Turbulent Pipe Flow - Numerical Solution

Author: Rajesh Bhaskaran, Cornell University Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation Exercises Comments

Useful Information

Click here for the FLUENT 6.3.26 version.

Numerical Solution

We'll use second-order discretization for the momentum equation, as in the laminar pipe flow tutorial, and also for the turbulence kinetic energy equation which is part of the *k-epsilon* turbulence model.

Solution > Solution Methods

Change the Discretization for *Momentum*, *Turbulence Kinetic Energy* and *Turbulence Dissipation Rate* equations to *Second Order Upwind* (if you do not see all of the equations scroll down to see them).

Solution Methods					
Pr	essure-Velocity Coupling				
	Scheme				
	SIMPLE	*			
Sp	patial Discretization				
	Gradient		^		
	Least Squares Cell Based	*			
	Pressure				
	Standard	~			
	Momentum				
	Second Order Upwind	~			
	Turbulent Kinetic Energy				
	Second Order Upwind	~			
	Turbulent Dissipation Rate				
	Second Order Upwind	*	v		
Tr	ansient Formulation				
	~				
	Non-Iterative Time Advancement				
-	Frozen Flux Formulation				
0	Default				

The order of discretization that we just set refers to the convective terms in the equations; the discretization of the viscous terms is always second-order accurate in FLUENT. Second-order discretization generally yields better accuracy while first-order discretization yields more robust convergence. If the second-order scheme doesn't converge, you can try starting the iterations with the first-order scheme and switching to the second-order scheme after some iterations.

Set Convergence Criteria

Recall that FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We'll iterate the solution until the residual for each equation falls below 1e-6.

Solution > Monitors > Residuals, Statistic and Force Monitors

Double click on *Residuals*.Notice that *Convergence Criterion* has to be set for the *k* and *epsilon* equations in addition to the three equations in the last tutorial. Set the *Convergence Criterion* to be 1e-06 for all five equations being solved.

Select *Print to Console* and *Plot* under *Options* (these are the defaults). This will print as well plot the residuals as they are calculated which you will use to monitor convergence.

Residual Monitors					×
Options Print to Console Plot Window 1 Curves Axes Iterations to Plot 1000	Equations Continuity x-velocity y-velocity k epsilon	V V V V	9 9 9 9 9	1e-06 1e-06 1e-06 1e-06	
Iterations to Store	Residual Values Normalize Iterations 5 Scale	Cor ab	nvergence Criteria solute	on V	
OK Plot	Renormalize		हेel He	lp	

Click OK.

Set Initial Guess

We'll use an initial guess that is constant over the entire flow domain and equal to the values at the inlet:

Solution > Solution Initialization > Standard Initialization

Problem Setup	Solution Initialization
General Models Materials Phases Cell Zone Conditions Boundary Conditions	Initialization Methods Hybrid Initialization Standard Initialization Compute from
Mesh Interfaces Dynamic Mesh Reference Values Solution	Reference Frame Relative to Cell Zone Absolute
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Initial Values Gauge Pressure (pascal) 0 Axial Velocity (m/s)

In the Solution Initialization menu that comes up, choose inlet under Compute From. The Axial Velocity for all cells will be set to 1 m/s, the Radial Velocity to 0 m/s and the Gauge Pressure to 0 Pa. The Turbulence Kinetic Energy and Dissipation Rate(scroll down to see it) values are set from the prescribed values for the Turbulence Intensity and Hydraulic Diameter at the inlet.

Solution Initialization	
Compute from	
inlet	~
Reference Frame	
Relative to Cell Zone Absolute	
Initial Values	
Gauge Pressure (pascal)	^
0	
Axial Velocity (m/s)	
1	
Radial Velocity (m/s)	
0	
Turbulent Kinetic Energy (m2/s2)	
0.00015	
Turbulent Dissipation Rate (m2/s3)	
2.156208e-05	
	×
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	

Click Initialize (this is easy to overlook).

This completes the problem specification. Save your project.

Iterate Until Convergence

Solve for 700 iterations.

Solution > Run Calculation

In the Iterate menu that comes up, change the Number of Iterations to 700. Click Calculate.

The solution converges in a total of about 220 iterations. You may get a different number of iterations to convergence depending on your mesh and software version.



Click here to see a higher resolution image.

We need a larger number of iterations for convergence than in the laminar case since we have a finer mesh and are also solving additional equations from the turbulence model.

Setup Data Export

In addition to the standard data quantities, we would also like to view the results for the Skin Friction Coefficient. This quantity is not transferred to the postprocessor by default; so we have to do it manually.

File > Data File Quantities

Under Additional Quantities, select Skin Friction Coefficient, which should be roughly half way down. Your window should now look like this:

Data File Quantities		X					
Many quantities are available for postprocessing in external applications through the standard data file. To include additional quantities in the data file for postprocessing in external applications, select them below.							
Standard Quantities Pressure X Velocity Y Velocity Mass Flux Body Force Wall Velocity Original Wall Velocity Wall Shear Mach Number Boundary Heat Flux Boundary Heat Flux Boundary Heat Flux Turbulent Kinetic Energy Turbulent Dissipation Rate Wall Yplus Wall Yplus Utau Density Laminar Viscosity Turbulent Viscosity Znd Grad Bc Source Distance From Wall	Additional Quantities Turbulent Viscosity Ratio Wall Ystar Wall Ystar Wall Yplus Turbulent Reynolds Number (Re_y) Molecular Viscosity Wall Shear Stress Radial-Wall Shear Stress Radial-Wall Shear Stress Skin Friction Coefficient Cell Partition Cell Element Type Cell Zone Type Cell Zone Type Cell Zone Index Partition Neighbors Cell Weight X-Coordinate Y-Coordinate Radial Coordinate Radial Coordinate Face Area Magnitude X Face Area						
[<	F					
OK Cancel Help							

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules