Analysis of Vertical Axis Wind Turbines via Meshfree Computational Fluid Dynamics Methods

Nicholas Taluzek, Peter Kozak, Dr. Dietmar Rempfer
Illinois Institute of Technology
Fluid Dynamics Research Center

In this study a method of data extraction is developed for quicker and easier analysis of the fluid dynamics for vertical axis wind turbines (VAWTs) modeled by two-dimensional (2D) computational fluid dynamics (CFD). A three bladed H-rotor or Savonius-style VAWT moves at a set rotational speed and a fluid passes over with a constant velocity to simulate operating air flow conditions. Data is probed from the simulation at quasi-random points in the domain, collecting relevant flow velocity data which is then processed by a meshfree approximation with a radial basis function (RBF) interpolation. This provides an interpolated set of velocity data across the entire domain that is then used to calculate the stream functions and velocity potentials of the fluid flow. At this point in the research, the stream function of a single VAWT has been calculated for multiple operating conditions. Future work includes superimposing these stream functions against themselves to replicate the interaction between two VAWTs in a coupled system. The results of this superimposition are to be compared to a separate set of double VAWT CFD simulations for validation.

I. INTRODUCTION

As our understanding of carbon emissions and greenhouse gases slowly continues to develop, global warming is becoming a more tangible threat to our modern way of life. The Intergovernmental Panel on Climate Change estimates that global average temperatures will increase 2.5 to 10 degrees Fahrenheit over the coming century [1], which can have many negative effects on agriculture, wildlife, and our own living environments. In the U.S. approximately 86.5% of our electricity is generated through non-renewable sources like coal, gas, petroleum, and nuclear energy [2]. These sources of electricity are all large contributors to our greenhouse gas emissions within the atmosphere. There is a growing need for alternative sources of energy like hydropower, solar, biofuels, and wind power to be utilized, if not for our own sake but for the livelihood of generations yet to come. Wind power currently accounts for about 4% of the USA's electricity generation. Initiatives are already in place for wind power technologies to grow to generate 20% of the U.S. electricity by 2030 [3]. Such an endeavor may be possible with current wind power technologies, but with further research into wind turbines a larger percentage of our power can be harnessed from the wind.

The turbine design that most people are familiar with are horizontal axis wind turbines (HAWTs). These are lift-based power generators and while there has been great success with this design thus far, it also has its shortcomings. HAWTs operate with their tips travelling at speeds that can be dangerous in the case of catastrophic failure [4], pose risks for flying animals [5], and generate noise not seemly for well-populated regions [6]. Another set of alternative turbine designs are called vertical-axis wind turbines (VAWTs) which include lift- and drag-based power generation schemes. VAWTs can operate at much slower tip-speeds, because of the vertical axis of rotation they can operate under any wind direction without the need for additional mechanics, and they are relatively easier to construct and maintain as most of the mechanical systems can be located at ground level. VAWT designs are better suited for large-scale construction since they do not experience the oscillating gravity loads that HAWTs do, which create significant bending moments [7]. Unfortunately, the aerodynamics of VAWTs is significantly more complicated due to factors like the blades not having constant angles of attack, and the effects of dynamic stall and wake dynamics. The VAWT's complexity makes modeling and experimentation more difficult and time consuming, and as a consequence there has not been as much research into their design as compared to HAWTs.

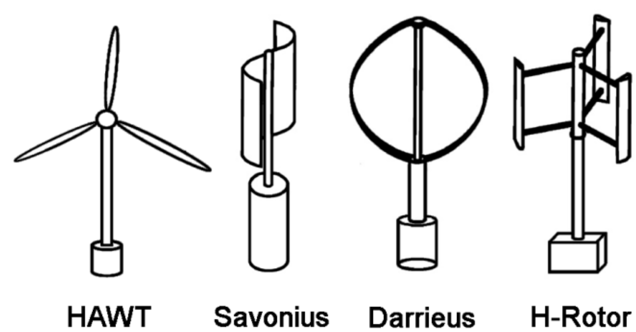

Figure 1 - The major wind turbine types [8]

Substantial work involving VAWT design has been done previously within The Fluid Dynamics Research Center, a division of the Mechanical, Materials, and Aerospace Engineering Department at the Illinois Institute of Technology (IIT). The work completed in this research is a continuation of some of the research done by Payam

Mehrpooya for his Master's thesis at IIT. Using computational fluid dynamics, he investigated the improvement of VAWT performance via turbine coupling – placing two VAWTs near each other to take advantage of the interactions with the surrounding air. The practice of analyzing systems of VAWTs is fairly new, so there is still much research to be done. As current research suggests that wind turbine performance can benefit from strategic positioning, further understanding of the potential power generation from VAWT arrays is critical in moving forward with renewable wind energy.

The main contribution that this study aims to make for VAWT development is the creation of a faster and easier method of data analysis. In this research a simplified CFD data collection and analysis process through meshfree approximation methods is developed so that future work studying VAWT stream functions can be done more efficiently. A large hindrance in CFD work is the vast quantities of calculations that computers must run to produce accurate simulations, but the results from this research should cut back on required computational time.

The topic of this proposed research fits directly under the Energy Theme within the Armour College of Engineering Themes program. This research into wind as a renewable energy source will further exemplify IIT's dedication to sustainability. Success from this research could potentially make for significant improvement in the speed at which future progress can be made in The Fluid Dynamics Research Center's VAWT efforts.

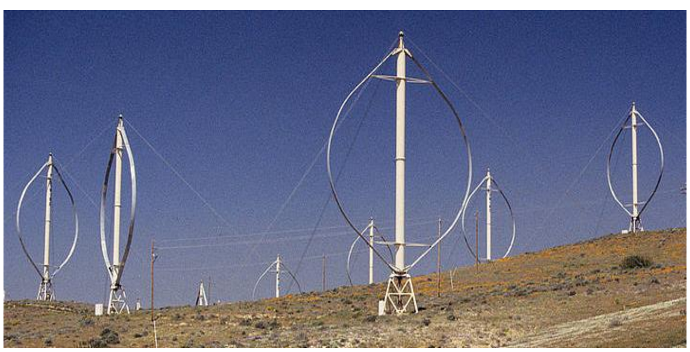

Figure 2 – FloWind Darrieus VAWT farm located in California [9]

II. METHODS

From work previously done by Peter Kozak and Payam Mehrpooya in their master's thesis work at IIT, CFD simulations were already developed for a single VAWT and a double VAWT system [10] [11]. These were created using the commercial software Star CCM+, a finite volume solver. The turbine in the single VAWT simulation is comprised of three NACA 0021 airfoils with chord length of 0.0858m, set around a common axis of rotation with a constant radius of 0.513m. The simulation domain is rectangular with the leftmost wall being set as a fluid inlet with a constant flow velocity of 9m/s. The geometry of the domains for Kozak's single VAWT simulation can be seen in Figure 4 and Payam's double VAWT simulation in Figure 5.

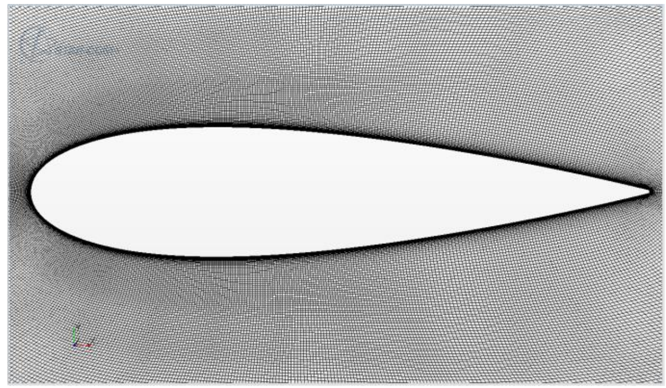

Figure 3 – VAWT NACA 0021Airfoil Mesh Geometry

The scope of this study has thus far only utilized the single VAWT geometry, and simulations have been run under different Reynolds numbers (Re) and Tip Speed Ratios (TSR). The TSR is the ratio of the turbine blade's induced velocity to the free-stream velocity.

$$TSR = \frac{U}{U_0} = \frac{\vartheta R}{U_0}$$

Where U is the induced velocity, U_0 is the free stream velocity, ϑ is the rotational speed in rad/s, and R is the turbine radius. The induced velocity equates to the rotational speed times the radius of the turbine. The TSR is a component in calculating the Re as follows,

$$Re = \frac{\rho U_0 c}{\mu} TSR$$

Where ρ is the fluid density, c is the airfoil chord length, and μ is the fluid viscosity. For the purposes of this study, Re was changed only by changing the value of the fluid viscosity. Information about the single VAWT simulations that have been completed is tabulated in Table 1.

Table 1 – Single VAWT Simulation Sets

Re	3750	375000	4500	450000
TSR	2.5	2.5	3.0	3.0
ϑ (rad/s)	43.86	43.86	52.63	52.63
μ (Pa ·s)	6.31E-4	6.31E-6	6.31E-4	6.31E-6

As can be seen in Figures 3, 6, and 7, the mesh grid in the CFD simulations is very fine. A fine mesh makes for more accurate calculations and while data can be extracted at every mesh node for analysis, it would demand a lot of computational time to process all of it. Through this study, a meshfree approximation method is employed to reduce the computational expense.

Figure 4 – Single VAWT CFD Geometry

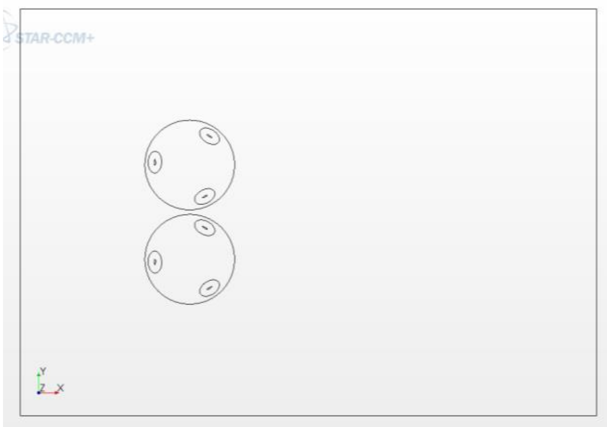

Figure 5 – Double VAWT CFD Geometry

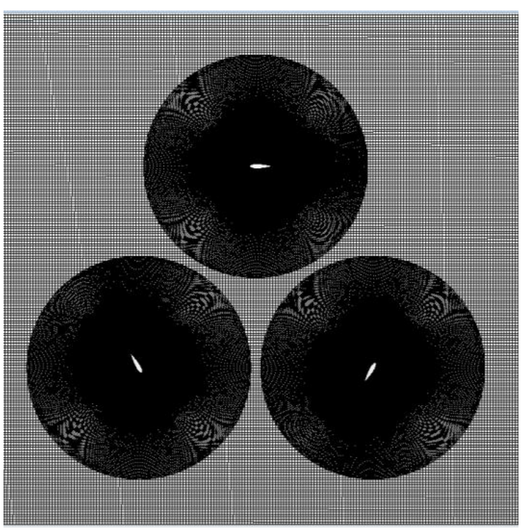

Figure 6 – Mesh surrounding VAWT airfoils

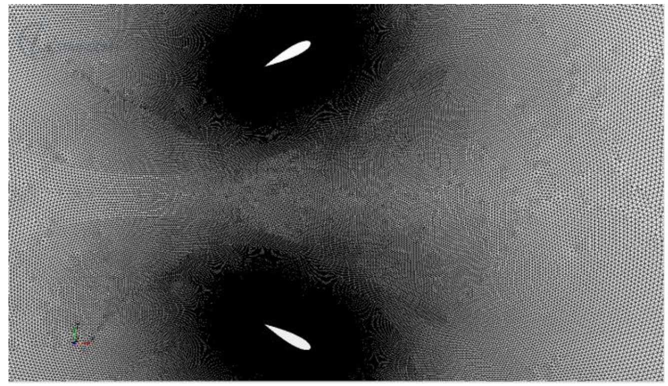

Figure 7 – Mesh interaction between double VAWT systems

Instead of extracting flow data from every mesh node in the simulation a smaller number of data points are collected. Star CCM+ allows for data to be extracted from the simulation at discrete user-defined points in the domain by use of point probes. Star CCM+ is able to execute Java files in order to complete functions for the user, and in this case one was written to populate the simulation with point-probes according to a list of coordinates.

Using MATLAB, a quasi-random list of 10,000 coordinates was exported into a comma-separated value file. This number of coordinates was estimated to be more than enough for our purposes with the meshfree approximation. The quasi-random coordinates were created from a Halton sequence, a deterministic sequence of numbers based on a prime number [12]. The domain of the probes was limited to a 6 by 6 meter region about the turbine. Figure 8 shows the distribution of the point-probes around the single turbine.

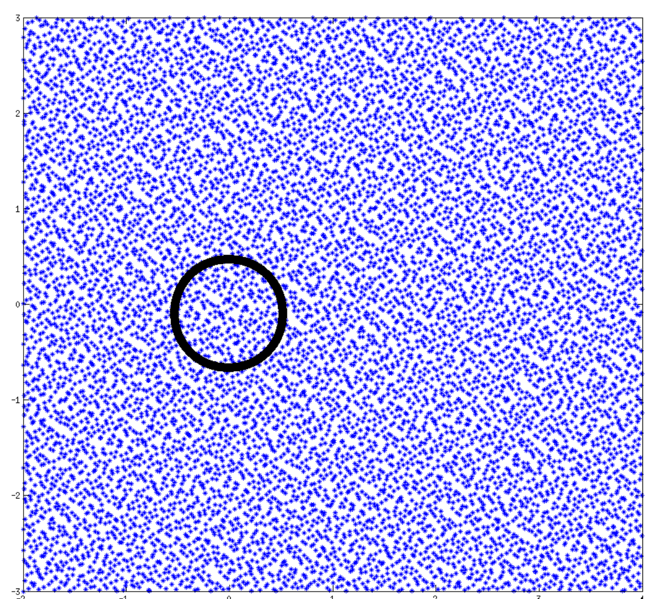

Figure 8 – Scatter plot of point-probe locations about single VAWT

As the simulation runs, data is set to extract to an external file for every 5 degrees the turbine rotates. This is set in Star CCM+ as a delta time trigger event, which is calculated from the rotational speed of the turbine. The data is saved to a new file that can be later processed in MATLAB.

$$\Delta t = \frac{5}{360}\frac{2\pi}{\dot{\theta}}$$

The data of interest for this study are the *x*- and *y*-velocity components of the fluid, as these will later be used in calculating the stream function and velocity potential of the flow.

With the scattered data collected from the CFD simulations, a radial basis function (RBF) interpolation is performed to compute a solution across the entire domain. The radial basis function interpolation utilizes the Gaussian function,

$$\varphi(r) = e^{-(\varepsilon r)^2}, \qquad r \in \mathbb{R}$$

where the shape parameter ε is related to the variance of the normal distribution function [Fasshauer]. Having the Gaussian as a function of Euclidean distance about a fixed center there is the function,

$$\phi_k(x) = e^{-\varepsilon^2 \|x - x_k\|_2^2}, \qquad r \in \mathbb{R}$$

where x_k is any fixed center of the interpolation. The interpolation is done separately for the *x*- and *y*-velocity components of the flow within MATLAB, and is done to create a uniform grid of point values.

From the interpolated velocity profiles the stream function and velocity potentials of the fluid flow can be calculated. Using Simpson rule summation the matrices of the *x*- and *y*-velocities over the domain are integrated and the potentials computed. The MATLAB functions used to calculate the stream function and velocity potential were written by Kirill Pankratov [13].

$$u = \frac{d\phi}{dx}, v = \frac{d\phi}{dy} \; for\ potential\ flows$$

$$u = -\frac{d\psi}{dy}, v = \frac{d\psi}{dx} \; for\ solenoidal\ flows$$

III. RESULTS

In order to develop the MATLAB scripts for the RBF interpolation a known test function was used.

$$F(x,y) = (x_{max} - x)(x_{min} - x)(y_{max} - y)(y_{min} - y)$$

Figure 9 shows a 3D plot of the exact test function, and figures 10-12 show the same function after RBF interpolation. A number of interpolations were performed by varying the shape parameter, ε, and the average relative error is plotted in Figure 14. In addition to error calculations from with coordinates from a Halton sequence, a Sobol sequence for quasi-random coordinate generation was also used for comparison.

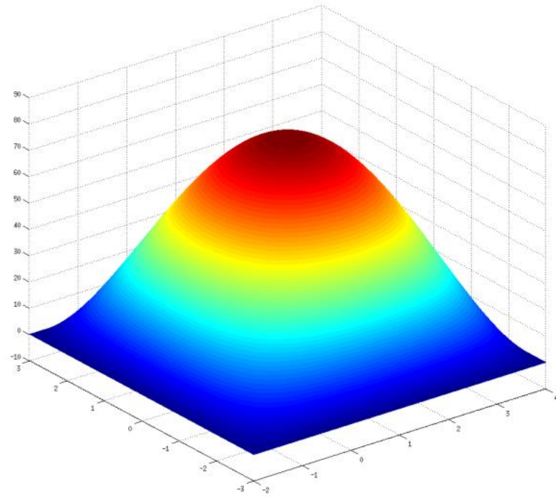

Figure 9 – 3D plot of exact test function

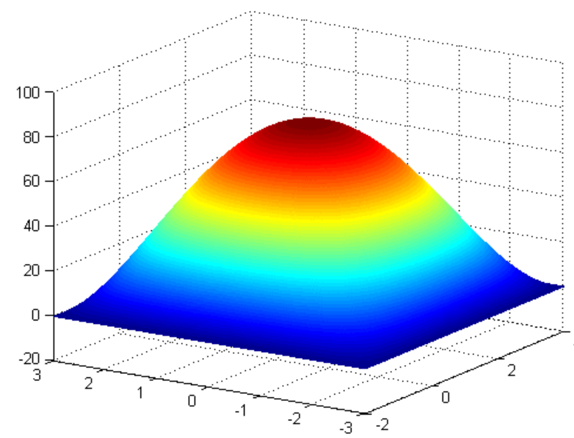

Figure 10 - 3D plot of interpolated test function with 1000 Halton points and ε = 1

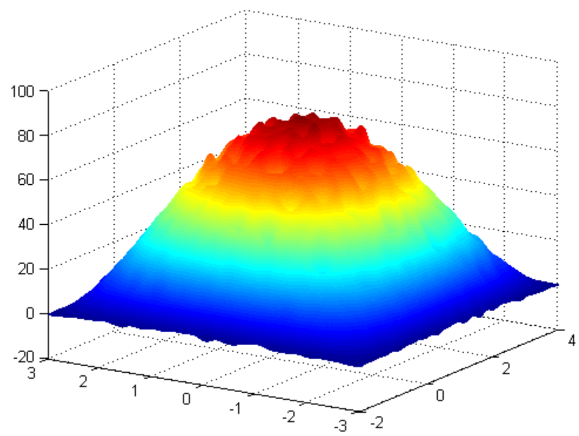

Figure 11 - 3D plot of interpolated test function with 1000 Halton points and $\varepsilon = 5$

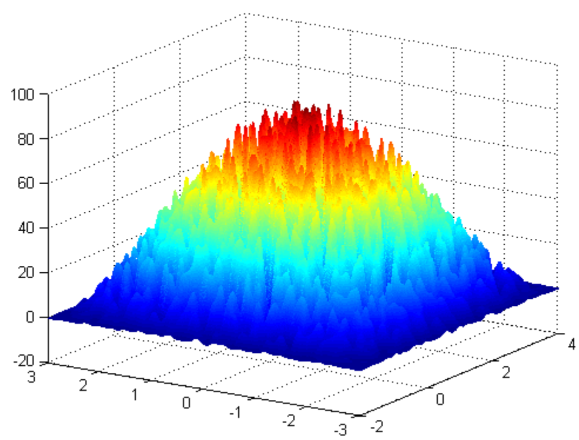

Figure 12 - 3D plot of interpolated test function with 1000 Halton points and $\varepsilon = 10$

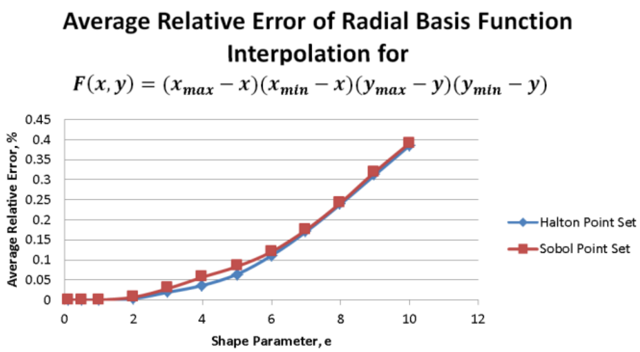

Figure 13 – Error analysis of Halton and Sobol RBF interpolation

10,000 point probes were used to extract data from the simulations. However with the intention of developing a data analysis process to reduce computational time, efforts were made to reduce the number of data points needed during analysis. Figure 15 shows the time average *x*-velocity contours of a single VAWT with varying numbers of data points used in the RBF interpolation. The relative error in velocity caused by using fewer data points was calculated and is shown in Table 2. The interpolations performed result in a uniform grid that is 200 by 200 data points across the domain.

Table 2 – Error calculations in mean x-velocity with RBF interpolations using different number of data points

Halton Points	10,000	5000	2000	1000	500
Mean X-Velocity (m/s)	8.35	8.47	8.43	7.99	6.60
% Change from 10,000 points	0%	1.5%	0.95%	4.22%	5.16%

Figures 16-23 show contour plots of the interpolated velocity data for a single VAWT. The one time step plots are interpolated data of the 20th delta time step, showing a fraction of the simulation. The phase average plots show the velocity data averaged for every 120 degrees the turbine spins, so that the average is calculated only from data when the three airfoils are in the same positions. From 72 simulation time steps, 3 are averaged together for the phase average plots. And the period average plots for *x*- and *y*-velocity depict the average values across the entire simulation time of one revolution period, the average of 72 time steps.

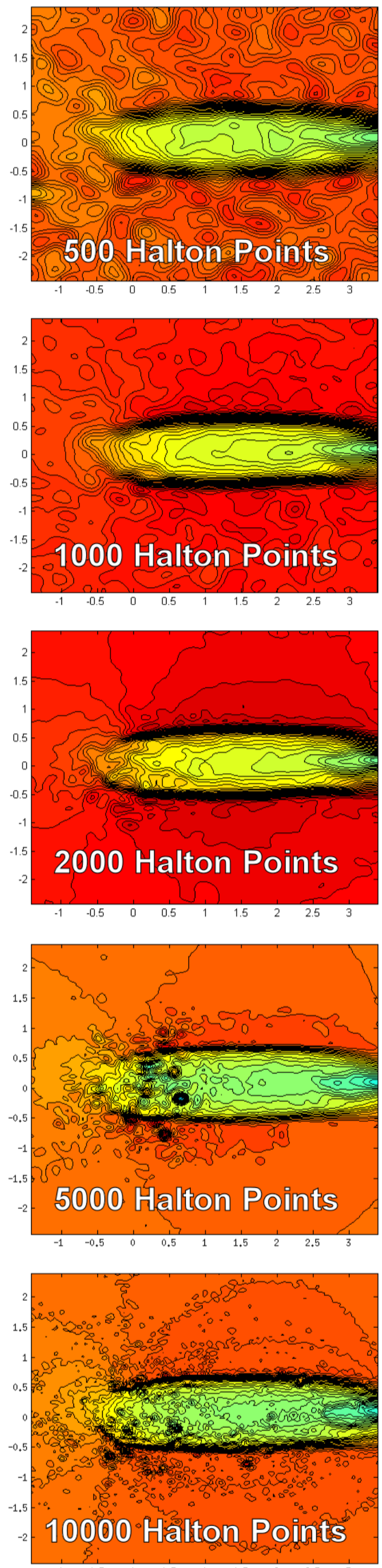

Figure 14 – Interpolated *x*-velocity contour plots with different numbers of data points used in calculations

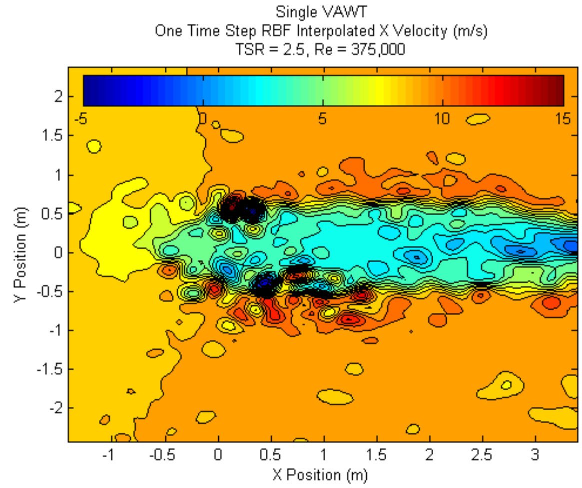

Figure 15 – One time step *x*-velocity contour plot, Re = 375,000

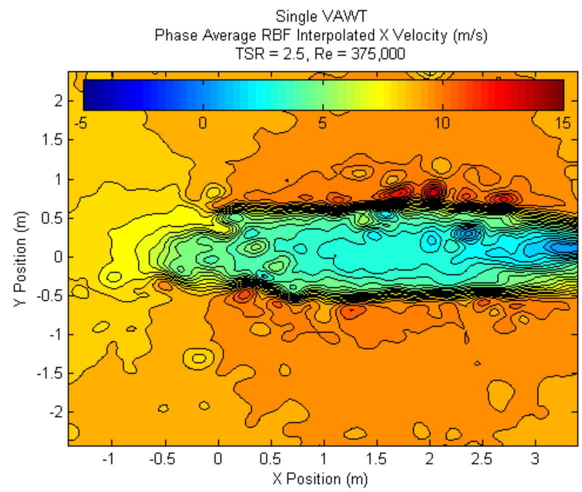

Figure 16 – Phase average *x*-velocity contour plot, Re = 375,000

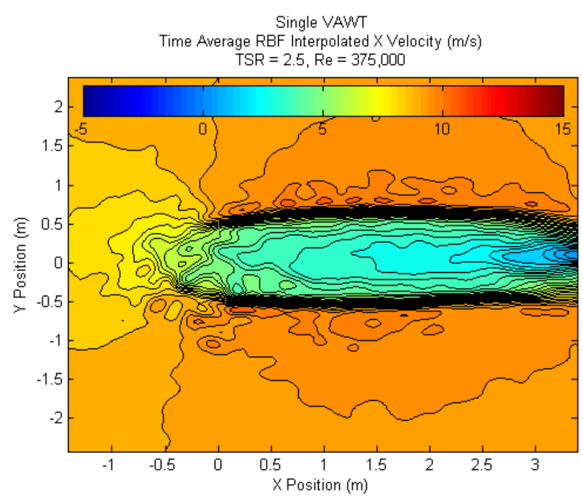

Figure 17 – Period average *x*-velocity contour plot, Re = 375,000

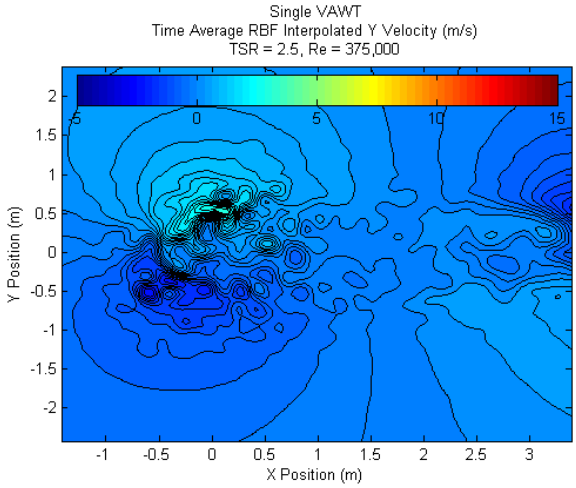

Figure 18 – Period average *y*-velocity contour plot, Re = 375,000

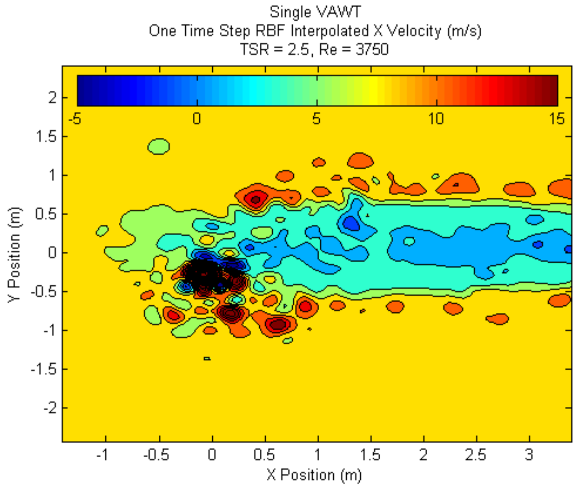

Figure 19 – One time step *x*-velocity contour plot, Re = 3750

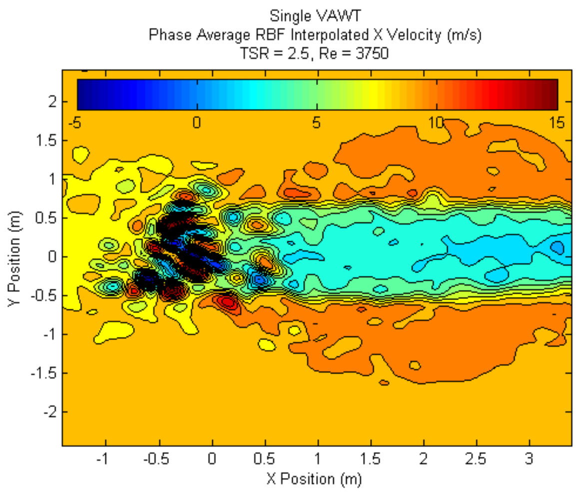

Figure 20 – Phase average *x*-velocity contour plot, Re = 3750

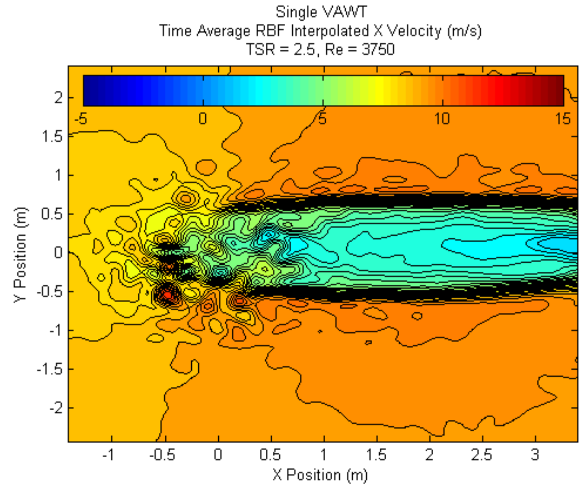

Figure 21 – Period average *x*-velocity contour plot, Re = 3750

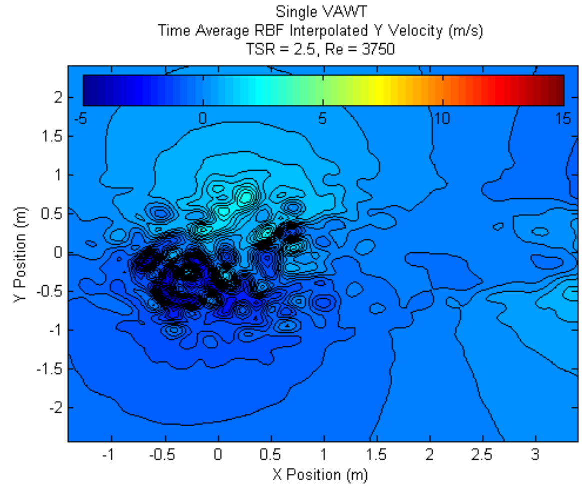

Figure 22 – Period average *y*-velocity contour plot, Re = 3750

From the time averaged x- and y-velocity data, the stream function (ϕ) and velocity potentials (ψ) were plotted. These are shown in Figures 24 and 25.

