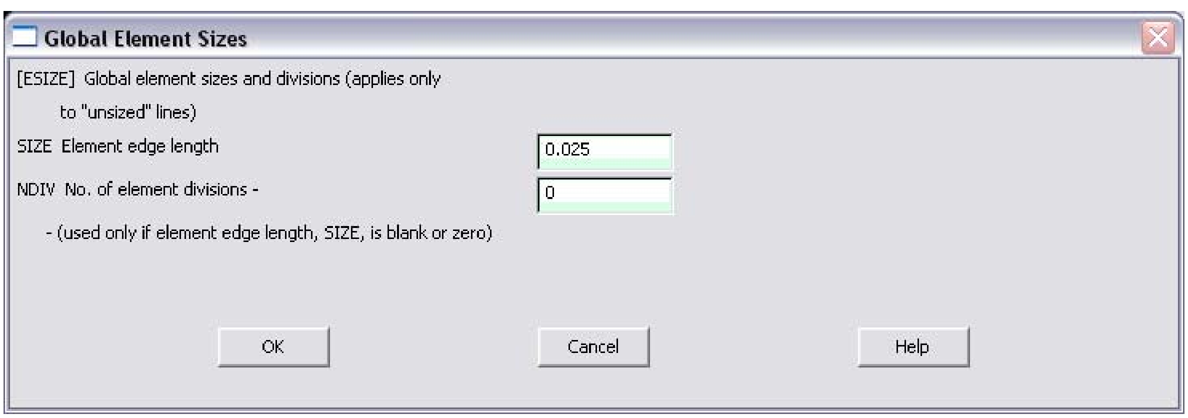


- Fill in 1 to set the viscosity of Water. Click **OK**
- Now the Material 1 has the properties defined in the above table so the Material Models window may be closed.

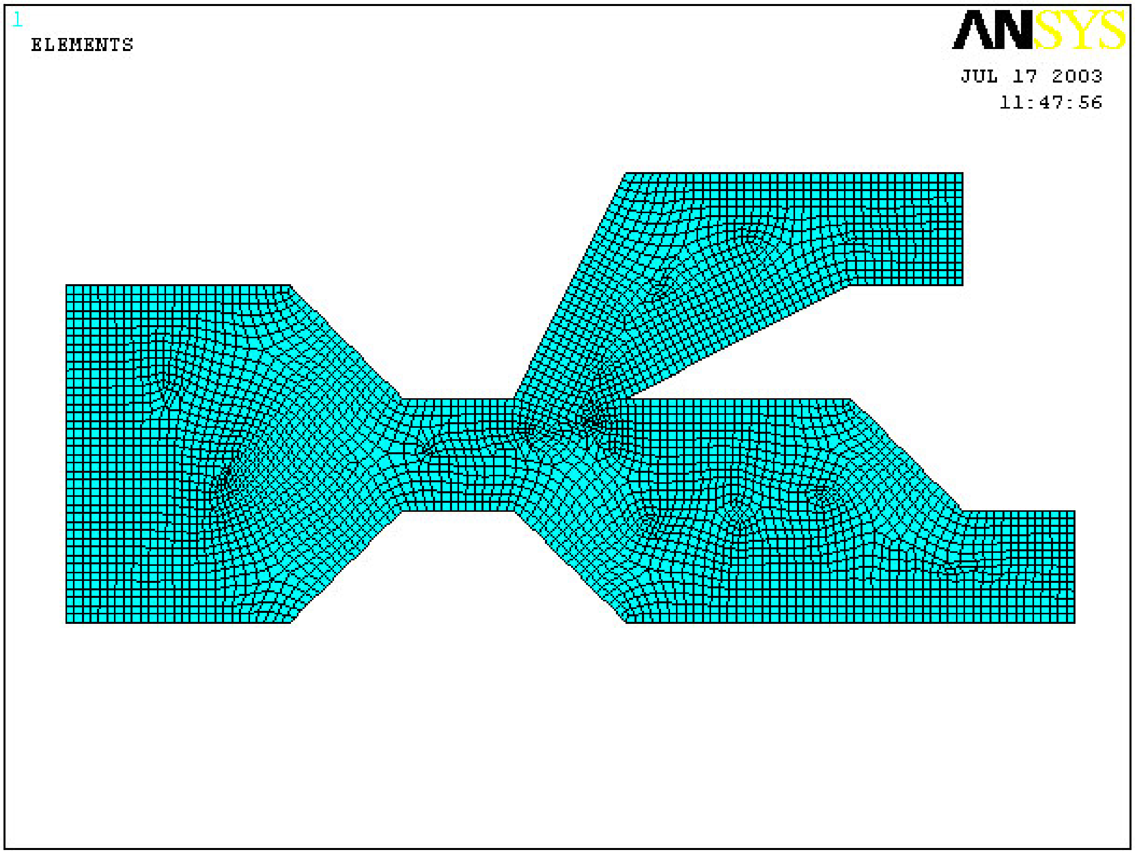
MESHING:

