Fluent – Laminar flow over a cylinder

Author: John M. Cimbala, Penn State University
Latest revision: 29 January 2008

Introduction and Instructions:
This learning module contains a procedure to solve for laminar flow over a cylinder with the CFD program, Fluent. *Note*: This set of instructions assumes that you have already run Gambit, and has generated a grid for the flow. The file cylinder.msh is assumed to exist on your directory.

Log on and launch Fluent:
1. Log on to a computer which has access to the Fluent software.
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).]
3. After a few seconds, the main *Fluent* window should appear on your screen.

Read the grid points and geometry of the flow domain:
1. Select File-Read-Case. Browse as necessary to locate the mesh file cylinder.msh (or whatever you named it) that was previously created by Gambit. [On Windows machines, the default may be the C:\Temp directory.] Highlight this file (click on it), and OK. Fluent will read in the grid geometry and mesh. Some information is displayed on the main screen. If all is read well, it should give no errors, and the word "Done" should appear.
2. Check the validity of the grid: Grid-Check. If the grid is valid, no errors should appear. If there are errors, you may have done something wrong in the grid generation, and will have to go back and regenerate the grid.
3. Look at the grid to make sure it is correct. Display-Grid-Display. A new graphical display window opens up showing the grid. If this window is too big or too small, re-scale it by dragging the edges of the window. It is best if the graphical display window is small enough that both it and the *Fluent* window are visible simultaneously. The *Fluent* window and/or the graphical display window may need to be moved to accomplish this.
4. The graphical display can be zoomed-in or zoomed-out with the *middle* mouse button. If you start on the *lower left* and draw a rectangle with the middle mouse button towards the upper right, the display will zoom in on what is included in the rectangle. Meanwhile, the *left* mouse button can be used to drag the image to a new location. If you draw a rectangle *backwards* with the middle mouse button, i.e., from right to the left, it will zoom out. Zoom in if necessary until the grid is shown nicely in the window. Close the *Grid Display* window; the display itself will remain.

Generate some rays for analyzing the boundary layer profiles in detail:
1. The boundary layer profile will be examined in detail at several rays extending from the circular cylinder, namely at angles of 15, 30, 45, 60, 75, 90, 105, 120, and 135 degrees from the front stagnation point. These rays need to be created within Fluent.
2. In the main *Fluent* window, Surface-Line/Rake. Type in the desired starting and ending *x* and *y* locations of the 45^o ray, i.e., a line going from the origin (0,0) to somewhere far from the cylinder along a 45^o ray (-5,5).
3. The *New Surface Name* should be assigned at this point. It is suggested that this line be called "ray_045" or something descriptive of its intended purpose.
4. Click on Create to create the line.
5. Similarly, create a ray at 30^o (use some simple trig to figure out the points needed to create this ray); a suggested label is "ray_030". Create other rays at 15^o, 60^o, 75^o, 90^o, 105^o, 120^o, and 135^o.
6. To view these newly created lines, return to the main *Fluent* window, and Display-Grid-Display. Unselect (by left mouse click) the default interior, and select the newly created lines instead. Display. The lines should be visible at the appropriate locations. If not, create them again more carefully.
7. *Note*: If you screw up, use the Manage utility to either delete or re-name the erroneous ray.
8. Close both the *Line/Rake Surface* window and the *Grid Display* window.

Record the properties of the fluid, air:
1. The default fluid is air, so we don't have to change it. However, it is wise at this point to record the fluid density and viscosity. In the main *Fluent* window, Define-Materials. Check that air is the default for *Fluid Materials*.
2. Write down the density and viscosity of air, which are needed later to calculate Reynolds numbers, etc.
3. Close the *Materials* window.

Define the boundary conditions:

1. Now the boundary conditions need to be specified. In Gambit, the boundary conditions were declared, i.e. wall, velocity inlet, etc., but actual *values* for inlet velocity, etc. were never defined. This must be done in Fluent. In the main *Fluent* window, click on Define-Boundary Conditions, and a new *Boundary Conditions* window will pop up.
2. Select fluid and Set. Verify that the default fluid is air, as appears in the *Material Name* drop-down list of material names. OK.
3. The default boundary condition for the cylinder (wall) is okay (stationary wall), so nothing needs done to it.
4. Likewise, the default boundary conditions for the symmetry plane (symmetry) and the pressure outlet (outlet) are okay, so nothing needs done to them.
5. Select inlet (a velocity inlet), which is the left side of the computational domain. Set. Change the *Velocity Specification Method* to Magnitude and Direction. Change the *Velocity Magnitude* to 1.0 m/s. The default flow directions (1 for *x* and 0 for *y)* are for zero angle of attack, which is what is desired here, so OK.
6. What is the Reynolds number based on cylinder diameter? Will this be in the laminar boundary layer regime?
7. Boundary conditions are complete, so Close the *Boundary Conditions* window.

