The Effect of Structural Shapes on Fluid Flows in Fluid-Structure Interactions

Sarah Pomerantz

Abstract

Computational Fluid Dynamics is the use of computer simulations to study fluid flows, and in this case Fluid-Structure Interactions, where fluids flow past and interact with various obstacles. Using Ansys Student software, this research studied the flow past 9 objects: a circular cylinder, a horizontal ovular cylinder, a vertical ovular cylinder, a cube, an equilateral triangular prism pointing towards the outlet, an equilateral triangular prism pointing towards the inlet, a cube rotated 90$\circ$, an equilateral triangular prism pointing towards the wall, and a right scalene triangular prism. In the end it was found that objects with similar properties create similar recirculation regions and flow patterns.

Acknowledgements

This work was conducted at Rensselaer Polytechnic Institute under the instruction of Lucy Zhang, Ph.D., and Chen Shen.

Introduction

Computational fluid dynamics (CFD) is the study of fluid flows using computer simulations. These simulations mimick real world conditions without the cost of a true experiment. Engineers use CFD to investigate how fluids will interact with their designs before they begin construction. For example, when building planes CFD is utilized to understand how air will flow above above and below a wing to decide if the wing design is the most optimal choice. It is also incredibly useful when building air conditioners and heaters because one can visualize how the air will flow through a room before they build. CFD is also used by medical doctors and biomedical engineers. The human body is filled with fluids. CFD can be used to see how blood flows through a blood vessel or even how the blood vessel may disintegrate as a result.

Fluid flows can appear in a multitude of different ways depending on their properties. Every fluid has a reynolds number. The reynolds number of a fluid is dependent on its velocity, density, and viscosity.

$$Re = \frac{velocity \cdot density}{viscosity}$$

Velocity refers to the displacement over time of a particle. Density refers to the mass of a substance divided by its volume. Viscosity displays the thickness of a fluid. For example, a high viscosity substance would be honey, which has difficulty moving. A substance with a low viscosity would be water, which flows very easily. Reynolds numbers then indicate how the fluid will flow. If a fluid's reynolds number is under 2000, then the flow is laminar. Laminar means the particles flow in straight lines, not interacting with each other, as seen in figure 1.

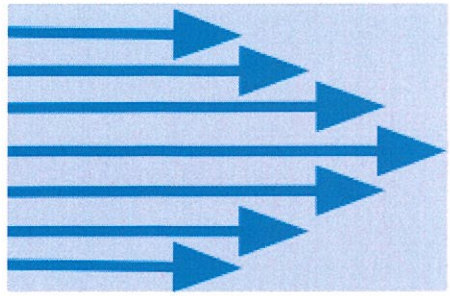

Figure 1: A laminar flow with a reynolds number below 2000. Notice the differences in arrow lengths when trying to understand how a fluid flows between walls. *Student Generated Image*

Although in a laminar flow the particles do flow in straight lines, the flow is still not always completely uniform. For example, if a fluid was flowing between two walls in a pipe or other similar object, the particles in the center would have the highest velocities. Meanwhile, particles by the walls would adhere to the walls, lowering their velocities. In between the center and the walls particle cohere with each other, causing a velocity between the high and low.

Not all flows are as simple as the laminar flow. Flows with reynolds numbers higher than 4000 are called turbulent flows. These flows are much more chaotic, the paths of the particles constantly intersect, as illustrated in figure 2.

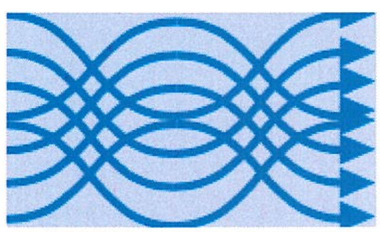

Figure 2: Turbulent flow with a reynolds number greater than 4000. Notice how the paths intersect. *Student Generated Image*

As evident in the diagram, the turbulent flows are much less predictable. This research focuses on laminar flows.

In order to understand the basis of how fluid flows work, miscellaneous shapes which can resemble real life objects may be placed in a fluid flow simulation to understand how the fluid flows past it. When designing something specific one can also put that specific object into a simulation, however for this research there was no specific design in mind. There are 2 important elements of a fluid flow past an object. The first would be the recirculation region behind the object. The recirculation region begins at a separation point, or where the fluid which had been adhering to the object stops adhering and enters into an eddy. An eddy is a flow structure, as drawn in figure 3. After recirculating through the eddies the flow regroups and continues its path.

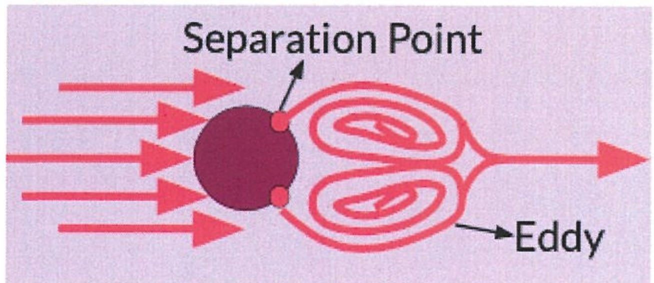

Figure 3: Recirculation region with separation point and eddies. *Student Generated Image*

The other very important element of a flow past an object is the portion of the flow past the recirculation region. Typically the flow will enter into a von karman vortex street. This is when the fluid enters flow structures called vortices repeatedly.

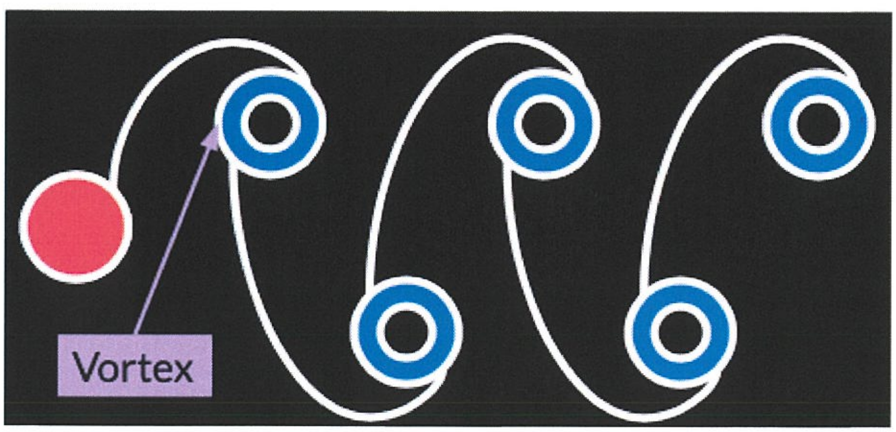

Figure 4: Von Karman Vortex Street. *Student Generated Image*

Fluids are governed by the Navier Stokes Equations. The first is Conservation of Momentum:

$$\frac{\partial}{\partial t}\int_V \rho u dV + \int_S (\rho u \otimes u - \sigma) \bullet n dS = \int_V \rho b dV$$

Then there is Continuity:

$$\frac{\partial}{\partial t}\int_V \rho dV + \int_S \rho u \bullet n dS = 0$$

Finally, there is Conservation of Energy:

$$\frac{\partial}{\partial t}\int_V \rho E dV + \int_S \rho E u \bullet n dS = \int_V s dV + \int_V \rho b \bullet u dV + \int_S \lambda \triangledown T \bullet n dS + \int_S (\text{��}\sigma \bullet u) \bullet n dS$$

The first intention of this research is to study fluid flows past numerous objects. The next step is to compare the various results between shapes with similar properties. This will help to understand the effect and significance of geometry on fluid flows in fluid-structure interactions.

Methodology

In this research flows past 9 different objects were studied. These objects were a circular cylinder, a horizontal ovular cylinder, a vertical ovular cylinder, a cube, an equilateral triangular prism pointing towards the outlet, an equilateral triangular prism pointing towards the inlet, a cube rotated 90 ∘ , an equilateral triangular prism pointing towards the wall, and a right scalene triangular prism. Each simulation was conducted at reynolds number 80. The velocity of each flow was 80 meters/second, the density was 1 kilogram/meter^3, and the viscosity was 1 pascal-second. Each flow lasted 2.5 seconds. All images were taken after these 2.5 seconds.

The fluid flows will not be created in a true experiment. They are simulated on a computer using the Ansys Student software to input the aforementioned parameters and then visualize the flows afterwards. They flows are shown in velocity streamlines so one can clearly see where the flow is faster and where it is slower.

Results

Rounded Objects

Circular Cylinder:

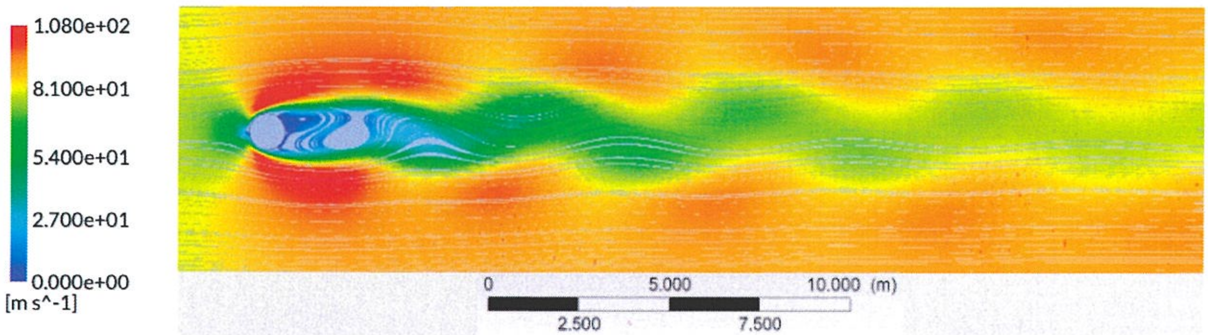

Figure 5: 1 large eddy in the recirculation region, clear von karman effect. *Student Generated Image*

Horizontal Ovular Cylinder:

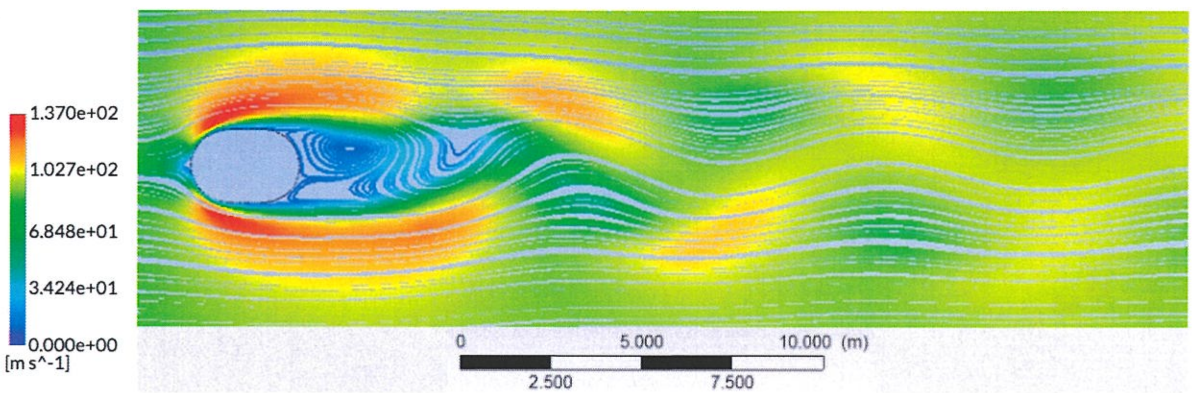

Figure 6: 1 large eddy in the recirculation region, clear von karman effect. *Student Generated Image*

Vertical Ovular Cylinder:

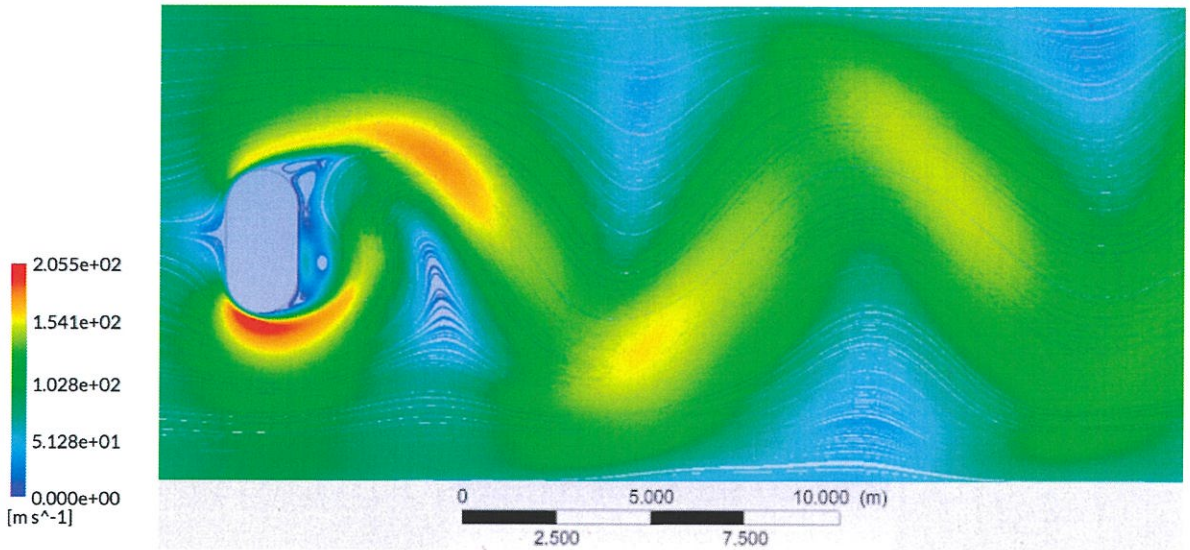

Figure 7: Recirculation remains tightly close to the object. Wide von karman vortex street is present. *Student Generated Image*

Symmetrical Objects with Straight Edges

Cube:

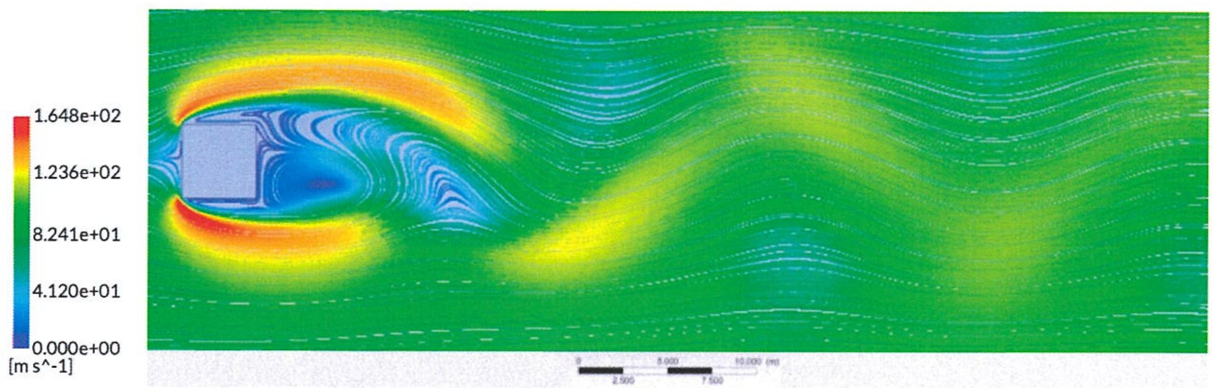

Figure 8: 1 major eddy in the recirculation region and 1 smaller one. Wide von karman effect. *Student Generated Image*

Equilateral Triangular Prism Pointing to Outlet:

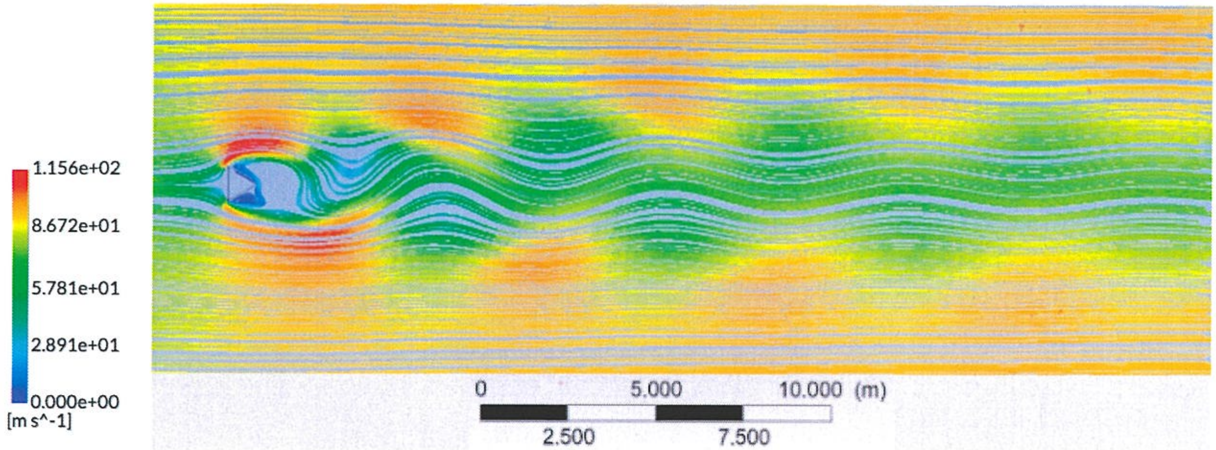

Figure 9: 1 major eddy and one minor eddy. Recirculation region greatly adhered to the object. Von karman effect present. *Student Generated Image*

Equilateral Triangular Prism Pointing to Inlet:

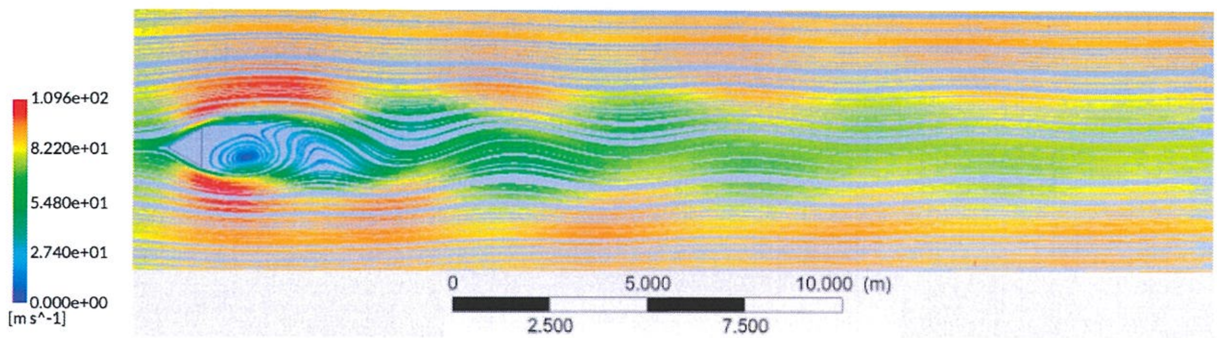

Figure 10: Recirculation region has one eddy and no adhesion to the object. Von karman effect is weak. *Student Generated Image*

Cube Rotated 90 ∘ :

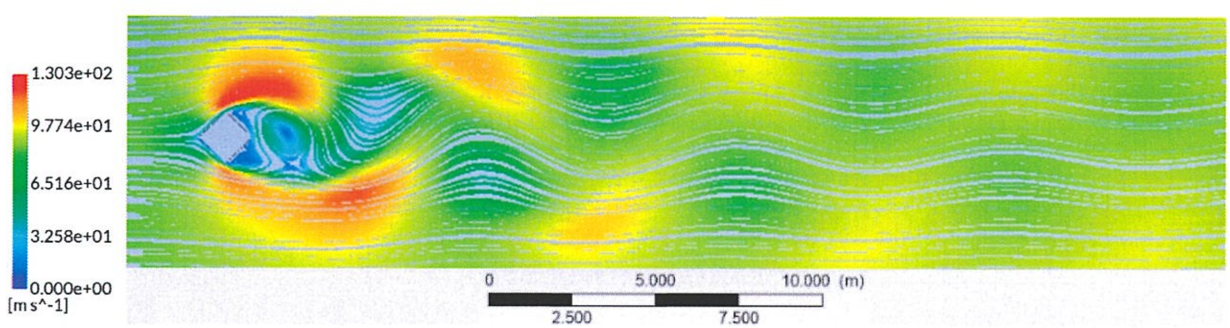

Figure 11: Recirculation region has one major eddy, one minor eddy, and one medium eddy. This is somewhat a mix of what was seem in figures 8-10. Von karman effect is present. *Student Generated Image*

<u>Asymmetrical Objects</u>

Equilateral Triangular Prism Pointing to Wall:

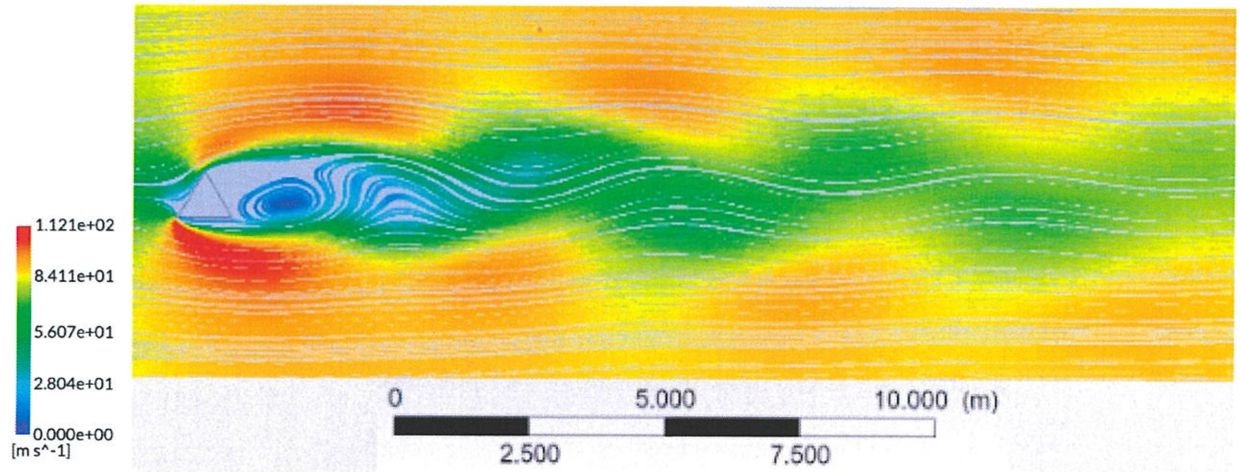

Figure 12: Recirculation region has one eddy and no adhesion to the object. The von karman vortex street is uneven. *Student Generated Image*

Right Scalene Triangular Prism:

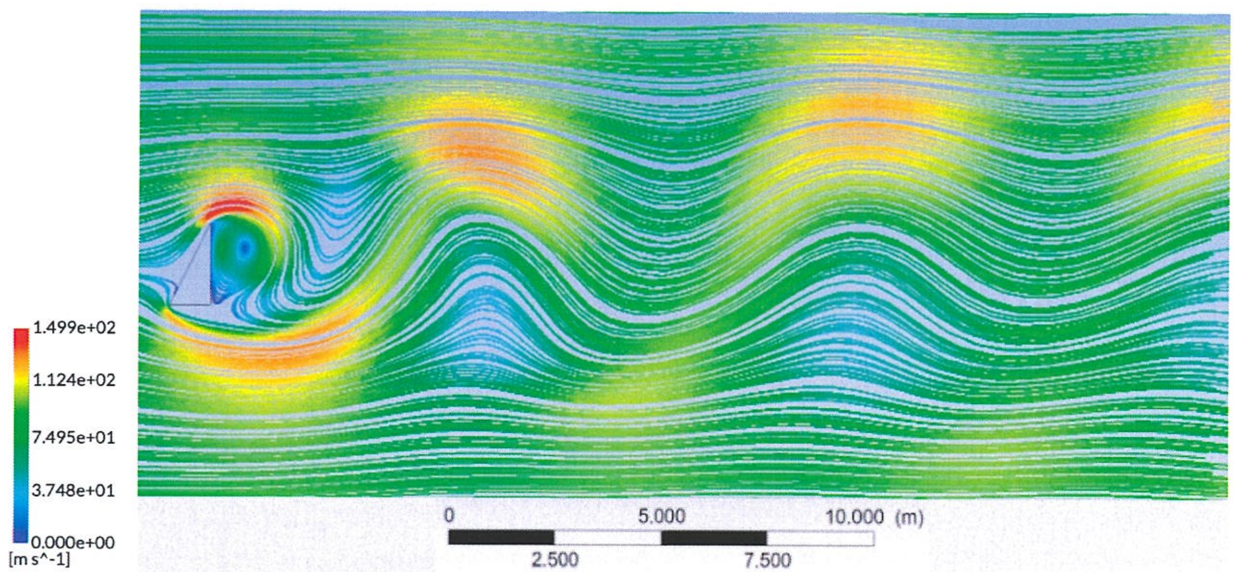

Figure 13: Recirculation region has one large eddy and adhesion to the object. The von karman vortex street is very uneven. *Student Generated Image*

Discussion

In the rounded objects, the circular cylinder and horizontal ovular cylinder have a clear similarity. Both have one large eddy and a similar low velocity region before entering into the von karman vortex street. The vertical ovular cylinder, however, mostly seems to have flow adhere to it or stay close in the recirculation region.

For the symmetrical objects with straight edges there were also multiple similarities. The cube and 2 equilateral triangles both had major eddies on the lower part of the recirculation region and minor eddies above. The cube and equilateral triangular prism pointing to the outlet both had the same separation points and in turn both had heavy adhesion between the fluid and the object. Meanwhile, the equilateral triangular prism pointing to the inlet has a separation point later in the flow and has no adhesion between the recirculation region and the object. The cube rotated 90 ° is somewhat of a compilation of the 3 previous structures, with the later separation point seen in the equilateral triangular prism pointing to the inlet, as well as the adhesion between the object and the recirculation region from the cube and equilateral triangular prism pointing to the outlet. All 4 had a von karman vortex street but it was very weak for the equilateral triangle point to the inlet.

The last two structures were asymmetrical in respect to the flow. Both recirculation regions only had one eddy and the von karman vortex streets were consistently uneven. The equilateral triangular prism pointing to the wall had no adhesion between the fluid and the object in the recirculation region, however the right scalene triangular prism did.

Altogether, shapes with earlier separation points exhibit adhesion between the fluid and the object in their recirculation region. Structures with similar properties always have similar recirculation regions. All the flows studied also had a von karman vortex street. Asymmetric shapes also always have asymmetrical flows.

Future Work

There are many more ideas to be studied using Computational Fluid Dynamics, as well as numerous applications to the methods used within this paper. There are an infinite number scenarios that could be simulated using this method. Researchers could also look into the effects of the strouhals numbers on fluid flows. Furthermore, any reynolds number could be used, one could even test with turbulent flows. There are also many practical applications to CFD. Any engineering project can use CFD because fluids are everywhere and therefore every structure must interact with them.

References

ANSYS Student [Computer software]. (2019). Retrieved from https://www.ansys.com/academic/free-student-products

Bisetti, F., Attili, A., & Pitsch, H. (2014). Advancing predictive models for particulate formation in turbulent flames via massively parallel direct numerical simulations. *Philosophical transactions. Series A, Mathematical, physical, and engineering sciences*, *372*(2022), 20130324. doi:10.1098/rsta.2013.0324

Cohen, I. M. and Kundu, P. K. (2008). *Fluid Mechanics.* Academic Press, San Diego, 4 edition.

Reynolds Number. (n.d.). Retrieved from https://www.grc.nasa.gov/www/BGH/reynolds.html

Van Dyke, M. (1982). *An Album of Fluid Motion.*

White, Frank M. (2017) *Fluid Mechanics.* McGraw-Hill Education.