Computational Lab 2 - Computation of Potential Flow

Connor Ott - 104690690

Introduction

Understanding how flow moves around objects and visualizing flow can prove to be a difficult task for scientists and engineers. Physical testing and analytical solutions can be clunky, time-consuming fixes to this problem. This makes numerical solving tools the ideal technique for visualizing flow. This lab focuses on representing flow properties as a fluid flows over a rectangular prism and two cylinders in a wind tunnel. Specifically, this lab will simulate velocity potential of the flow using MATLAB's PDE solver, *pdetool.m*. From the potential, information on the flow direction, velocity, and pressure can be derived and analyzed. In addition, a qualitative analysis of the simulation's validity will be performed. This will help solidify the capabilities, and limitations, of numerical PDE solutions.

I. Procedure

This section will focus mainly on the set-up of MATLAB's *pdetool.m*. Recall that the main purpose of this lab is to simulate velocity potential at all points in the flow for the given scenario. Already, this means the flow must be assumed to be irrotational, as the velocity potential is not defined for rotational flows. In addition, incompressibility is assumed. This means that the divergence of velocity must be equal to zero. Since velocity is defined to be the gradient of the velocity potential,$\nabla\phi$, the equation $\nabla^2\phi = 0$ must hold everywhere in the flow. This, the Laplacian of the velocity potential, is used to govern the flow at all points in the simulation.

One of the most important parts of this analysis is ensuring boundary conditions are correctly defined. The boundaries for the given simulation include the walls, inlet and exit, and the edges of the shapes in the flow. Fig.(1) provides visual reference for the boundaries being described. In the figure, the large rectangle represents a sort of test section for the flow. The square and circles represent the objects interacting with the flow. The upper and lower walls of the test section, square, and circles must be given boundary conditions such that no flow moves through them. This is characterized by the normal derivative of the velocity potential. Essentially, the rate of change of the potential, ϕ, with respect to the normal of the boundary, $\hat{n}$, should be zero. I.e. $\partial\phi/\partial n = 0$. The inlet of the test section is where the initial flow velocity is defined. Here, flow *should* move through boundary, and at a rate of 25 m/s normal to the boundary. For this reason, at the inlet, $\partial\phi/\partial n =$ -25. The normal derivative is negative due to the positive direction of the normal being oriented outward, opposite the direction of the flow. The exit of the test section is defined such that, regardless of the flow's velocity, the fluid will flow perpendicular to the exit boundary. This is defined by saying the dot product of $\mathbf{V}$, the velocity of the flow, and $\hat{s}$, the tangent of direction of the boundary, is zero. Written simply, $\mathbf{V}\cdot\hat{s} = 0$. Now that the boundary conditions are set, the velocity potential can be solved for. For more information on how to implement the PDE and boundary conditions into MATLAB's *pdetool.m*, please consult the videos "Part 1 - Problem Statement - Vimeo" and "Part 2 - MATLAB - Vimeo" by John Evans.

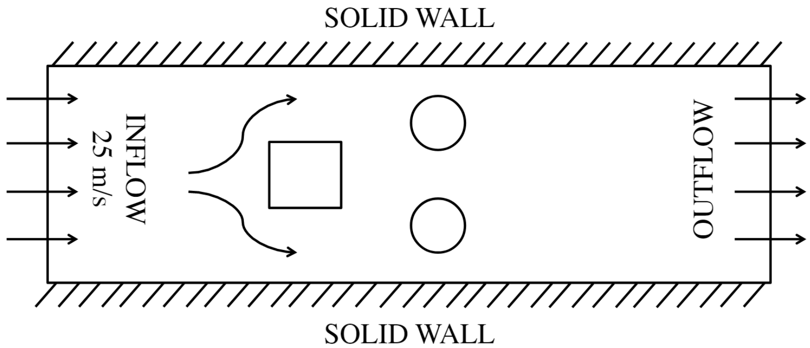

Figure 1. Reference for boundary condition descriptions.[3]

Results

Fig.(2) shows the numerical results of *pdetool*. The contour lines here represent equipotential lines in the flow, while the arrows indicate flow direction. Note that velocity always points in the direction perpendicular to equipotential. The color gradient indicates the actual value of the velocity potential at each point. This shows that the velocity also points in the direction of increasing velocity potential.

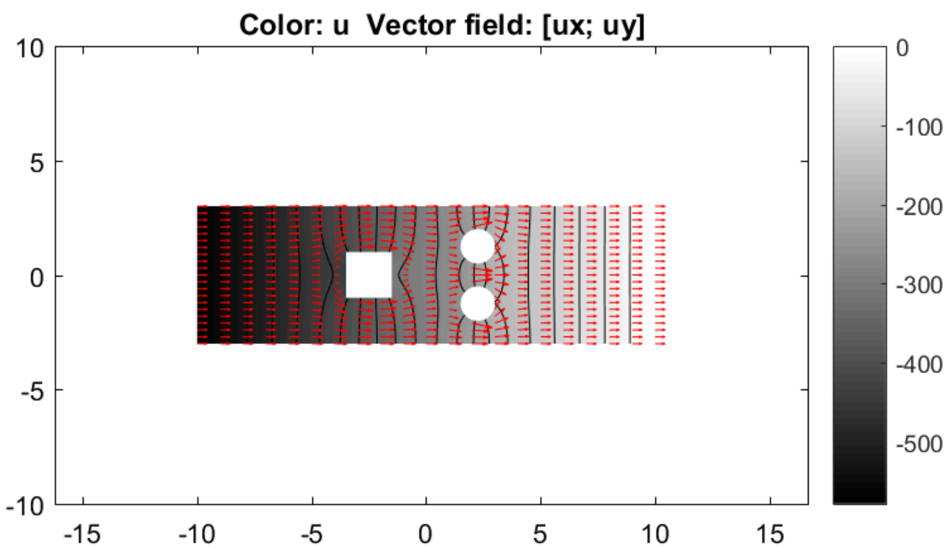

Figure 2. Velocity potential gradient overlaid with equipotential lines and arrows indicating velocity direction.

Fig.(3) was derived from velocity potential using Bernoulli's equation relating pressure and velocity at different points in a flow. Here, it is apparent that locations where the flow was forced around the sides of the object incurred decreases in pressure. Fig.(4) illustrates how these pressure drops correspond to increases in velocity. This is consistent with Bernoulli's equation, which dictates that increases in velocity must be met with decreases in pressure along a streamline.

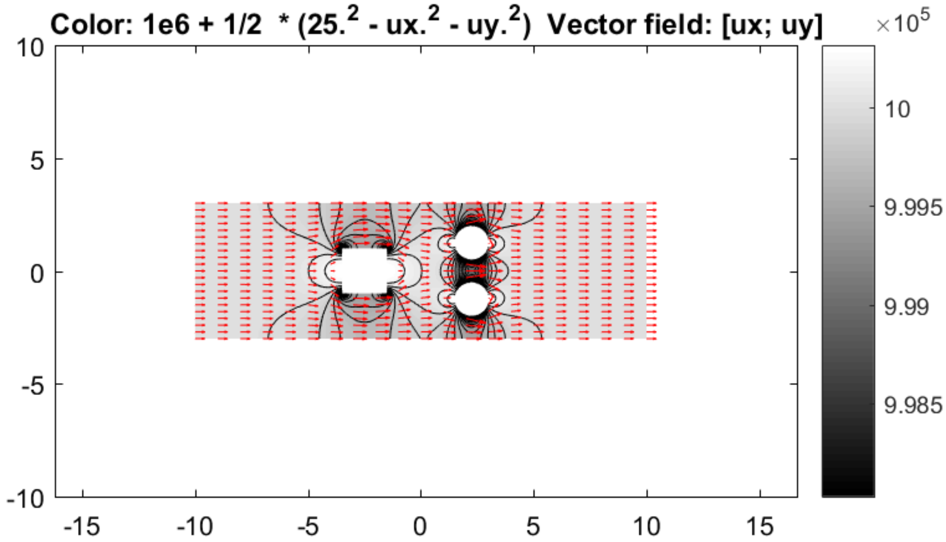

Figure 3. Pressure distribution gradient and contour lines overlaid with velocity arrows.

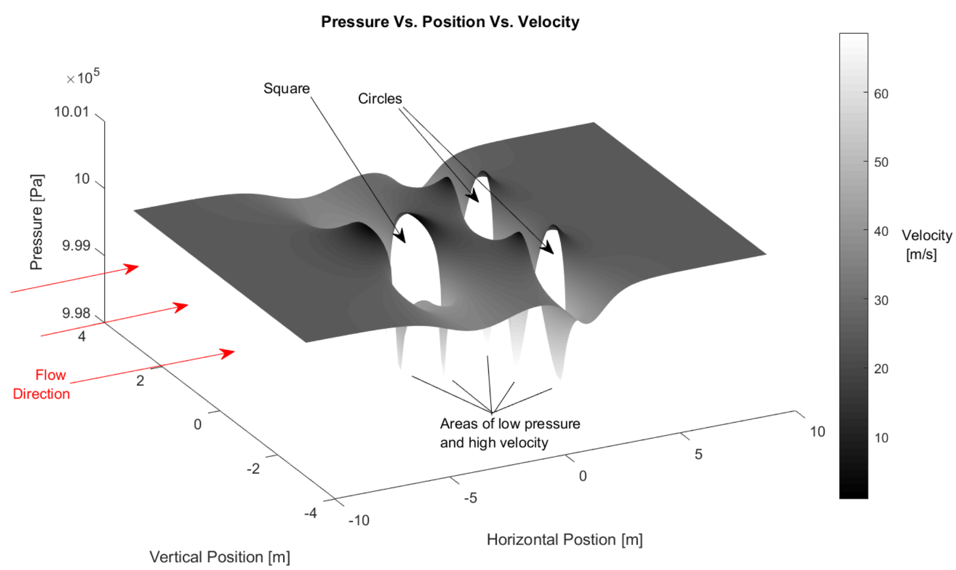

Figure 4. 3D representation of flow pressure with velocity color gradient. Indicates decreased pressure at areas of increased velocity.

Discussion

The results of the pressure distribution in Fig.(3) allow for some qualitative analysis about the forces acting on the objects. For instance, notice that the pressure values in front of and behind the objects are identical. Fig.(6) shows this more clearly without the clutter of the contour lines and arrows (located in the Appendix).

Similarly to the pressure ahead of and behind the objects, the pressure distribution above and below the objects are equivalent. Lastly, it's apparent that there are no pressure increases along the walls of the test section (which would indicate lower velocity due to friction with the wall). This implies that no objects in this scenario experience lift or drag forces. These finding are not only counterintuitive, but conflict with thousands of wind tunnel tests in which, at the very least, drag is measured acting on test models.

The reason for the discrepancy between actual and simulated results is based in the fundamentals of the simulation. Recall that potential flow must be irrotational, and irrotational flows are inviscid by definition. This is explained by the lack of shear forces on fluid elements in the flow, which would incur shear strains and angular velocities in the flow. Since fluid elements in an irrotational flow have no angular velocities, there must be no shear forces, and subsequently no friction. This is sufficient to explain the lack of drag on the walls of the test section, but more discussion is required for the drag on the objects.

A secondary result of the inviscid flow is the lack of boundary layer effects, and more specifically, the lack of flow separation. The lower velocity flow in a boundary layer experiences difficulty moving through the adverse pressure gradient created by a body. This results in the flow reversing direction and separating entirely from the body, causing a low pressure area behind the object. This is not apparent in this simulation, resulting in the lack of pressure drag, and overall drag on the objects in the flow.

Comparing plots with different levels of mesh refinement also provides insight to the limitations and capabilities of the PDE solver. Fig.(5) shows how refining the mesh used for solving the PDE drastically changes the results. In Subfigure (a), the contour lines are jagged and asymmetric. Also the color gradient is very shallow, not indicating sharp pressure drops. Subfigure (d) then shows much smoother contour lines with a steeper color gradient, which is more consistent with the actual flow fields. This indicates that increased mesh density leads to more accurate simulation results.

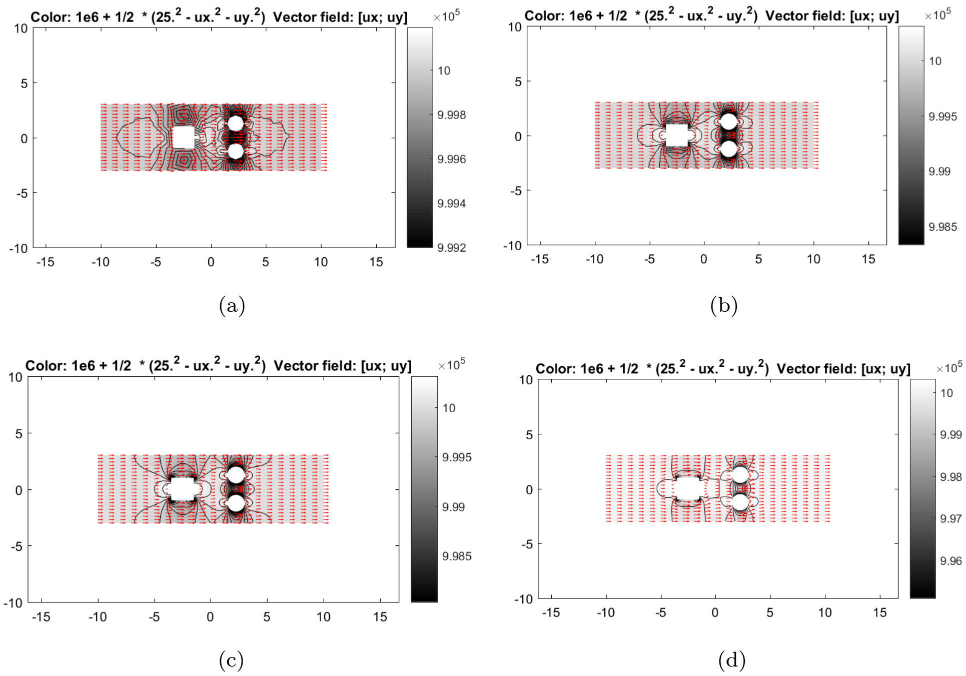

Figure 5. The four figures show different levels of mesh refinement: (a) shows mesh with 0 refinements; (b) shows mesh with 2 refinements; (c) shows mesh with 4 refinements; and, (d) shows mesh with 6 refinements.

Appendix

A. Extra Plots

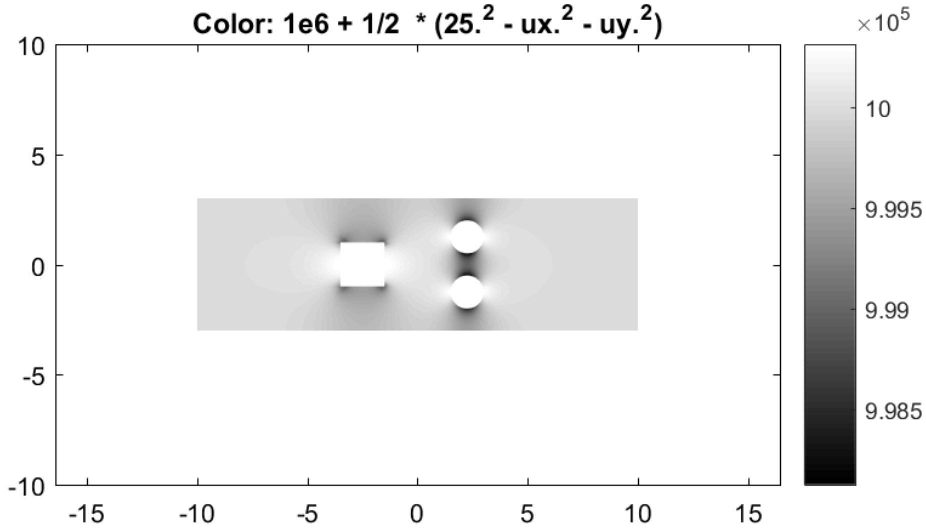

Figure 6. Pressure gradient sans contour lines and arrows.

References

[1]Evans, John. 'Part 1 - Problem Statment - Vimeo'. 2017. Available: `https://vimeo.com/233924231`.

[2]Evans, John. 'Part 2 - MATLAB - Vimeo'. 2017. Available: `https://vimeo.com/233925502`.

[3]Evans, John. *ASEN 3111 Computational Lab #2: Computation of Potential Flow.* D2L. Available: `https://learn.colorado.edu/d2l/le/content/217338/viewContent/3191473/View?ou=217338`