Turbulent Pipe Flow (LES) - Physics Setup

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

Physics Setup

Regardless of whether you downloaded the mesh and geometry files or if you created them yourself, you should have checkmarks to the right of *Geometry* and *Mesh*. Your current *Workbench Project Page* should look comparable to the following image.

•	A	
1	😨 Fluid Flow (FLUE	NT)
2	🔞 Geometry	 Image: A second s
3	🧼 Mesh	× .
4	🍓 Setup	? ,
5	💼 Solution	2
6	🥪 Results	? .
	TurbuentPipeLE	5

A question mark should appear to the right of the Setup cell. This indicates that the Setup process has not yet been completed. This means that the mesh and the geometry data need to be read into FLUENT.

Launch Fluent

Double click on **Setup** in the **Workbench Project Page** which will bring up the **FLUENT Launcher**. When the **FLUENT Launcher** appears change the Options to "Double Precision", and Processing Options to "Parallel (Local Machine)" with Number of Processes equal to "4" or to the available number of processors at your end. Click **OK** as shown below.

er renter i sus sha
FLUENT Launche
Options
P Double Precision
Use Job Scheduler
Use Remote Linux Nodes
C Serial
(Parallel (Local Machine)
Number of Processes

Higher Resolution Image

Twiddle your thumbs a bit while the FLUENT interface starts up. This is where we'll specify the governing equations and boundary conditions for our problem. On the left-hand side of the FLUENT interface, we see various items listed under *Problem Setup*. and *Solution*. We will work from top to bottom of both these items to setup the physics of our boundary-value problem. On the right hand side, we have the *Graphics* pane and, below that, the *Commanc* pane.

Check and Display Mesh

First, the mesh will be checked to verify that it has been properly imported from *Workbench*. (*Click*) *Mesh* > *Check* and make sure that the minimum volume is positive. It is a good practice to check if x/y/z - domain extents are according to the dimensions given in the problem specification.

In order to obtain the statistics about the mesh (Click) Mesh > Info > Size, as shown in the image below.

🔁 A:	Turbu	entPipe	LES Pai	allel FL	UENT	l@ca	itslogin	.tc.cornel
File	Mesh	Define	Solve	Adapt	Surf	face	Display	Report
	Che	eck				ب	* 6	a) ⊻t El
	Info)			•	Q	uality	
Prot	Poly	vhedra			•	Si	ize	
						M	lemory U	sage
M.	Mer	rge				Z	ones	
M.	Sep	arate			►	P	artitions	i i
Pł	Fus	e			-			
G	Zon	e			►	<u> </u>		
Bo	Rep	blace						
M	Bec	rdor						
		bruer				h-Ba	red	Velocity Fo
Re	Sca	le				-Bas	ed .	C Relativ
Solu	Tra	nslate				1.00	00	- Rolden
So	Rot	ate						
So		He / C	_					
		ootn/Swa	p			ht		
	Rec	orded Me	esh Ope	rations				

Then, you should obtain the following output in the *Command* pane.

Mesh Si	ize			
Level	Cells	Faces	Nodes	Partitions
0	843380	1978811	332780	4
1 cell	L zone, 4	face zon	es.	

In order to bring up the display options (Click) General > Mesh > Display. Make sure you have inlet, outlet, pipewall and interior-solid in Surfaces as shown in the figure below.

💶 Mesh Display	×	
Options Edge Type Surfaces	<u>=</u> =	1: Mesh
Image: Second secon		
Shrink Factor Feature Angle		
Surface Name Pattern Match Surface Types	= =	
Outline Interior axis clip-surf exhaust-fan fan	×	F
Display Colors Close	Help	Mesh

Please review the "Laminar Pipe Flow" tutorial to understand how to rotate, zoom-in and zoom-out the geometry in the Graphics Window.

Define Solver Properties

In this section the various solver properties will be specified in order to obtain the proper solution. On the left side of the window (*Click*) *Problem Setup*> *General.* Make sure that *Pressure-Based* is selected under *Type* and *Transient* is selected under *Time* in the *Solver* section. Note: LES is a transient simulation where the solution is marched in time.

