Fluent – Laminar flow over a sphere

Author: John M. Cimbala, Penn State University
Latest revision: 16 January 2008

Introduction and Instructions:
This learning module contains a procedure to solve for laminar flow around a sphere with the CFD program, Fluent. Minor modifications of this module enable one to solve for flow around other bodies as well. This set of instructions assumes that you have already run the Gambit program, and have generated a grid for the body. The file cylinder.msh is assumed to exist on the user's Fluent directory. You are referred to the companion learning module on laminar flow over a 2-D cylinder, since the procedure is basically identical except for a few minor changes as shown below.

Launch Fluent:

1. If Fluent is already running, with the 2-D cylinder case already loaded, this section and the next can be skipped. Instead, skip to the section below called **Define axisymmetry and the boundary conditions**, and then re-initialize the flow and start iterating.
2. Start Fluent, and select 2ddp, the two-dimensional, double precision option. Run. [*Note*: If you are using a Unix or Linux machine, go to the desired directory, and enter the command fluent 2ddp & (the & symbol lets Fluent run in background mode so that the Unix shell is still usable).] After a few seconds, the main Fluent window should appear on your screen.

Read the grid points and geometry of the flow domain:

1. The *same* grid that is used for the 2-D cylinder will be used for the axisymmetric case. Select File-Read-Case. Browse as necessary and select the file called laminar_cylinder.cas.gz).
2. OK. Fluent will read in the grid geometry, boundary conditions, etc. that were previously defined in the circular cylinder run, along with the extra rays we created, etc. Some information is displayed on the main screen. If all went well, it should give no errors, and the word Done should appear.

Define axisymmetry and the boundary conditions:

1. To change from 2-D to axisymmetric, Define-Models-Solver. Under *Space*, select Axisymmetric and OK.
2. Now the boundary conditions need to be specified. In the main *Fluent* window, click on Define-Boundary Conditions.
3. Select inlet, and Set. The *Velocity Specification Method* should be Magnitude and Direction. The *Velocity Magnitude* should still be 1.0 m/s, since that was used for the cylinder case. The default flow direction is in the axial direction, which is what is desired here, so OK.
4. The default boundary conditions for the wall and the pressure outlet are okay, so nothing needs to be done to those.
5. The 2-D symmetry boundary condition, however, needs to be changed to an axis boundary condition, so that the axisymmetric equations know where the axis of symmetry is defined. In the *Boundary Conditions* window, select symmetry. Change *Type* to axis. Yes.
6. In the *Axis* window that opens up, change the *Zone Name* to "axis". OK.
7. Close the Boundary Conditions window.

Initialize:

1. In the main *Fluent* window, Solve-Initialize-Initialize. The default initial values of velocity and gage pressure are all zero. These are good enough for this problem. Init and Close.

Complete the calculations:

1. From this point on, everything is identical to the 2-D cylinder case except for a couple things:
2. When writing the case and data files, choose a different file name, like "laminar_sphere.cas.gz" so that the cylinder data are not overwritten.
3. Re-define the reference area for calculation of drag coefficient; ***use the frontal area of the sphere***. *Note*: You do not need to be concerned with the factor of 2 – even though only a half-slice of the sphere is modeled, Fluent knows to integrate the entire 360^{o} around the axis.
4. Iterate as in the cylinder learning module. Make the same plots, calculate drag coefficient, etc. as in the cylinder case. The rays which were previously created should still be there for your use.