Turbulent Pipe Flow (LES) - Numerical Solution

Author: Ranjith Tirunagari, 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

Numerical Solution

Solution Methods

Click on Solution Methods then click on Momentum and select Bounded Central Differencing as shown in the image below. Also choose, Second order for Pressure and Second Order Implicit under Transient Formulation. as

🧉 • 🖬 • 📧 🔞	S本QQ∥Q先膳•□•
Problem Setup General Models Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Presh Interfaces Dynamic Nesh Reference Yalues Solution Solution Solution Controls Monitors Solution Initialization Colo Jotion Articities	Solution Methods Pressure-Velocity Coupling Scheme SDPLE Spatial Discretization Gradent Least Squares Cell Based Pressure Pressure Second Order Nomentum Bounded Central Differencing
Results Graphics and Animations Pilots Reports	Transient Pormulation Second Order Implicit Non-Renative Time Advancement Prozen Flux Pormulation High Order Term Relaxation Default

Click Here for Higher Resolution

Set Initial Guess

Here, the flow field will be initialized to the values at the inlet. In order to carry out the initialization click on *Solution Initialization* then click on *Compute from* and select *inlet* as shown below.

💶 A:TurbuentPipeLES Par	rallel FLUENT@catslogin.tc.cornell.edu [3d, dp, pbns, l
File Mesh Define Solve	Adapt Surface Display Report Parallel View Help
🐸 • 🖬 • 🗟 🔞	S∲曳€∥∥奧洗腸・□・
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Calculation Controls Monitors Calculation Activities Run Calculation Results Graphics and Animations Flots Reports	Solution Initialization Initialization Methods C Hybrid Initialization Compute from al-zones Initial Values Gauge Pressure (pascal) 0 X Velocity (m/s) 0 Y Velocity (m/s) 0 Z Velocity (m/s) 0
	Reset DPM Sources Reset Statistics

Click Here for Higher Resolution

Then click *initialize* to initialize the domain with an initial guess. This completes the initialization.

Set Convergence Criteria

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. In order to specify the residual criteria (*Click*) *Monitors > Residuals > Edit...*, as shown in the image below.

💶 A:TurbuentPipeLES Pa	rallel FLUENT@catslogin.tc.cornell.edu [3d, dp, pbns
File Mesh Define Solve	Adapt Surface Display Report Parallel View Help
) 🐸 • 🖬 • 🗟 🔘	5.∲Q⊄∥ 奧先開・□・
Problem Setup General	Monitors Residuals, Statistic and Force Monitors
Models Materials Phases	Residuals - Print, Plot Statistic - Off
Cell Zone Conditions Boundary Conditions Mesh Interfaces	
Reference Values Solution	Create V EdR Delete Surface Monitors
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Due Controline	
Results	Create Edit Delete
Graphics and Animations Plots Reports	
	Create Edt Delete
	Нер

Click Here for Higher Resolution

Next, change the residual under Convergence Criterion for continuity, x-velocity, and y-velocity, all to 1e-6, as can be seen below.

Options	Equations				
Print to Console	Residual	Monitor C	heck Converge	ence Absolute Criteria	1
Plot	continuity	9	1	1e-06	
Window	x-velocity	4	2	1e-06	
Iterations to Plot	y-velocity	9	1	1e-06	
1000	z-velocity	9	V	1e-06	
	Residual Values			Convergence G	Rerion
1000	☐ Normalize		Iterations	absolute	-
	Compute L	ocal Scale			

Click Here for Higher Resolution

Lastly, click OK to close the Residual Monitors menu.

Execute Calculation

Note that, we have to run the transient simulation to a statistically stationary state and then collect statistics. Click on *Run Calculation* then choose the *Tim e Step Size(s)* as 1e-05 and the *Number of Time Steps* as 20000. Also choose *Extrapolate Variables* option (refer to FLUENT documentation for information) and leave the *Max Iterations/Time Step* as 20 (default). The corresponding image is shown below.

Problem Setup	Run Calculation				
General Models	Check Case	Preview Mesh Motion			
Materias Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Deference Valuer	Fixed	Time Step Size (s)			
	Settings	Number of Time Steps			
sourcion Solution Methods Solution Conkrols Monitors Solution Initialization Calculation Activities Run Calculation	Extrapolate Variables Data Sampling for Tim Sampling Interval I Time Sampled (e Statistics Sampling Options			
Results Graphics and Animations Plots	Max Iterations/Time Step	Reporting Interval			
Reports	Profile Update Interval	+			
	-	-			

Click Here for Higher Resolution

As a safeguard save the project now. Now, click on *Calculate* in order to run the calculation. The residuals for each iteration are printed out as well as plotted in the graphics window as they are calculated. The residuals decrease during the inner iterations and jump again when we advance by one time step as shown in image below.

From the above inputs, it is clear that 20 inner iterations are carried out to advance the simulation by one time step and we requested for 20,000 time steps. So, typically these simulations are very long even if we use 4 processors. To reduce the simulation time one can opt to High Performance Computing (HPC) resources. The calculations performed for this tutorial are based on the simulations run on an HPC cluster using 48 cores.



Click Here for Higher Resolution

In this simulation, it is verified that the statistically stationary state is reached by advancing the simulation from 0s to 0.1s in physical time i.e. using only 10,000 time steps with 1e-05s time step size with 20 inner iterations. When statistically stationary state is reached, statistics have to be collected by advancing the physical time from 0.1s to atleast 0.15s. Use 5,000 time steps with 1e-05 time step size with 20 inner iterations. This is done as shown in the figure below. Select the *Data Sampling for Time Statistics* to start collecting the statistics. You will see the physical time for which the statistics have been collected across *Time Sampled* (s).



Click Here for Higher Resolution

Collect the statistics for 0.05s (this corresponds to marching from 0.1s to 0.15s in time) as shown in the above figure. At this point save the project once again.

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules