Development of Aerodynamic Coefficients for Multisided Cylinders

Ostap Lisowitch
Civil Engineering
Dr. Byungik Chang

Abstract

Wind tunnel testing is routinely used to study various aerodynamic phenomena and determine aerodynamic parameters of engineering structures. However, the required design aerodynamic coefficients for a complex shape may not always be available from the wind tunnel testing or standards. In the last decade, a computer-aided computational fluid dynamics code has been widely used in fluid mechanics and wind engineering for simulation of complex structures. Computational Fluid Dynamics (CFD) provides a quicker and virtually a free alternative to modeling complex systems in comparison to wind tunnel testing. The primary objective of this study is to verify and develop aerodynamic coefficients for multisided cylinders, in particular, drag coefficients (C_D) using CFD (ANSYS-CFX) [1]. There are numerous numbers that describe the regime of flow. One dimensionless hydrodynamic numbers that directly describe flow past a structure is the Reynolds Number (Re). Re is classified as a ratio of inertial forces to viscous forces which enumerates the relative importance of the forces inflicting the object or structure. Drag coefficient (C_D), indirectly related to the Re, is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment, such as air in this study. The output will be able to predict buffeting loads for the fatigue design of multisided cross section slender support structures without field or wind tunnel test, which costs much more compared to computer modeling.

***Index Terms*—drag coefficient; wind tunnel, computer-aided simulation, cylinders**

I. INTRODUCTION

Aerodynamic coefficients such as drag and lift are crucial inputs in the design on various structures. When used correctly, the optimized design yields a conservative and sufficient option that may be implemented on projects. Over the past few years, aerodynamic coefficients for smooth cylinders have been widely tested, (James, Bosh, and Chang), however, there is very minimal testing of multisided cylinder and other complex shapes. Technological advances provided engineers an option of testing and experimenting with CFD, a faster, inexpensive, and accurate method of simulating real life systems.

II. BACKGROUND

A. Reynolds Number

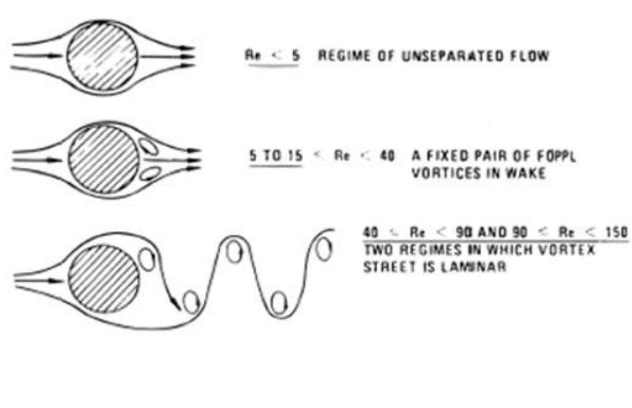
When fluid flow passes around a structure, in this case, cylindrical structures, the structure experiences various aerodynamic phenomena. In this study, drag and lift are analyzed. The drag and lift forces are essential elements when designing structures. There are numerous numbers that describe the regime of flow, one dimensionless hydrodynamic numbers that directly describe flow past a structure is the Reynolds Number (Re). Re is classified as a ratio of inertial forces to viscous forces which enumerates the relative importance of the forces inflicting the object or structure. The widely used ratio can be formulated via various relationships, in this study Reynolds Number is formulated as

$$Re=\rho Vd/\mu \qquad (2.1)$$

Manuscript received July 30, 2014.
O. Lisowitch is an undergraduate student with the Civil, Mechanical, and Environmental Engineering Department, University of New Haven, West Haven, CT 06516 USA (e-mail: oliso1@unh.newhaven.edu)

Where ρ is the fluid density, V is the flow velocity, d is the cylinder diameter, and μ is the dynamic viscosity of the fluid.

Flow contours are attained as the result of enormous degree of change of the Reynolds Number. As the Reynolds Number fluctuates, the continuous flow around the cylinder creates vortices in the wake of the cylindrical structure. Dependent on the Reynolds number, the wake region of the cylinder varies. At a small Reynolds Number (Re < 5), the wake typically shows a creeping flow, meaning no separation. As Re increases, the wake region become more turbulent and unstable, displaying a phenomenon called Vortex Shedding as displayed in Figure 1.



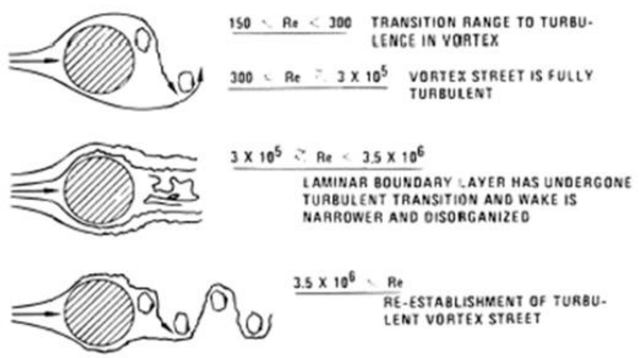


Figure 1 – Flow contour past a smooth cylinder [6]

B. Drag and Lift

Drag coefficient (C_D), indirectly related to the Re, is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment, such as air in this study. In order to compute the C_D, several parameters are required and are formulated as

$$C_D = \frac{F_d}{\frac{1}{2}\rho v^2 A} \tag{2.2}$$

Where F_D is the drag force, by definition is the force component in the direction of the flow velocity, ρ is the mass density of the fluid, v is the speed of the fluid relative of the object, and A is the projected area of the cylinder.

