Modeling Two Dimensional Incompressible Fluid Flow with the Navier Stokes Equations

Mark Zimmerman

Physics Department, The College of Wooster, Wooster, Ohio 44691, USA

(Dated: May 9, 2008)

The flow of a fluid around a barrier was simulated by numerical integration of the Navier-Stokes equations. The methods utilized include forward time central space finite differencing, a staggered computational grid, and the Euler-Cromer algorithm, which allowed for the observation of well known effects such as reflection, interference, and vortices. More complex and striking features, such as low velocity zones, vortex eyes, and high velocity tails preceding the vortices, were also revealed by the simulation.

I. INTRODUCTION

The properties of fluids are so different from the less elusive solid environment that people normally see, touch, and otherwise experience that the flow of fluids is of almost universal fascination. People create fountains in cities, seek out waterfalls in the wilderness, sit by lakes or streams, and raft down raging rapids. All of these interactions involve a fascination with water, whether the flow is controlled or natural.

Fluid flow can be classified by three main regimes. There is the regime of laminar flow, which involves the most gradual passage of water, such as waves on a lake. An intermediate regime develops as more complex structures form, including Von Kármán vortex streets, which are trails of whirlpools caused by interaction with an object in the flow (Fig. 1). The last regime is that of turbulence (Fig. 2).

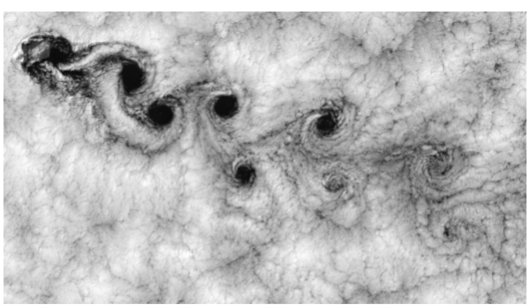

FIG. 1 Photograph of Von Kármán vortex street off the coast of Chile near the Juan Fernandez Islands as seen from space. The vortices are formed in clouds. Image obtained from reference (1).

These regimes of fluid flow can be characterized by different ranges of Reynolds numbers $\mathcal{R}$, which are approximately equal to the inertia of the fluid divided by its viscosity. For laminar flow, the Reynolds number is low. An increase in the Reynolds number leads to more complex behavior as the flow of the fluid moves toward turbulence.

Turbulence is a truly different state than the laminar flow of a fluid, which is well organized and symmetric. Similarly, Von Kármán vortex streets are symmetric. One eddy emerges on the left of an object, then one on the right, then one on the left, and so on. Unlike these previous variations of fluid flow, turbulence appears completely chaotic and not symmetric. The questions emerge: why does the symmetry break? Where does the turbulence come from?

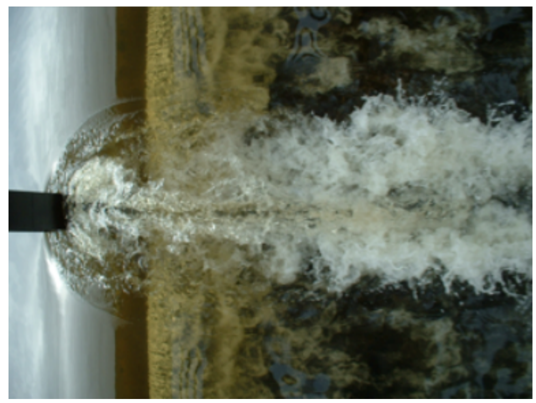

FIG. 2 Photograph of turbulence in water after passing an obstacle. Image obtained from reference (1).

The answer to these questions is believed to reside somewhere in two innocuous looking second order partial differential equations called the Navier-Stokes equations for an incompressible fluid:

$$\frac{d\vec{v}}{dt} = -\overrightarrow{\nabla} p + \frac{1}{\mathcal{R}}\nabla^2 \vec{v} \quad (1)$$

$$\overrightarrow{\nabla} \cdot \vec{v} = 0. \quad (2)$$

Eq. 1 is Newton's law of conservation of momentum, and Eq. 2 is a conservation of mass equation for an incompressible fluid; together, these equations are the mathematical representation of the physical reality of fluids. Somewhere in these equations is the answer to the question of why turbulence occurs, why chaos ensues, though where precisely is yet a mystery.

This project approaches this mystery from a computational position with the intention of investigating the nature of fluid flow by observation of the transition from laminar flow to turbulence. To accomplish this aim a two dimensional virtual wind tunnel was created, the concept of which is fairly simple. It is a rectangle which allows for the passage of fluid from one end to the other. In the center of the wind tunnel, there is an object (vertical line) for the fluid to flow around. By observation of the behavior of the fluid in this tunnel information can be gained about the nature of fluid flow around an obstacle, which with high enough Reynolds numbers, produces Von Kármán vortex streets and turbulence.

II. THEORY

Deeper discussion of fluid flow requires additional understanding of the characteristics and motion of fluids, particularly with respect to their compressibility and response to shear and pressure forces.

A. Compression

The first of these three topics, compressibility, asks the question of whether the density of the fluid can change. Since density is the ratio of mass to volume, this is asking if one can fit more fluid into a space than was previously present by application of pressure. Considering an everyday example of a fluid, water, it is observed that it is difficult to fit more water into a space than fills that space naturally. This fluid is not perfectly, but nearly, incompressible. The opposite of an incompressible fluid is a compressible fluid, for example, any common gas. All fluid is somewhat compressible, which becomes particularly evident when a fluid travels a significant portion of the speed of sound, as there is compression due to shock waves. However, in this simulation, the simplifying choice of using an incompressible fluid will be made, which implies that the density of the fluids studied will be constant and the fluid will travel less than the speed of sound.

B. Shear

Though essential to this simulation, the incompressibility of the fluid is more a description of how it does not move than of how it does. To broach the idea of how fluid does move, shear force is considered, which is the force resultant from action tangential to the fluid. The strength of the shear force depends heavily on the viscosity, which is the amount of clinginess that each molecule exerts on all of the others by friction. The more attachment each molecule has on the others around it the more the fluid will remain together and the greater the influence of shear force. Fluids of lower viscosity, such as water or gasoline, are observed to be less effected by shear force in comparison to fluids of greater viscosity, such as molasses and pudding.

C. Pressure

Further description of the motion of a fluid may be obtained through the consideration of the pressure force, which is a force acting perpendicularly on a fluid and resulting from the objects and other fluid surrounding the fluid in question. Since the area surrounding each bit of fluid is not uniform in most situations, there is a gradient of zones with higher and lower pressures. These differences in pressure cause the fluid from higher pressure zones to be forced into areas of lower pressure, which then places more pressure on the fluid in the lower pressure areas, causing movement throughout the fluid in a chain reaction until the pressure reaches equilibrium. If the fluid is not contained but moves continuously, such as in a stream or wind tunnel, then equilibrium is never reached and the fluid will continually experience pressure force.

D. The Navier-Stokes Equations

The first of the Navier-Stokes equations for an incompressible fluid represents the incompressibility of the fluid, and may be stated that the divergence of the velocity must equal zero

$$\overrightarrow{\nabla} \cdot \vec{v} = 0. \tag{3}$$

This equation also signifies conservation of mass for the fluid, which is recognized more easily when this equation is multiplied by the density of the fluid, and thus refined to state that the divergence of the mass current equals zero.

The second of the Navier-Stokes equations comes from Newton's law of conservation of momentum. As only shear and pressure forces are at work on the fluid, this can be written for a unit volume of fluid as

$$\rho \frac{d\vec{v}}{dt} = \vec{f}_p + \vec{f}_s, \tag{4}$$

and rewritten as

$$\rho \frac{d\vec{v}}{dt} = -\overrightarrow{\nabla} p + \eta \overrightarrow{\nabla^2} \vec{v} \tag{5}$$

by use of derived equations for shear and pressure forces.

By recognizing that the total derivative of the velocity receives contributions from the change of the velocity of the particle and also the velocity of the fluid at a point, Eq. 5 can be expanded to complete the Navier-Stokes equations for an incompressible fluid

$$\begin{aligned} \frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \overrightarrow{\nabla} \vec{v} &= -\frac{1}{\rho} \overrightarrow{\nabla} p + \frac{\eta}{\rho} \overrightarrow{\nabla^2} \vec{v} && (6) \\ \overrightarrow{\nabla} \cdot \vec{v} &= 0. && (7) \end{aligned}$$

For convenience, this initial derivation the Navier-Stokes formulas may be manipulated into a variety of forms. The form of choice in this simulation consists of three coupled, non-dimensional, partial differential equations proposed by Hoffmann and Chiang (2). The first equation describes the conservation of mass and the other two equations are conservation of momentum equations from Newton's law as follows

$$\begin{aligned} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \\ \frac{\partial u}{\partial t} + \frac{\partial}{\partial x}\left(u^2 + p\right) + \frac{\partial}{\partial y}(uv) &= \frac{1}{\mathcal{R}}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \\ \frac{\partial v}{\partial t} + \frac{\partial}{\partial x}(uv) + \frac{\partial}{\partial y}\left(v^2 + p\right) &= \frac{1}{\mathcal{R}}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right), \end{aligned} \tag{8}$$

and where the velocity has been broken into components such that $\vec{v} = \hat{x}u + \hat{y}v$ and $\mathcal{R} = \rho_\infty V_\infty L/\mu_\infty$. Here, ρ_∞, V_∞, and μ_∞ are the density, velocity, and viscosity at an infinite distance from the barrier interaction, and L is a length scale. Together, these three equations are equivalent to Eqns. 6 & 7.

III. DEVELOPMENT

The implementation of what has been developed in the theory began with the staggered computational grid. The second major aspect was the forward time central space differencing scheme (FTCS) used to numerically integrate the Navier-Stokes equations by the Euler-Cromer algorithm. Next, the boundary conditions were implemented with periodic boundaries along the sides of the tunnel and fans pointed in the x direction at the ends. Lastly, a barrier was inserted into the middle of the tunnel, as the goal of this simulation is to study flow around a barrier as a way to investigate the origins of turbulence.

The perhaps least known method was the implementation of a staggered computational grid of which a diagram is shown in Fig. 3. The basic concept of the staggered grid is that different types of information are recorded for different vertices on the grid, serving as a complicated coordinate system. In this simulation, pressure information is recorded on the primary grid (blue solid lines), while the velocity is recorded for intersections of the secondary grid (dashed red lines) with the primary grid. The x components of velocity u are recorded for the 1/2 grid lines in the $\hat{\imath}$ direction and the whole grid lines in the $\hat{\jmath}$ direction, while the y components of velocity v are recorded oppositely.

By using this scheme, the stability of the program is improved over a more simple single grid, through a strengthening of coupling between the pressure and velocity variables (2). Also advantageously, this grid simplifies the definition of pressure and velocity boundaries, such as the barrier which the fluid flows around. While the velocity of the fluid at the surface of a solid is well known to be zero, the pressure at the intersection of a fluid with a solid is not. Rather than attempt to answer that question, it is avoided by the use of a staggered grid, which at the boundaries does not require pressure definition.

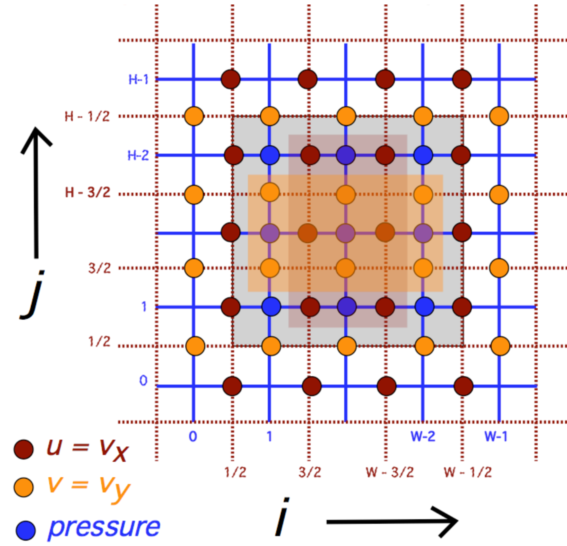

FIG. 3 Diagram of the staggered grid. The primary grid is shown as solid blue lines, whose vertices hold the pressure. The secondary grid is shown as dotted red lines, and the intersections of this grid with the primary grid hold the velocity information. The area, excluding the boundaries, over which pressure is defined is shown as a blue square, the area of definition for the y component velocities in an orange horizontal rectangle, and the area for the x component velocities as a red vertical rectangle. Points outside these rectangles are boundaries for the system. Diagram provided by reference (3).

A second consideration in the construction of the simulation was the set of initial conditions to use for the fluid. To remain simple, the initial pressure everywhere was defined to equal one. The most economical option for the velocity was to begin the fluid flowing at the velocity of the fan, immediately starting the system rather than necessitating a period of waiting before the collision of the fluid with the barrier.

IV. DATA & RESULTS

The basic progression of the program is that the fluid interacts with the barrier, then vortices form, and lastly these become more pronounced as the fluid flow across the entire simulation is more effected.

The flow provided in Figure 4 for observation was simulated at a Reynolds number of 280. Each point is colored based on the magnitude of the velocity $\sqrt{u^2 + v^2}$. The dark blue vertical line is the barrier, at which the velocity is by physical requirement zero.

The wave that propagates back from the barrier as a result of the interaction with the wall in the middle picture of Fig. 4 is worth noting, and a testament to the reality of this simulation, because a wave would propagate back from the barrier radiating away from the collision in reality. On the left edge of the simulation, one reflected wave interacts with another producing constructive and destructive interference. Additionally, it is interesting that at the center of the barrier there is a small bubble of very low velocity, and another small low velocity circle slightly after and between the vortices (seen as the brown loops).

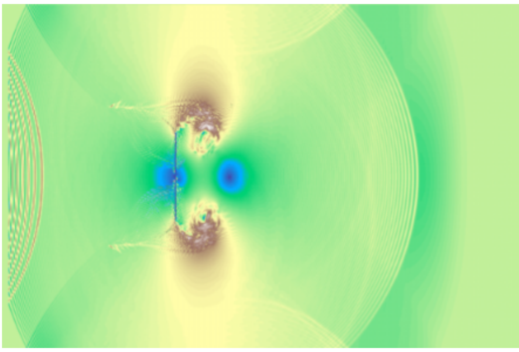

FIG. 4 The colored speed field resulting from a Reynolds number of 280. Blue represents low velocities, green middle velocities, brown higher, and white highest.

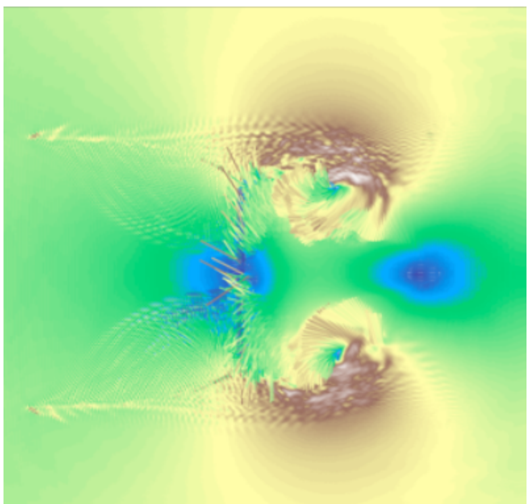

FIG. 5 Closer view of the same picture as Fig. 4. The image is drawn with the addition of velocity vectors, with length proportional to the velocity. High velocity trails are visually identified by yellowish lines preceding the vortexes and punctuated by a small brown speck.

Taking a closer look in Fig. 5, other locations of low velocity are observed at the center of the vortices, which seem particularly interesting as points of stability in an increasingly wild environment. Of additional note is the observation of higher velocity tails coming from the vortexes and culminating in a peak at the furthest end of the tail.

A last image to share is from a higher Reynolds number simulation (500) than the previous, and also with a smaller barrier than previously. Features of the last series of pictures are also very present in this higher Reynolds number case shown in Fig. 6. Two vortices have formed along each corner of the obstacle, a green wave is propagating to the left and a yellow wave to the right, and two low velocity zones are seen, one before the barrier and one immediately after the vortices. There are also little zones of calm in the vortex centers, similar to an eye of a hurricane, and small high velocity tails preceding the vortex.

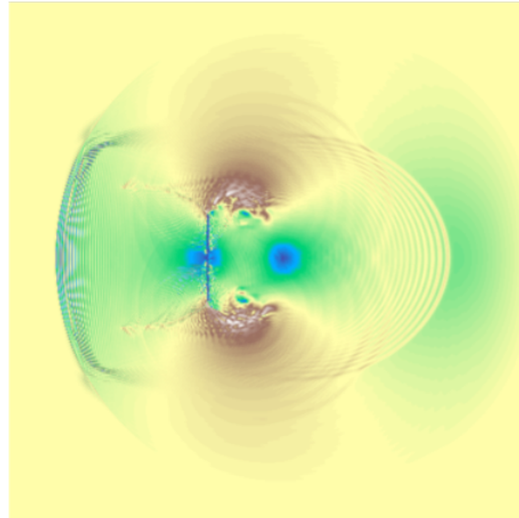

FIG. 6 Picture of a flow with Reynold's number 500. Demonstrates features of vortices, reflected waves, low velocity zones, and vortex eyes.

V. CONCLUSIONS

This simulation culminated in the observation of vortices resulting from fluid flow around a barrier. Additionally, low velocity eyes and high velocity tails were associated with the vortices, and low velocity areas were observed before and after the barrier interaction.

Overall, this project seems to have provided a successful initial visualization of fluid flow around a barrier. Further investigation of the high velocity trails would be interesting, but more important than this is increasing the Reynolds numbers to observe the beginning of turbulence. Future improvements would involve optimization of the integration and stability conditions for speed and accuracy. Other future work could also involve reworking the boundary conditions so that there would be no bouncing back from the sides of the simulation or the fan.

References

[1] *Wikipedia*. 2008. Wikipedia. 25 March 2008. `<http://en.wikipedia.org/>`.

[2] K.A. Hoffmann and S.T. Chiang, *Computational Fluid Dynamics*, Vol. 1, (Engineering Education System, Whichita, KS, 1998).

[3] J. Lindner, personal communication April 2008.