Wind Flow Model of Area Surrounding the Case Western Reserve University Wind Turbine

Matheus C. Fernandes[1], David H. Matthiesen PhD[*2]
[1]Case Western Reserve University Dept. of Mechanical Engineering,
[2]Case Western Reserve University Dept. of Material Sciences and Engineering
*10900 Euclid Ave. Cleveland, Ohio 44106 – david.matthiesen@case.edu

Abstract: The Case Western Reserve University Wind Turbine (NorthWind®100kW) is a research turbine located on an urban campus in Cleveland, Ohio. Due to the tall surrounding buildings, this location may create turbulence, resulting in a possible loss in energy generation. This work attempts to answer this question. Depending upon the direction of the wind, the surrounding buildings, which vary in height from 20 to 40 meters, may affect the wind patterns at the hub height of the turbine that is 37 meters from ground. In order to investigate this issue, a COMSOL Multiphysics model of the immediate area was generated. Reasonable agreement was found when the results of the model were compared against real wind measurements and the maximum difference found was about 12%. These measurements were made by cup anemometers placed at three different locations and heights, which surrounded the wind turbine. Based on these results the wind turbine output is not expected to be affected by the close proximity of the buildings in most cases.

Keywords: CFD, Wind, Turbine, Flow, Energy

1. Introduction

The Case Western Reserve University Wind Turbine is a NorthWind®100kW research turbine that partially powers the adjacent Veale Convocation Center. This turbine was erected in November 2010 and is useful as an example of an urban wind turbine due to its location. It is in the center of the University campus in which the surrounding building heights vary from 20 to 40 meters high. In order to analyze and visualize the wind flow from every possible direction, a Computational Fluid Dynamics (CFD) model using COMSOL Multiphysics was created. By using local wind direction measurements, the analysis of the model focused mainly on the prevailing wind directions. These wind measurements were captured by cup anemometers, which were located at three different heights (28, 31, 40 meters from ground)

and locations. The model includes dimensions of the buildings as well as their footprint location relative to the turbine as shown in *Figure 1*. However, the model does not include buildings that are not immediately surrounding the turbine, nor does it include the buildings complex shapes.

Figure 1 Google aerial view of Case Western Reserve University wind turbine and surrounding buildings

2. Governing Equations

This model uses the Navier-Stokes iterations and the continuity equation as implemented in the COMSOL Multiphysics laminar flow package. The boundaries of the model are set to at least 60 meters from the top or the sides of any building in order to avoid any tunnel effect created by the fictitious boundaries. Also, in order to calculate the type of fluid flow, the Reynolds number calculation was used.

Eq. (1): Reynolds Number

$$\mathrm{Re} = \frac{\rho \mathbf{v} L}{\mu} = \frac{\mathbf{v} L}{\nu}$$

3. Methods

3.1 Geometry

In order to create this model, a footprint sketch consisting of each buildings location and dimension was drawn using the 2-D work plane feature of COMSOL Multiphysics. Each building was then extruded to a bit larger than its dimensional height. The reason for broadening the buildings dimensions in the model is to overshoot any effects the buildings may have on the turbine at hub height. If the buildings with these dimensions do not impact the turbine, then the smaller dimensions will not impact the turbine either. To create the model universe, an arbitrary square was also drawn on the 2-D work plane and extruded to a proper height. This square was centered in the model in order to be easily rotated using the rotation angle feature of COMSOL Multiphysics. The rotation feature allows for easy change in the direction of the induced wind flow permitting the analysis on the effect of the buildings on the wind turbine as a function of wind direction. The turbine is constructed by creating a union of regular blocks, cylinders and a vertical extruded sketch to create the shape of the blades.

3.2 Boundary Conditions

Once the model universe is created, all walls are set to slip conditions except for the ground which is set to slip condition, one face is set to inflow and the opposing face is set as the outflow. After creating these boundaries the cubes material is set to air at room temperature (27° C). Once all of these settings are defined for the model universe, each building is cut from the cube using the difference feature of COMSOL Multiphysics. This action recreates the bottom boundary of the model by taking into account the shapes and locations of the buildings. The wind turbine geometry is then also cut using a similar method as to the buildings.

3.3 Laminar Flow

After the geometry was fully created, the wind was uniformly induced through the face at a steady rate with zero initial values. This model uses the laminar flow feature of COMSOL Multiphysics and therefore uses time invariant stationary solvers.

4. Numerical Model

4.1 Wind direction analysis

Using the three years of wind direction and velocity data acquired from the three cup anemometers, the wind was found to prevail from the West North West direction. This wind varies slightly also in a few other directions depending on the season as seen in *Figure 2*.
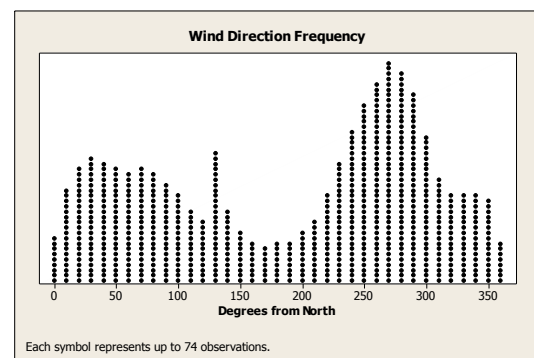

Figure 2 Frequency plot of wind direction acquired from cup anemometers

4.2 Model Validation

To validate the model, real wind measurements made by the cup anemometers were compared to nodes created at the same location within the model. Given four specific different wind directions, the model was compared between its nodes and the actual wind data collected by the anemometers. As seen in *Table 1* trough *Table 4*, the Garage Cup (Node 3) was used as the control node because its location and height are beyond the effect of most buildings. The v*elocity % Difference* for these tables was calculated by dividing the difference between *COMSOL calculation velocity* and *the Measured Wind Velocity* by the *Measured Wind Velocity.* The d*irection % Difference* for these tables was calculated by dividing the angular difference between the *COMSOL Calculation direction* and the *Measured Wind Direction* by 360 degrees.

North

	COMSOL Calculation	Measured Wind by Cup Anemometers	% Difference (Velocity, Direction)
North Cup (Node 1)	(4.3 m/s, 350.2 N)	(4.0 m/s, 321.9 NW)	(7.5%, 7.8%)
South Cup (Node 2)	(5.1 m/s, 13.2 N)	(4.9 m/s, 34.1 NE)	(4.1%, 5.0%)
Garage Cup (Node 3)	(5.5 m/s, 0.1 N)	(5.5 m/s, 0.1 N)	(0%, 0%)

Table 1 Wind direction and speed comparison table between COMSOL results and Cup Anemometers data. Wind induced from 0.1 degrees from north at 5.5 m/s.

South

	COMSOL Calculation	Measured Wind by Cup Anemometers	% Difference (Velocity, Direction)
North Cup (Node 1)	(6.2 m/s, 120.2 SE)	(5.1 m/s, 109.3 SE)	(5.9%, 2.1%)
South Cup (Node 2)	(4.2 m/s, 160.2 S)	(3.9 m/s, 152.8 SE)	(7.7%, 4.5%)
Garage Cup (Node 3)	(6.5 m/s, 180.5 S)	(6.5 m/s, 180.5 S)	(0%, 0%)

Table 2 Wind direction and speed comparison table between COMSOL results and Cup Anemometers data. Wind induced at 180.5 degrees from north at 6.5 m/s.

East

	COMSOL Calculation	Measured Wind by Cup Anemometers	% Difference (Velocity, Direction)
North Cup (Node 1)	(5.5 m/s, 83.6 E)	(5.2 m/s, 64.6 NE)	(5.8%, 5.3%)
South Cup (Node 2)	(4.3, 95.5 E)	(4.0 m/s, 102.6 SE)	(7.5%, 2.0%)
Garage Cup (Node 3)	(5.7 m/s, 90.9 E)	(5.7 m/s, 90.9 E)	(0%, 0%)

Table 3 Wind direction and speed comparison table between COMSOL results and Cup Anemometers data. Wind induced at 90.9 degrees from north at 5.7 m/s.