Reliant on the object, drag coefficients vary as displayed in Figure 2.

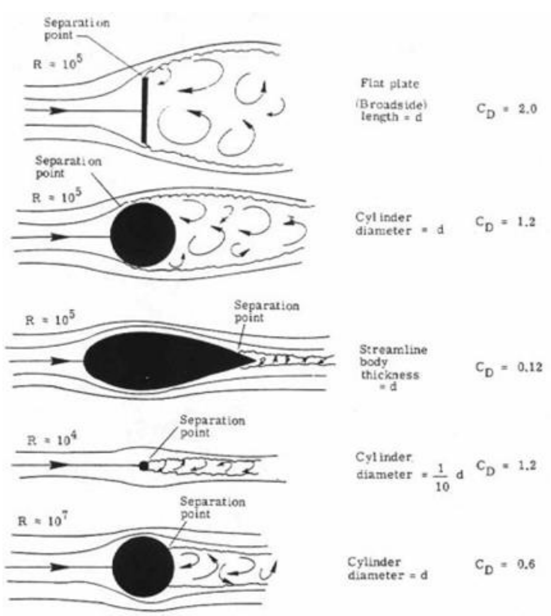


Figure 2 – Drag Coefficients of various objects [7]

Drag and lift forces, force components that can be divided into in-line and cross-flow directions, periodically change depending on the pressure distributions due to the range of Reynolds Number. This pressure distribution is what causes vortex shedding, and when this phenomena occurs, lift forces are generated. Both, the drag and lift forces are formulated as follows:

$$F_D = \bar{F}_D + \hat{F}_D \sin(2\omega_S t + \phi_S) \tag{2.3}$$
$$F_L = \hat{F}_L \sin(2\omega_S t + \phi_S) \tag{2.4}$$

Where $\hat{F}_D$ and $\hat{F}_L$ are both amplitudes of the vortex shedding frequency which is represented by $\omega_S = \frac{2\pi}{f_v}$, and ϕ_S is the phase angle between the oscillating forces and the vortex shedding with t being time.

III. COMPUTATIONAL FLUID DYNAMICS (CFD)

A. Pre-Processor

1) Object's Domain

The CFD analysis begins with the geometry, the cylinder and walls of the wind tunnel. The primary objective of this study is to verify and develop aerodynamic coefficients for multisided cylinders, therefore a virtual wind tunnel is designed that will encompass the various cylinders. The tunnel dimensions and radius of various cylinders were constant in each analysis. The wind tunnel dimensions are shown in Figure 3.

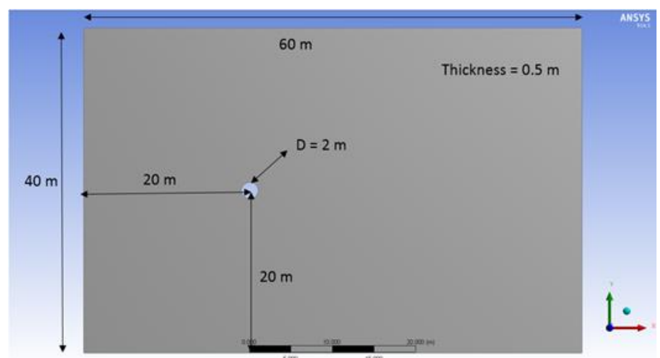


Figure 3 – Geometry of Modeled Wind Tunnel

2) Meshing

Meshing is a crucial step in receiving accurate results. The type of mesh and mesh quality is an integral part of CFD. Meshing is breaking up a geometric object into sets of finite elements for computational modeling. A too fine mesh will result in long solution time and will require large CPU capabilities. Too coarse of a mesh will lead to error and inaccuracy. The mesh utilized for the analysis is shown in Figure 4, where heavily dense mesh is surrounding the cylinder and less dense mesh around the rest of the geometry.

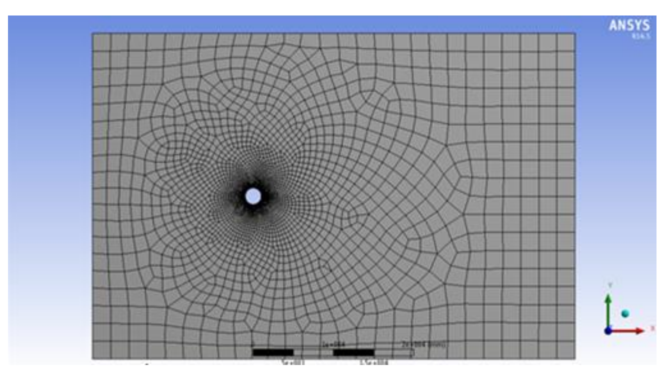


Figure 4 – Meshing

3) Boundary Conditions

To establish the inlet and outlet locations for the wind in the wind tunnel, boundary conditions are required. The boundary conditions specify the walls and whether or not slip is adjusted for. The cylinder is a boundary as well, labeled as a smooth wall. Figure 3.3 shows the inlet and outlet location of the wind tunnel.

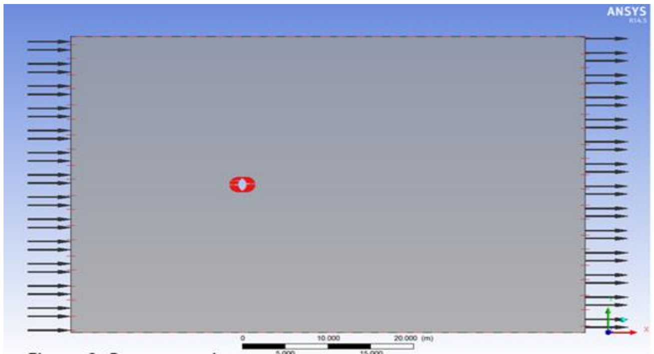


Figure 5 – Boundary Conditions

IV. RESULTS

Five different cylinders were analyzed: A square, hexagon, octagon, dodecagon, and circle. Using ANSYS – CFX, the drag coefficient computed at a Reynolds Number of a 100 was 1.92. According to Scruton's, "An Introduction to Wind Effects on Structures" the actual wind tunnel test yielded a drag coefficient of 2.0[2]. For a six sided cylinder, a drag coefficient of 1.83 was computed, however, due to the lack of actual data on such object, no comparison was found. The octagonal cylinder yielded a drag coefficient of 1.59 compared to actual wind tunnel testing where the coefficient varies from 1.0 to 1.6. Figure 6 shows results and actual test comparison. Dodecagon, a twelve-sided cylinder, was computed to have a drag coefficient of 1.33 compared to design code of 1.2. Ultimately the circular cylinder was computed to have a drag coefficient of 1.23 and compared to multiple tests of having a drag coefficient of 1.2. Figure 6 shows the summarized results and comparisons.

	Drag Coefficients of Multisided Cylinders (Re 100)			
	Number of Sides	Wind Tunnel	Design Code [5]	ANSYS-CFX
	Square - 4	2.0 [2]	1.7	1.92
	Hexagon - 6	NA	NA	1.83
	Octagon - 8	1.0 ~ 1.6 [3]	1.2	1.59
	Dodecagon - 12	1.2 ~ 1.6 [2, 3, 4]	1.2	1.33
	Cirlce - 1	1.2 [2,3]	1.1	1.23

Figure 6 – Drag Coefficients

V. CONCLUSION

It was determined that CFD is a fast, inexpensive, and accurate method of modeling complex fluid to structure interaction. With the resulting data aerodynamic coefficients were verified and developed, as well as, predicted vortex shedding and buffeting loads.

VI. FUTURE WORKS

In CFD, Aerodynamic coefficients were developed at a Reynolds Number of a 100. The following task is to model the various cylinders with a Reynolds Number range of $100 < Re < 1.0 \times 10^6$, which covers typical wind velocity in reality. This task involves analyzing 20 various turbulence options. Going further into the project, analyzing vibration induced vortices would become the third objective.

References

[1] "ANSYS - Simulation Driven Product Development." ANSYS - Simulation Driven Product Development. N.p., n.d. Web. 05 Sept. 2014.

[2] Scruton, An introduction to wind effects on structures, Engineering design guides 40, BSI&CEI, 1981.

[3] James, W.D. Effects of Reynolds number and corner radius on two-dimensional flow around octagonal, dodecagonal and hexdecagonal cylinders, PhD Dissertation, Iowa State University, 1976

[4] Chang, B. A time-domain model for predicting aerodynamic loads on a slender support structure for fatigue design cylinders, PhD Dissertation, Iowa State University, 2007

[5] American Association of State Highway and Transportation Officials, Standard Specifications for Structural Support for Highway Signs, Luminaries, and Traffic Signals. Washington, D.C., 2010.

[6] Sunden, Bengt. "Tubes, Crossflow over." Tubes, Crossflow over. 2, 2 Feb. 2011. Web. 2 Oct. 2014.

[7] "Drag Coefficient." Drag Coefficient. NASA, n.d. Web. 26 Oct. 2014.

Acknowledgements

Special thanks to my faculty advisor Dr. Byungik Chang for guiding me in the right directions throughout my research and Dr. Daniels for ANSYS Tutorials and Lectures.

Biography

Ostap (Ozzy) Lisowitch, E.I.T., is currently a senior at the University of New Haven studying Civil Engineering. He is currently the Construction and Safety Manager of ASCE. Ozzy has a passion for engineering and futbol. After graduation, Ozzy plans to work in Construction Management.