Lab 3: Computational Fluid Dynamics

Justin Francis
Tuesday, February 4

Overview

Imagine that you are an engineer on a team that has been tasked with designing a more efficient windfarm. Your team is looking for ways to minimize the wake effect of upstream wind turbines on power generation from the downstream wind turbines in the farm. Figure 1 illustrates the wake interaction in a large offshore windfarm. One team member has the idea of mounting turbines on hexagonal towers, as shown in Figure 2. The project leader has asked you, the team expert in fluid dynamics, to investigate the effect that hexagonal towers would have on the downstream wind pattern. Based on your thorough understanding of engineering principles, you decide that it would also be important to estimate the drag coefficient of the hexagonal tower, as this will impact the structural integrity of the tower.

1 Figures

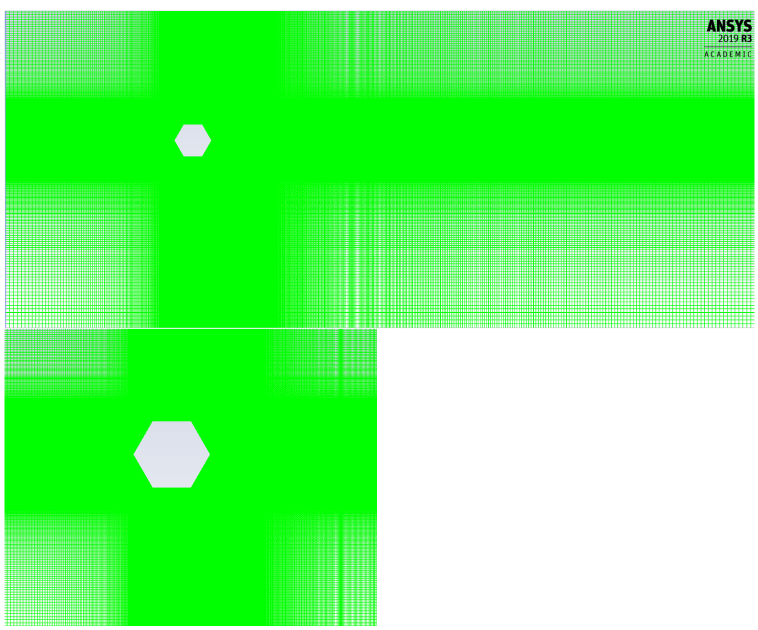

Figure 1: Computational mesh used in the numerical simulation. (top) Entire domain. (bottom) Enlarged region near the hexagon.

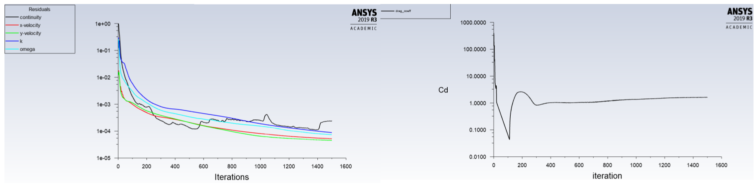

Figure 2: Convergence plots from the numerical simulation. (left) Residuals of the velocities, continuity equation, and turbulence quantities as a function of iteration number. (right) Value of the drag coefficient as a function of iteration number.

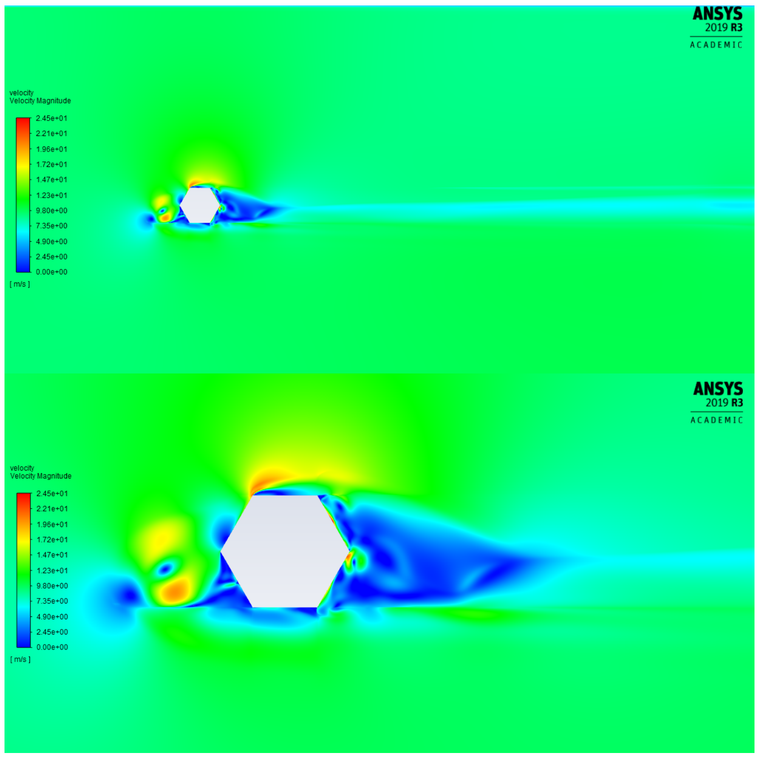

Figure 3: Velocity contour plots from the numerical simulation. The colorbar indicates velocity magnitude in m/s. (top) Entire domain. (bottom) Enlarged region near near the hexagon.