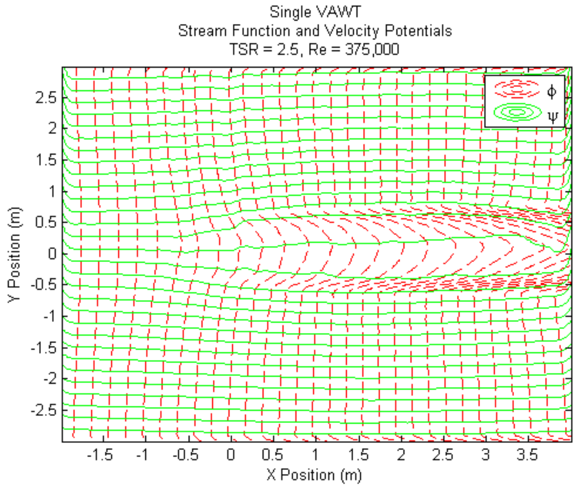

Figure 23

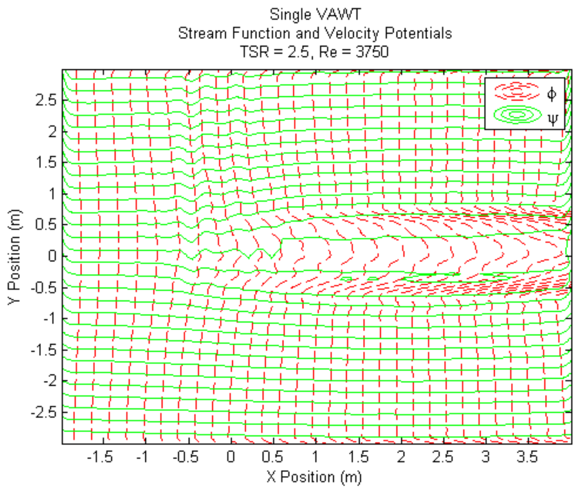

Figure 24

IV. DISCUSSION

With the shape parameter and number of points used during the RBF interpolation, the accuracy of the results can be adjusted. From the error analysis of the test function the shape parameter is seen as important, and the lower the shape parameter the more accurate the interpolation can be. The difference is visualized in Figure 10, as the peaks of the Gaussian curves can be seen (points where the interpolated values match exactly to the test function). The relative error for the interpolation sets using Halton and Sobol sequences for quasi-random coordinate generation both show the same error behavior.

The test function used to develop the interpolation code is very smooth, and the RBF interpolation can function properly even at very small shape parameter values. The data extracted from the simulation is not ideal for the interpolation, as boundary conditions along the airfoils create concentrated regions of high and low velocities. These regions limit the functionality of the RBF interpolation, and in running test calculations it was found that the shape parameter could not be set below 5. Shape parameters below 5 resulted in truncation errors in the interpolation, resulting in plots like Figure 26. Being limited to that value for the shape parameter, the test function analysis shows that errors of at least 0.1% can be expected from the VAWT interpolations.

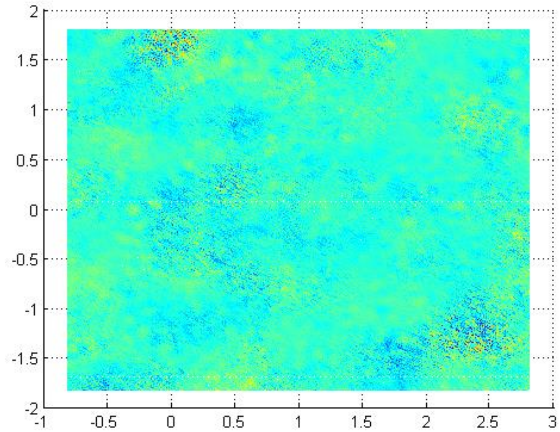

Figure 25 – VAWT interpolated x-velocity contour plot error with $\varepsilon = 4$

Inserting 10,000 point-probes into the simulation and collecting data from each was a burden at the beginning of this study, but it is now understood that using so many points may not be necessary. Granted the more data that is collected the more accurate the results can be assumed to be, but again this comes at the expense of computation time. In Figure 15 the time average *x*-velocity contour plots using 500 to all 10,000 point-probes is seen. With every point used in the interpolation an excess amount of noise is present throughout the fluid flow and reducing the number of points being used in the calculation appears to smooth out the resulting interpolated data. When only 1000 and 500 points are used the resulting mean x-velocity error relative to 10,000 points is much too high being at 4.22% and 5.16%, respectively. From this analysis, it was decided that 2000 points provided the desired balance between accuracy and computing time. All future interpolations were performed using data from 2000 point probes, outputting calculations for a uniform grid that is 200 data points square.

Much of the work completed thus far qualitatively compares the interpolation results. In looking at the contour plots for a single time step from the simulation (Figure 16 and 20), some notable features are the regions around the airfoils of both high or low velocities and the non-uniform velocity profile in the wake of the turbine. When the phase average is considered (Figure 17 and 21), when data is averaged at points in time when the turbine airfoils overlay every 120 degrees, the wake of the turbine appears smoother but the airfoil regions are still notable on the contour plots.

