FLUENT - Turbulent Flow Past a Sphere - Step 5

UNDER CONSTRUCTION

Author: Daniel Kantor and Andrew Einstein, Cornell University

Problem Specification
1. Create Geometry in GAMBIT
2. Mesh Geometry in GAMBIT
3. Specify Boundary Types in GAMBIT
4. Set Up Problem in FLUENT
5. Solve!
6. Analyze Results
7. Refine Mesh
Problem 1

Step 5: Solve!

We'll use a second-order discretization scheme.

Main Menu > Solve > Controls > Solution ...

Take a look at the options available. Under *Discretization*, set *Pressure* to *Standard*, set *Momentum* to *Second Order Upwind*, set *Turbulent Kinetic Energy* to *Second Order Upwind*, and set *Specific Dissipation Rate* to *Second Order Upwind*. All other values should remain at their default.

Solution Controls			×				
Equations	Under-Relaxation Factors						
Flow Turbulence	Pressure	0.3					
	Density	1					
	Body Forces	1					
	Momentum	9.7					
Pressure-Velocity Coupling	Discretization						
SIMPLE	Pressure	Standard +	-				
	Momentum	Second Order Upwind +					
	Turbulent Kinetic Energy	Second Order Upwind Second Order Upwind					
	Specific Dissipation Rate	Second Order Upwind •	w				
OK Default Cancel Help							

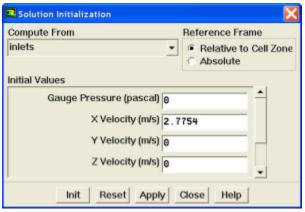
Click OK.

Set Initial Guess

Initialize the flow field to the values at the inlet:

Main Menu > Solve > Initialize > Initialize...

In the Solution Initialization menu that comes up, choose *inlet* under *Compute From*. The *X Velocity* for *all* cells will be set to 2.7754 m/s, the *Y Velocity* to 0 m/s and the *Gauge Pressure* to 0 Pa. These values have been taken from the inlet boundary condition.



Click Init. This completes the initialization. Close the window.

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.

Main Menu > Solve > Monitors > Residual...

Change the residual under Convergence Criterion for continuity, x-velocity, and y-velocity, all to 1e-6.

Also, under Options, select Plot. This will plot the residuals in the graphics window as they are calculated.

💶 Residual N	onitors						
Options	Storage	Plotting					
Print Plot	Iterations	000 +	Window 0				
	Normalization	Iterations 1000					
	Normali	Axes Curves					
	Convergence Criteri	ion					
	absolute	-					
Residual	Check Monitor Converge	Absolut Ince Criteria					
continuit	y 🛛 🖓	1e-06					
x-velocit	y v v	1e-06					
y-velocit	y 🛛 🖓	1e-06					
z-velocit	y 🔽 🔽	1e-06					
k	X	1e-06	•				
OK Plot Renorm Cancel Help							

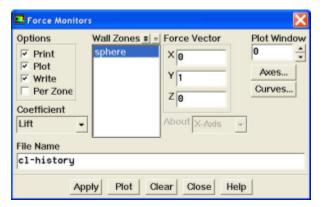
Click OK.

Monitor also the drag and lift coefficient on the sphere.

Main Menu > Solve > Monitors > Force...

Select Sphere under Wall Zones. Under Options, select Print, Plot and Write. Under Coefficient, choose Drag. Click Apply.

Choose Sphere under Wall Zones. Under Options, select Print, Plot and Write. Under Coefficient, choose Lift. Click Apply.



Setting Reference Values

To plot C $_{\rm d}$ and C $_{\rm l}$ we need to set the reference value.

Main Menu > Report > Reference Values...

Under Reference Values, change Area to 28.274, Density to 1.225, Velocity to 2.7754 and Viscosity to 1.7894E-05.

Reference Values					
Compute From					
-					
Reference Values					
Area (m2) 28.274					
Density (kg/m3) 1.225					
Enthalpy (j/kg)					
Length (m) 1					
Pressure (pascal)					
Temperature (k) 288.16					
Velocity (m/s) 2.7754					
Viscosity (kg/m-s) 1.7894e-05					
Ratio of Specific Heats 1,4					
Reference Zone					
•					
OK Cancel Help					

Click OK.

This completes the problem specification. Save your work:

Main Menu > File > Write > Case...

Type in SingleSphere.cas for *Case File*. Click *OK*. Check that the file has been created in your working directory. If you exit FLUENT now, you can retrieve all your work at any time by reading in this case file.

Iterate Until Convergence

Main Menu > Solve > Iterate ...

In the *Iterate Window* that comes up, change the *Time Step Size* to 1, change the *Number of Time Steps* to 100, and change *Max Iterations per Time Step* to 10. Click *Iterate*.

💶 Iterate	X					
Time						
Time Step Size (s) 1						
Number of Time Steps 100						
Time Stepping Method						
 Fixed Adaptive Variable 						
Options						
Data Sampling for Time Statistics						
Iteration						
Max Iterations per Time Step 10	*					
Reporting Interval 1	+					
UDF Profile Update Interval 1	÷					
Iterate Apply Close Help						

The residuals and drag coefficients for each iteration are printed out as well as plotted in the graphics window as they are calculated.

Save the solution to a data file:

Main Menu > File > Write > Data...

Enter SingleSphere.dat for *Data File* and click *OK*. Check that the file has been created in your working directory. You can retrieve the current solution from this data file at any time.

Go to Step 6: Analyze Results

See and rate the complete Learning Module

Go to all FLUENT Learning Modules