Honors Theses   Honors College

Spring 5-2016

Momentum Transfer to a Simplified Wind Turbine Blade

Myer D. Milbrath
University of Southern Mississippi

Follow this and additional works at: https://aquila.usm.edu/honors_theses

Part of the Fluid Dynamics Commons

The University of Southern Mississippi

Momentum Transfer to a Simplified Wind Turbine Blade

by

Myer Milbrath

A Thesis
Submitted to the Honors College of
The University of Southern Mississippi
in Partial Fulfillment
of the Requirements for the Degree of
Bachelor of Science
in the Department of Physics and Astronomy

May 2016

Approved by

_____

Michael Vera, Ph.D., Thesis Advisor
Associate Professor of Physics and Astronomy

_____

Chris Winstead, Ph.D., Chair
Department of Physics and Astronomy

_____

Ellen Weinauer, Ph.D., Dean
Honors College

Abstract

This project looks into the potential of simplified simulations as a supplement to wind turbine blade designs. The idea is to take a two-dimensional representation of a wind turbine blade and attempt to use a fluid flow simulation to determine which "aspect ratio" is the most efficient in the transfer of momentum. This is then compared to the current requirements for wind turbine blades to judge how accurate the simulation was. It was found that an aspect ratio of 16.66 percent was most efficient, however when compared to the requirements for a wind turbine blade this ratio only falls into the requirements for a wind turbine tip. The largest reason for a disparity is due to the research not considering the requirements for the load that parts of the blade must hold, which means that the root section and wing section, to an extent, sacrifice efficiency in order to be able to hold the necessary weight. Due to this, the simulation tested here is insufficient for testing a root or blade section. However if the load-bearing requirement was taken care of in the design process or if the design being tested does not have such a requirement, then a simple simulation is sufficient for testing efficiency.

Key Terms: Computational fluid dynamics, wind turbine blade, energy, momentum

Dedication

To my parents, Bruce and Sherry Milbrath, for being constantly supportive throughout the past four years of university life. Thanks also to many of the professors in the USM physics department such as Dr. Michael Vera, Dr. Khin Maung Maung, and Dr. Chris Winstead for the excellent classes and willingness to help me succeed. Additional thanks to the USM Honors College for providing many educational opportunities over the years such as this project.

Acknowledgements

Table of Contents

List of Tables

List of Figures

Chapter 1: Introduction

Computational fluid dynamics, abbreviated as CFD, is the use of numerical analysis with computers to simulate the movement of fluids and their interactions with objects. CFD has seen use in many research projects to do a variety of tasks such as comparing wind turbine tips[3], investigating rotor designs[4], and estimating angles of attack from blade pressure[6]. However, not all CFD projects consider only complex modeling and complicated measurements. One such project was completed in 2010 by Dr. Bharath Kandula and Dr. Michael Vera in which a simulation was created with the intention of being relatively simple. The goal of the project was to create a simulation of fluid flow around a simple tilted obstruction[5] and output the velocity, vorticity (spin), and stream function values into an array to be manipulated as desired. From the final results the simulation appeared to be successful, but the possibilities of the simulation were not fully explored.

The idea behind this project was to expand upon the simple simulation, see what kind of results can be found, and discover if said results correspond to previous, more advanced simulations of a similar nature. More specifically, this project attempted to discern if the simple simulation can extract information on the transfer of momentum between the fluid and the simplified wind turbine blade. As such, the main goals behind this project were to modify the simulation created by Dr. Kandula to be able to easily change the aspect ratio of the turbine blade in order to test different designs, to extract a momentum transfer value from each ratio, and to compare the momentum transfer values of many different ratios both to each other and to previous research. From this information it may be possible to determine the efficiency of different blade designs relative to each other before running them through more accurate simulations, saving not just time but also effort.

Chapter 2: Literature Review

There has been much research and development of CFD techniques. Generally, all fluids follow a set of equations that may be used to simulate a fluid system. These equations are the Navier-Stokes equation (equation 2.1), the equation of continuity (equation 2.2), and the fluid's own equation of state which relates density, pressure, and other parameters[9].

$$\rho[\frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla)\vec{v}] = -\nabla p - \rho\nabla\phi + \eta\nabla^2\vec{v} + \eta_2\nabla(\nabla \cdot \vec{v}) \tag{2.1}$$

$$\nabla \cdot (\rho\vec{v}) = -\frac{\partial \rho}{\partial t} \tag{2.2}$$

For these equations in particular p is pressure, η and η_2 are fluid viscosity parameters, ρ is density, $\vec{v}$ is velocity, and ϕ is a scalar potential. The uses of these equations and variations thereof are seen in quite a few projects. Some particularly relevant projects are the more recent ones regarding CFD simulations of wind turbine blades, such as a paper that estimates the angle of attack from blade pressure measurements[6]. In particular, these CFD simulations can be used in the design of wind turbine blades and the testing of their efficiencies[4]. The energy efficiency is especially important in the designing of wind turbine blades, having been put under scrutiny since at least 1928[2] while the actual energy output and feasibility of the turbines has also been researched[1].

However, while the uses of CFD are evident in the ability to model fluid flow, there is still the issue of the complexity of the modeling itself. All of the CFD research mentioned involve using complex, three-dimensional models that, despite being highly accurate, are quite difficult to use. More importantly, they take quite a long amount of computational processing time as well. Luckily, not all CFD simulations require these difficult three-dimensional models or even the complex two-dimensional models as

seen in other projects[8]. One project completed in 2010 produced a simple simulation of a tilted obstruction in a two-dimensional fluid flow simulation[5]. Most importantly, this tilted obstruction takes a form that can be used as an extremely simplified wind turbine blade. The original project only models the flow of the fluid around the obstruction, however the output contains values for velocity, vorticity, and a stream function. It is from this output that it may be possible to extract a value for how efficient different turbine blade ratios are. Doing so successfully might prove that even extremely simple simulations such as this have some merit for testing designs before moving onto the more accurate three-dimensional models.

In order to see if this simple model actually does produce values with some measure of accuracy then there needs to be some results to compare them to. There are two articles that were used for this purpose. The first uses complex, three-dimensional models to test different rotor designs[4] while the second details the actual design of turbine blades, including required ratios of the different sections of the blade[7]. As already mentioned, the idea is to confirm if an incredibly simple model can obtain results are comparable to those from the complicated models. In particular, results of how efficiently the turbine designs transfer momentum at different aspect ratios will be considered. It is due to this that the simulations created by Gómez-Iradi and Barakos were used as a comparison for how close the simple model is to their complex ones. The ability to compare the two is not perfect, however, as the ratios considered by Gómez-Iradi are not exactly the same the ratios that were tested. The measure for energy efficiency used in these two projects also differ, with Gómez-Iradi using aerodynamic power in kilowatts while this project considered the momentum transfer. Additionally, the final results were compared to actual requirements on turbine blade designs to see if they fall into the current requirements. For this purpose the paper by Schubel and Crossley from 2012[7] was used as the basis for "state-of-the-art" turbine blade design.

Chapter 3: Methodology

As mentioned at the start of the previous chapter the Navier-Stokes equation (eq. 2.1) forms the basis for almost all CFD simulations and this project is no different. However, the equation was not used in its full form, instead multiple assumptions were made that significantly simplify the simulation without causing much loss in accuracy. The first and biggest assumption is that the fluid settled into a steady state. In other words all derivatives of time were set to zero. Additionally, the fluid was assumed to be incompressible so that the density at all points was constant, the second viscosity effect could be ignored, and the continuity equation became $\nabla \cdot \vec{v} = 0$. Two alternate parameters were also used in the final Navier-Stokes equation, the vorticity ($\vec{\Omega} = \nabla \times \vec{v}$) and a "stream function". This stream function, Ψ, is defined such that the velocity, $\vec{v} = (u, v)$, can be obtained by $u = \frac{\partial \Psi}{\partial y}$ and $v = -\frac{\partial \Psi}{\partial x}$. Finally, all characteristic scales that could affect the problem have been set into a single value, the Reynolds number R. This is important because flow fields with the same Reynolds number obey the same equation, so many systems with different physical extents can be compared.

So, after making these assumptions and adjusting for alternate parameters the final Navier-Stokes equation and the continuity equation became

$$\frac{\partial \Psi}{\partial y} \frac{\partial \Omega}{\partial x} - \frac{\partial \Psi}{\partial x} \frac{\partial \Omega}{\partial y} = \frac{\nabla^2 \Omega}{R} \tag{3.1}$$

$$\nabla \cdot \vec{v} = 0 \tag{3.2}$$

$$\nabla^2 \Psi = -\Omega \tag{3.3}$$

It is from these equations that a technique called "relaxation" was applied. The basic idea is that every time an iteration of the simulation is run every single grid point is

adjusted by its neighbors. In order to do this "cross" and "square" sums are taken around a point and used to update the point's values. A "cross" sum, in this case, is simply a summation of the points directly above, below, and to the sides of the point being adjusted. A "square" sum is the summation of the points diagonally from the initial point, the "corners" of a square surrounding the point. So, the update function for the stream function looks like

$$\Psi_{ij} = \frac{1}{5}[S_C + \frac{1}{4}S_S] + \frac{1}{5}\Omega_{ij} + \frac{1}{40}[S_C^\Omega] \tag{3.4}$$

With a similar idea being applied for the vorticity. Normally when relaxation is done a technique called "overrelaxation" is used where the relaxation is adjusted by a weight value, w in this case, that is chosen to accelerate the rate of convergence of the iterations. Unfortunately this causes instability for the simulation being attempted, so a different technique called "underrelaxation" is used. This idea is the same as overrelaxation except the weight value is chosen to decelerate the rate of convergence of the iterations to provide more stability. So, now that the technique is in place, starting conditions for the simulation are required. First, the initial velocity, vorticity, and stream function values are set to 0 at every point to ensure there are no errant values. From there, the points are then set based on a linear interpolation of the edges where edge conditions are based on horizontal flow[9]. One concern is for how long, or rather how many iterations, the simulation would need to be run. After testing multiple different run times, it was found that 200k iterations [Fig. 3.1] was long enough to allow the field to settle into an acceptable state. A run with 300k iterations [Fig. 3.2] was tested and the differences between it and the 200k run were negligible, so 200k iterations was used as the baseline.

In order for any of this to be useful an actual obstruction is still needed. This particular obstruction has some difficulty in its creation as it requires that the boundary
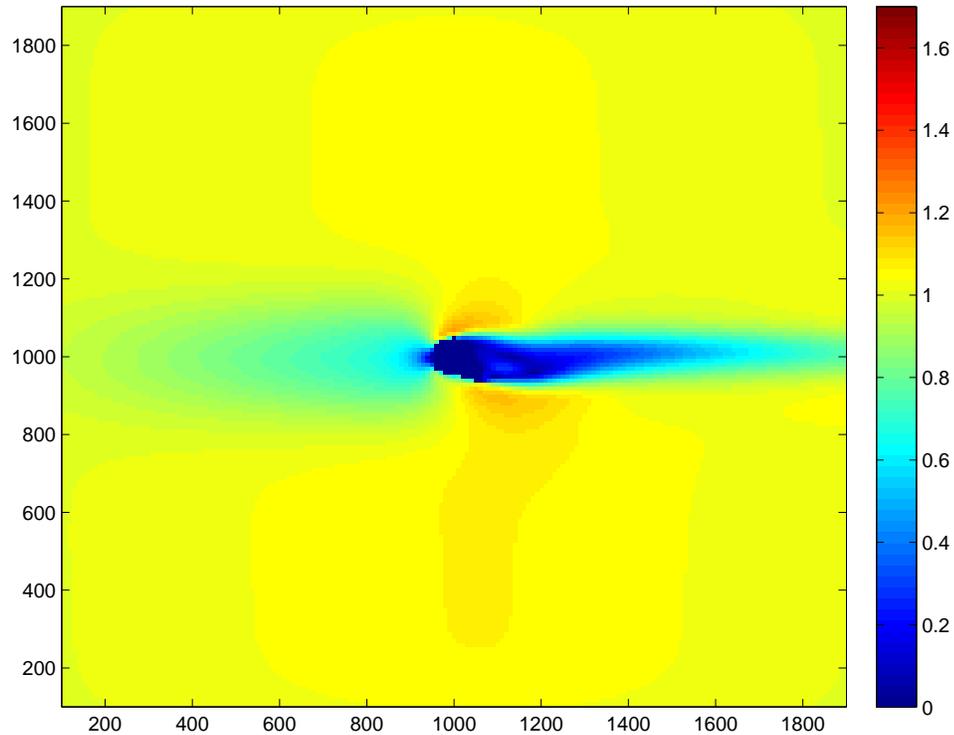
Figure 3.1: A tilted obstruction of aspect ratio 1 run through 200k iterations of the simulation is shown.

condition is applied between grid points. This problem was solved in a long series of checks that decide if a point is actually inside the solid and, if it is, places the point into a boundary condition array for later use. This array is then used to reset the tilted obstruction to have no vorticity, velocities, or stream function in between each iteration of the relaxation method. However, the methods and settings that were initially being used were not ideal and needed adjustment.

One of the first issues that had to be dealt with was the problem of domain size. The original version of the program ran with a 1000x1000 domain (Fig. 3.3) which, based on color plots of the speed, seemed to be large enough that the boundary conditions on the edges would not affect the flow near the object. However, after doing a more thorough examination of the output it became obvious that the flow was being forced through unnatural changes due to the boundary conditions as far as 300 to 400 away from the edge. Unfortunately, this meant that the area being used
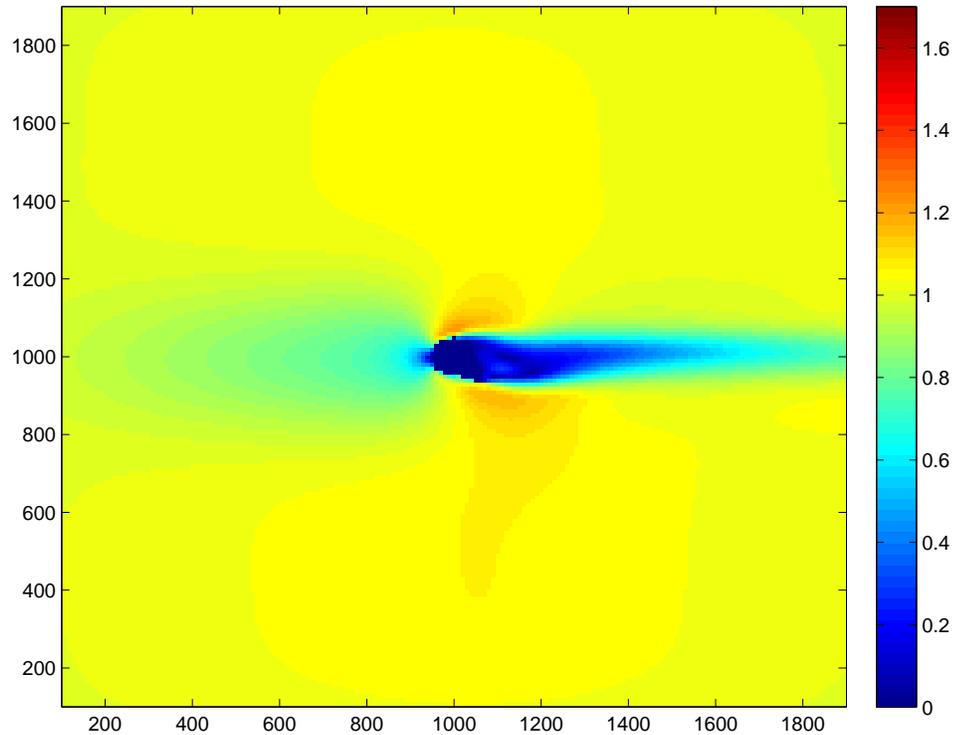
Figure 3.2: A tilted obstruction of aspect ratio 1 run through 300k iterations of the simulation is shown.

to obtain some measure of the momentum was also being polluted by the boundary conditions. This went unnoticed for quite a long time due to how misleading the speed color plot turned out to be (Fig. 3.6, note how suddenly vertical velocities disappear towards the edges). While the flow seemed acceptable in speed, checking the direction it was found that the flow was forced into a more horizontal direction when it should have still been more vertical. The most obvious solution to eliminating this contamination was to simply increase the size of the domain. In this case, the domain was doubled on each side and set to 2000x2000 (Fig. 3.4). This did create a slight problem with some of the code as the placement of the obstruction was set by the user and not automatically derived from the initial variables. This problem was temporarily fixed by recalculating the values needed to center the obstruction and later permanently fixed when the code was overhauled. Another slight change to the code to improve the efficiency was to alter the value used for the relaxation
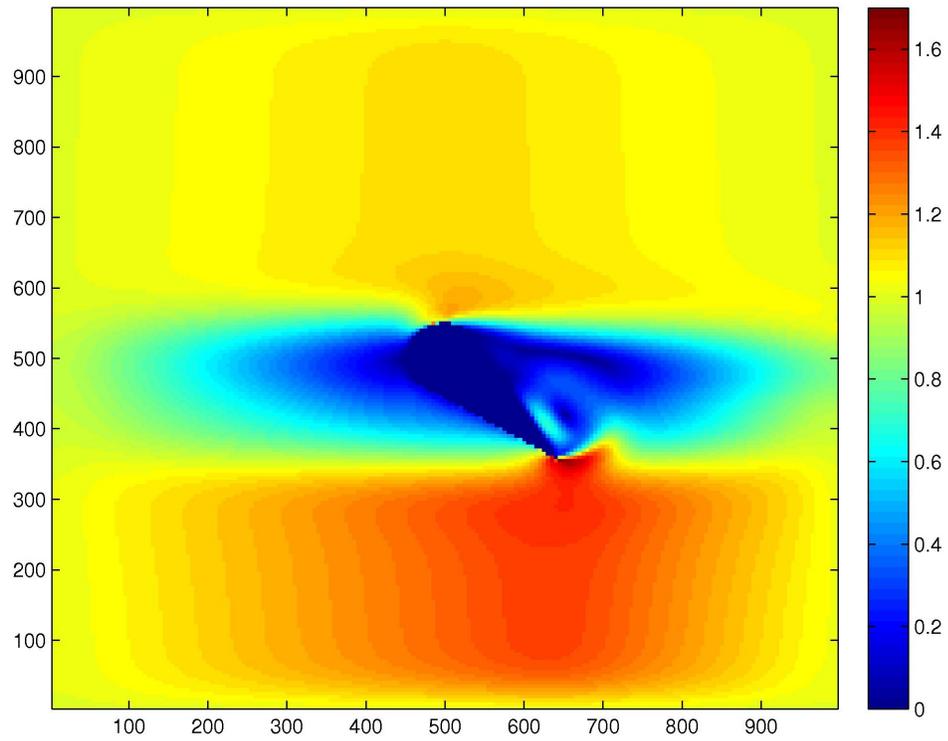
Figure 3.3: Pictured is the original speed plot of a tilted obstruction with an aspect ratio of 1 in the domain of 1000x1000.
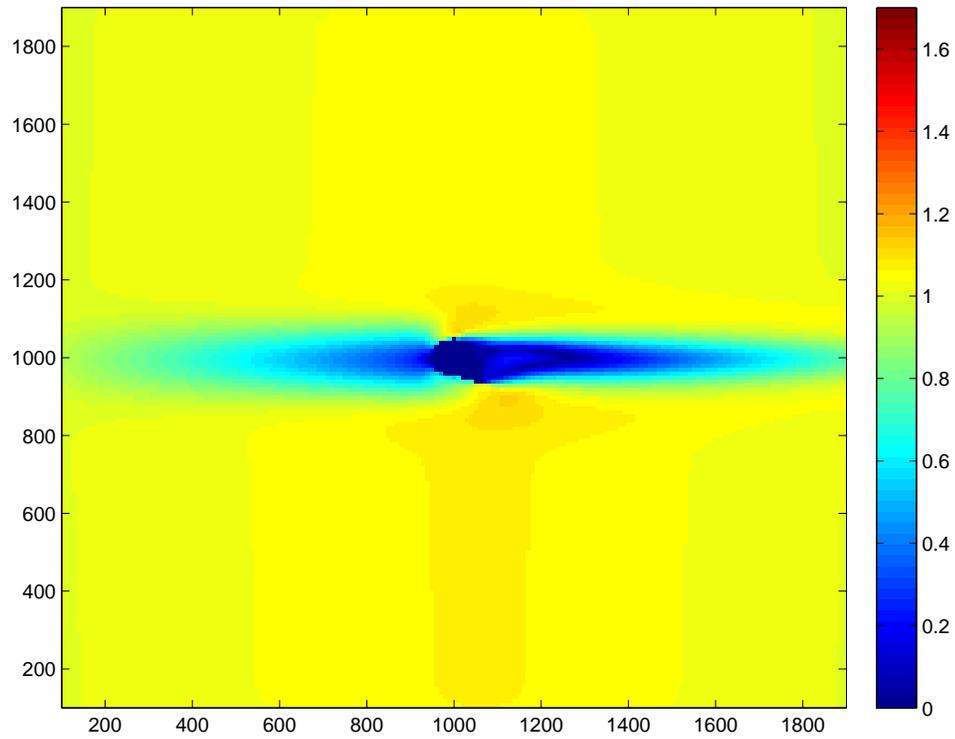
Figure 3.4: Pictured is the revised speed plot of a tilted obstruction with an aspect ratio of 1 in the modified domain of 2000x2000.
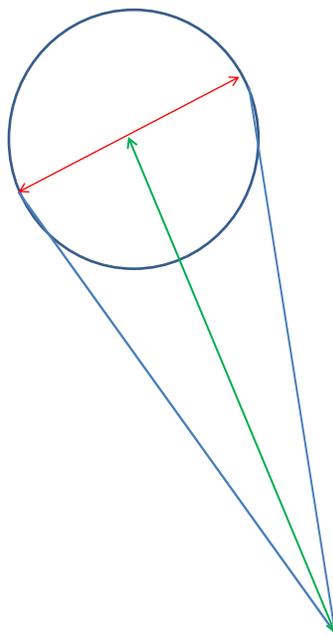

Figure 3.5: A simple diagram of the obstruction. The red line represents the circle diameter while the green line represents the chord length.

method. Initially, the values were set to 0.05 and then switched over to 0.1 after the first tenth of the iterations was complete. Essentially this guaranteed that the runs were stable at the start and eventually switched to making larger changes each pass in the later stages. However, after some brief tests it was found that this could be further modified so that the first tenth is still the initial value of 0.05, which is then doubled after one-tenth of the iterations are complete. Halfway through the iterations it then switches to triple the initial value, 0.15. This was found to be stable and allowed more work to be done for the same processing time. The final change was re-hauling the entire program in order for changes to the aspect ratio in terms of the length of the tail from the center of the circular top to the diameter of the circle ($\frac{\text{chord length}}{\text{circle diameter}}$ see Fig. 3.5) to be simple. This simulation went from needing changes in every subroutine to only requiring the changing of a single slope value at the start of the program to change the aspect ratio.