The time average contour plots show the data collected every 5 degrees over one full revolution averaged together. The wakes are much cleaner and more defined, giving a better impression of the complete fluid region that the turbine influences. Also, the drastic velocity changes located at the turbine airfoils are no longer distinguishable from the surrounding flow. It is also interesting to note from the time average y-velocity plots that the fluid above the turbine has a slightly higher velocity. This is to be expected since the turbines spin counterclockwise, creating a region of fast moving fluid on the outside edge of the airfoils.

V. CONCLUSION

The complexities of calculations that are completed within the CFD simulations demand a significant computational expense. With the advent of improved computer hardware and software, there is an increased reliance on CFD to study new engineering technologies. Research of VAWTs will slowly improve with the advancement of computer hardware, but improved methods of data processing may lead to an even faster rate of progress. The goal of this study is to develop a method of data extraction and processing that simplifies CFD analysis. With successful results at the end of this study, future research of VAWTs that is planned within The Fluid Dynamics Research Center may continue at a faster rate of progress.

Within the scope of this study a system to extract data from a 2D Star CCM+ CFD simulation was created. Using MATLAB a set of code was written for processing the CFD data through interpolation so that fluid dynamic values can be calculated for any point within a domain. The interpolation employed utilizes a Gaussian function in terms of Euclidean distance, called a radial basis function.

A test function was used to develop the MATLAB codes for the RBF interpolation and the expected error was determined. In applying the RBF interpolation to the simulation data it was found that the minimum shape parameter value that could be used without error is 5. From the test function study the resulting VAWT simulation interpolations can be expected to have at least a 0.1% average error.

Sets of data originally extracted and used for analysis of the single VAWT included 10,000 data points. Through testing sets of interpolations with fewer data points it was found that the preferred balance between accuracy and computation time was using only 2000 points.

From the interpolated velocity data, stream functions and velocity potentials were created. These plots are critical for the future steps in this research. All of the work up to this point has only involved the single VAWT simulations created by Kozak. Being able to quickly extract and process data from the VAWT simulations will allow for easier analysis to compare against a coupled VAWT system. Mehrpooya created a set of double VAWT simulations that test different distances between turbine centers, testing the hypothesis that VAWTs can interact together to harness more power from the wind than if they were far apart. Work will be done with that set of simulations to create stream function and velocity potential plots.

The plots from the single VAWT simulations completed so far will be superimposed upon one another in such a way as to replicate the same coupled VAWT systems that Mehrpooya's simulations model. With success in superimposing the stream functions to replicate two VAWTs interacting, further research into VAWT arrays may be simplified and completed at a faster rate than relying on CFD simulations alone.

VI. ACKNOWLEDGEMENTS

Thank you to the IIT Armour College of Engineering for creating the PURE program, providing undergraduate students valuable research opportunities. The financial support from the college makes it possible for students to commit time towards research programs. And a big thanks to Dr. Dietmar Rempfer and Peter Kozak for allowing me to work alongside them on this important research within the Fluid Dynamics Research Center at the Illinois Institute of Technology. This work would also not have been possible without the work that Payam Mehrpooya completed during his Master's program at IIT.

VII. REFERNCES

[1] Intergovernmental Panel on Climate Change. (2007).Summary for Policymakers, in *Climate Change 2007: Impacts, Adaptation and Vulnerability. Contribution of Working Group II to the Fourth Assessment Report of the Intergovernmental Panel on Climate Change*, Cambridge University Press, Cambridge, UK, p. 17.

[2] U.S. Energy Information Administration. (2014). *Short-Term Energy Outlook.*

[3] Lindenberg, S., Smith, B., & O0Dell, K. (2008). 20 Percent Wind energy by 2030. National Renewable Energy Laboratory (NREL), US Department of Energy, Renewable Energy consulting Services, Energetics Incorporated.

[4] Seifert, H., Westerhellweg, A., & Kroning, J. (2003). Risk analysis of ice throw from wind turbines. BOREAS, 6 (9).

[5] Baerwald, E. F., D'Amours, G. H., Klug, B. J., & Barclay, R. M. (2008). Barotrauma is a significant cause of bat fatalities at wind turbines. Current Biology, 18 (16), R695-R696.

[6] Pedersen, E., & Waye, K. P. (2004). Perception and annoyance due to wind turbine noise-a dose-

response relationship. The Journal of the Acoustical Society of America, 116, 3460.

[7] Musgrove, P. J. (1987). Wind energy conversion: recent progress and future prospects. Solar and wind technology, 4 (1), 37-49.

[8] Eriksson, S., Bernho_, H., & Leijon, M. (2008). Evaluation of different turbine concepts for wind power. Renewable and Sustainable Energy Reviews, 12 (5), 1419-1434.

[9] Gipe, Paul. FloWind: *The World's Most Successful VAWT*. WindWorks.org. February 2013.

[10] Kozak, Peter. *Effects of Unsteady Aerodynamics on Vertical-Axis Wind Turbine Performance*. Illinois Institute of Technology. Chicago, IL. May 2014.

[11] Mehrpooya, Payam. *Improvement of Vertical-Axis Wind Turbine Performance via Turbine Coupling.* Illinois Institute of Technology. Chicago, IL. July 2014.

[12] Kuipers, L.., Niederreiter, H., *Uniform distribution of sequences*. Dover Publications. p. 129. 2005.

[13] Pankratov, Kirill. *How to compute a streamfunction?* Massachusetts Institute of Technology. Cambridge, MA. March 1994.