💶 A:TurbuentPipeLES P	arallel FLUENT@catslogin.tc.cornell.edu [3d, dp, pbns				
File Mesh Define Solve	e Adapt Surface Display Report Parallel View Hel				
	ऽ⊹҇҇€⋞∥ℚӼ╠∙□᠇				
Problem Setup	General				
General	Mesh				
Models Materials	Scale Check Report Quality				
Phases Cell Zone Conditions	Display				
Boundary Conditions Mesh Interfaces	Solver				
Dynamic Mesh Reference Values	Type Velocity Formulation Pressure-Based Absolute				
Solution	C Density-Based C Relative				
Solution Methods Solution Controls Monitors Solution Initialization	Time © Steady © Transient				
Calculation Activities Run Calculation	Gravity Units				

Next, the Viscous Model parameters will be specified. In order to open the Viscous Model Options (Click) Problem Setup > Models > Viscous - Laminar > Edit.... Click Large Eddy Simulation under Model and WMLES under Subgrid-Scale Model. Click OK.

1odel	User-Defined Functions
C Inviscid	Subgrid-Scale Turbulent Viscosity
🔿 Laminar	none
😳 Spalart-Allmaras (1 eqn)	
🔘 k-epsilon (2 eqn)	
🔆 k-omega (2 eqn)	
Transition k-kl-omega (3 eqn)	
C Transition SST (4 eqn)	
C Reynolds Stress (7 eqn)	
C Dehead Eddy Circulation (SAS)	
 Detached Eddy Simulation (DES) Laves Eddy Simulation (DES) 	
Carge Edgy Simulation (LES)	
Subgrid-Scale Model	
C Smagorinsky-Lilly	
C WALE	
• WMLES	
C Kinetic-Energy Transport	

An *Information* box will appear as shown below, click *OK*. Basically, FLUENT switches the discretization scheme for momentum equation to *Bounded Central-Differencing*. It also urges to change the order to *Second Order Implicit* for *Transient Formulation* in the *Solution Methods*, which we will do in the later stages.



For incompressible flows, the energy equation is decoupled from the continuity and the momentum equations. So the energy equation is not solved. Make sure that *Energy* is set to *Off* in *Problem Setup > Models > Energy*.

Define Material Properties

Now, the properties of the fluid that is being modeled will be specified. The properties of the fluid were specified in the Problem Specification section. In order to create a new fluid (*Click*) Problem Setup > Materials > Fluid > Create/Edit... as shown in the image below.

File Mesh Define Solve	eGrid Parallel FLUENT@catslogin.tc.cornell.edu [3d, Adapt Surface Display Report Parallel View Hel
) 😂 • 🛃 • 💽 🙆 📗	ऽфℚጚ∥ℚӼ閒・□・
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 Solution Initialization Calculation Activities Run Calculation	Materials Fluid air Solid aluminum
Results Graphics and Animations Plots Reports	Create/Edit Delete

In the Create/Edit Materials menu set the Density to 1.331 kg/m^3 (constant) and set the Viscosity to 2.34e-05 kg/(ms) (constant) as shown in the image below.

Create/Edit Ma	terials			
Name at		Material Type		Order Materials by
Chemical Portula		FLUENT Fluid Materials	د 	C Chemical Portuals
		Moture .	-	User-Defined Database
Properties		norm	<u>*</u>	
Density (kg/m3)	constant	• E8	-	
	1.331			
Vacanty (kg/m-a)	constant	• E0		
	2.34e-05			
J			J	
	Change/C	reate Delete Close	Help	

Higher Resolution Image

Click Change/Create. Close the window.

Define Boundary Conditions

At this point the boundary conditions for the three Named Selections will be specified.

Inlet Boundary Condition

In order to start the process (Click) Problem Setup > Boundary Conditions > inlet > Edit... as shown in the following image.



