



Preparing people to lead extraordinary lives

Modeling Wind Flow Around Single Obstructions to Aid in Wind Turbine Placement Decisions

Engineering Science Capstone Design Project

Stefano Favuzzi, Jacklyn Hoppe, Carolyn Judy, Briannah Smith

Sponsor: One Energy Faculty Advisor: Dr. Gaj Sivandran



Background and Problem Statement:

Wind energy is usually created by large-scale wind farms with hundreds of turbines. Our sponsor, One Energy, takes a different approach and provides power to manufacturing plants by installing a small number of turbines on their property. This creates a challenge because there are no industry-wide standards to determine how small, single obstructions impact the wind flow. One Energy currently uses the Perera Obstacle Model, which is outdated. Our team was tasked with creating a new modeling method to model wind flow around single small obstructions to aid in One Energy's wind turbine placement decisions.

Perera Obstacle Model:

The Perera Obstacle Model [1] is based on a study from 1981 of the same name which estimates the percent wind speed reduction based on the ratio of the height of the obstruction and the height of the turbine hub. This is the model that One Energy currently uses. They would like to replace it because it is outdated and because One Energy must modify it to find useful because the hub height to obstruction height is too large for the Perera Obstacle Model to use.

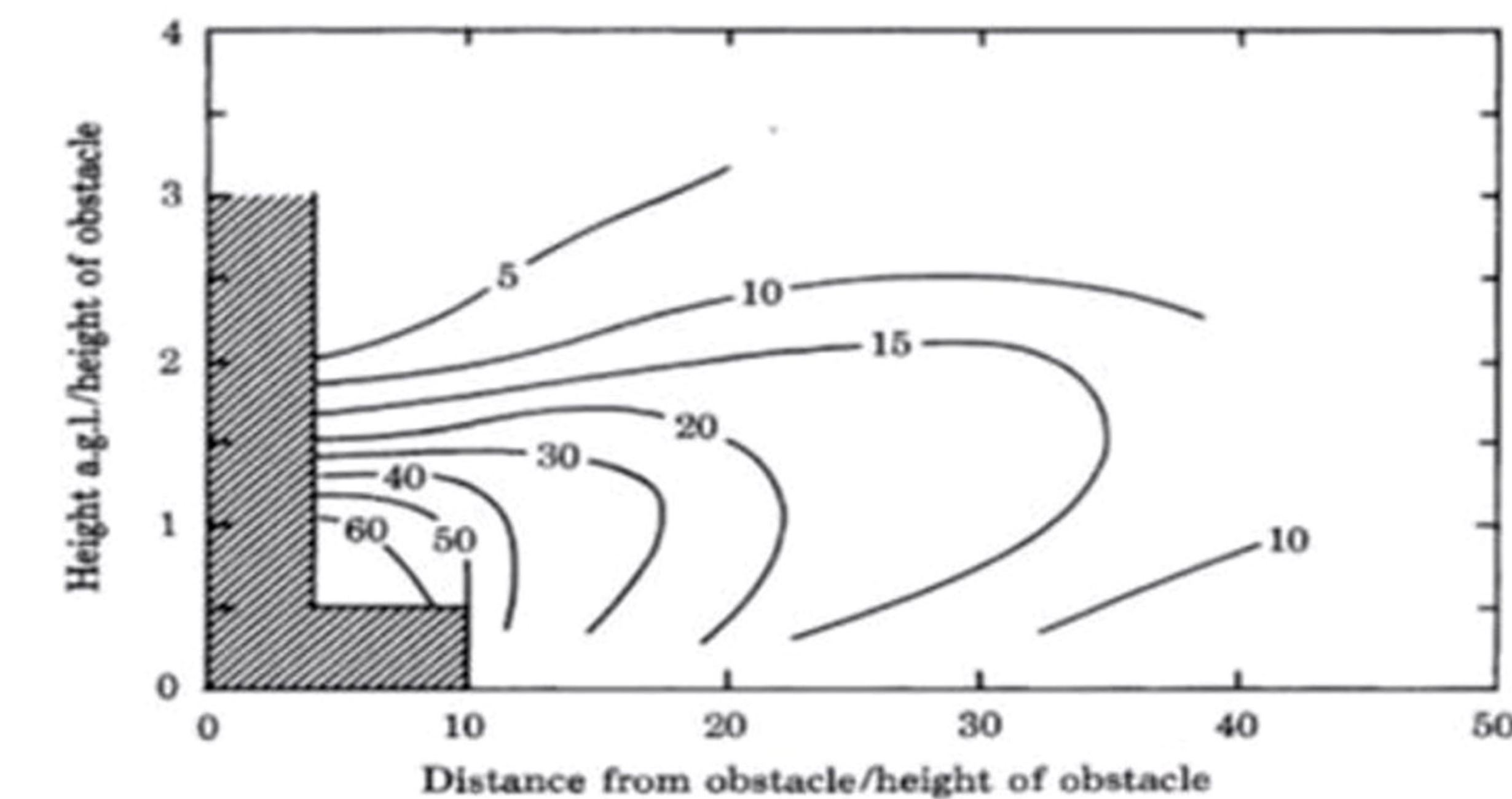


Fig. 1: The results of the Perera Obstacle Model study which shows the percentage wind speed reduction caused by an obstruction by distance from the ratio of the distance from the obstruction to the height of the obstruction and ratio of the turbine hub height to obstruction height. [1]

Turbulence:

Modeling wind flow poses a challenge because there is no mathematical way to calculate turbulence; we can only estimate it. Wind flow is estimated by calculating the Navier-Stokes equations, a series of six equations which describe motion in fluid [2]. Turbulence is then calculated by different models to account for turbulence kinetic energy and dissipation of eddies, which can be seen in Figure 2. Eddies are areas of circular flow moving against the flow.

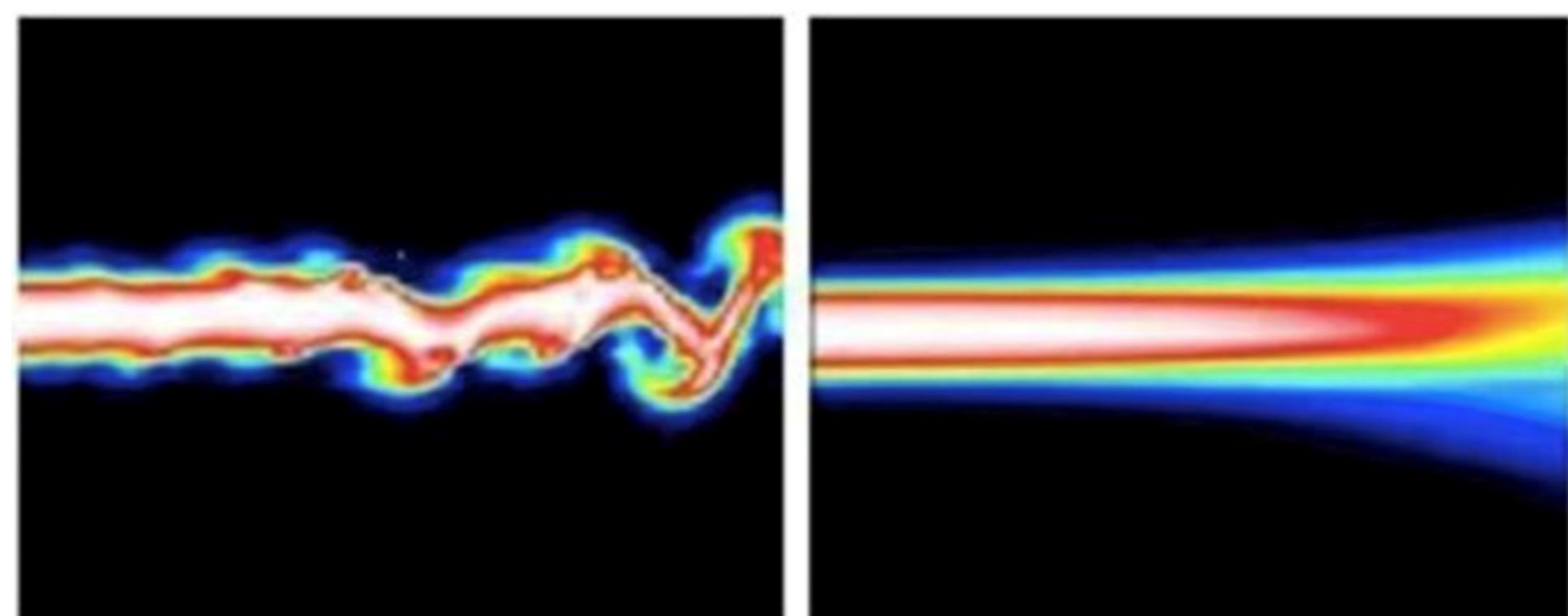


Fig. 2: The image on the right shows a laminar (non-turbulent) wind flow. The image on the left shows turbulence and eddies. [3]

Modeling:

Due to other turbulence modeling software in the industry having poor equation visibility, we decided to use a software called FEATool (Finite Elemental Analysis Toolbox) Multiphysics [4]. Using the FEATool CFD (Computational Fluid Dynamic Simulation) Toolbox, we were able to set up and solve multiple turbulence models, with the upmost equation visibility and adjustability. Using the open source OpenFOAM CFD toolbox in FEATool, we were able to run multiple turbulence models in the FEATool environment. Through research, we decided which turbulence models they wanted to compare to determine the best modeling method for flow around a single building. In FEATool we were able to run the chosen k-ε and k-ω turbulence models, and in the standalone OpenFOAM software we were able to run models using the LES (Large Eddy Simulation) turbulence model. Differences in how these turbulence models ran were compared to real world wind tunnel data to determine which turbulence model to proceed testing with. We decided to continue testing with the k-ε turbulence model in FEATool and ran case studies to determine the effects of turbulence in different circumstances.

Our client, One Energy, requested that the simulations be able to run on a desktop computer. Due to long computation times and instability when modeling in 3D, we decided to do the turbulence modeling in two dimensions. The major limitation of 2D modeling includes the forcing of flow around the sides over the top of the object, but despite this limitation we were still able to validate the model using similar wind tunnel data and draw useful conclusions from the modeling.

Acknowledgements and References:

We would like to thank Ben Malterre and Erin Roelke from One Energy for their expertise and guidance through this project. Dr. Gaj Sivandran, Dr. Gail Baura, Dr. Jason Streeter, and Dr. Sarah Ali, for their support and guidance through this project, and Jihan Hying from FEATool for his guidance in modeling. [1] M. Perera, "Shelter behind two-dimensional solid and porous fences," Journal of Wind Engineering and Industrial Aerodynamics, vol. 5, no. 1-2, pp. 93-104, 1981. [2] "Steady-state Navier-Stokes equations," Navier-Stokes Equations, pp. 105-165, Oct. 2003. [3] Xie, Charles. "The Reynolds Number." The Concord Consortium, National Science Foundation, 2010. [4] Hying, Jihan S. "MATLAB Finite Element FEM Simulation Toolbox." FEATool Multiphysics, 2013. www.featool.com/ [5] B. San, Y. Wang, and Y. Qiu, "Numerical simulation and optimization study of the wind flow through a porous fence," Environmental Fluid Mechanics, vol. 18, no. 5, pp. 1057-1075, 2018. [6] B. Leitl, Quasi 2D Flow Across a Square Shaped Beam, Hamburg: Meteorological Institute, 1999 [Dataset]. Available: https://mji-pub.com.uni-hamburg.de/index.php?id=428. [Accessed: 7 April 2020]. [7] NASA Langley Research Center. "CPEX_DAWN_DCS." 24 June 2017.

Turbulence Equations:

Turbulence calculations are necessary to model wind flow around obstructions. Turbulence calculates the steadiness of the wind and determines how wind is altered when it flows over or around an object. After conducting a thorough literature review, the CDP team established multiple turbulence models that could calculate wind speed reduction around an object. The software chosen to build the model, FEATool, utilizes the standard k-ε equation to solve for the fluctuations in wind speed. Because the Perera model does not utilize specific turbulence equations, the CDP team will compare the results of their model's wind speed reduction with the Perera model process to determine if inclusion of turbulence equations develop more accurate results. The turbulence model utilized within FEATool is explained below.

Standard k - ε equation [5]

- Model is frequently utilized in computational fluid dynamic models
- The Reynolds number must equal or equal to 4000 to demonstrate turbulent flow.
- There are two transport equations that make up this model. k, kinetic turbulent energy and (ε), energy dissipation rate.
- This model is governed by the Reynold's-Averaged Navier-Stokes momentum equation and the Continuity equation
- The two equations are derived from the Reynolds stress transport equation to account for the directional effects of the Reynolds stresses and complex interactions in turbulent flow.

- k is the kinetic energy:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\frac{\mu}{\rho} \frac{\partial u_i}{\partial x_j} - u_i' u_j' \right)$$

- ε is the dissipation rate

$$\frac{\partial \epsilon}{\partial t} + u_j \frac{\partial \epsilon}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left(\alpha_\epsilon \mu_{eff} \frac{\partial \epsilon}{\partial x_j} \right) + \frac{1}{\rho} C_{\epsilon 1} G_k \frac{\epsilon}{k} - \left(C_{\epsilon 2} + \frac{C_\mu \rho \eta^3 \cdot (1 - \eta/\eta_0)}{1 + \beta \eta^3} \right) \frac{\epsilon^2}{k}$$

- The equations variables are:

- μ_{eff} : effective turbulent viscosity
- α_k : inverse effective Prandtl number for k
- α_ε : inverse effective Prandtl number for ε
- η : S^k/ε
- S : scalar measure of deformation tensor
- η₀ : constant, =4.39
- β : constant, =0.012
- C_μ : constant, =0.0845
- C_{ε1} : constant, =1.42
- C_{ε2} : constant, =1.68
- G_k : turbulent kinetic energy production
- u_i : fluid velocity in the jth direction
- x_j : the jth direction
- t : time
- ρ : density of the air
- k : turbulent kinetic energy
- ε : kinetic energy dissipation rate

Validation:

In order to validate the efficacy of the FEATool model, the results are quantitatively compared to data that has already been proven valid. Hamburg University published CEDVAL (Compilation of Experimental Data for Validation)[6], composed of several data sets from their wind tunnel that will serve this purpose. Since the data comes from a physical wind tunnel, it is considered valid.

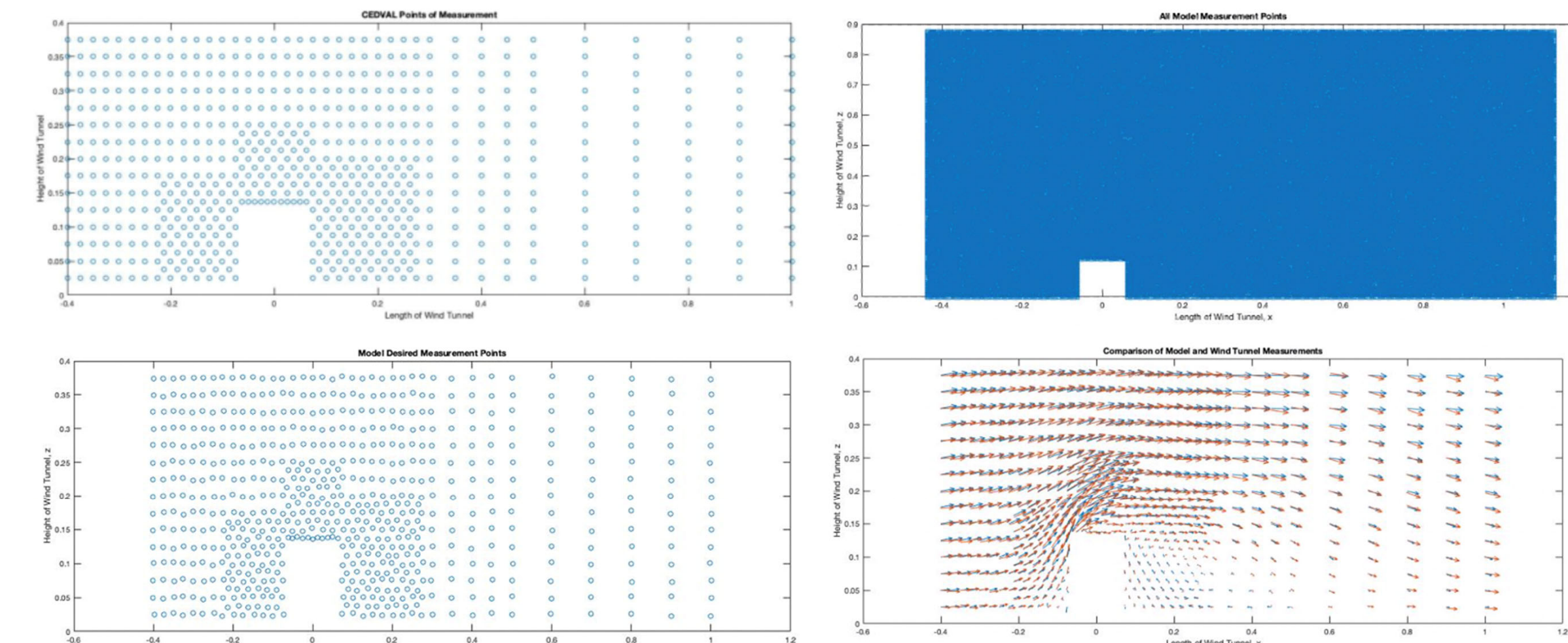


Fig. 3: MATLAB results of validation protocol. Top left: CEDVAL points of measurement, top right: Model points of measurement, bottom left: points of comparison, bottom right: vector field comparing CEDVAL wind velocities and directions and model wind velocities and directions. The CEDVAL data is in blue and the model data is in red. All axis measurements are in meters.

The CEDVAL data set used for validation comes from an experiment using a square beam across the width of the wind tunnel. A beam was used rather than a block because in FEATool's 2D geometry, a block acts as a beam. Each arrow represents one of the 670 points of measurement. Each measurement has an associated horizontal and vertical wind velocity. These values are used to compute wind direction. In order to compare the CEDVAL results to those of the model, the CEDVAL results must be replicated in MATLAB.

First, the CEDVAL measurement points are mapped (see Figure 3, top left). To make an accurate comparison, the model version is evaluated at or near these points. FEATool takes velocity measurements at more than 65,000 points on the plane (see Figure 3, top right). In order to refine the large model data set, only the 670 points closest in Euclidean distance to CEDVAL's measurement points are considered (see Figure 3, bottom left). Once the points have been selected, a quiver plot is constructed to show the similarities and differences between CEDVAL's data set and the model's data set (see Figure 3, bottom right).

Testing:

In order to gain useful insights from our model, tests need to be performed. Our team decided on four tests; one using real world data and three case studies. Our sponsor felt that it was important to see how the model interacts with real world data. We found a data set from a NASA study called the Convective Processes Experiment Doppler Aerosol Wind (CPEX DAWN)[7] study. This study used a Doppler Aerosol wind instrument attached to a low flying plane to measure wind profiles across the Florida Peninsula. The profiles contain the wind magnitude and direction in 50-meter vertical increments at each specific point. To use this data set, we used the corresponding Google Earth files to identify points where wind data was taken that are near a small, single obstruction in an otherwise flat, unobstructed area. We then found a nearby unobstructed point to build in input profile from. Next, we measured the dimensions of the obstruction and the distance to the wind data point. We then ran the input profile through the model.

In addition to the real-world study, we created three case studies to analyze the impact of changing the obstruction height, input velocity, and obstruction length. In each case study, we ran ten tests, changing the variable of interest by a set amount each time. After we ran each of the models, we applied the k-ε turbulence model and analyzed the results.

