

# Computer Modeling of the First Critical Reynolds Number of Flow around a Two Dimensional Cylinder

Brad Steele  
Physics Senior Seminar  
Berea College

Fall 2009

## Abstract

In this paper, we will show how to model the flow around a cylindrical object on a computer by approximating the Navier Stokes equations with the finite difference method. We will do this to find the Reynolds number where the first transition to turbulence occurs. Our results are consistent with what has been found elsewhere that the first transition from creeping laminar flow exists from  $(0 < Re < 4)$  and laminar flow with steady separation exists from  $(4 < Re < 50)$ . Transitions at higher Reynolds number was not achieved because of lack of computational power and the lack of sophisticated computational techniques. A square two-dimensional lattice was implemented on MatLab with a fixed ratio of the diameter to the length of the lattice of  $\frac{D}{L} = \frac{1}{6}$ . The transitional Reynolds number at different ratios could have been found, but there was not enough time to complete a detailed analysis of such variations.

## 1 Introduction

Despite numerous efforts made to model the flow around cylindrical objects there are still fundamental questions that remain unanswered. Most of these

questions involve investigating what happens to the flow at high velocities or low viscosities (high Reynolds number). Other research going on involves flow that is perpendicular to the stationary axis of the cylinder and the associated problems of heat and mass transport [2]. At the heart of the problem lies the infamous Navier Stokes equations that mathematicians have yet been able to prove solutions always exist. The fact that the Navier Stokes equations have not been proven to always have solutions without singularities and other various problems raises an interesting question about the equation's validity. Not to mention that the equations yield continuous solutions at all points in space when we really live in a discrete world, but that is besides the point. The equations can also model turbulent flow to a degree despite the chaotic nature of turbulence. Investigating how accurate these solutions are at higher velocities/lower viscosities is important to verify the equation's validity and usefulness.

## 1.1 Reynolds Number

The Reynolds number is a dimensionless quantity that measures the ratio of the velocity of the fluid to the viscosity of the fluid. It is defined as,

$$Re = \frac{Rv}{\nu}. \quad (1)$$

Where  $\nu$  is the kinematic viscosity defined as  $\nu = \frac{\mu}{\rho}$  where  $\mu$  is the viscosity of the fluid (explained later) and  $\rho$  is the density of the fluid.  $R$  as the characteristic length, which is the radius of the cylinder for flow around a cylinder. The characteristic length would be different for flow around other objects such as the diameter of a pipe for flow through a pipe. It does not matter whether we are talking about fluid flowing around a cylinder or a cylinder moving through a fluid the two situations are identical. The Reynolds number is extremely useful because it describes the relationship between inertial forces like the velocity and resistive forces like the viscosity. Higher Reynolds number could be achieved either by having a large velocity or a low viscosity. A high viscosity would be a fluid such as glycerin and a low viscosity would be like water. When the Reynolds number gets large enough the flow make a transition from smooth laminar flow to turbulent flow and the Reynolds number at which this transition takes place is what is being found here.

## 1.2 Five Flow Regimes

Recent literature [2] has shown experimentally and computationally that there exists five different states of the flow around cylinders. The first type of flow is appropriately called creeping laminar flow that is for  $(0 < Re < 4)$  where no vortices exist in the region just beyond the cylinder (called the wake). The next type of flow is also still considered laminar flow but with steady separation [2]. It contains two contra-rotating vortices in the wake and exists from  $(4 < Re < 48)$ . These two types of flows and the transition between them have been confirmed by this paper as well as the fact that the vortices begin to become longer and thinner with higher Reynolds number. The third flow regime  $(48 < Re < 180)$  have vortices that have created other smaller vortices, in a process called vortex shedding. The next regime is called the transition in wake regime where the transition to full turbulence exists  $(180 < Re < 400)$  with the last regime being complete turbulence with anything beyond a Reynolds number of 400. The last two regimes were too complicated for this simple program to model because of the high degree of precision needed. The Reynolds number that is reported in the literature will vary because of the different ratio of the diameter to the lattice used. As the radius of the cylinder to the length of the lattice decreases the Reynolds number where a transition is made will decrease.

## 2 Background

Anyone that has taken an introductory course in physics has probably done the Stokes-flow experiment where you drop balls of different diameter in a viscous fluid such as glycerin. This is the exact same situation that is being modeled in this numerical simulation except here we consider only the two-dimensional case and a range of velocities much greater than what is used in this experiment. This experiment demonstrates Stokes Law where the drag force on the ball is proportional to the velocity,

$$F_D = 6\pi\mu Rv. \tag{2}$$

Where  $\mu$  is the fluid's dynamic viscosity,  $R$  is the radius of the ball, and  $v$  is the velocity of the ball. This equation is only valid for low Reynolds number where the flow is purely laminar. The viscosity and density at room temperature for glycerin are about 1.49 Pas and  $1261 \frac{kg}{m^3}$  respectively, the

density of a steel ball that is typically used in this experiment is  $8000 \frac{kg}{m^3}$ , and the diameter of the ball is about 0.005m. By using this information and equating the gravitational force to the buoyant force plus the drag force the terminal velocity of the steel ball can be calculated to be about  $0.0625 \frac{m}{s}$ . This yields a Reynolds much smaller than one which is small enough to guarantee creeping laminar flow. The fact that the Reynolds number is significantly less than one is exactly what makes this experiment work so well because with the Reynolds number small enough laminar flow is guaranteed and turbulence is avoided. If a larger steel ball was used or if the glycerin was replaced with water, then the Reynolds number would be much higher and turbulence could become a problem. Finding where this transition takes place was the goal of this project because at higher Reynolds number the flow becomes turbulent and Stokes Law no longer applies.

## 2.1 The Navier Stokes Equations

The Navier Stokes equations are very powerful set of equations that describe the velocity of a Newtonian Fluid. A Newtonian Fluid is simply one that obeys the relation,

$$\tau = \mu \frac{du}{dy}. \quad (3)$$

Where  $\tau$  is the sheer stress applied to the fluid defined as force per area. What this means is that when a stress, or force is applied to a fluid it causes there to be a change in the velocity. This is in contrast to a solid where a stress will cause a deformation that will resist the stress until an equilibrium is reached. So fluids are more complicated than solids because describing them requires describing dynamics not statics. Most fluids are Newtonian fluids so they obey this relation such as water. However blood does not so it is not a Newtonian fluid and requires a more complex set of equations to model its dynamics. Here we only consider Newtonian fluids that are also incompressible. Incompressibility just means the density of the fluid is constant in time and space. By using the conservation of mass, or more specifically the continuity equation it can be shown that this just means,

$$\nabla \cdot \mathbf{v} = 0. \quad (4)$$

So the velocity of the fluid must be divergence less everywhere. As you will see this simplifies the problem greatly. The Navier Stokes equations

are thus derived from conservation of mass, momentum, and energy as well as assuming the fluid stress is proportional to the viscosity plus another pressure term. More information about this can be obtained in [1]. The resulting equation becomes,

$$\rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla P + \mu \nabla^2 \mathbf{v} + \mathbf{f}. \quad (5)$$

This combined with (4) form the holy grail of computational fluid dynamics. They are the equations we must approximate the solution to model the flow on a computer.  $\mathbf{f}$  is any external body forces per volume such as gravity exerted on the fluid, for the situation considered here  $\mathbf{f} = 0$ . This is a vector differential equation so there are actually three (in three dimensions) equations that need to be solved. Furthermore, it also involves the pressure field of the fluid which presents difficulty because the pressure varies with velocity. To top it off, it is a partial differential equation involving partial derivatives. Clearly this is a nasty equation.

Stokes Law is actually a solution to the partial differential equation at low velocities. By looking at (1) you can see that the velocity is directly proportional to the Reynolds number, so for small velocities you will have small Reynolds number. By looking at the convective term in (5),  $\mathbf{v} \cdot \nabla \mathbf{v}$ , notice it is proportional to  $\mathbf{v}^2$ . So when the velocity is small this term can be ignored. By ignoring this term the solution is approximately Stokes Law. This is why Stokes Law is only applicable for low Reynolds number because it is only the solution to the Navier Stokes equations at low Reynolds number or low velocity.