Higher resolution Image

Note that the *Boundary Condition Type* should have been automatically set to *velocity-inlet*. Now, the velocity at the *inlet* will be specified. In the *Velocit y Inlet* menu set the *Velocity Specification Method* to *Magnitude, Normal to Boundary*, set the *Velocity Magnitude (m/s)* to 6.58 m/s and set the *Fluct uation Velocity Algorithm* to *Spectral Synthesizer* (this is needed to fluctuate the velocity at the inlet). Also set the *Turbulence Specification Method* to *Intensity and Hydraulic Diameter*. Set the value of *Turbulent Intensity (%)* to 10 % and *Hydraulic Diameter (m)* to 0.0127 m. Finally set the *Reynolds-Stress Specification Method* to *K or Turbulent Intensity* as shown below. Click *OK* to close the *Velocity Inlet* menu.

Pelocity Inlet				×
Zone Name				
inlet				
Momentum Thermal Radiation Species	DPM Multip	bhase UDS	1	
Velocity Specification Method	Magnitude, Norr	nal to Boundary	γ	•
Reference Frame	Absolute			•
Velocity Magnitude (m/s)	6.58	Cor	nstant	•
Supersonic/Initial Gauge Pressure (pascal)	0	Cor	nstant	•
Fluctuating Velocity Algorithm	Spectral Synthe	sizer		•
Turbulence				
Specification Method Ir	itensity and Hyd	Iraulic Diameter		•
	Turbulent Ir	ntensity (%)	10	P[
	Hydraulic (Diameter (m)	0.0127	
Reynolds-Stress Specification Method K	or Turbulent Int	ensity		-
OK	Cancel	Help		

Outlet Boundary Condition

First, select outlet in the Boundary Conditions menu, as shown below.

🐸 • 🖬 • 🖬 🛞	\$ 秦風風∥風久陽•□•
Problem Setup General Models Habriale Habriale Habria Bondowy Considerer Bondowy Considerer Habriale Habrians Solution Solution Controls Solution Controls Solution Controls Solution Controls Monitors Solution Controls Monitors	Boundary Conditions Zone Inter- Interior-solid Codds ppercel
Results	1
Graphics and Animations Plots Reports	Friest Type 10 Friedure 1 Foresure-outlet 1 Fore

Higher Resolution Image

As can be seen in the image above the *Type* should have been automatically set to *pressure-outlet*. If the *Type* is not set to *pressure-outlet*, then set it to *pressure-outlet*. Now, no further changes are needed for the *outlet* boundary condition.

Pipe Wall Boundary Condition

First, select *pipewall* in the *Boundary Conditions* menu, as shown below.

AlTurbuentPipeLES Pa File Mesh Define Solve	rallel FLUENT@catslogin.tc.comell.edu [34, dp, pbrs Adapt Surface Deplay Report Parallel Verv Help
🖉 • 🖬 • 🕥 🛞	5∻QQ∥∥&洗腸・□・
Problem Setup General Nodels Phaterals Cellizare Conditions Bostofic Conditions Dynamic (Mech Partinemes Values Solution Solution Controls Membran Solution Initialization Calulation Activities Nam Calulation	Boundary Conditions Zone International International Control Control Control
Results Graphics and Animations Plats Reports	Finoc Type ID Finoture V wal V 6 Edit. Copy Profiles Parameters Dipley Meh Privide Conditions Highlight Zone

Higher Resolution Image

As can be seen in the image above the *Type* should have been automatically set to *wall*. If the *Type* is not set to *wall*, then set it to *wall*. Now, no further changes are needed for the *pipe_wall* boundary condition. Also make sure that the boundary condition *Type* for interior.

It is a good practice to change the *Reference Values* now, these values can be useful when we are postprocessing results later on. (*Click*)Problem Setup > Reference Values. Select Compute from as inlet.

Save

In order to save your work (Click)File > Save Project as shown in the image below.

22 A	:Turbu	entPipe	LES Par
File	Mesh	Define	Solve
Refresh Input Data			
Save Project			
Read			•
Write			•
Import			•
Export			•
Solution Files			
Interpolate			
EM Mapping			•
F:	FSI Mapping		
Sa	ave Pict	ure	
Data File Quantities,			
Close FLUENT			

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules