

# Applications of the Navier-Stokes Equations and Simulation of Taylor-Couette Flow

Dana Lucas

The applications within the field of fluid mechanics are relevant to daily life all around us. The behavior of fluid flow can be described by an equation known as the Navier-Stokes equation, and by analyzing the behavior of a fluid as it flows across a surface, the velocity and pressure of the fluid at certain points can be determined. This project works closely with fluid mechanics and the Navier-Stokes equations to describe fluid flow specifically related to Taylor-Couette flow, which is the flow of a fluid between two surfaces where one surface is at rest and the other is in motion. An animation to describe the velocity of particles traveling through two stationary plates was created and compared to a simulation describing Taylor-Couette Flow. Additionally, the velocity and pressure fields were drawn and compared by solving the Navier-Stokes equations for Taylor-Couette flow and also a more complicated velocity vector field. The importance of the no-slip boundary condition was described in the analysis of this project, as it greatly affected the fluid flow that was seen. The ANSYS Fluent software was also used in collaboration with this project to further explore fluid flow and verify some of the results that were determined.

## 1. Introduction

The Navier-Stokes equation is a differential equation that describes the motion of fluids using variables that influence flow such as pressure and velocity (1). The equation is defined as:

$$\rho\left(\frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla)\vec{v}\right) = -\nabla P + \rho \vec{g} + \mu \nabla^2 \vec{v} \text{ (Equation 1)}$$

where  $\rho$  is the density,  $\vec{v}$  is the velocity vector,  $t$  is time,  $P$  is pressure,  $\vec{g}$  is the acceleration due to gravity, and  $\mu$  is the viscosity of the fluid. The equation takes into account the gradient, curl, and divergence of variables such as the velocity and pressure in order to precisely describe the motion of a fluid and therefore makes the equation very complex. The gradient, often specified by the del symbol,  $\nabla$ , takes the partial derivative in respect to each dimensional direction. As one may recall, the derivative is a way of finding the slope, or rate of increase of a function, so taking the gradient is simply a way to find the vector that describes the greatest rate of increase of a function in multiple dimensions. The curl, specified by the cross product of a vector and the del operator, is a vector operation that describes the amount of rotation particles at a specific point will undergo. Similarly, the divergence is specified by the dot product of a vector and the del operator, and describes the amount of fluid that flows through a specific point and whether that fluid is moving towards or away from the point (2).

Most often, nonetheless, the Navier-Stokes equation can be simplified and used to derive certain relationships that one may want to study within the field of fluid mechanics. The acceleration due to gravity found in the equation, for example, can be assumed to be 0 when describing motion in two dimensions because gravity doesn't act in either the x or y directions.

This equation can be used to analyze anything from shock waves from airplanes to turbulence in water flow (3). Taylor-Couette Flow, specifically, is a topic that uses the understanding of the Navier-Stokes equations to analyze the effects of a fluid when it is enclosed by a stationary plate and a moving plate. Traditionally, a highly viscous fluid such as corn syrup is placed in between two hollow and concentric cylinders, representing the plates, and drops of food coloring are placed in the corn syrup to use as a tracker. As the inner cylinder is rotated, the

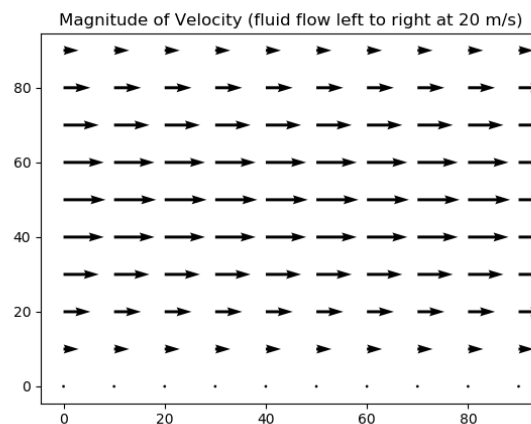
corn syrup close to the inner cylinder flows or curls around the cylinder while the corn syrup near the outer cylinder remains motionless as it adheres to the cylinder that remains at rest. When the inner cylinder is rotated in the opposite direction, it is seen that the process is reversible with very little diffusion of the food coloring. This process creates an interesting flow pattern that can be described with the Navier-Stokes equation as well as a continuity equation describing the velocity and several boundary conditions.

The Navier-Stokes equations offer the ability to look at various pressure gradients and velocity in fluids. The strategies taught in class will be implemented to create a 2D visualization of the velocity and pressure differences at various positions of a fluid contained in specific boundaries. The two major sets of boundaries will be tested. The first will be two stationary plates and the second will be a stationary plate and a moving plate. An animation will be created to describe the flow as a function of time in both a Cartesian and cylindrical coordinate setup. The areas of maximum velocity and pressure will be observed and the physics and engineering behind the results will be explained.

Additionally, many complex equations arise when analyzing the boundary conditions of fluid flow when it is contained within some set of walls. In order to effectively describe more advanced fluid flow, these boundary conditions must be accounted for. ANSYS Fluent software is a software that takes into account the mathematics and physics behind fluid flow and its boundaries, including those of irregular shapes, and will be used alongside the modeling to further understand some of the important engineering concepts that go along with the subject that have not yet been learned in engineering coursework.

## 2. Results

The quiver plot in Figure 1 below shows the velocity of particles at various locations between two stationary, horizontal plates that are set a distance 100 units apart. Each arrow represents the magnitude of the velocity at that particular location. The code used to produce this plot is located in Appendix 1.

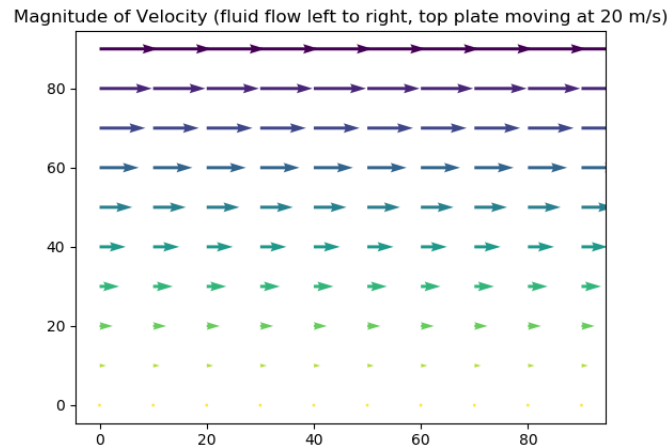


**Figure 1.** Quiver plot created on Spyder describing the velocity at various locations of a fluid flowing between two plates in laminar flow. Both plates are stationary

Figure 1 suggests that the velocity of a particle near the stationary plates, which are located at  $y=0$  and  $y=100$ , is close to zero. This is due to a condition called the no-slip condition, which says that a fluid in contact with another surface tends to adhere to that surface, and therefore the fluid at that position travels at the same velocity as the surface (1). At the exact

center between the two plates, the velocity is highest because the fluid particles are furthest from the stationary plates.

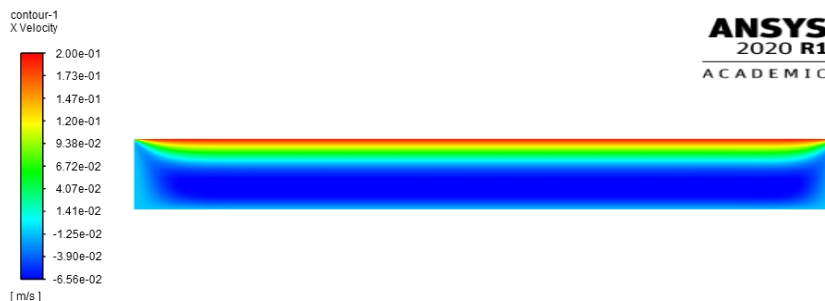
The quiver plot in Figure 2 below shows the effect of Taylor-Couette flow on the velocity of particles between two plates where the top plate is moving at some speed and the bottom plate is stationary. The code used to produce this plot is also located in Appendix 1.



**Figure 2.** Quiver plot created on Spyder describing the velocity at various locations of a fluid flowing between two plates in laminar flow. The top plate is moving, while the bottom plate is stationary.

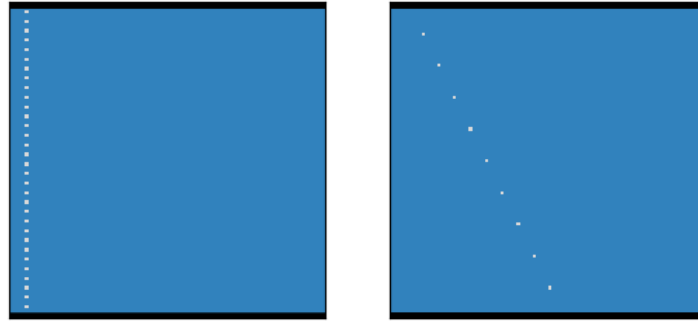
Similar to what was seen in Figure 1, Figure 2 suggests that the velocity near the bottom, stationary plate is 0. Additionally, the velocity of particles near the top, moving plate is the velocity of that plate. Once again, this is due to the no-slip condition. The plot further suggests that the velocity of a particle at any vertical position in between decreases linearly from the top plate to the bottom plate. In laminar flow, which is the idea that fluids flow smoothly across a surface with very little turbulence, this is an accurate depiction of what may be observed.

This trend was verified using ANSYS Fluent as shown in Figure 3. A sketch was created of a two-dimensional surface with a top moving wall and bottom stationary wall, similar to how the plot in Figure 2 was set up. Calculations were made through the software, and the results showed that the velocity of the fluid particles close to the moving surface matched the velocity of that surface exactly. Likewise, the velocity near the bottom, stationary plate was zero. Near the corners of the plate, however, the velocity tended to slow down, likely due to the fluid particles being obstructed by the boundary and momentarily being stopping as it either enters or leaves the plates.



**Figure 3.** Contour plot of velocity on ANSYS Fluent for Taylor-Couette flow between a top moving plate and stationary bottom plate. The top plate was moving at 0.2 m/s.

An animation was created on Spyder to visualize the position of particles in fluid flow through two horizontal stationary plates and the flow through a Taylor-Couette setup where one of the plates is moving. Figure 4 below shows a single frame from the animation, where the left image shows two stationary plates and the right image shows the bottom plate in motion. In the figure on the left, the fluid flows from left to right across the screen at a rate of 1 unit per time step between two plates that are depicted as the top and bottom boundaries of the box. In the right Taylor-Couette flow image, the bottom plate, or bottom boundary, is moving to the right at a speed of 10 units per time step in addition to the fluid moving from left to right across the screen at a rate of 1 unit per time step. The position of a single particle as it travels horizontally through the plates, assuming there is no movement of the particle in the vertical direction, is depicted by each of the white dots that move across the screen as the animation progresses.



**Figure 4.** One frame of an animation comparing the position of particles between two stationary plates (on the left) and one stationary plate and one moving plate (on the right) at a single time step.

As the simulation is played, it is seen that each of the particles in the setup on the left move at the same velocity during each frame, and therefore reach the right edge at the same time. In the setup on the right, however, the particles near the moving bottom plate accelerate and reach the right end sooner than the particles near the stationary top plate. Although the positions of the particles at each time step follow a linear trend, assuming that no turbulent flow occurs as each particle progresses through the plates, the time at which the particles reach the right end increases exponentially as particles closer to the top plate are considered.

The above simulation does not take into account the no-slip condition that states that particles tend to adhere to the boundaries it is contained in. Therefore, in the simulation where both plates are stationary, it will be expected that the particles near each of the plates will not move at all while the particles in the exact middle between the two plates will move at the largest velocity. Figure 5 below shows what the particles will look like if the no-slip condition is accounted for. In this figure, the profile of the velocity has a quadratic shape, verifying that the fluid near the center is largest and then falls off to a velocity of 0.



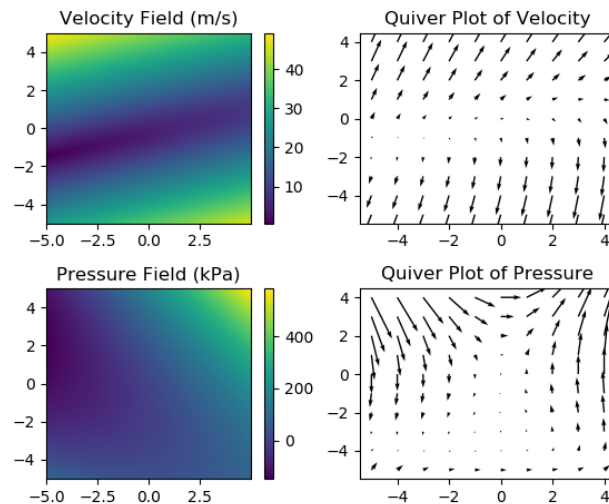
**Figure 5.** One frame of an animation showing the position of particles between two stationary plates at a single time step where the no-slip boundary condition is accounted for.

This calculation is true as long as the flow between the plates is considered laminar, meaning there is a smooth flow with no turbulence, and that the flow is fully-developed, meaning that the no-slip condition has already effected the velocity profile completely and that the fluid is no longer transitioning from being in open space to being in between two plates.

A lot of the analysis of the project thus far has focused on the velocity within the fluid. Although visualizing the velocity of fluid flow is a key step to understanding how fluids act within certain boundaries, the pressure at various points can also provide some insight as to how fluids flow. Given a certain velocity vector field, the pressure field within the fluid can be determined by analyzing the Navier-Stokes equation. Unfortunately, in Taylor-Couette flow an interesting pressure field does not occur since there is only a constant x-velocity near the top plate. Instead, a more complicated vector field was applied and Python was used to calculate what the pressure field would look like. Given an arbitrary velocity field of:

$$\mathbf{V} = (3x + 5)\hat{i} + (-2y + 7x)\hat{j}$$

the pressure field was determined. Figure 6 shows both the velocity and pressure fields graphed side by side on both a color map that shows the intensity by color and a quiver plot that shows the x and y components of the vector in addition to the magnitude.



**Figure 6:** The associated pressure field that goes along with an arbitrary vector field.

### 3. Methods

#### Modeling the Velocity Field Between Stationary Plates and in Taylor-Couette Flow

The quiver plots shown in Figures 1 and 2 describe the magnitude of velocity at various points within channel flow. In order to determine what the velocity at each location was, x and y coordinate points were fed into a definition that contains the formula for velocity with respect to location.

In Figure 1, the velocity profile equation written as Equation 8-17 in the Fluid Mechanics textbook was used [1]. This equation, describing the velocity as a function of distance from the center of the ‘pipe’, was:

$$u(r) = 2V_{avg} \left(1 - \frac{r^2}{R^2}\right) \text{ (Equation 2)}$$

where  $V_{avg}$  is the average velocity of the incoming fluid,  $R$  is half the distance between the two plates, or the radius if the figure is thought of as a section cut out of a circular pipe, and  $r$  is the distance from the point halfway between the two plates, or the center of the ‘pipe’, to the fluid particle. Because the  $r$  value is referenced from the halfway point in between the plates, some algebra had to be done to determine what the velocity would be with respect to the vertical distance from the top plate so that the distance can be related to the y coordinate of the array used to create the image.

The quiver plot in Figure 2 was created by feeding in first the coordinate points to identify the location of a certain particle within the fluid, and then the calculated velocity at that particular coordinate point. Therefore, the velocity at each point was quantified as the length of the arrow at that point. In order to easily access the magnitude of velocity at any point within the channel, the velocity at each point was saved to a dictionary as it was calculated. Upon entering a coordinate point as indexes of the dictionary, the velocity associated with that point was outputted.

#### Creating a Simulation Comparing Particles Flowing Through Two Plates

The animation described in Figure 4 was created by importing animation from matplotlib, a Python library that helps create visualizations of data through figures such as graphs and charts. A 100x100 array was created to describe various x and y locations of particles. At time step 0, the position in the first column of every few rows in the array was colored white to track individual particles through the plates without overcrowding the screen. In order to do this, the modulus operator was used.

Once the simulation was run, the animation was updated after each time step to display the new position of each particle as it traveled from left to right through the channel. When both plates were stationary, the new location of each particle was simply one column to the right since it was specified that the fluid was flowing at one unit per time step. When one of the plates was moving, a formula had to be created to determine where the new location of the particle will be because not only was the fluid itself being forced through at a certain velocity, but the moving plate was dragging the fluid forward as well. The amount a fluid particle moves as a result of the moving plate is described linearly as a function of how far away from the moving plate the particle is. Therefore, the new location is specified by the addition of the speed at which the fluid is being forced through, just as with the two stationary plates, and the vertical distance (or row that the particle is in) with respect to the moving plate multiplied by the speed of the moving plate. Additionally, since some of the particles were destined to reach the end of the channel

before the rest, code was devised to stop these particles in their place once they reach the end so that they don't return to the beginning of the channel and start again.

Because each position of the array is updated one by one, the update order had to be reversed so that the right-most columns were updated before the left columns. The column associated with each particle traveling through a certain row on the screen was increased by a certain amount each time step. Since the particles traveled from left to right, if the order updated from left to right as well, particles that were already updated during that time step would be updated again and would cause the animation to not work properly.

To clean up the graphs, the x and y axis marks were removed, and thick, horizontal lines were added to the top and bottom of each plot to signify the plates that the fluid is traveling through. In order for the horizontal axis lines to appear in every time step of the simulation, this portion of code had to be written within the definition that updates the animation after every time step. If it were written near the end of the code near the lines that remove the axis marks, then the thick, horizontal axis lines would only appear during the first time step. The code for the simulation is shown in Appendix 2.

### **Accounting for the No-Slip Condition in the Simulation**

The no-slip condition was applied to the simulation in describing fluid particles flowing between two stationary plates by modifying the code used to produce Figure 4. Equation 2 was used to determine how far each particle should advance through the plates, or the velocity of each particle, at each additional timestep. A new simulation was created as shown in Figure 5 and the code used to produce it is shown in Appendix 3.

### **Deriving a Pressure Field from a Given Velocity Field**

The graphs in Figure 6 were created by a direct analysis of the Navier-Stokes equations. Given the Navier-Stokes equation, which is Equation 1 in the introduction section, the acceleration due to gravity is zero since the x and y directions are only being considered. If each component of the vector field is of order 1 or less, then the viscosity term in the Navier-Stokes equation will also be zero because taking the 2<sup>nd</sup> partial derivative of the velocity with respect to each component will be zero, and this is essentially what the square of the gradient does. Therefore, the partial derivative of the pressure with respect to each the x and y component can be determined as follows:

$$\frac{\partial P}{\partial x} = -\rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) \text{ (Equation 3)}$$

where u is the x component of the velocity field and v is the y component of the velocity field. The partial of pressure with respect to y can be determined through a similar analysis. By inputting the vector field in Python, SymPy can be used to find the partial derivative of each term to plug in to Equation 3. From there, the integral can be taken of both to determine the pressure field.

Once this was done, an equation for both the velocity field and the pressure field were known. Each were graphed on the same figure using Matplotlib to easily visualize the pressure field that is associated with each vector field. Both color mesh plots and quiver plots were shown, as each effectively showed different aspects of the field: the color mesh plot worked best to specify to magnitude of the field at a certain location while the quiver plot was better at showing the direction of the vector at each location. The code used to create this portion of the

project is in Appendix 4.

#### 4. Discussion

Fluid flow was described in two scenarios throughout this project: one describing the flow between two stationary plates and one describing Taylor-Couette flow where one of the plates is moving. The velocity and pressures at various points within the fluid were analyzed to understand the behavior of the fluid as it flows. It was also observed that boundary conditions play a large role in analyzing the behavior of fluids and although these may involve long and tedious mathematics, they can be effectively modeled for this flow.

By analyzing the Navier-Stokes equations, the velocity field of any fluid flow can be related to its pressure field. This concept was analyzed near the end of the project using velocity fields that were more complicated than that created through Taylor-Couette flow. The computations for this would have been lengthy by hand but were much easier when solved in Python. Graphs of the velocity and pressure vector fields were created side by side in multiple forms to visualize what each looks like. Analyzing the locations of maximum pressure is an important aspect of mechanical design. For example, it is important to know how much pressure a pipe wall can endure before it explodes. Infrastructure is designed and built around these areas of high velocity and pressure and simulations similar to the one created here that uses complex mathematical formulas makes finding these values much more efficient.

A lot of the project relies on analyzing boundary conditions that describe how the fluid acts near the walls it is contained in rather than at the center of the system. The no-slip boundary condition was taken into account in the above figures, but this condition will vary slightly depending on factors such as the surface material and viscosity fluid assuming that the fluid particles will adhere to the surface differently. In general, if both plates are stationary, the fluid near the plates would be fairly close to the velocity of the plates and the maximum velocity in the channel was at the center of the channel where the fluid is furthest away from both the plates. Therefore, a more quadratic velocity profile was seen rather than the constant profile that would be expected if the boundaries weren't there.

The Navier-Stokes equation is very useful for fluid mechanics because it can be used to describe a large range of situations. Additionally, it can be used along with various other engineering relationships and concepts to analyze other variables related to fluid flow. For example, the viscosity of the fluid can be found by analyzing the shear stress on the fluid near the cylinders as learned in engineering courses. Because the torque is equivalent to the shear stress multiplied by the area, a relationship between viscosity and torque can be described as:

$$\mu = T_{applied} \frac{(R_o - R_i)}{2\pi\omega R_i^3 L}$$

where  $T_{applied}$  is the torque applied to rotate the inner cylinder,  $R$  is the radius of the outer or inner cylinder,  $\omega$  is the angular velocity of the inner cylinder, and  $L$  is the depth of the cylinder. These derivations can be used alongside the analysis of the Navier-Stokes equations to further describe the physics behind the behavior of fluid flow. Therefore, the extent to which the Navier-Stokes equation can be used to model fluid flow is endless and there can always be an extension to the code that will give a better understanding of fluid mechanics.



## 5. References

1. Y. A. Cengel, J. M. Cimbala, “Differential Analysis of Fluid Flow” in Fluid Mechanics: Fundamentals and Applications (McGraw-Hill Education, New York, ed 3, 2014), pp. 8-9, 437-497.
2. P. Dawkins, “Curl and Divergence” (Lamar University, 2020; <http://tutorial.math.lamar.edu/Classes/CalcIII/CurlDivergence.aspx>).
3. W. L. Hosch, Navier-Stokes equation. (Britannica, 2018; <https://www.britannica.com/science/Navier-Stokes-equation>).

## 6. Appendices

Appendix 1: *Lucas\_SciModProject\_QuiverPlot.py* Quiver plots showing the magnitude of velocity at various coordinates between a moving plate and a stationary plate and two stationary plates.

Appendix 2: *Lucas\_SciModProject\_FlowVelocityComparisons.py* An animation of the fluid flow between two plates that compares traditional flow and Taylor-Couette flow.

Appendix 3: *Lucas\_SciModProject\_No-SlipStationaryPlates.py* An animation of the fluid flow between two stationary plates that takes into account the no-slip condition.

Appendix 4: *Lucas\_SciModProject\_VelocityField-PressureField.py* A figure of four graphs that show color mesh plots and quiver plots of a velocity field and its associated vector field.