In order to actually obtain a measure of the efficiency of each aspect ratio then some metric needed to be decided. For this, the transfer of momentum from the fluid to the turbine blade is being used. Once again, keeping with simplicity, the values are being obtained from the basic equation of momentum, $p = mv$. Since the density is being kept constant throughout the entire simulation this makes obtaining a value of momentum exceptionally easy; only a value of velocity needed to be found. Specifically, a value for the vertical velocity was used as the primary means of determining the efficiency. The reasoning behind this is that the simulation initially starts with only horizontal velocities and, therefore, any vertical velocities must have been produced due to interaction with the obstruction. Obtaining these values was a simple matter of extracting the vertical velocities, summing them together, and then normalizing by the surface area of the turbine blade (n_d is the diameter of the
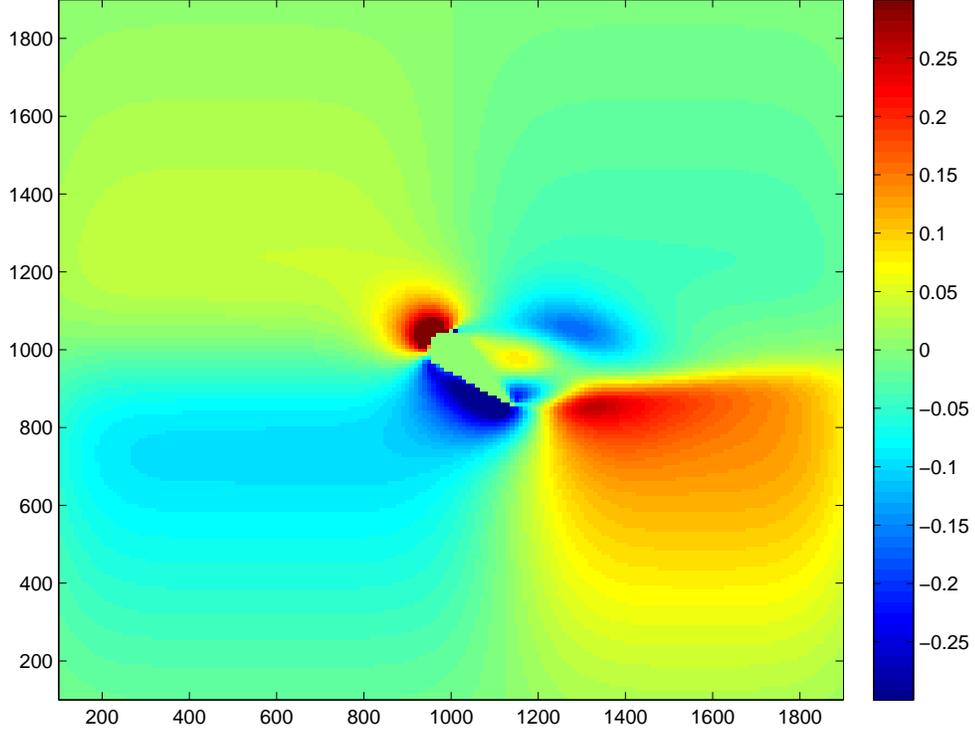
Figure 3.6: Color plot of vertical velocities for a tilted obstruction of aspect ratio 2 showing that the obstruction does not have significant influence at y=400.

semicircle and n_l is the chord length). The surface area is given by the following.

$$\text{SA} = \frac{\pi(\frac{n_d}{2})^2}{2} + n_l(\frac{n_d}{2}) \tag{3.5}$$

There were some specific adjustments that had to be made on a per aspect ratio basis. Namely the vertical location from which the velocity values would be taken had to be determined. The first method attempted simply had a fixed location, y=400 in this case, and found the vertical velocities only along that line. This had the unfortunate effect of favoring the larger aspect ratios as they were simply closer to where values were being taken (see figures 3.6 and 3.7). In order to remedy this, the y-coordinate for each aspect ratio was adjusted such that it would look at a line exactly one chord length vertically downward from the center of the obstruction (1000 - chord length). The width was also adjusted to cover two different lengths, 400 to
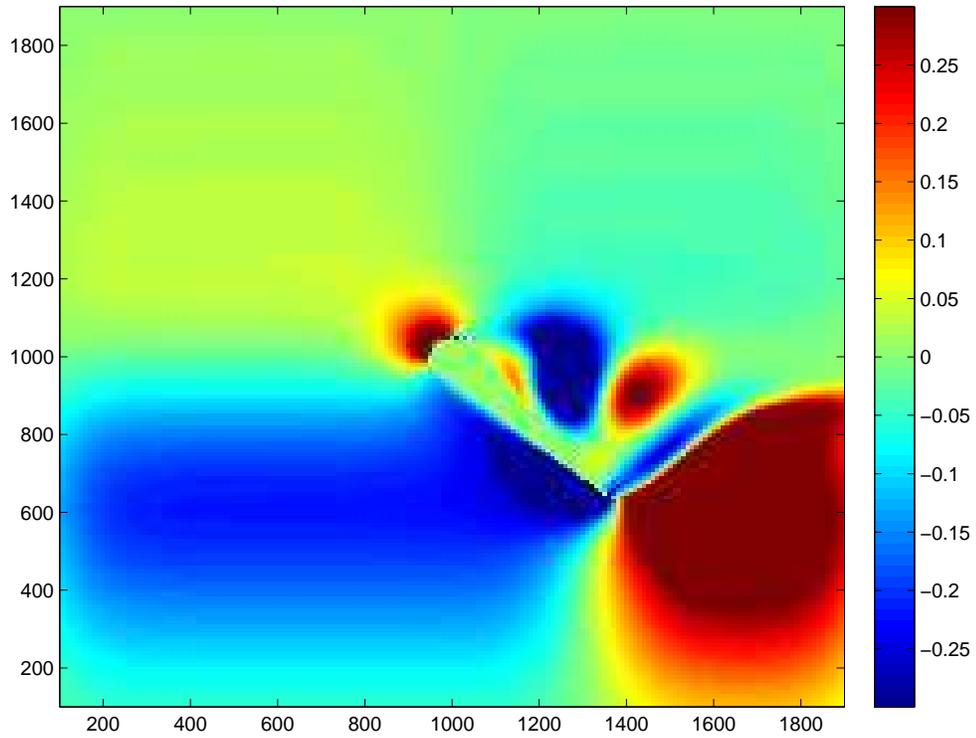
11

Figure 3.7: Color plot of vertical velocities for a tilted obstruction of aspect ratio 5 showing that the obstruction now has a significant influence at y=400. Compare this to figure 3.6 and it is obvious that a fixed y-value inherently favored larger ratios.

1600 and 200 to 1800. The width, however, did not make any significant difference in momentum values as it was simply gathering more data points and not necessarily favoring any ratio over another.

Chapter 4: Results

The results of the normalized momentum sums for the region of x from 400 to 1600 and 200 to 1800 are shown in table 4.1 and the magnitudes are also plotted in a simple line plot (Fig.4.1). The collected vertical velocities of the fluid are negative since interaction with the solid drives the fluid downward as seen in Fig.3.7. Additionally, speed plots of aspect ratios 2 (Fig.4.2), 4 (Fig.4.3), 6 (Fig.4.4), and 8 (Fig.4.5) are given to show the final flow fields of the different aspect ratios. The goal here was to find if any of these different aspect ratios caused a "valley" (or "peak" in terms of the magnitude) in the momentum metric. From the results seen in the table and the line plot, this maximum momentum transfer "peak" is seen at an aspect ratio of 6, when the chord length is 600 and the diameter is 100.