DIVIDING THE CHANNEL INTO ELEMENTS:

- Go to **Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Size**. In the window that comes up type **0.025** in the field for 'Element edge length'.

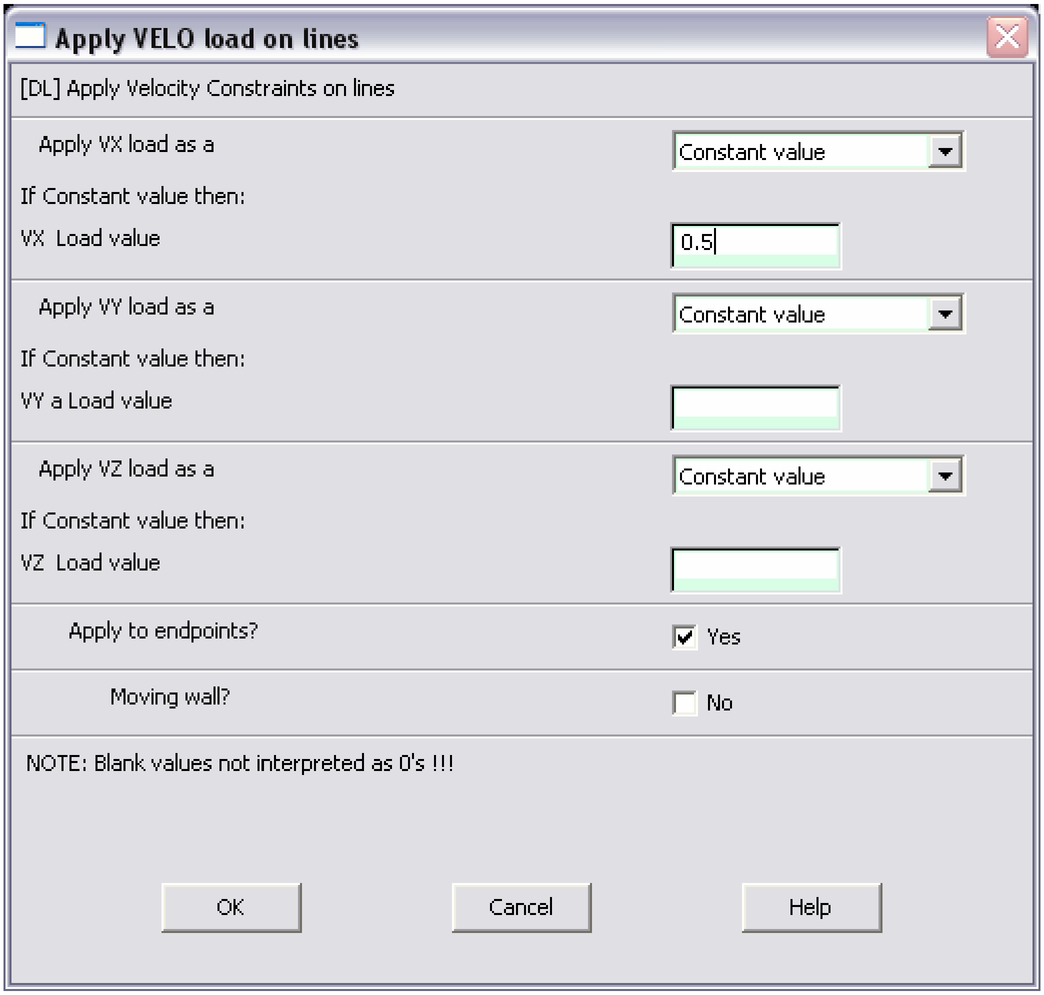


- Click on OK. Now when you mesh the figure ANSYS will automatically create a mesh, whose elements have a edge length of **0.025 m**.
- Now go to **Preprocessor>Meshing>Mesh>Areas>Free**. Click Pick All. The mesh will look like the following.