## 2.2 Bernoulli's Equation

To provide more confidence in the Navier Stokes equations as well as to show the sheer power of the equations we will show that the classic Bernoulli's equation is just another simplification of the Navier Stokes equations. The simplification is made by assuming the flow is inviscid,  $\mu = 0$  and the velocity does not change with time  $\frac{\partial \mathbf{v}}{\partial t} = 0$ . Notice that this does not mean that the velocity does not vary in space, it just does not vary in time at a particular point in space. By making this simplification (5) becomes,

$$\rho (\mathbf{v} \cdot \nabla \mathbf{v}) = -\nabla P + \mathbf{f},$$

where  $\mathbf{f} = \frac{mg}{V} = \rho g$  due to gravity in the negative  $z$  direction. So this can be written as minus the gradient of  $mgz$ . After simplifying the equation becomes,

$$\rho(\mathbf{v} \cdot \nabla \mathbf{v}) + \nabla(P + \rho gz) = 0.$$

By writing out the convective term it can be shown that  $\mathbf{v} \cdot \nabla \mathbf{v} = \frac{1}{2} \nabla v^2 - \mathbf{v} \times (\nabla \times \mathbf{v})$ . When the flow is rotation less  $\nabla \times \mathbf{v} = 0$ . This simplifies the equation to,

$$\nabla \left( \frac{1}{2} \rho v^2 + P + \rho gz \right) = 0.$$

This just implies that,

$$\frac{1}{2} \rho v^2 + P + \rho gz = C,$$

where  $C$  is a constant. Which is the classic Bernoulli equation.

### 2.3 The Stream Function

So the task at hand is to solve simultaneously equations (5) and (4). As stated before, the fact that the velocity field is divergence less simplifies the problem greatly because according to vector algebra when a field is divergence less, there must exist a function call it  $\Psi$  such that  $\nabla \times \Psi = \mathbf{v}$ . The function  $\Psi$  is also called the stream function because it when it is plotted on a contour plot it makes stream lines. To calculate it you just need to write out the curl as the determinant of the matrix below. Since we are only in two dimensions  $v_z = 0$ . Also to make our life simpler we can make the stream function in only one dimension, the dimension perpendicular to our velocity field (the  $z$ -dimension). So  $\mathbf{v}$  will be given by the matrix,

$$\begin{pmatrix} v_x \\ v_y \\ 0 \end{pmatrix} = \begin{vmatrix} i & j & k \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ 0 & 0 & \Psi \end{vmatrix}$$

This implies,

$$v_x = \frac{\partial \Psi}{\partial y} \quad v_y = -\frac{\partial \Psi}{\partial x}.$$

Where  $\Psi$  is now much easier to deal with because it is a scalar field. Not only that, but by giving  $\Psi$  values we automatically ensure that the velocity field is divergence less. Furthermore we can write (5) in terms of our new stream function with a little bit of algebra. By first writing the x and y components of the equation,

$$\rho \left( \frac{\partial v_x}{\partial t} + v_x \frac{\partial v_x}{\partial x} + v_y \frac{\partial v_x}{\partial y} \right) = -\frac{\partial P}{\partial x} + \mu \left( \frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} \right). \quad (6)$$

$$\rho \left( \frac{\partial v_y}{\partial t} + v_x \frac{\partial v_y}{\partial x} + v_y \frac{\partial v_y}{\partial y} \right) = -\frac{\partial P}{\partial y} + \mu \left( \frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} \right). \quad (7)$$

Then by taking  $\frac{\partial}{\partial y}$  of (6) minus  $\frac{\partial}{\partial x}$  of (7) it can be shown that the equation simplifies to,

$$\frac{\partial \nabla^2 \Psi}{\partial t} + \frac{\partial \Psi}{\partial y} \frac{\partial \nabla^2 \Psi}{\partial x} - \frac{\partial \Psi}{\partial x} \frac{\partial \nabla^2 \Psi}{\partial y} = \left( \frac{\mu}{\rho} \right) \nabla^4 \Psi. \quad (8)$$

As you can see the term involving the pressure is gone and the equation only depends on the scalar  $\Psi$ . This is also the reason why we define the kinematic viscosity to be  $\nu = \frac{\mu}{\rho}$ . An example of what the stream function would be you we can imagine a velocity field where the flow is all pointing in the positive x-direction, so  $\mathbf{v} = v_0 \hat{x}$ . In this case you can find the stream function by just integrating over the velocity. The stream function would then just be  $\Psi = v_0 y + C(x)$ , where  $C(x)$  is any function that depends on x. Since  $v_y$  is zero in this case,  $\frac{\partial \Psi}{\partial x} = 0$  so  $C(x)$  must also be zero.

## 2.4 Finite Difference Method

The way that we will be approximating the differential equation (8) is by using a technique called the finite difference method. The finite difference method is a way of approximating derivatives and should not be confused with the finite element method which is different. If the stream function is differentiable over the region we will be approximating it then we can Taylor expand it like so,

$$\Psi(x) = \Psi(x_0) + \frac{\partial \Psi(x_0)}{\partial x} \Delta x + \dots$$

If  $\Delta x$  is small enough than the Taylor is expansion is nearly exact and we can approximate the first derivative as,

$$\frac{\partial\Psi(x_0)}{\partial x} = \frac{\Psi(x)-\Psi(x_0)}{\Delta x}.$$

Depending on whether  $\Delta x$  is positive or negative will determine whether you are doing a *forward* approximation or a *backward* approximation. By combining the forward and backward approximations you can get an even better approximation. Written in terms of the discrete space coordinates on a lattice  $(i, j)$  the equation becomes,

$$\frac{\partial\Psi(i,j)}{\partial x} = \frac{\Psi(i+1,j)-\Psi(i-1,j)}{2\Delta x}.$$

It can then be shown as is done in [4] that higher order derivatives follow a predictable pattern. The formulas used here are;

$$\begin{aligned} \frac{\partial^2\Psi(i,j)}{\partial x^2} &= \frac{\Psi(i+1,j) - 2\Psi(i,j) + \Psi(i-1,j)}{2\Delta x^2} & (9) \\ \frac{\partial^3\Psi(i,j)}{\partial x^3} &= \frac{\Psi(i+2,j) - 3\Psi(i+1,j) + 3\Psi(i-1,j) - \Psi(i-2,j)}{4\Delta x^3} \\ &\vdots \\ &\vdots \end{aligned}$$

By using these approximations we can approximate equation (8) to find how the stream function changes with time. In this paper,  $\Delta x = 1$  to simplify things.

### 3 Method

In order to approximate the stream function we will use a square lattice with a section in the center where the stream function will be initialized to zero to represent a cylinder blocking the flow. We will initialize the stream function so that the velocity field will be a constant in the positive x-direction far away from the cylinder and zero inside of the cylinder and around the outer boundary of the lattice. In order for the function to converge quickly we will initialize the stream function around the cylinder to be approximately what it should be. We will do this by using the dimensionless function,

$$\Psi = Re \left( y \left( 1 - \frac{R^2}{x^2 + y^2} \right) \right). \quad (10)$$

Where  $Re$  is the Reynolds number and  $R$  is the radius of the cylinder. This function should be proportional to the initial velocity in the x-direction because (if you recall) the derivative with respect to  $y$  is equal to the velocity

in the x-direction. As you can see, when  $x^2 + y^2 \gg R^2$  the function is just equal to the Reynolds number times  $y$ . This is why we use the Reynolds number because it is proportional to the velocity times distance in dimensionless units (1). We use (10) to initialize the field at all points except for on the outer boundary and inside the circle. The resulting velocity field can be seen in figure (1) and the stream function in figure (2)

In most of the current research that is done on this topic they use a much more efficient lattice instead of the square lattice used here. As can be seen in [3] p.300 they are using a form of a polar lattice. This would be much more applicable because we are using a cylinder that has polar symmetry. In equation (10) you can see  $\Psi = 0$  on the cylinder. However, using a square lattice this would not be true, but on a polar lattice it would be so a polar lattice would be much better, but a square lattice was used for simplicity.

### 3.1 The Dynamic

After initializing the velocity field the next step is to find out what happens at a later time. We do this by solving for  $\frac{\partial \nabla^2 \Psi}{\partial t}$  in (8) using the finite difference method to approximate the derivatives. Once we find what the time rate of change is of  $\nabla^2 \Psi$  we can approximate what the new  $\nabla^2 \Psi$  will be by assuming it is linear over some small  $\Delta t$ . Keep in mind that since the derivative does not exist on the outer boundary we only do this for all the points within one lattice point of the boundary. Thus creating somewhat of a buffer zone. It is ok to do this because the outer boundary does not change much anyway. Now that we have the new  $\nabla^2 \Psi$  at some later time, we must now construct what  $\Psi$  actually is. By looking back at (9) it should be clear that this essentially amounts to solving the system of equations,

$$\begin{pmatrix} \frac{\partial^2 \Psi(1)}{\partial x^2} \\ \frac{\partial^2 \Psi(2)}{\partial x^2} \\ \cdot \\ \cdot \\ \frac{\partial^2 \Psi(N)}{\partial x^2} \end{pmatrix} = \frac{1}{2} \begin{pmatrix} -2 & 1 & 0 & 0 & \cdot & \cdot & 0 \\ 1 & -2 & 1 & 0 & \cdot & \cdot & 0 \\ \cdot & & \cdot & & & & \cdot \\ \cdot & & \cdot & & & & \cdot \\ 0 & 0 & \cdot & \cdot & 0 & 1 & -2 \end{pmatrix} \begin{pmatrix} \Psi(1) \\ \Psi(2) \\ \cdot \\ \cdot \\ \Psi(N) \end{pmatrix}$$

This is not an easy task for a 60 by 60 matrix. So what we do instead is diagonalize the matrix by using a Fourier transform. By using the Fourier transform the derivative can be calculated easily because the derivative is just a constant multiplied by the Fourier transform itself. Written algebraically,

$$F(k) = \int_{-\infty}^{\infty} f(x)w_N^{kx} dx \quad ikF(k) = \int_{-\infty}^{\infty} \frac{df(x)}{dx}w_N^{kx} dx.$$

Where we have used MatLab's convention where  $w_N = e^{-\frac{2\pi i}{N}}$ . The way we will go about solving for  $\Psi$  is by using the discrete inverse Fourier Transform of  $\nabla^2\Psi$  to solve for  $\Psi$ . That means we can calculate  $\nabla^2\Psi$  is, call it *Phi*. We will also call the inverse Fourier Transform of just  $\Psi$ ,  $\phi(k_x, k_y)$ . So *Phi* becomes,

$$\Phi(k_x, k_y) = \frac{1}{N^2} \sum_{i=1}^N \sum_{j=1}^N \nabla^2\Psi w_N^{-(k_x-1)(i-1)-(k_y-1)(j-1)}$$

Where if we write the Del squared out using finite differences,

$$\Phi(k_x, k_y) = \frac{1}{N^2} \sum_{i=1}^N \sum_{j=1}^N \left( \frac{\Psi(i+1,j)-2\Psi(i,j)+\Psi(i-1,j)}{2} + \dots \right) w_N^{-(k_x-1)(i-1)-(k_y-1)(j-1)}$$

Where we have excluded the y part because it is exactly analogous. Now if we make a substitution to evaluate the shifted inverse Fourier transform of  $\Psi$  we get,

$$\Phi(k_x, k_y) = \frac{1}{N^2} \sum_{i=2}^{N+1} \sum_{j=1}^N \frac{\Psi(i,j)}{2} w_N^{-(k_x-1)(i-1)-(k_y-1)(j-1)} w_N^{(k_x-1)} - \phi(k_x, k_y) + \frac{1}{N^2} \sum_{i=0}^{N-1} \sum_{j=1}^N \frac{\Psi(i,j)}{2} w_N^{-(k_x-1)(i-1)-(k_y-1)(j-1)} w_N^{-(k_x-1)} + \dots$$

Where the shifted ifft is the same as  $\phi(k_x, k_y)$  including an extra part and excluding another part,

$$\begin{aligned} \Phi(k_x, k_y) &= \left( \frac{\phi(k_x, k_y)}{2} + \frac{1}{N^2} \sum_{j=1}^N \frac{\Psi(N+1,j)}{2} w_N^{-(k_y-1)(j-1)} - \frac{1}{N^2} \sum_{j=1}^N \frac{\Psi(1,j)}{2} w_N^{-(k_y-1)(j-1)} \right) w_N^{(k_x-1)} - \\ &\phi(k_x, k_y) + \left( \frac{\phi(k_x, k_y)}{2} + \frac{1}{N^2} \sum_{j=1}^N \frac{\Psi(0,j)}{2} w_N^{(k_x-1)-(k_y-1)(j-1)} - \right. \\ &\left. \frac{1}{N^2} \sum_{j=1}^N \frac{\Psi(N,j)}{2} w_N^{-(k_x-1)(N-1)-(k_y-1)(j-1)} \right) w_N^{-(k_x-1)} + \dots \end{aligned}$$

combining the terms,

$$\begin{aligned} \Phi(k_x, k_y) &= \phi(k_x, k_y) \left( \cos\left(\frac{2\pi}{N}(k_x - 1)\right) + \cos\left(\frac{2\pi}{N}(k_y - 1)\right) - 2 \right) \\ &+ \frac{1}{2N} \left( [\mathbf{ifft}(\Psi(N+1, j)) - \mathbf{ifft}(\Psi(1, j))] w_N^{(k_x-1)} + [\mathbf{ifft}(\Psi(i, N+1)) - \mathbf{ifft}(\Psi(i, 1))] w_N^{(k_y-1)} \right) + \\ &\frac{1}{2N} [\mathbf{ifft}(\Psi(0, j)) - \mathbf{ifft}(\Psi(N, j)) + \mathbf{ifft}(\Psi(i, 0)) - \mathbf{ifft}(\Psi(i, N))]. \end{aligned}$$

Where  $\mathbf{ifft}(f)$  is the inverse Fourier transform of a function. By solving the above equation for  $\phi(k_x, k_y)$  we can find the original stream function  $\Psi$  because  $\phi$  is the 2-D ifft of  $\Psi$ . So all we need to do is take the 2D

Fourier Transform of  $\phi$  and we obtain our stream function  $\Psi$ . Notice in the above equation we must use  $\Psi$  to calculate  $\phi$ , which might seem like backwards logic. However, we only need the stream function on the outer boundaries which do not change with time. So we can just the function (10) to approximate what  $\Psi$  should be along those boundaries. Also notice that there is a singularity at  $k_x = 1$  and  $k_y = 1$  making it impossible to find  $\phi(1, 1)$ . It should be  $\sum_{j=1}^N \sum_{i=1}^N \Psi(i, j)$  because the exponential part goes away at one, one. This matrix does not depend on  $k_x$  and  $k_y$  so it a constant throughout the whole matrix where the constant is equal to the summation over the whole stream function. Th effect this has on the stream function is it just shifts it by some constant value everywhere in the lattice even in the cylinder! This is why at the end of the dynamic we make the stream function inside of the cylinder zero again. Shifting the entire stream function by a constant is not a problem though because we are only interested in the rate of change of the stream function, so a constant has no effect. That is all there is to the program. It is actually quite simple the equations are just long and complicated.

## 4 Results and Analysis

Creeping laminar flow was observed between ( $0 < Re < 5$ ). The transition to the next state can easily be seen by looking at the resulting velocity field. If there are a pair of vortices in the wake, then the transition to the next state has been made. A detailed analysis of the appropriate time that the field takes to converge to the steady state was not made, but is not absolutely necessary to notice the transition because as shown below the vortices have clearly developed at  $Re = 5$ . All of these simulations were run with the ratio of the diameter to the length of the lattice set to  $\frac{1}{6}$  on a  $60 \times 60$  lattice with a time step of  $\Delta t = 0.0001$ . The ratio of  $\frac{1}{6}$  was chosen simply because results could be seen using this ratio and because the radius should be somewhat small compared to the whole lattice.  $\Delta t$  was chosen because it needed to be at least smaller than one and small enough to approximate the differential equation. It was found that 0.0001 gave the best results so it was used throughout the simulation. Usually results were obtained over a relatively small time scale of  $T = 10,000$ . This is relatively small because of the small time step used. The transition can be seen in figures (3), (4), and (5).

One of the things researchers look at in the wake is the size of the vortice.

As is shown in [3] the length of the vortice will grow larger while the width of it does not change much with increasing Reynolds number making it thinner and thinner. This has also been noticed in this program in the second phase of the flow ( $10 < Re < 40$ ). As can be seen in the figures 6, 7, 8, 9 in the same time span  $T = 10,000$ , the vortices that have developed are indeed growing with increasing Reynolds number. After a Reynolds number of forty the program seemed to be unstable. Possibly because of the error involved when we calculate the derivatives. If the actual function varies greatly over the interval when we approximate the derivative, then the approximation will not be close at all. This would be magnified by the square lattice we are using instead of a polar-based lattice.

## 5 Conclusion

A more concrete way of measuring the turbulence than just looking at the velocity field is done in research. Usually a quantity that is called the vorticity defined as,

$$\omega = \nabla \times \mathbf{v}$$

is used. This quantity is zero around any closed loop when the velocity is all pointing in one direction. When the velocity is rotating about a point the vorticity is certainly not zero, but neither is it when the velocity is not all exactly pointing in one direction. There is no telltale sign involving the vorticity that will show definitively that the velocity is rotating about a point so there was no detailed analysis of the vorticity.

The simplicity of this program and the fact that it can determine the transitional Reynolds number with fairly good accuracy according to [2] at low Reynolds number demonstrates the usefulness of the Navier Stokes equation. Even with a rough approximation it can still model the beginning stages of turbulence accurately. It remains to be seen whether it can accurately model the last stage of turbulence with Reynolds number exceeding 400.

## References

- [1] John J. Bloomer. *Practical Fluid Mechanics for Engineering*. Library of Congress, 2000.

- [2] A. Kandasamy B.N. Rajania and Sekhar Majumdar. Numerical simulation of laminar flow past a circular cylinder. *CTFD Division, National Aerospace Laboratories, India*, 2007.
- [3] Bengt Fornberg. Steady viscous flow past a circular cylinder up to reynolds number 600. *Journal of Computational Physics*, November 1985.
- [4] Eric W. Weisstein. Finite difference. from mathworld—a wolfram web resource. <http://mathworld.wolfram.com/FiniteDifference.html>.

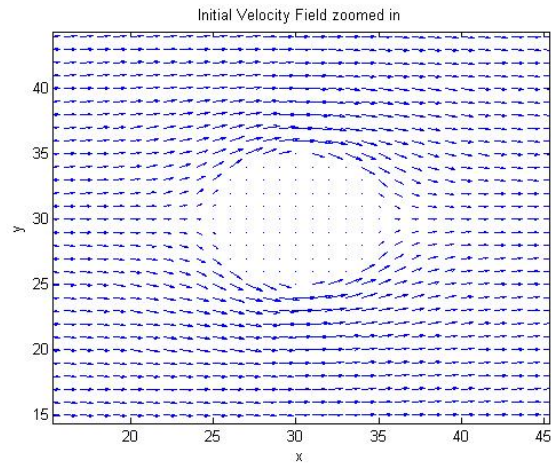


Figure 1: The initial velocity field around the cylinder

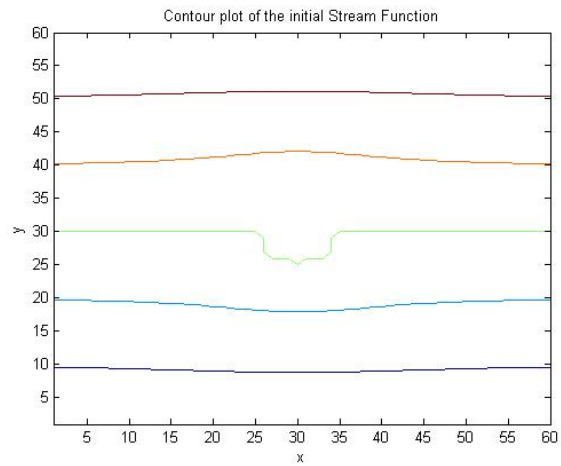


Figure 2: A contour plot of the initial stream function  $\Psi$

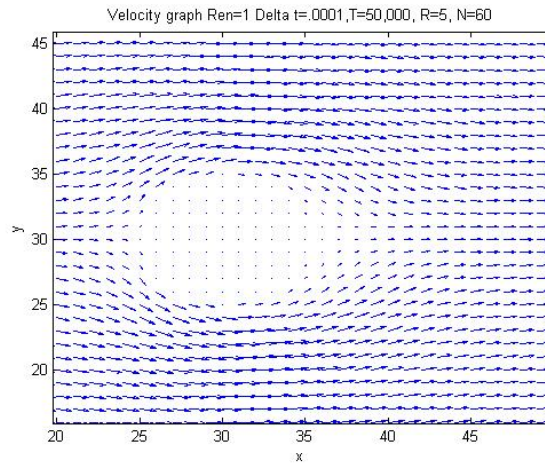