	AR1	AR2	AR3	AR4	AR5	AR6	AR7	AR8
x=400,1600	$-.43\times10^{-4}$	-1.59×10^{-4}	-1.78×10^{-4}	-2.36×10^{-4}	-3.02×10^{-4}	-4.15×10^{-4}	-3.8×10^{-4}	-3.56×10^{-4}
x=200,1800	$-.21\times10^{-4}$	$-.86\times10^{-4}$	$-.89\times10^{-4}$	-1.2×10^{-4}	-1.54×10^{-4}	-2.12×10^{-4}	-2.04×10^{-4}	-2.08×10^{-4}

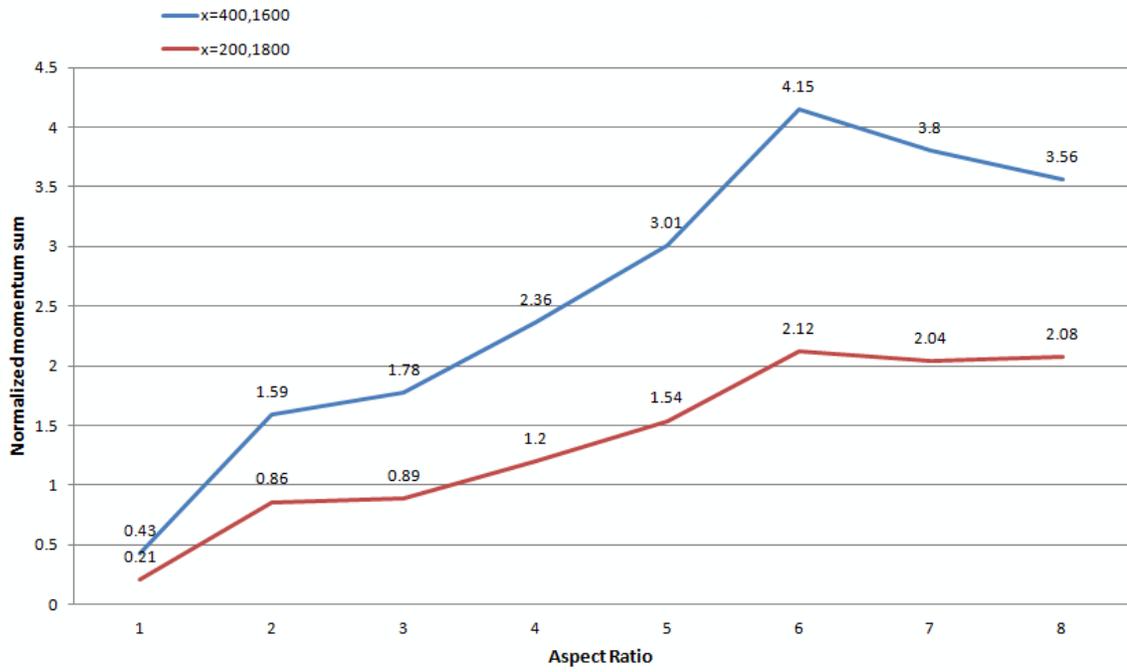Table 4.1: The normalized momentum sum values

Figure 4.1: Normalized momentum values

Figure 4.2: A speed color plot of tilted obstruction with aspect ratio of 2 is shown.