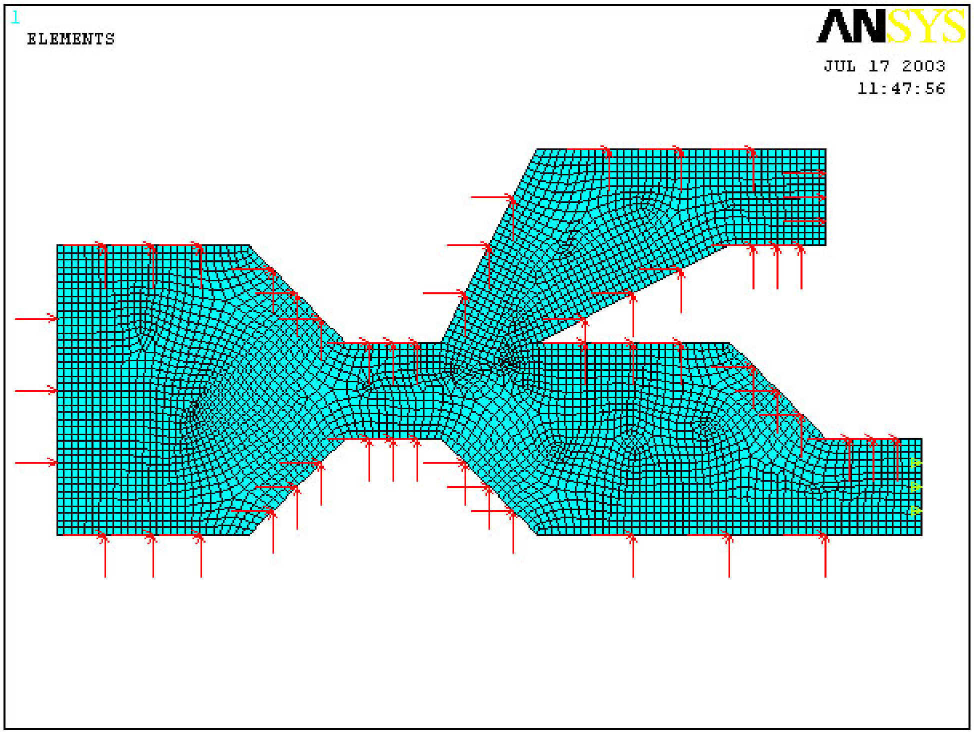


BOUNDARY CONDITIONS AND CONSTRAINTS

- Go to **Preprocessor>Loads>Define Loads>Apply>Fluid CFD>Velocity>On lines**. Pick the left edge of the block and Click **OK**. The following window comes up.



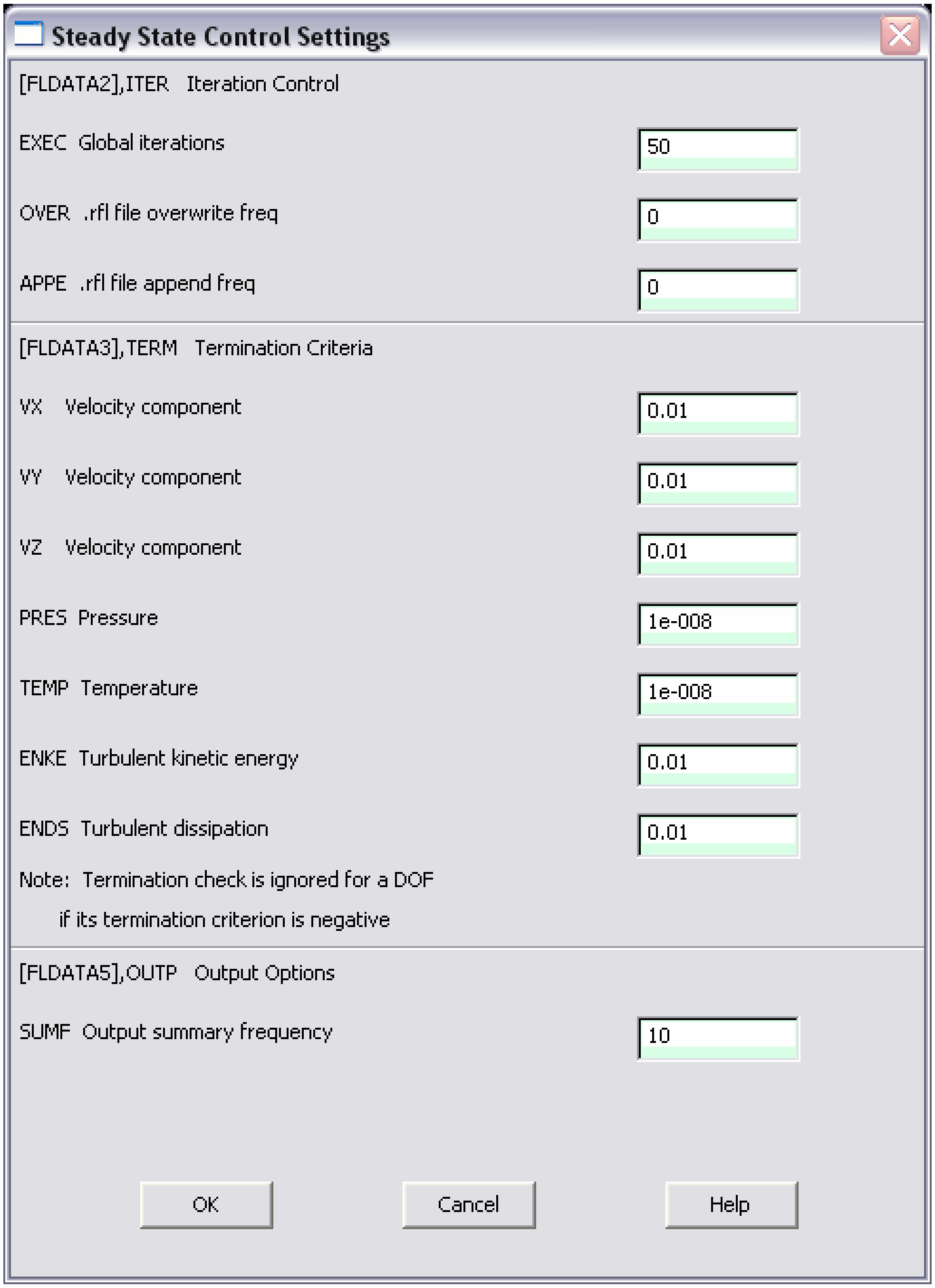
- Enter **0.5** in the VX value field and click OK. The 0.5 corresponds to the velocity of 0.5 meters per second of air flowing into the pipe grid.
- Repeat the above and set the **velocity into the upper pipe** as -1 meter/second. This is because the flow is traveling to the left, or the negative direction.
- Then, **set the Velocity to ZERO** along all of the edges of the pipes. This is because of the "No Slip Condition" (VX=VY=0 for all sides)
- Go to **Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid CFD>Pressure DOF>On Lines**. Pick the bottom pipe outlet and click **OK**.
- Once all the Boundary Conditions have been applied, the pipe grid will look like this:



- Now the Modeling of the problem is done.

