ELECTRIC ROOTS

Electric Roots Experimental Report

14th of April, 2023

Ciana David, Cameron Fox, Gina Kay, Olivia Pereyra, Audrey Wilson

Oregon State University

Experimental Report

Introduction

Our team at Electric Roots has been working with our clients at RUTE SUNTRACKER to evaluate the distribution and influences of wind forces on an array of photovoltaic (PV) panels. More specifically, our clients, David McFeeters-Krone and Dr. Frank Chaplen would like the Electric Roots team to evaluate the convoy effect (See Definitions pg.19) on the array for the purpose of future design optimization. This will be achieved using the computational fluid dynamics (CFD) program Ansys Fluent.

The RUTE SUNTRACKER design currently consists of an array of 20ft x 20ft panels that will stand 9 ft above ground at the lowest point of the panel edge. Each panel is connected via stabilizer cables (Figure 1 pg.2 and Figure 4 pg.14). The cable-supported PV panels are to be placed within a pastureland and will function as an energy source and help conserve water. The system will provide shade for the grazing animals and crops, which will increase efficiency of water use by reducing evapotranspiration from pastureland.
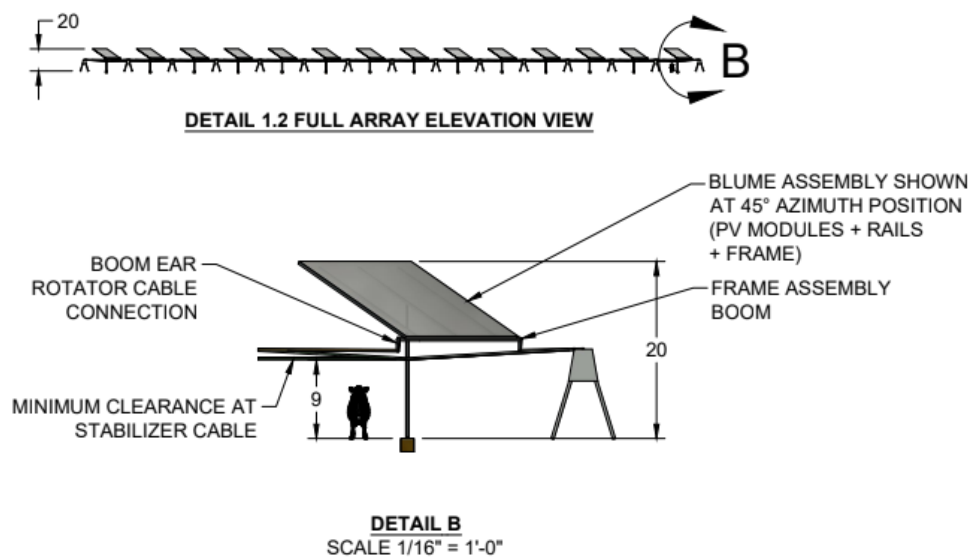

Figure 1: RUTE SUNTRACKER photovoltaic panel design [1].

Methods

Ansys Fluent was used to study the effects of wind forces on the RUTE SUNTRACKER PV panels. The project was completed in 5 steps:
1. A study conducted by NASA Ames Laboratory that provided data from a wind tunnel experiment on heliostats [2] was evaluated.
2. Two wind loading equations were studied as a reference point for calibration and validation of models created in Ansys and to inform on relevant parameters.
    a. The American Society of Civil Engineers (ASCE) equations for wind loads on solar panels using the ASCE handbook section ASCE 7-16 [3] were explored.

      b. The drag force equation and force coefficients (See Definitions pg.19) were explored using python with input data from the NASA Ames Laboratory study [2].

3. A model of a single heliostat from the NASA Ames Laboratory study [2] was developed in Ansys Fluent for use in future calibration of the RUTE SUNTRACKER model.
4. A model of a single RUTE SUNTRACKER PV panel was built in Ansys Fluent.
5. A model of a series of three RUTE SUNTRACKER PV panels were built in Ansys Fluent.

Alternatives to Ansys Fluent were using other CFD programs or conducting a wind tunnel experiment. Ansys Fluent was selected due to accessibility.

Python NASA Ames Analysis

A regression was run on the data from the NASA Ames Wind Tunnel study. The scaled heliostat panels were square with each side being 200 mm and the width being 5 mm. The velocity of the wind, the temperature of the facility, and the density of the air was kept constant at 18.2 m/s, 23 ℃, and 1.20 kg/m^3 respectively. The wind tunnel study had model heliostat panels tilted at angles to the ground (α) from 90° to 0° in increments of 15°. At each 15° increment, a uniform load of wind was directed onto the panel at the constant velocity, temperature, and density provided above. The wind was blown at angles to the PV panels (*β*) from 0° to 360° in 5° increments. In the data from the NASA Ames Wind tunnel study, the force and moment coefficients for each cartesian coordinate was provided instead of the force, given that the coefficients should be constant based on the direction of the wind while the force can change due to factors such as velocity and air density (See Figure 5 pg.14) [2].

