Steady Flow Past a Cylinder - Physics Setup

Authors: John Singleton and 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

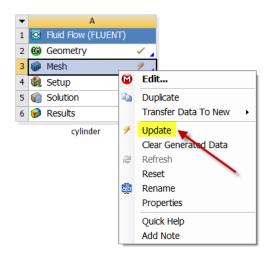
Comments
```

Physics Setup

Your workbench project should look like this.

•		А		
1	3	Fluid Flow (FLUENT)		
2	00	Geometry	× .	4
3	۱	Mesh	1	4
4		Cotup		
4		Setup	, 1	4
4 5		Solution	?	
-			2	4

We are done with all the meshing steps but for some reason, a tick mark doesn't appear next to Mesh in the project page. To get the tick mark next to mesh , right-click on it and select Update as shown below.



Launch Fluent

(Double Click) Setup in the Workbench Project Page.

When the *FLUENT Launcher* appears change options to "Double Precision", and then click *OK* as shown below. The Double Precision option is used to select the double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision, but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.

💶 Fluent Launcher (Setting Edit Only)			
ANSYS	Fluent Launcher		
Dimension	Options Double Precision Processing Options Serial Parallel		
💽 Show More Options			
OK <u>C</u> ancel <u>H</u> elp •			

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 boundary-value problem. On the left-hand side of the FLUENT interface, we see various items listed under *Problem Setup*. We will work from top to bottom of the *Problem Setup* items to setup the physics of our boundary-value problem. On the right hand side, we have the *Graphics* pane and, below that, the *Command* pane.

Check Mesh

(Click) Info > Size

You should now have an output in the *command* pane stating that there are 18,432 cells.

Eile Mesh Define Solve Adapt Surface Display Report Parallel Vie Mesh Info I	A:cylinder Fluent [2d, dp, pbns, lam] [ANSYS Academic Teaching Advanced						
Info Quality Mesh Polyhedra Me Polyhedra Solut Merge Solut Merge Solut Merge Mo Fuse Max Partitions Phi Zone Cel Replace Boo Reorder Me Scale Translate 2D Space Solut Rotate	<u>F</u> ile	Mesh Define Solve Adapt Su	S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> ie				
Sol Mo Sol Recorded Mesh Operations	Mest Me Solut Ge Mo Mo Mo Mo Ret Solut Solut Solut	Check Info Polyhedra Merge Separate Fuse Zone Replace Reorder Scale Translate Rotate Smooth/Swap	文 『唐 □ Quality Size Memory Usage Zones Partitions Velocity Formulation ④ Absolute ⑦ Relative 2D Space ④ Planar △ Axisymmetric △ Axisymmetric Swirl				

(Click) Check > Perform Mesh Check

You should see no errors in the *command* pane.

Specify Material Properties

Solution Setup > Materials > Fluid > Create/Edit.... Then set the Density to 1 kg/m^3 and set Viscosity to 0.05 kg/m*s. Click Change/Create then click Close.

Eile Mesh Define Solve Adapt Surface Display Report Meshing Meshing Image: Im	A:cylinder Fluent [2d, dp, pbns, lam] [ANSYS Academic Teach				
Meshing Materials Mesh Generation Materials Solution Setup General Models Materials Materials Fluid Materials Solid Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Dynamic Mesh Reference Values Solution Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	<u>File Mesh Define Solv</u>	e <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport			
Mesh Generation Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	i 📖 i 📂 🖌 🖌 🐼 🞯	ら 🕀 🗨 🗶 🖉 🔍 🍭 🏷 📲 🖛 🗔 🖞			
General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	-				
Waterials Solid Models Materials Materials Solid Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Solution Setup	Fluid			
Created Editin Sciete	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 Results Graphics and Animations Plots	air Solid			

Boundary Conditions

FarField1

Solution Setup > Boundary Conditions > farfield1. Set Type to velocity-inlet. Click Edit.... Set Velocity Specification Method to Components, set X-Velocity to 1 m/s, and set Y-Velocity to 0 m/s.

Meshing	Boundary Conditions		
Mesh Generation	Zone		
Solution Setup	cylinderwall		
General	farfield1 farfield2		
Models	interior-surface_body		
Materials Phases	surface_body		
Cell Zone Conditions			
Boundary Conditions			
Mesh Interfaces			
Dynamic Mesh Reference Values			
Solution			
Solution Methods			
Solution Controls			
Monitors			
Solution Initialization Calculation Activities			
Run Calculation	Phase Type ID		
Results	mixture velocity-inlet 6		
Graphics and Animations	axis		
Plots	Edit		
Reports	Parameters Op intake-fan		
	Display Mesh		
	outflow		
	outlet-vent Help pressure-far-field		
	pressure-inlet		
	pressure-outlet symmetry		
	velocity-inlet wall		
	, YUM	X	
Velocity Inlet			
Zone Name			
farfield 1			
Manager	de la la classificada	l una l	
Momentum Thermal Radiation Species DPM Multiphase UDS			
Velocity Specification Method Components Reference Frame Absolute Supersense (Initial Caurae Brane trace (Specific))		▼	
Supersonic/Initial	Gauge Pressure (pascal) 0	constant 👻	
	X-Velocity (m/s) 1	constant 👻	
Y-Velocity (m/s)			
		constant 👻	
	OK Cancel H	elp	

FarField2

Solution Setup > Boundary Conditions > farfield2.. Set Type to pressure-outlet.

Note that pressures in the FLUENT interface are specified in terms of gauge values where: gauge pressure = absolute pressure - reference pressure. The reference pressure is 1 atm by default. So in our case the gauge pressure at the outlet is 0 atm which is also the default. The following video from the laminar pipe flow module in our free online course explains the advantages of working in terms of gauge pressures. The video discusses this in the context of laminar pipe flow but the same ideas apply for our cylinder flow too.

Cylinder Wall

Solution Setup > Boundary Conditions > cylinderwall. Set Type to wall.This is the default.

Reference Values

Solution Setup > Reference Values. Set the Density to 1 kg/m^3. The other default values will work for the purposes of this simulation.

Save Project

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules