Unsteady Flow Past a Cylinder - Exercises

Authors: John Singleton and Rajesh Bhaskaran, Cornell University

Exercises

1. **Base Case – Unsteady Cylinder**
    a. How many steps were there per lift coefficient oscillation once the oscillation reaches an apparently steady state? What do these oscillations represent physically?
    b. Why did you include the patched velocity found in the tutorial? What happens when you don't include it in your calculated solution? Does anything change if you increase the patch velocity?
2. **Varying Timestep - Unsteady Cylinder**
    a. Plot the Strouhal number: the dimensionless frequency oscillation (f*d/U) vs. the square of time step, and compare to the results in Fig. 5 of "Implicit Multigrid Computation of Unsteady Flows with Applications to Aeroelasticity" (Caughey 2001).
3. **Altering Grid – Unsteady Cylinder**
    a. Returning to a timestep of 0.2s, compare a mesh of 128 x 64 cells and a mesh of your choosing to the original mesh. Maintain the same cell height by altering the bias factor yourself.
    b. Once the solution is computed, calculate the lift oscillation frequency, and use Richardson extrapolation to compute the limiting frequency corresponding to zero mesh spacing (similarly to the procedure done in the paper by Caughey in part 2).
4. **Altering Reynolds Number – Unsteady Cylinder**
    a. Compare the results for Reynolds numbers of 60, 120, and 80 at a timestep of 0.2s and mesh 192x96. Create a graph of the Strouhal number (dimensionless frequency oscillation) as a function of these Reynolds numbers.

5. **Turbulent Cylinder - The goal of this exercise is to compute the turbulent flow around a cylinder at Re = 10^6 and compare to the paper by P. Catalano et al.**
    a. Change the Reynolds Number to be 1e6. Since the flow is now turbulent, you will need to change the mesh, so that you can resolve the turbulent boundary layer. Set the edge sizing around the "farfields" to be 96 divisions. Set the number of divisions for the imprinted edges to be 100. This will give you about 19200 cells.
    b. In Setup, change the solver from Steady to Transient and change the Model to Standard K-e and leave everything else as the default value. Patch the region behind the cylinder as you did in the unsteady tutorial. This is again so we can quickly get to the steady state solution.
    c. You will run the case with a time step size of .01s for 300 timesteps for a total of 3 seconds.

        i. Plot the pressure distribution vs. theta on the cylinder surface and compare to Fig 3 of the Catalano paper.
        ii. Plot the vorticity contours and compare to Fig. 4 of the Catalano paper
        iii. Compare the Strouhal number, drag coefficient and the base pressure coefficient to the values found in Table 1 of the Catalano paper.

Go to Comments

Go to all FLUENT Learning Modules