Stevens Institute of Technology ME345 Modeling and Simulation – Prof. Frank Fisher

Analyzing Flow in a System of Pipes USING FLOTRAN (Carnegie Mellon) Last updated September 2008 – Allen Umali

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.



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

WP Setting	şs
 Cartesia Polar 	an
Grid and Grid On Grid On	d Triad ly nly
Snap Incr Snap Ang	5nap 0.33 5
Spacing Minimum Maximum	0.33
OK	Apply Cancel
Help	[

- 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	
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?	I No
[FLDATA12],PROP,VISC	
Viscosity	Constant
[FLDATA13],VARY,VISC	
Allow viscosity variations?	I No
[FLDATA12],PROP,COND	
Conductivity	Constant
[FLDATA13],VARY,COND	
Allow conductivity variations?	I No
[FLDATA12],PROP,SPHT	
Specific heat	Constant
[FLDATA13],VARY,SPHT	
	Cancel Help

CFD Flow Properties	$\overline{\times}$
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

• 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

🔺 Define Material Model Behavior		
Material Edit Help		
Material Models Defined	Material Models Available	
Material Model Number 1 ▲	Structural Struct	

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

al Number 1	_	
Number 1		
T1		
1		
1000		
Delete Temperature	f	Graph
Delece reinperacure]	- Graph
ок и	Cancel	Help
	al Number 1 Number 1 T1 1000 Delete Temperature	Al Number 1 Number 1 T1 1000 Delete Temperature

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

iscosity for Mate	rial Number 1	
Viscosity for Materi	al Number 1	
а // г	T1	
Temperatures	0	
VISC	1	
Add Temperature	Delete Temperature	Graph
	0K C	1 1 1-1-1-

- 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		
[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)	
10		
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.

[DI] 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?	I 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. The pressure window comes up. Be sure the Pressure applied on the lines are at Constant Value and click OK.

Apply PRES on lines				K
[DL] Apply PRES on lines as a		Constant v	/alue	
If Constant value then:				
PRES Pressure value				
Apply to endpoints?		🔽 Yes		
ОК	Apply	Cancel	Help	

• 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	X
[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	Help

- Go to Solution>Run FLOTRAN.
- Wait for ANSYS to solve the problem. ٠
- 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:

ontour Nodal Solution	Data		
Item to be contoured			
🚰 Favorites			<u>^</u>
💅 Nodal Solution			
💅 DOF Solution			
🍘 Pressure	,		
🎓 X-Compo	onent of fluid velocity		
🧭 Y-Compo	onent of fluid velocity		
🎓 Fluid vel	ocity		
🍘 Turbuler	nt kinetic energy		
🤗 Turbuler	t energy dissination		
	ic onorgy application		
Contraction of the second seco	N Quantities		
i Other FLOTRA	N Quantities		
💓 Tabaa 🧑 Other FLOTRA	N Quantities		
Viter FLOTRA	N Quantities		
Conter FLOTRA	N Quantities		T
Other FLOTRA	N Quantities		•
Other FLOTRA	N Quantities		•
Other FLOTRA	N Quantities		×
Other FLOTRA Jundisplaced shape key	Deformed shape only		
Other FLOTRA Other FLOTRA Judisplaced shape key Cale Factor	Deformed shape only Auto Calculated		▼ ▶
Control of the second of the s	N Quantities Deformed shape only Auto Calculated		

- 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:

Vector Plot of Predefined Vectors	
[PLVECT] Vector Plot of Predefined Vectors	
Item Vector item to be plotted	DOF solution Velocity V
Mode Vector or raster display	
	Vector Mode
	🔘 Raster Mode
Loc Vector location for results	
	Elem Centroid
	C Elem Nodes
Edge Element edges	☐ 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 **OK** to accept the defaults. This will display the vector plot of the velocity gradient.