SOLUTION

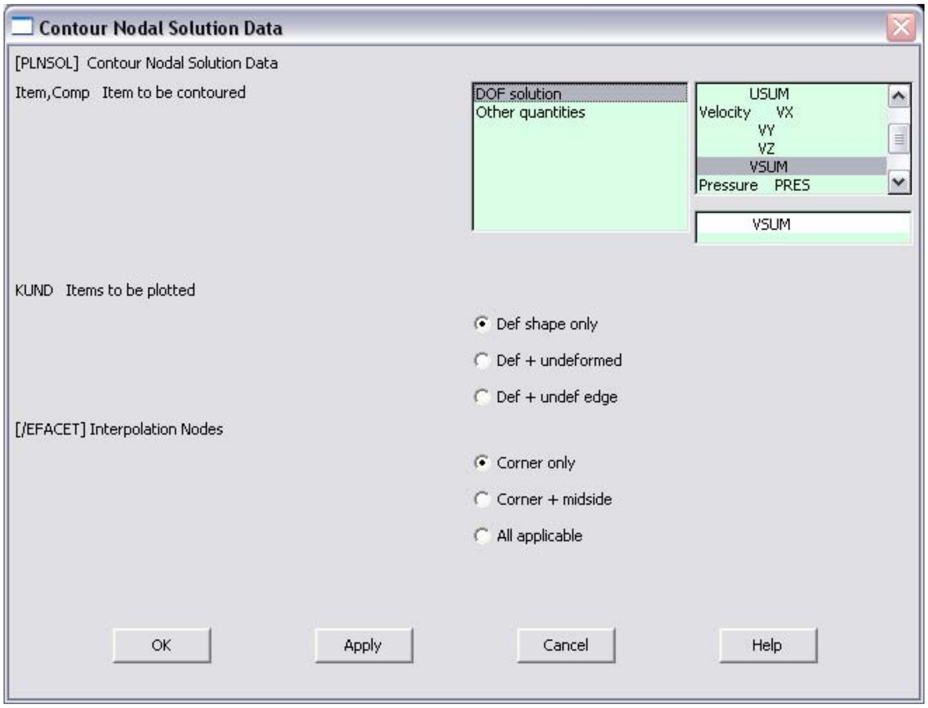
- Go to ANSYS **Main Menu>Solution>Flotran Set Up>Execution Ctrl**.
- The following window appears. Change the first input field value to **50**, as shown. No other changes are needed. Click **OK**.



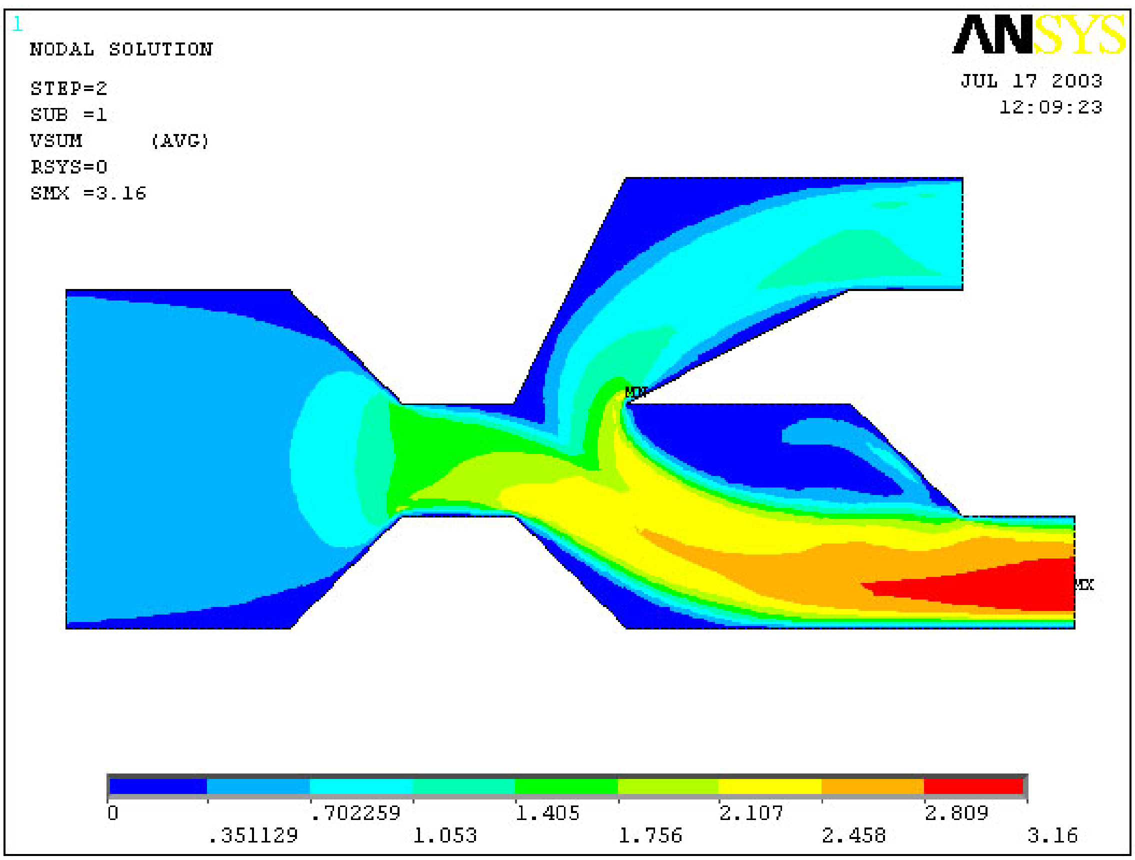
- Go to **Solution>Run FLOTRAN**.
- Wait for ANSYS to solve the problem.
- Click on OK and close the 'Information' window.

POST-PROCESSING

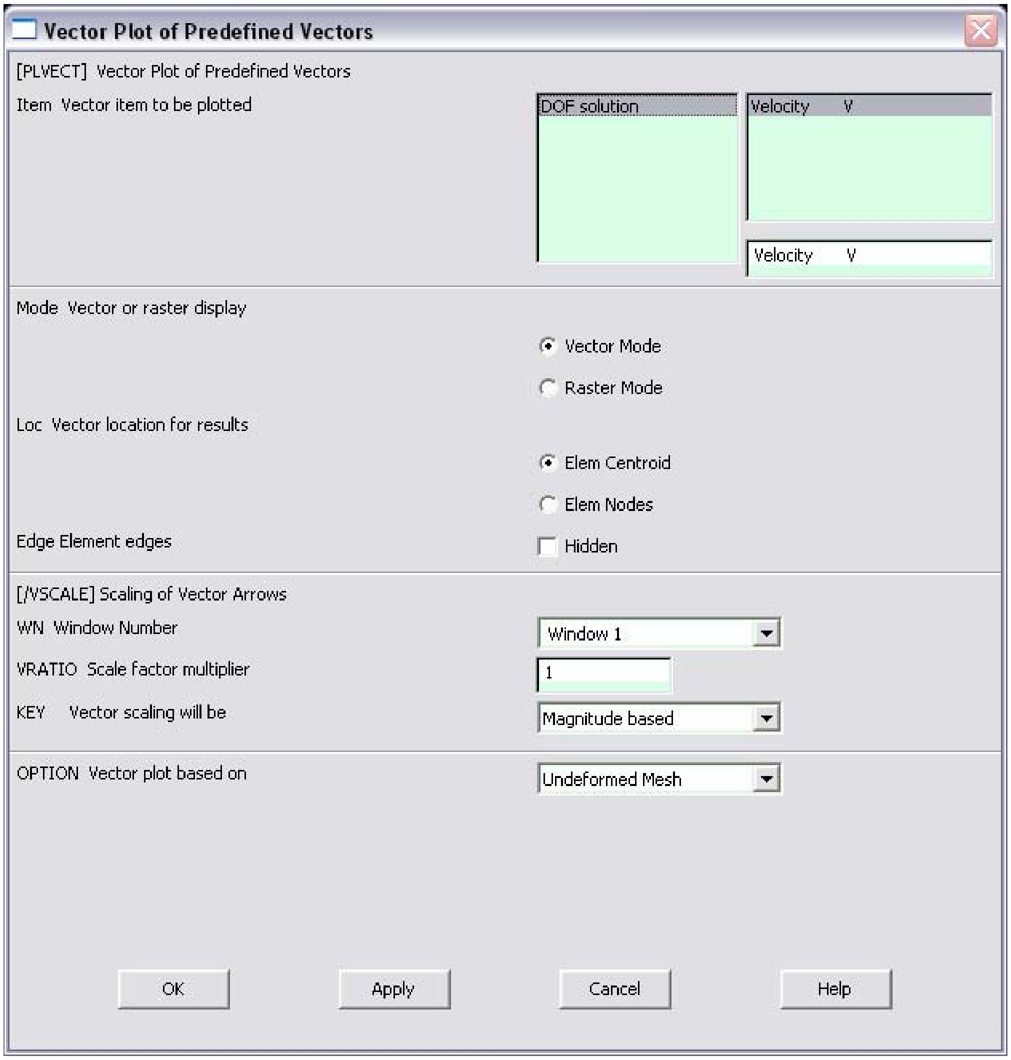
- Plotting the velocity distribution...
- Go to **General Postproc>Read Results>Last Set**.
- Then go to **General Postproc>Plot Results>Contour Plot>Nodal Solution**. The following window appears:



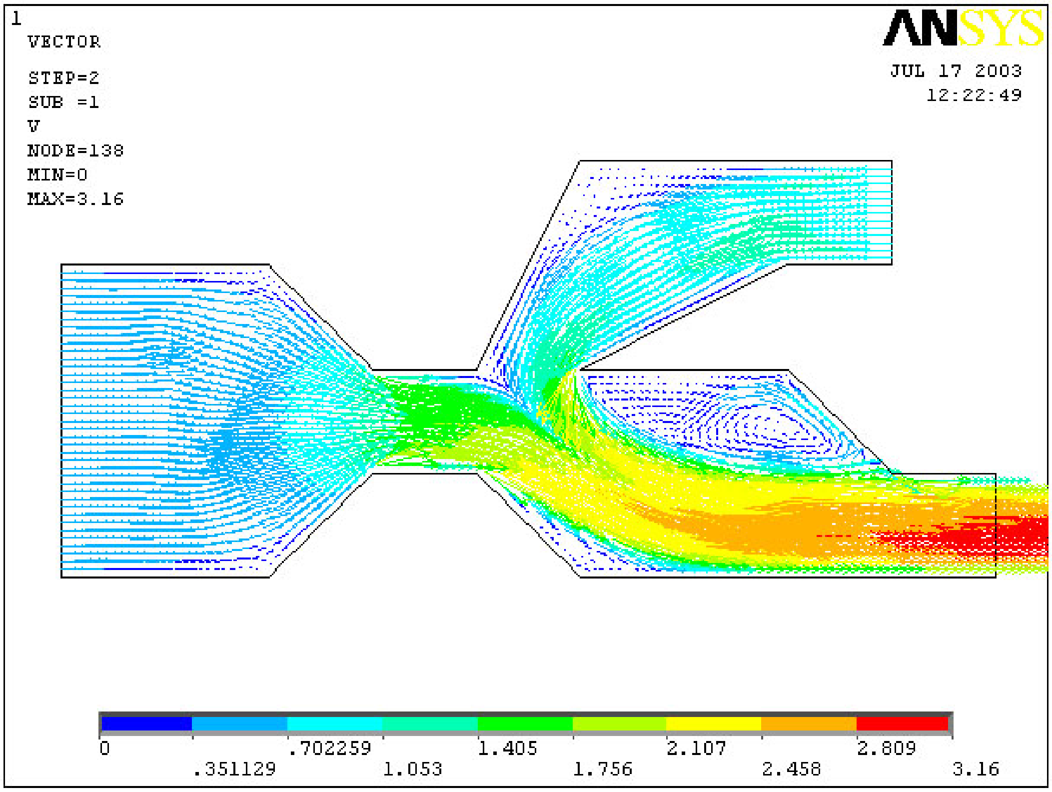
- Select **DOF Solution** and **Velocity VSUM** and Click **OK.**
- This is what the solution should look like:



- Next, go to **Main Menu>General Postproc>Plot Results>Vector Plot>Predefined.** The following window will appear:



- Select **OK** to accept the defaults. This will display the vector plot of the velocity gradient.


1
VECTOR
STEP=2
SUB =1
V
NODE=138
MIN=0
MAX=3.16
ANSYS
JUL 17 2003
12:22:49
0
.351129
.702259
1.053
1.405
1.756
2.107
2.458
2.809
3.16