Figure 4.3: A speed color plot of tilted obstruction with aspect ratio of 4 is shown.
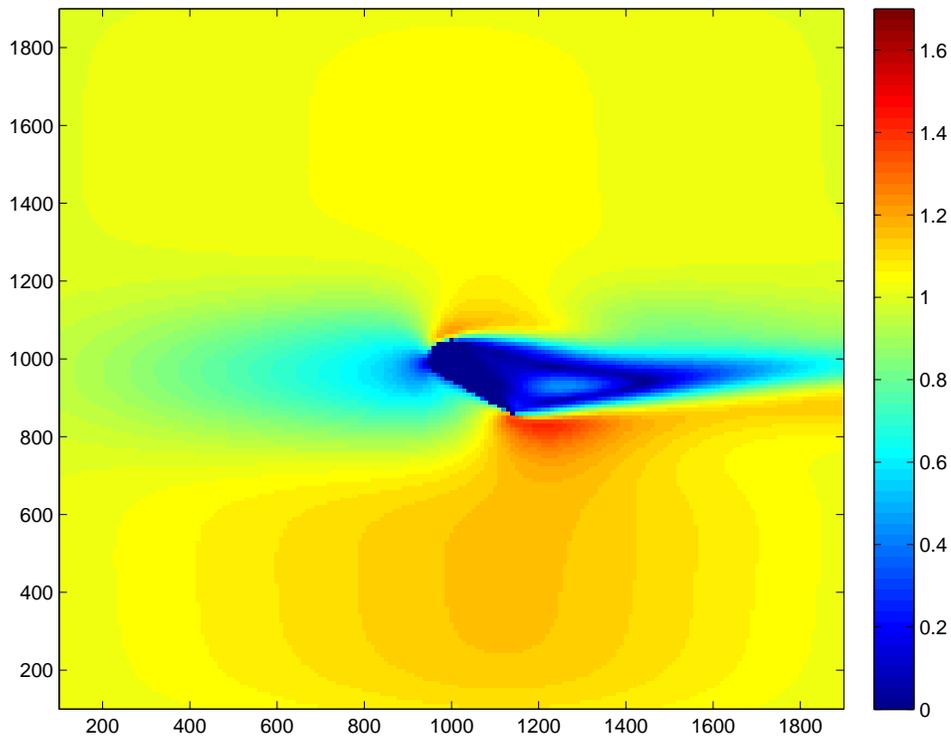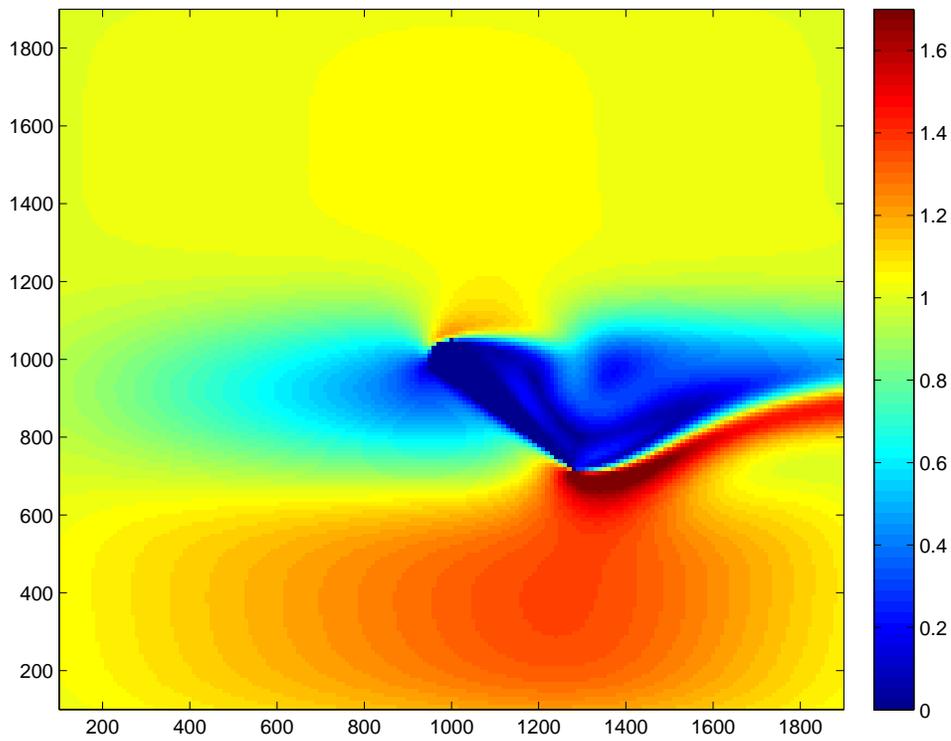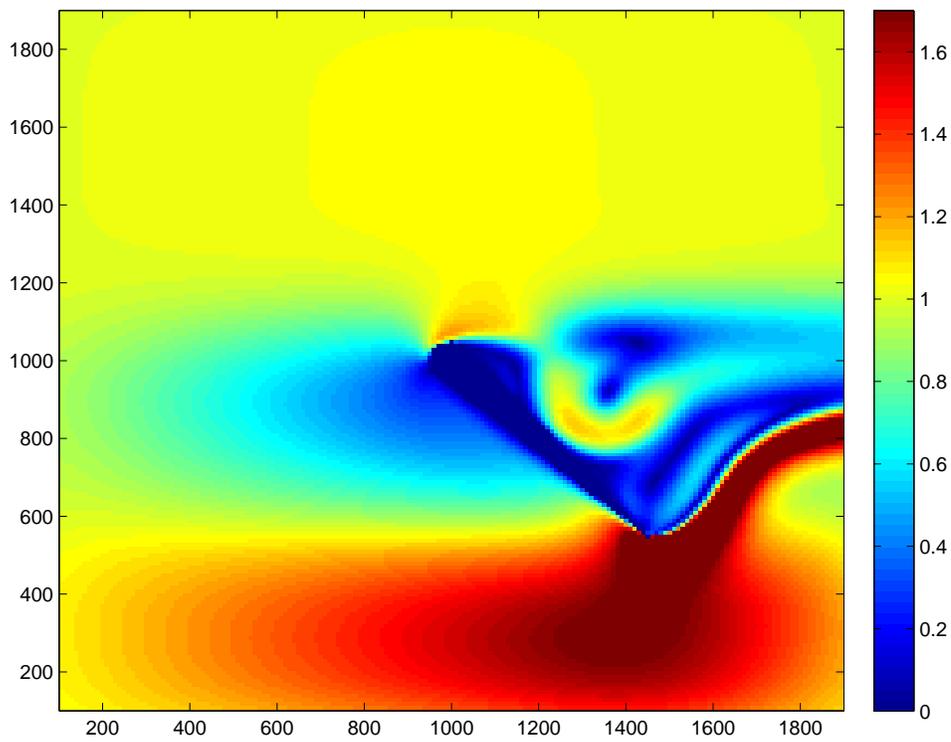

Figure 4.4: A speed color plot of tilted obstruction with aspect ratio of 6 is shown.
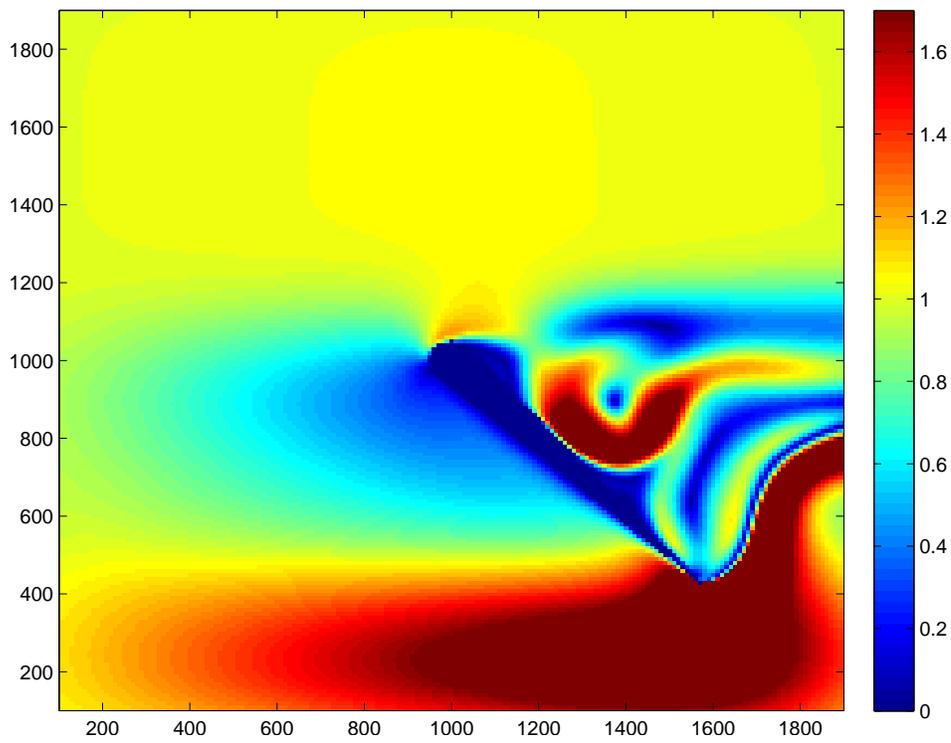
Figure 4.5: A speed color plot of tilted obstruction with aspect ratio of 8 is shown.

Chapter 5: Discussion

As said in the results, these simulations show a maximum momentum transfer efficiency for the turbine blade to be near an aspect ratio of 6. Having a large ratio is not unexpected, as shown in cases 1 and 2 of the CFD investigation of wind turbine rotor design parameters[4] a blade with a larger ratio does produce more "aerodynamic power". However, this does not necessarily mean that some ratio of 6 is the absolute best nor that it is even correct, just that it is an expected result. Indeed, when comparing this result to the requirements for aerofoil designs of different regions[7] there seems to be a large disparity between the found value for best efficiency (16.6 percent in their terms which is given by $[\frac{\text{diameter}}{\text{chord length}} * 100]$) and the requirements for a root section, a ratio greater than 27 percent. This is primarily due to the fact that this simple model does not consider all of the requirements for what a root section needs to accomplish. Primarily, this simulation does not consider the fact that a root section must be able to handle the entire load of the turbine blade. This means that the most efficient ratio in terms of momentum transfer is not necessarily the most efficient in terms of its ability to carry the blade. However, it is interesting to note that while root sections do require a thickness to chord ratio of greater than 27 percent an optimal ratio can still be found inside that area. For example, the difference between aspect ratios 2 (50 percent) and 3 (33 percent) is only 0.2 at worst, a 3 percent difference. So then, if looking at only root sections that fall into the requirements for aerofoil design then this simple simulation does not accurately produce results as it does not consider the additional requirements of the root sections.

Instead, the tip and blade sections of the turbine blade might be considered as they focus more on the efficiency of the design over its need to support a load. For this particular simulation the geometry is not of a shape that would properly represent a

blade. However, if it is considered as a way to check the efficiency of the tip, then there is some correlation between the results and the requirements for the aspect ratio of a tip. The tip of the blade is essential to the aerodynamic efficiency of the blade. This is especially evident from Gómez-Iradi and Barakos where a flat-tip and a rounded tip produced powers of 5.74 kW and 5.81 kW[4] respectively. Thus, if we take the required ratio of the tip to be between 21 and 15 percent then the found result of maximum efficiency seems to fall into that range. However, this model was not designed to simulate only a tip of a blade but rather be an extremely simple view of the entire turbine blade. Considering this, the aspect ratio of 6 does not seem to conform to the expected requirements for a blade, with aspect ratios of 21 to 27 percent being expected.

The primary reason for this is due to the fact that this simulation does not consider the load bearing requirements at all as mentioned earlier. If this is taken into consideration then a slightly smaller ratio, around 5, would likely be efficient in balancing both load requirements and efficiency in aerodynamic power. This would be in confirmation with the requirements as presented by Schubel and Crossley and also makes sense as a ratio in the area of 5 is still close to the maximum possible effiency as seen in figure 4.1. So, while simple simulations do not completely encompass the requirements for a turbine blade simulation they are still worthwhile for sections where load bearing capacity is not essential (the tip). However, if considerations for the load bearing were already taken into account then a simulation such as this one can still be used to check the efficiency and flow fields of the different designs.

Chapter 6: Conclusion

The goal behind this project was to see if a simple simulation could produce results consistent with more complex models. This was done with the purpose of seeing if a simple approximation of complex models could be used to check designs before running them through the more accurate simulations. To do this, a tilted obstruction which represents a wind turbine blade was created and then run through a simple fluid flow simulation. From this, a metric of the transfer of momentum to the blade was used as a representation of the energy that the blades can produce. The momentum was then normalized by the surface area of the blade and multiple different "aspect ratios" were used to find a most efficient ratio. These results were then compared to the currently accepted values for turbine blade sections to check for accuracy. The final results showed that an aspect ratio of 6 (16.6 percent) was found to be the most efficient in this simulation.

However, this value does not fall into the expected ratio of 27 percent or higher that is required for the root section of a wind turbine blade. However, this value does fall into the expectation for the tip of a wind turbine blade. This shows that a simple simulation such as this is not adequate in finding the most efficient design for a root section by itself, as it does not account for requirements in being able to bear the load of the blade's weight. However, if such considerations were enforced during the designing of multiple blade sections it seems that a simulation such as this should still be able to determine which is more efficient for obtaining energy from the wind. However there is more that may be possible from this type of modelling. For instance, it should be possible to obtain the pressure values along the different sections of the blade as well, allowing more detail in an analysis of a particular blade design.

Bibliography

[1] Rangan Banerjee. Capacity building for renewable energy in India. *Indian Renewable Energy Development Agency*, 2(1):19–25, 2005.

[2] A. Betz. Windmills in the light of modern research. Technical Report 474, National Advisory Committee for Aeronautics, August 1928.

[3] E Ferrer and X Munduate. Wind turbine blade tip comparison using CFD. In *Journal of Physics: Conference Series*, volume 75, page 012005. IOP Publishing, 2007.

[4] S Gómez-Iradi and GN Barakos. Computational fluid dynamics investigation of some wind turbine rotor design parameters. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 222(5):455–470, 2008.

[5] B. Kandula. Two-dimensional simulations of fluid flow with an arbitrary obstacle. *Journal of the Mississippi Academy of Sciences*, January 2010.

[6] Tonio Sant, Gijs van Kuik, and GJW Van Bussel. Estimating the angle of attack from blade pressure measurements on the national renewable energy laboratory phase vi rotor using a free wake vortex model: yawed conditions. *Wind Energy*, 12(1):1–32, 2009.

[7] Peter J Schubel and Richard J Crossley. Wind turbine blade design. *Energies*, 5(9):3425–3449, 2012.

[8] Wen Zhong Shen, Martin OL Hansen, and Jens Nørkær Sørensen. Determination of the angle of attack on rotor blades. *Wind Energy*, 12(1):91–98, 2009.

[9] Michael Vera. *Physics 711: Computational Methods for Physical System II.* University of Southern Mississippi, 2012.