Set up some parameters and initialize:

1. In the main *Fluent* window, Define-Models-Viscous. Laminar flow is the default, so we really don't need to do anything here. Later on, however, we will try turbulent flow calculations; this is where the turbulence models are specified in Fluent. OK.
2. Now the convergence criteria need to be set. As the code iterates, "residuals" are calculated for each flow equation. These residuals represent a kind of average error in the solution - the smaller the residual, the more converged the solution. In the main *Fluent* window, Solve-Monitors-Residual. In the *Residual Monitors* window that pops up, turn on Plot in the *Options* portion of the window. The Print option should already be on by default. Here, Print refers to text printed in the main *Fluent* window, and Plot causes the code to plot the residuals on the screen while the code is iterating.
3. Since there are three differential equations to be solved in a two-D incompressible laminar flow problem, there are three residuals to be monitored for convergence: continuity, *x*-velocity, and *y*-velocity. The default convergence criteria are 0.001 for all three of these. Experience has shown that this value is generally not low enough for proper convergence. Change the *Absolute Criteria* for each case from 0.001 to 1.E-08.
4. To apply the changes, OK, which will also close the *Residual Monitors* window.
5. 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.
6. At this point, and every so often, it is wise to save your work. In the main *Fluent* window, File-Write-Case & Data. If not on by default, turn on the option to Write Binary Files (to save disk space). To save even more disk space, the files can be compressed by adding a "gz" at the end of the file name. In the *Select File* window that pops up, the default file name should be changed to something more descriptive ("laminar_cylinder.cas.gz" is suggested to distinguish this case from the turbulent case to be calculated later). Note that "Case & Data" refers to both the *case* (the grid plus all boundary conditions and other specified parameters) and the *data* (the velocity and pressure fields calculated by the code). The code actually writes out *two* files, laminar_cylinder.cas.gz and laminar_cylinder.dat.gz.
7. Click OK to write the file onto your directory.

Iterate towards a solution:

1. In the main *Fluent* window, Solve-Iterate to open up the *Iterate* window. Change Number of Iterations to 200, and Iterate. The main screen will list the residuals after every iteration, while the graphical display window will plot the residuals as a function of iteration number. The residuals may rise at first, but should slowly start to fall. It is normal for the residuals to fluctuate up and down. Do not be concerned if there are reverse flow warnings; these will disappear in time.
2. At the end of 200 iterations, check to see how the solution is progressing. In the main *Fluent* window, Display-Vectors-Display. The graphical display window will show the velocity vectors. Zoom in with the middle mouse, as described above, to view the velocity field in more detail if desired. In particular, view the boundary layer along the front portion of the cylinder wall. Is it starting to look like a boundary layer profile?
3. Zoom in near the top of the cylinder. Can you see where the boundary layer separates from the wall? *Note*: A click of the middle mouse button centers the view at the location of the cursor. This is useful to "migrate" around the cylinder while zoomed in.

Save your velocity profiles and your calculations:

1. In the main *Fluent* window, File-Write-Case & Data. In the *Select File* window which pops up, the default file name should be "laminar_cylinder.cas.gz", as previously entered. OK to write the file onto your directory. OK again since it is okay to overwrite these files.

Iterate towards a final solution:

1. Iterate some more. (To restart the iteration, either find the *Iterate* window, which is probably hidden under some other windows at this point, or click again on Solve-Iterate to re-open the *Iterate* window.) In the *Iterate* window, set Number of Iterations to about 800, Apply, and Iterate.
2. Check the velocity vectors, as described above after these iterations. It is wise to move the *Iterate* window someplace out of the way of the other windows so you can easily restart the iteration.
3. Iterate some more if necessary. The residuals may go up sometimes. This is normal, as the code attempts to zero in on a solution - after a while the residuals should again decay or level off. If the residuals all go below the convergence criteria, the calculations will stop. In some cases, however, the residuals reach a lower limit, and further iterations don't improve the solution.
4. When the solution does not converge, it sometimes helps to lower the under-relaxation factors. Solve-Controls-Solution. Lower the pressure under-relaxation factor by a factor of two or three – this usually helps convergence, especially since the conservation of mass equation is often the culprit inhibiting convergence.
5. OK, and then iterate some more, hopefully to convergence. In this particular case, any more than a couple thousand iterations is probably a waste of time.
6. Save (overwrite) your case and data files after the solution has converged.