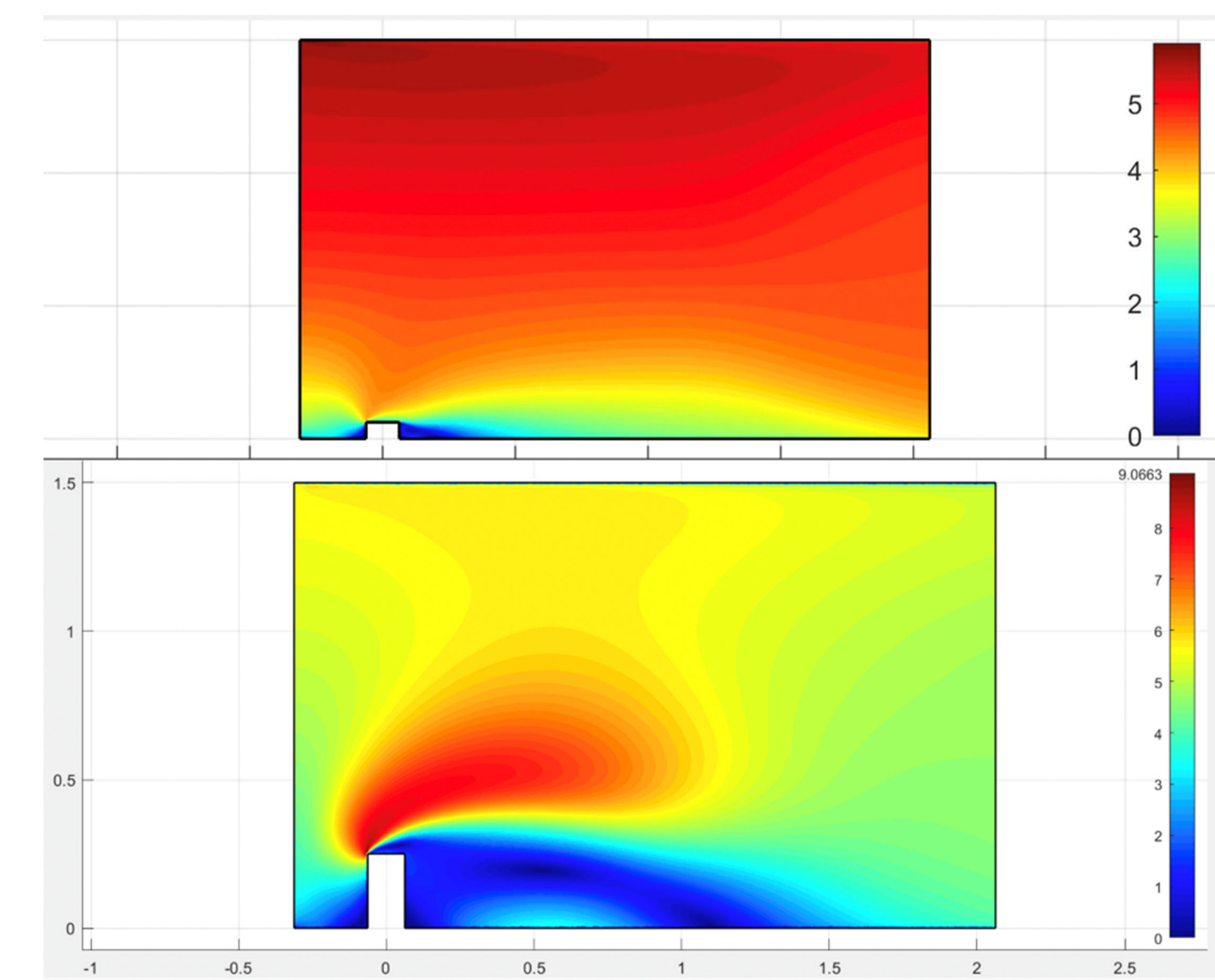


Fig. 4: The visual result of two case outputs of the height case study. The top image shows the half height study and the bottom image shows the double height study. For each variable of interest, studies were run in 0.25 increments. The colors represent wind velocity, with red being high wind velocity and blue being low wind velocity.

Results:

For each case study (obstruction height, obstruction length, input velocity) we analyzed the results at 1 turbine fall length, 1.5 turbine fall lengths, and hub height. The turbine fall length is the height at the tallest point on the turbine and the hub height is the height of the tower, where each blade is attached to the turbine. We did not analyze closer to the obstruction than 1 turbine fall length away because One Energy would not place a turbine closer than that to a building.

Each case allowed us to understand the relationship between the variable of interest and the resulting wind speed in the u-direction, or the wind flowing perpendicular to the ground. Figure 5 shows the results of all three case studies. The percentage is the percentage of the original inlet wind speed that remains at each point, measured from the hub height. From this table, we can see that, at 1 turbine fall length away and beyond, the obstruction does not have a significant impact on the wind velocity.