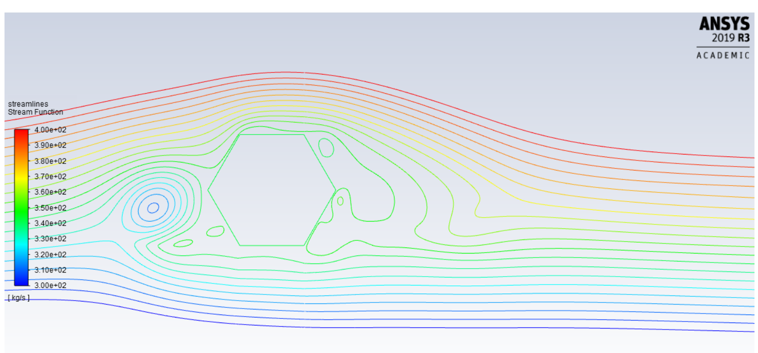

Figure 4: Streamline plot of the flow in the immediate vicinity of the hexagon.

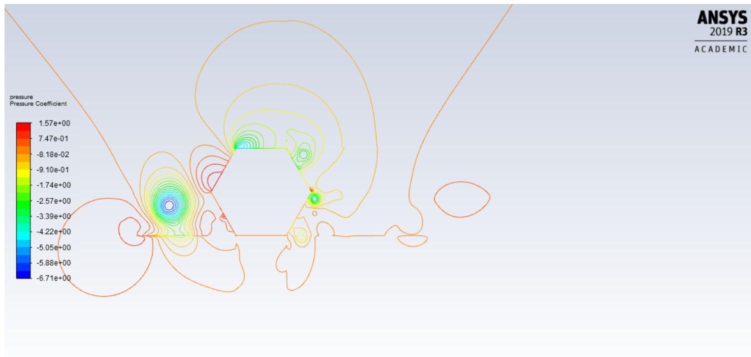

Figure 5: Pressure contour plot in the immediate vicinity of the hexagon. The colorbar indicates pressure in Pa.

```
Forces
                          Forces (n)
Coefficients
Zone                      Pressure                                       Viscous                                        Total
Pressure                                      Viscous                                       Total
hexagon                   (513.0603 570.12015 0)                         (-1.6005123 1.3861436 0)                       (511.45979 571.5063 0)
(1.6108644 1.7900162 0)                       (-0.0050251565 0.0043520994 0)                (1.6058393 1.7943683 0)
------------------------- ------------------------------------------- ---------------------------------------------
--------------------------------------------- --------------------------------------------- ---------------------------------------------
---------------------------------------------
Net                       (513.0603 570.12015 0)                         (-1.6005123 1.3861436 0)                       (511.45979 571.5063 0)
(1.6108644 1.7900162 0)                       (-0.0050251565 0.0043520994 0)                (1.6058393 1.7943683 0)

Forces - Direction Vector (1 0 0)
                          Forces (n)                                      Coefficients
Zone                      Pressure        Viscous         Total           Pressure        Viscous         Total
hexagon                   513.0603        -1.6005123      511.45979       1.6108644       -0.0050251565   1.6058393
------------------------- --------------- --------------- --------------- --------------- --------------- ---------------
Net                       513.0603        -1.6005123      511.45979       1.6108644       -0.0050251565   1.6058393
```

Figure 6: Values of the drag coefficient of the hexagon and drag force in Newtons as calculated by Fluent.

2 Short-Answer Questions

1. State, and justify, the simplifications and assumptions you made to the real-life problem in order to create a tractable CFD problem.

   Answer: A few assumptions I made were that 10 m/s air flow was sufficient for approximating the coefficient of drag. This is because the minimum operating speed is 6 m/s and maximum is 25 m/s. It is very hard to simulate uniform flow, in this lab we were able to work around it by setting the initial conditions which helps make the difference negligible. Our simulation results also neglects the fact that these wind turbine bases are quite large and can experience different conditions at differing heights (ground effect).

2. Discuss the similarities and differences between the flow around the hexagonal tower and that around the square obtained from the Tutorial.

   Answer: The flows were very similar in the fact that there was vortex shedding on the vertices of the two shapes, this makes sense intuitively because of the way the no-slip boundary condition works. There's also a similar high and low pressure zone on the inlet an outlet sides respectively. One very strange difference is the hexagonal flow is not symmetrical. This could, and most likely is due to my error in setting up the simulation.

3. Discuss how you would validate the simulation results (specifically, the drag coefficient) from your numerical simulation of the wind turbine tower.

   Answer: I would validate these results by conducting a small-scale experiment set up using the Buckingham Pi Theorem. This would allow us to build a cheap replica and match dimensionless numbers to create an accurate experiment. This would then allow us to directly measure the drag coefficient and the empirical data would either validate or invalidate my simulation.

4. Calculate the Reynolds number of your simulated flow (use a characteristic velocity scale based on the inlet velocity, and a characteristic length scale based on the hexagon side dimension). When stating the Reynolds number of any flow, it is good practice to indicate the quantities that were used for the characteristic scales. Therefore, your response to this question should be of the following form (where XX denotes the values for your simulation): "The Reynolds number of the flow around the hexagon tower is XX, based on an inlet flow velocity of XX m/s and a hexagon side length of XX m". Also, state whether you expect the flow around the hexagon tower to be laminar or turbulent based on this Reynolds number. Finally, discuss whether your CFD results support this expectation (be specific in your statement, i.e., what exactly about your results does or does not support your expectation).

   Answer: The Reynolds number of the flow around the hexagon tower is $3.56 \times 10^6 [dimensionless]$, based on an inlet flow velocity of $10[m/s]$ and a hexagon side length of $5.2[m]$. Based on this Reynolds number, I would expect the flow around the tower to be turbulent, this is because $3.56^6 > Re_{crit} \approx 1 \times 10^5$. My simulation supports this hypothesis because if one analyzes the contour plots in this paper, one can see that the flow around the hexagon is very erratic and turbulent in nature.