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.

 \blacksquare To graph the variation of velocity out the bottom pipe.

 \overrightarrow{P} 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.

Interactive 6.1	
	NSYS/University Low
Enable ANSYS Parallel Perfor	mance 🔲 Use ANSY'S Drop Test Module
Working directory	nd Settings\user\Desktop\ansyks
Graphics device name	win32
Initial jobname	file
MEMORY REQUESTED (megabyte	s) —
for Total Workspace	64
for Database	32
Read START.ANS file at start-up	? Yes 💌
Parameters to be defined (-par1 val1 -par2 val2)	
Language Selection	[english]
Execute a customized ANSYS ex	ecutable
Run Close	Reset Cancel About

MODELING THE STRUCTURE

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

WP Setting	s	
Cartesian		
C Polar		
🔿 Grid and	Triad	
💿 Grid Only	,	
🔿 Triad On	ly	
🔽 Enable 9	inap	
Snap Incr	0.33	
Snap Ang 5		
Spacing	0.33	
Minimum	0	
Maximum 3.66		
Tolerance 0.001		
OK	Apply	
Reset	Cancel	
Help		

- •
- 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:

Library of Element Types		×
Library of Element Types	Solid Shell ANSYS Fluid FLOTRAN CFD Magnetic Vector Scalar Interface Magnetic - Edge	
Element type reference number	1 Cancel Help	

- •
- 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**.

Fluid Properties	
[FLDATA12],PROP,DENS	
Density	Constant
[FLDATA13],VARY,DENS	
Allow density variations?	, □ No
[FLDATA12],PROP,VISC	
Viscosity	Constant
[FLDATA13],VARY,VISC	
Allow viscosity variations?	☐ No
[FLDATA12],PROP,COND	
Conductivity	Constant
[FLDATA13],VARY,COND	
Allow conductivity variations?	□ No
[FLDATA12],PROP,SPHT	
Specific heat	Constant 💌
[FLDATA13],VARY,SPHT	
OK Apply Cano	tel Help

CFD Flow Properties	×
Density property type CONSTANT	
Constant value	1000
Viscosity property type CONSTANT	
Constant value	1
-1: Implies conductivity need not be set	
Conductivity property type CONSTANT	
Constant value	-1
-1: Implies specific heat need not be set	
Specific Heat Property Type CONSTANT	
Constant value	-1
OK Cancel	Help
	, iciti

• 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

<u> </u>

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

Density for Material Number 1 🛛 🛛 🖾		
Density for Material Number 1		
T1		
Temperatures 0		
DENS 1000		
Add Temperature Delete Temperature	Graph	
ок са	ancel Help	

- ٠
- Fill in 1000 to set the density of Water. Click **OK**. Now choose **CFD>Viscosity**. The following window appears: •

Viscosity for Material Number 1 🛛 🛛			
Viscosity for Mate	erial Number 1		
	T1		
Temperatures	0		
VISC	1		
	1	. 1	
Add Temperatur	e Delete Temper	ature	Graph
	0K	Consul	
	OK	Cancel	Help

- 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'.

Global Element Sizes	$\overline{\mathbf{X}}$
[ESIZE] Global element sizes and divisions (applies only	
to "unsized" lines)	
SIZE Element edge length	0.025
NDIV No. of element divisions -	0
- (used only if element edge length, SIZE, is blank or zero)	
ок	Cancel Help

- 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.

Apply VELO load on lines	
[DL] Apply Velocity Constraints on lines	
Apply VX load as a	Constant value
If Constant value then:	
VX Load value	0.5
Apply VY load as a	Constant value
If Constant value then:	
VY a Load value	
Apply VZ load as a	Constant value
If Constant value then:	
VZ Load value	
Apply to endpoints?	Ves
Moving wall?	☐ No
NOTE: Blank values not interpreted as 0's !!!	
OK Cancel	Help

- 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**.

Steady State Control Settings		
[FLDATA2],ITER Iteration Control		
EXEC Global iterations	50	
OVER .rfl file overwrite freq	0	
APPE .rfl file append freq	0	
[FLDATA3], TERM Termination Criteria		
VX Velocity component	0.01	
VY Velocity component	0.01	
VZ Velocity component	0.01	
PRES Pressure	1e-008	
TEMP Temperature	1e-008	
ENKE Turbulent kinetic energy	0.01	
ENDS Turbulent dissipation	0.01	
Note: Termination check is ignored for a DOF		
if its termination criterion is negative		
[FLDATA5],OUTP Output Options		
SUMF Output summary frequency	10	
OK Cancel	Help	

- 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:

Contour Nodal Solution Data	×
[PLNSOL] Contour Nodal Solution Data	
Item,Comp Item to be contoured	DOF solution Other quantities Velocity VX VZ VSUM Pressure PRES VSUM
KUND Items to be plotted	
	Of shape only
	O Def + undeformed
	C Def + undef edge
[/EFACET] Interpolation Nodes	
	Corner only
	C Corner + midside
	C All applicable
OK Apply	Cancel Help

- 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:

Vector Plot of Predefined Vectors	×
[PLVECT] Vector Plot of Predefined Vectors	
Item Vector item to be plotted	DOF solution Velocity V
	Velocity V
Mode Vector or raster display	
	Vector Mode
	C Raster Mode
Loc Vector location for results	
	Elem Centroid
	C Elem Nodes
Edge Element edges	T Hidden
[/VSCALE] Scaling of Vector Arrows	
WN Window Number	Window 1
VRATIO Scale factor multiplier	1
KEY Vector scaling will be	Magnitude based
OPTION Vector plot based on	Undeformed Mesh
	Cancel Help

- Select \mathbf{OK} to accept the defaults. This will display the vector plot of the velocity gradient.