West

	COMSOL Calculation	Measured Wind by Cup Anemometers	% Difference (Velocity, Direction)
North Cup (Node 1)	(4.7 m/s, 201.3 SW)	(4.4 m/s, 173.5 S)	(6.8%, 7.7%)
South Cup (Node 2)	(7.1 m/s, 252.2 W)	(7.8 m/s, 208 SW)	(9.0%, 12.1%)
Garage Cup (Node 3)	(5.9 m/s, 270.2 W)	(5.9 m/s, 270.2 W)	(0%, 0%)

Table 4 Wind direction and speed comparison table between COMSOL results and Cup Anemometers data. Wind induced at 270.2 degrees from north at 5.9 m/s.
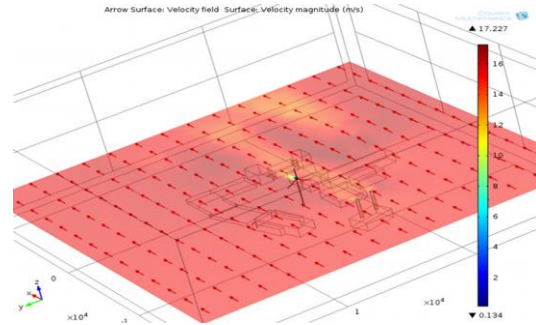
5. Experimental Results

Figure 3 Hub height wind velocity plot; 15 m/s induced wind from the East (90° non-prevailing) direction; note the slight yellow affecting the turbines blades at hub height.
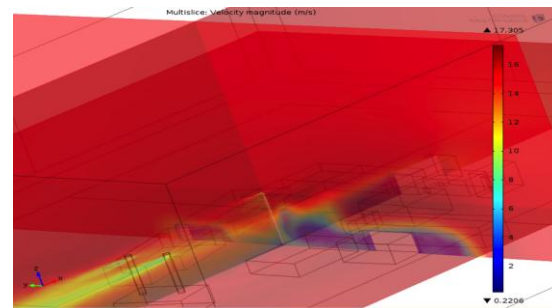

Figure 4 Wind velocity multi-surface; airflow induced from West (270° prevailing) 15 m/s; note the blue boundary is well below hub height.
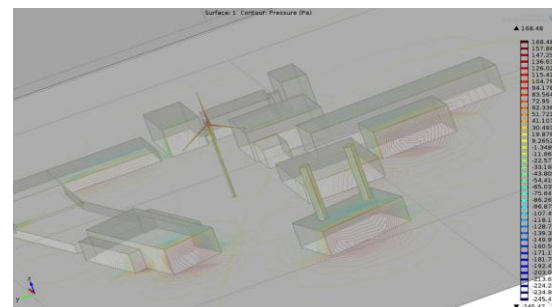

Figure 5 Pressure gradient of turbine area; airflow induced at East (90°non-prevailing) 15 m/s; note the high pressure build up on the blades of turbine.
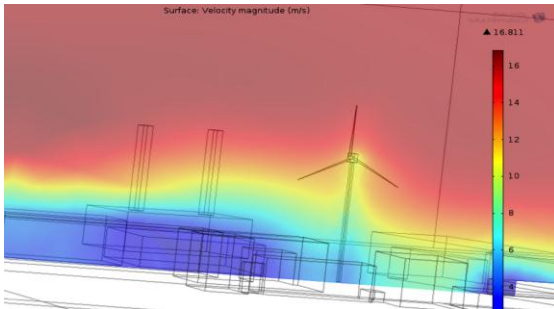
Figure 6 Wind velocity surface of CWRU turbine area; air flow induced from West (270°prevailing) 15 m/s; note the yellow boundary is below the blades.
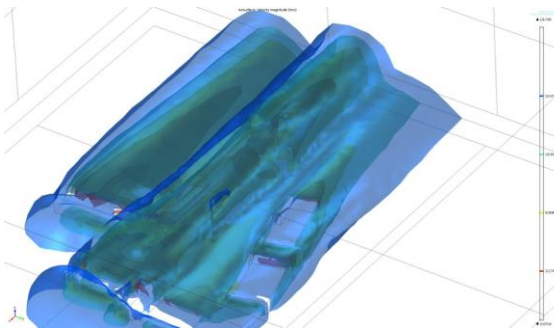