Figure 3: The velocity around the wake at  $Re=1$ . No vortices

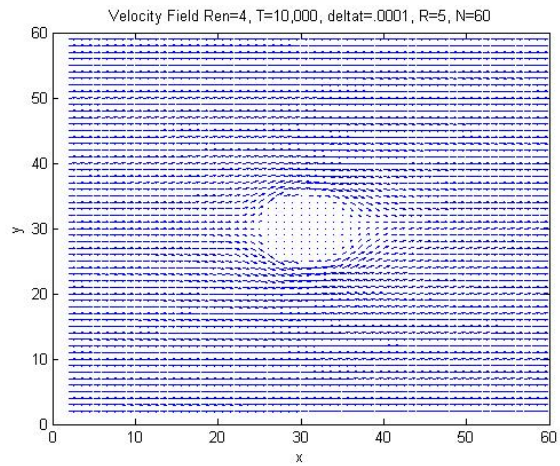


Figure 4: The velocity around the wake at  $Re=4$ . No vortices

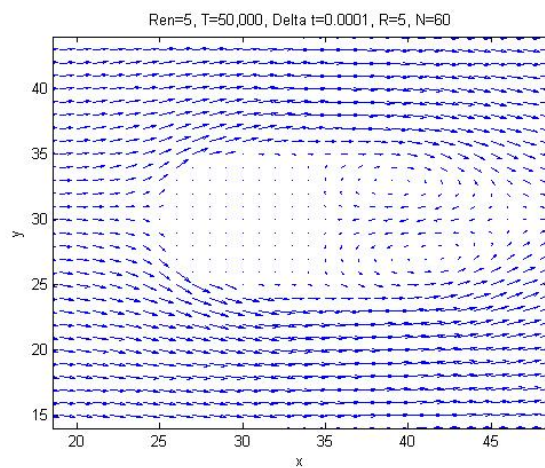


Figure 5: The velocity around the wake at  $Re=5$ . Vortices have developed

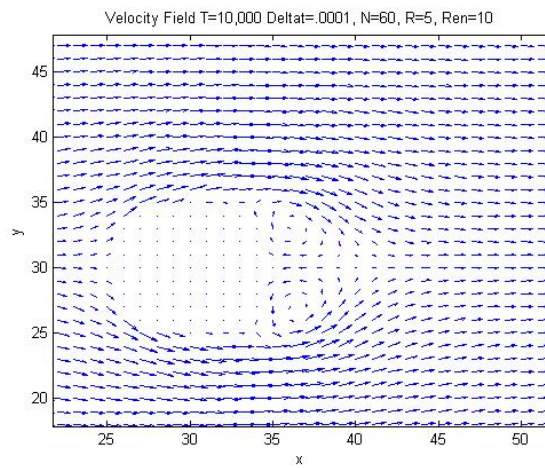


Figure 6: The velocity around the wake at  $Re=10$ . Vortices

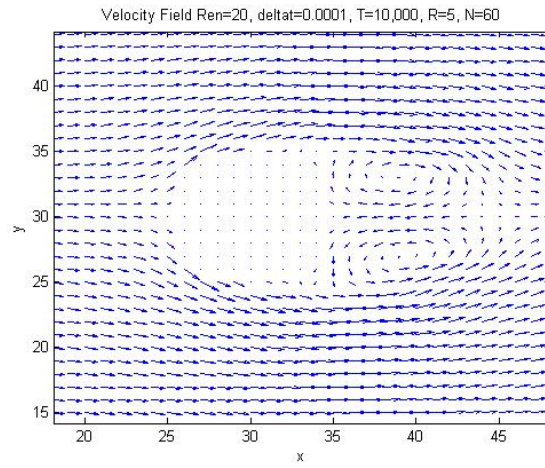


Figure 7: The velocity around the wake at  $Re=20$ . Larger vortices

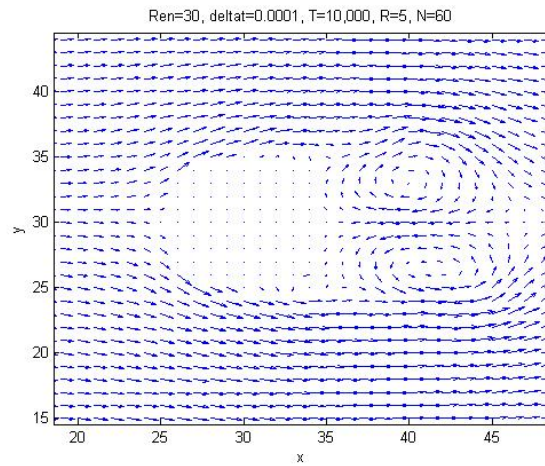


Figure 8: The velocity around the wake at  $Re=30$ . Even larger vortices

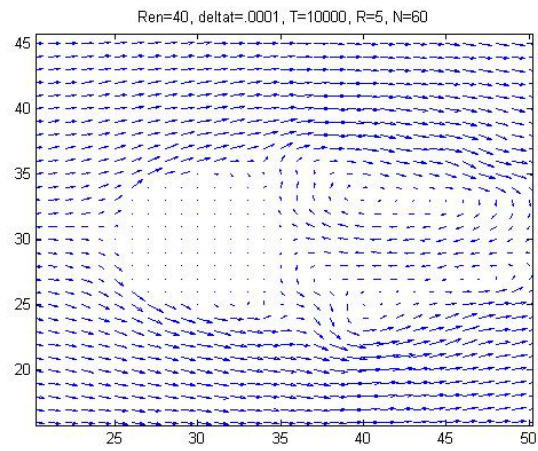


Figure 9: The velocity around the wake at  $Re=40$ . Largest vortices