Examine the velocity profiles in detail:

1. At this point, the velocity profiles along the rays previously generated will be plotted and examined in detail.
2. In the main *Fluent* window, Display-Vectors. In the window called *Vectors*, at the lower right corner, select (highlight) all the rays we created above, the ones called ray_015, ray_030, etc.
3. Change the *Scale* to 4 or 5 or so in order to see the velocity profiles more clearly, but without overlap from one ray to the other. Display.
4. Zoom in or out such that the boundary layer profiles along all the rays are clearly seen. If done correctly, the development of the boundary layer with downstream distance should be apparent. The flow should be separated well before the 135^o ray.

Create a hardcopy image file of the velocity vector plot:

1. When the velocity vector plot is to your liking, and clearly shows the boundary layers along the cylinder at all the rays, the vector plot of the boundary layer profiles will be saved.
2. First, we add a label. Click on the area just below the title at the bottom of the velocity vector plot. A cursor will appear which will allow you to enter a more descriptive title. Put your name here, along with the word "Cylinder", and perhaps the date if you have room.
3. In the main *Fluent* window, File-Hardcopy. Change *Coloring* to Monochrome if you have only a black-and-white printer. If you have access to a color printer, select Color. Select the desired format. Generally, TIFF or JPEG work best on Windows machines, but PostScript is often better for Unix or Linux machines. Save.
4. Name the file (something like "laminar_cylinder_profiles.*ext*" is appropriate, where the file extension *ext* should automatically be inserted). OK, and Close.
5. Print out the image file you just created. *Note*: This can be done later on a different computer if you do not have access to a printer at the present time.

Generate and plot streamlines around the body:

1. In the main *Fluent* window, Display-Contours. Choose *Contours of* Velocity and below that, select Stream Function. Make sure the option *Filled* is off. Display.
2. Adjust the number of levels (100 is the maximum allowed) so that you can see several streamlines close to the wall of the cylinder. It is also necessary to turn off *Auto Range*, and set your own upper limit. Experiment with different values until you get lots of streamlines close to the wall, and can clearly see some closed contours in the recirculation zone at the back end of the cylinder. (A maximum value of 0.1 or so is recommended for the cylinder case; a much smaller maximum value is needed for the sphere case.)
3. Zoom in as necessary to get a good view of how the streamlines behave along the cylinder surface.
4. In the main *Fluent* window, select File-Hardcopy. Save. Name the file "laminar_cylinder_streamlines.*ext*" or something equally meaningful, and OK. Close.
5. Print out the streamlines image file to attach to your homework.

Calculate the drag force and drag coefficient on the body:

1. In order to calculate drag coefficient correctly, the proper reference values for area, velocity, etc. need to be defined. In the main *Fluent* window, Report-Reference Values. A window called *Reference Values* will open. The proper reference area to use is the frontal area of the cylinder. Frontal area is defined as the area seen from upstream. Here we are solving for only half of the cylinder, so in this case the frontal area is *R* times *b*, where *R* is the cylinder radius and *b* is the depth or span of the cylinder. Fluent assumes unit depth (b = 1.0 m) by default for convenience so that drag force is per unit depth. *Depth* should already be set to 1.0 m by default.
2. Change *Area* to *R* times *b*, change *Velocity* to the freestream velocity of the flow, and change *Length* to *D*, the diameter of the cylinder.
3. The default reference values for density and viscosity are those of air, the default fluid, so nothing needs done to these. OK.
4. In the main *Fluent* window, Report-Forces. In the window that pops up, the only available *Wall Zone* is wall, which is the surface of the cylinder, so the default is okay. You will need to enter the components of the force vector, however. These are simply the *x* and *y* components of a unit vector pointing in the direction of the desired force. For example, setting the *x*-component to 1 and the *y*-component to 0 would cause Fluent to calculate the force acting on the wall in the *x*-direction. In our problem, we want drag force, which is defined as parallel to the freestream direction. Enter the appropriate *x* and *y* components so that the result is the drag force. *Note*: The units of the calculated force can be thought of as N/m since this is a 2-D problem (force per unit span).
5. Click on Print to perform the calculation. The results will be printed to the main *Fluent* window. You may need to widen that window to see the entire length of the line. Notice that the force is broken into a pressure component and a viscous component. Does most of the drag come from pressure or viscous forces? Discuss.
6. Write down the total drag force and drag coefficient. Does the calculated drag coefficient at this Reynolds number agree with published data?
7. To exit the *Force Reports* window, Close.

Save your calculations and exit Fluent:

1. In the main *Fluent* window, File-Write-Case & Data. OK. It is okay to overwrite the files, so OK again.
2. You don't have to exit Fluent at this point if you are going to immediately run the axisymmetric (sphere) case. Otherwise, exit Fluent by File-Exit.
3. Proceed to the next module, which solves for laminar flow over a *sphere*, using the same mesh.