HEIGHT	1 TURBINE FALL	1.5 TURBINE FALL	VELOCITY	1 TURBINE FALL	1.5 TURBINE FALL	LENGTH	1 TURBINE FALL	1.5 TURBINE FALL
x2.0	110.01%	98.56%	x2.0	100.85%	99.43%	x2.5	100.70%	98.76%
x1.75	105.22%	96.88%	x1.75	100.65%	97.15%	x2.0	100.67%	99.36%
x1.5	103.64%	97.48%	x1.5	100.96%	100.43%	x1.75	100.87%	100.13%
x1.25	102.04%	100.46%	x1.25	100.97%	100.45%	x1.5	101.01%	100.21%
x1.0	101.37%	101.59%	x1.0	101.37%	101.59%	x1.0	101.17%	100.25%
x0.75	100.40%	100.16%	x0.75	100.90%	99.51%	x0.75	101.20%	99.71%
x0.5	99.78%	97.24%	x0.5	100.93%	99.55%			
x0.25	99.66%	98.35%	x0.25	101.14%	100.23%			

Fig. 5: Table showing the results of each study at 1 turbine fall length and 1.5 turbine fall lengths away. The results show the percentage of the original inlet wind speed that remains at each point, measured from the hub height.

The results of the obstruction height case study can be seen in Figure 6. At lower heights, the wind velocity is slowed by the obstruction but higher above the obstruction the wind stream returns close to free stream. The third image in Figure 6 show the percent wind speed change at the hub height for every x point. A taller height corresponds to faster wind velocity close to the obstruction, but the wind speed slows down further away from the obstruction.

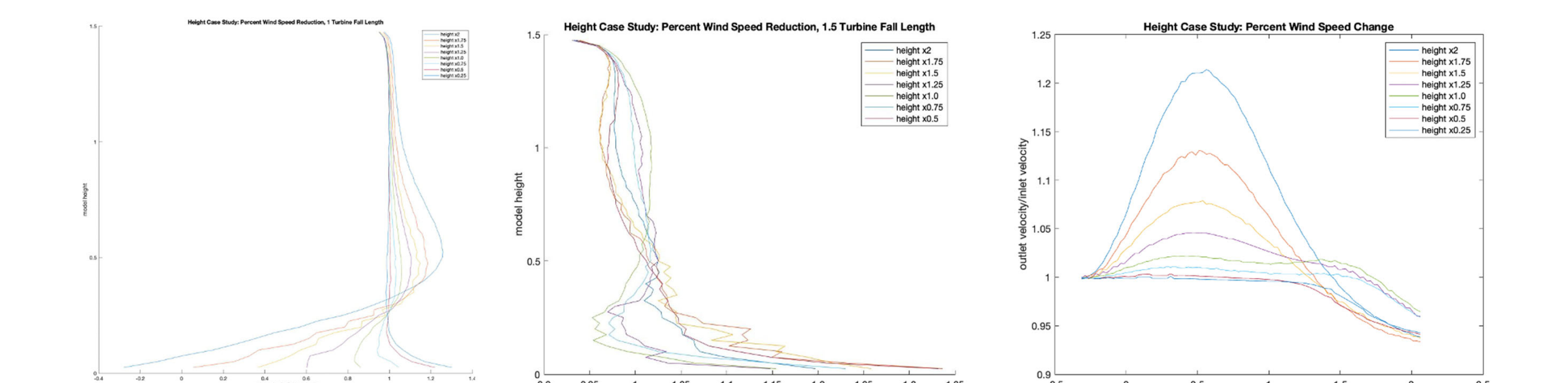


Fig. 6: Results of the height case study. The first image shows all the points of wind speed reduction in the y-direction at 1 turbine fall height from the obstruction. The second image shows all the points in the y-direction at 1.5 turbine fall heights. The third image shows the wind speed reduction at hub height for every point in the x-direction.

The results of the obstruction length and input velocity are not shown here but still important. The obstruction length impacts wind velocity at lower and higher altitudes but around the hub height there is no significant decrease in wind velocity. At the hub height, the wind is faster at all points before 1 turbine fall height and equal or slower after. The graphed results of the input velocity are very close together, meaning that changing the velocity of the wind does not impact how the wind interacts with the obstruction.