Figure 7 Wind velocity isosurface; airflow induced from West (270° prevailing)15 m/s; note the blue layer does not hit hub height until the air reaches the tower.
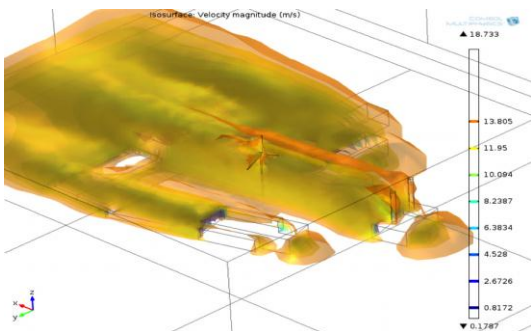

Figure 8 Wind velocity isosurface of turbine area; airflow induced at East (90°non-prevailing) 15 m/s; note the yellow slightly going over the blades.
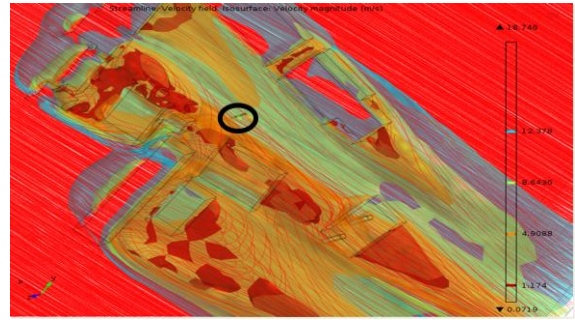

Figure 9 Wind velocity surface with line tracing, bottom view, circled in black is bottom of the turbine; airflow induced from West (270° prevailing)15 m/s; note the straight flow pattern in the flow tracking streaks.

6. Discussion

6.1 General Discussion

From the CFD results created by COMSOL Multiphysics, it can be seen that there is a very small effect due to the buildings in most cases, especially when the wind is induced from the North West direction, the prevailing direction. In *Figure 4, Figure 6, and Figure 7* it can be seen by the colors at hub height that there is no effect on the wind turbine.

6.2 Laminar Flow Validation

For this project the COMSOL Multiphysics basic package was used which only solves laminar flow problems. For air at room temperature with velocity of 15 m/s the Reynolds number was calculated to be around 1.1×10^6. However, a test to validate using laminar flow was conducted by looking at the difference in boundary layer as a function of the Reynolds number. In order to conduct this experiment, the Reynolds number was decreased by increasing the dynamic viscosity of air by keeping all else constant. The dynamic viscosity was increased by an order of magnitude each time resulting in the Reynolds number increasing an order of magnitude. The COMSOL model was run seven times and the boundary layers were analyzed by using the surface view feature. The difference between boundary layers was minimal. There was a 0.4 meters change in the boundary layer when the Reynolds number was changed from 1.1×10^6 to 11.0.

7. Conclusion

- ✓ The model had reasonable agreement with the wind measurements
- ✓ The model visually demonstrated the effects of the buildings on the wind profile, thus on the turbine
- ✓ For the prevailing wind direction the wind did not have an effect on the turbine at hub height
- ✓ From a very unlikely wind direction (North East), the turbine was slightly affected by the MCCO smoke stacks

8. Acknowledgements

This project could not have happened without the generous contribution of:

- Case School of Engineering Office of the Dean
- Support of Undergraduate Research and Creative Endeavors of Case Western Reserve (SOURCE)
  - o Dominion Foundation
  - o Case Alumni Association
  - o Case Western Reserve University Facilities Dept.