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)
- 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.

STARTING ANSYS
- Click on ANSYS 11.0 in the programs menu.

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.
If the Cartesian grid does not appear, click on **Workplane>-Display Working Plane->** Use the grid to create the key points.
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:

![Library of Element Types](image)

- 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. The pressure window comes up. Be sure the Pressure applied on the lines are at Constant Value 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:
Under **Nodal Solution**, click on **DOF solution**
Select **Fluid Velocity** 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.