FLUENT - Turbulent Flow Past a Sphere - Step 6

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 6: Analyze Results

For all of our analysis we will be looking at the Sphere surface under Surfaces, unless otherwise noted.

Plot Velocity Vectors

Let's plot the velocity vectors obtained from the FLUENT solution.

Display > Vectors

Set the Scale to 1 and Skip to 0. Click Display.



If we look closely at the sphere we can start to see where the separation occurs.

Zoom in the cylinder using the middle mouse button.

Now, let's take a look at the Contour of Velocity magnitude around the sphere.

Display > Contours

Under Contours of, choose Velocity... and Velocity Magnitude. Select the Filled option. Increase the number of contour levels plotted: set Levels to 100.

Contours		X	
Options	Contours of		
Filled	Velocity	-	
Node Values Global Range	Velocity Magnitude		
Auto Range	Min (m/s) Max (m/s)		
Clip to Range	0 3.518425		
Draw Profiles	Surfaces		
Levels Setup 100 1 1 Surface Name Pattern	default-interior		
	inlets		
	outlet		
	sphere		
	Surface Types	==	
Match	axis	^	
	clip-surf	-	
	exhaust-fan		
	Itan	×	
Display Co	mpute Close Help		

Click Display.



We see the flow is mostly what we would expect in this case.

Now, let's take a look at the Contour of Turbulent Intensity around the sphere. This will give us a picture of the turbulence that is occurring around the sphere.

Display > Contours

Under Contours of, choose Turbulence... and Turbulent Intensity. Select the Filled option. Increase the number of contour levels plotted: set Levels to 100.

Click Display.



Pressure Coefficient

Pressure coefficient is a dimensionless parameter defined by the equation the reference pressure, and q_{ref} is the reference dynamic pressure defined by

Unable to find DVI conversion log file.

The reference pressure, density, and velocity are defined in the Reference Values panel in Step 5.

Let's plot pressure coefficient vs x-direction along the cylinder.

Plot > XY Plot ...

Change the Y Axis Function to Pressure..., followed by Pressure Coefficient. Then, select Sphere under Surfaces.



Click Plot.



We see that there is a lot of scatter in our data, so we will be creating a line along the sphere to try and get a better picture of the pressure coefficient. To accomplish this we will create a surface of Z-axis position zero, and plot the line $y^2+(x-12)^2-9$, which is the equation of a ring around the sphere in the x-y plane.

Surface > Iso-Surface

Under Surface of Constant choose Grid... and Z-Coordinate. Under Iso-Values input 0. This will create a plane in our flow in which the Z coordinate is zero everywhere.

where p is the static pressure, p_{ref} is

Unknown Attachment

Surface of Co	nstant	From	n Surface	=
Grid		defa inle	ault-interior ts	
Z-Coordinate		 outl 	et	
Min (m)	Max (m)	plar	ne_split	
-15	15	sph	ere	
lso-Values (m))	From	n Zones	
0		fluid	1	
•				
New Surface	Name			
z-coordinat	te-6			

Call this Zero_Plane and click Create.

Define > Custom Field Functions

Here we input our formula $y^2+(x-12)^2 - 9$ under *Definition*. To do use the *Field Functions* section and choose *Grid...* and choose *X-Coordinate* and *Y-Coordinate* for the "x" and "y" in the above formula.

y ^ 2	* (x	12)	2 - 1	9			
+	-	×	1	у'х	ABS	Select Operand Field Functions	from
INV	sin	005	tan	In	log10	Field Functions	
0	1	2	3	4	SORT	Grid	•
5	6	7	8	9	CEVC	X-Coordinate	-
1)	PI			DEL	Select	

Call this Ring and click Define.

Surface > Iso-Surface

Now under Surface of Constant choose Custom Field Functions and choose our function Ring. Keep the default Iso-Values to 0. Then, within From Surfaces select Zero_Plane.

💶 Iso-Surfa	ce			×
Surface of Custom Fie	Constant ald Functions	•	From Surface default-interior	==
ring Min 0	Max 0	•	outlet plane_split sphere zero_plane	
lso-Values Ø			From Zones fluid	
New Surfac	e Name			
0	reate Compute M	lanag	e Close Help	

Call this CpLine and click Create.

We have now created the necessary line around the sphere to view the data better. Follow the same steps as before to plot the Pressure Coefficient, except that under *Surfaces* choose *CpLine*.

We now see that the data is has less scatter (this effect, however, is far more significant on a more refined mesh).



Comparison

With our simulation data, we can now compare the Fluent with experimental data. Click HERE to download the experimental data. The two sets of data for Pressure Coefficients are shown here:



The Red dots are the experimental data (labeled "sphere1"), while the white dots are our data (labeled CPline).

Lets display the coefficient of drag:

Report > Forces

Make sure that under Force Vector the X has a 1 by it (this represents the drag on the sphere), and that under Wall Zones, sphere is highlighted.

Force Reports		X
Options	Force Vector	Wall Zones = =
Forces Moments Center of Pressure	×1 Ye	sphere
Genter of Treasure	Z	
Wall Name Pattern		
	Match	
Print Writ	te Close	Help

Click Print.

The two sets of data for drag coefficient are shown here:

Experimenta I	0.1
Simulation	0.448 5

Go to Step 7: Refine Mesh

See and rate the complete Learning Module

Go to all FLUENT Learning Modules