The data from the NASA Ames Wind Tunnel study has multiple parts, including a single heliostat, an array of heliostats, and heliostats with various barriers. For the regression analysis, the team has so far focused on data from a single heliostat tilted at 35° to the ground per recommendation from the client.

Python coding language was used in JupyterHub to run the regression. Regression, statistical, and graphing packages *pandas*, *curvefit* (from *scipy.optimize*), *r2_score* (from *sklearn.metrics*), *numpy*, and *matplotlib.pyplot* were used. Data was first graphed using *matplotlib.pyplot* to see what type of regression it represented. Visual inspection of the graphs determined that the data most represented a sinusoidal wave. Given that the angle of the wind was changed from 0°-360°, it would stand to reason that the force coefficients (CFx, CFy, CFz) would also change sinusoidally with the angle. For the CFx, CFy, and CFz force coefficients the team modeled an equation on JupyterLab. With B being the force coefficient, t being the the angle of the wind, and b, e, a, c, f, g, and d being constants determined by regression, the equation for each coefficient was generically

$$B = b * sin(t^c) + e * cos(t^f) + a * cos(t^g) + d.$$

Most force coefficients had a slightly different form than the generic form based on manual guessing and checking (see Results pg.6).

The *scipy.optimize* function *curve-fit* was used to determine the constants from the force equation. *curve-fit* needs 5 inputs to run: the model equation, the x data (angle of the wind), the y data (force coefficients), initial guesses for each constant, and the maximum amount of times the

model is run to determine the constants. Once the models were determined from curvefit, the constants were entered into the model equation to make an equation specific for each coefficient. Wind angles were entered into the model equation and the R^2 value was determined from the *sklearn.metrics* function *r2_score*. *r2_score* needs the actual y values (force coefficients from data) and the modeled y values (force coefficients from the model equation) to determine the R^2 value. Data from the heliostats and the modeled y-values were graphed for a visual comparison, and to determine which angles reduced the force of the wind.

ASCE 7-16 Standards

To familiarize ourselves with the physics of the situations being studied ASCE equations for wind loads on solar panels using the ASCE handbook section ASCE 7-16 were studied [3]. We were able to access the full ASCE 7-16 text through an OSU interlibrary loan. The ASCE model for predicting wind forces on the panels includes parameters for variable wind gusts, wind speeds, topographical factors, ground elevation, and force coefficient standards for solid structures. These coefficients were all taken from various ASCE tables that identify a coefficient based on wind angle, speed and panel height, and angle. Factors for topographical factors and wind gusts were not taken into account as a simplifying assumption for the case of comparing the wind loads gained from this equation to the NASA wind tunnel experiments. The force outputs for the ASCE equations were categorized by the amount of wind force per square foot on the windward side and the leeward sides of the structure (See Figure 6 pg.15) [3].

ANSYS Fluent - NASA Ames Wind Tunnel Study

Ansys Fluent is a CFD program that can be used to analyze the force of wind on one or multiple PV panels. It was first used to model a single heliostat from the NASA Ames Laboratory study [2]. As mentioned before, the heliostat was 200 mm x 200 mm x 5 mm and was set 130 mm from the ground to the centerline of the heliostat. The single heliostat model geometry (See Definitions pg.19) was created in Ansys DesignModeler (See Figure 7 pg.15). A square control volume (See Definitions pg.19) with side lengths of 1210 mm was created to represent the wind tunnel. The panel was set at a 35° angle. The heliostat surface, control volume ground, and control volume ceiling was set as an impermeable surface, to match the wind tunnel that had a ceiling. The left control volume side was designated as the inlet for wind and the right side was designated as the outlet.

The quadrangle method was used for meshing (See Figure 8 pg.16). Within Ansys Fluent, the inlet and outlet were designated a velocity-inlet and outflow, respectively. The Reynolds Stress Viscous model was used, air density was set to 1.20 kg/m^3, and all other model parameters were kept at their default setting. The model was then run for a wind velocity of 42.6 m/s matching the NASA Ames Laboratory wind tunnel experiment [2].

ANSYS Fluent - RUTE SUNTRACKER Design

For the client's design, the team created two meshes (See Definitions pg.19). The first mesh modeled a single PV panel and the second mesh modeled an array of three PV panels. Both meshes were 2D. No poles or cables were modeled on the mesh as wind forces on them were assumed to be negligible.

Single Panel Geometry

The geometry of the single panel mesh involved two rectangles created using DesignModeler in Ansys. A model with a panel length of 20 ft tilted at 35° to the ground with the lowest edge 9 ft

above the ground was created (Figure 9 pg.17) (See Details pg.19). The thickness of the panel was 4 in. Once the rectangles were created the team generated a surface skin of the geometry and imported it into Fluent Meshing.

Triple Panel Geometry
The geometry of the three panel mesh involved four rectangles created using DesignModeler (Figure 10 pg.17). Like the first mesh, the first rectangle modeled the control volume of the system. It had a length of 210 ft and a height of 60 ft. The following three rectangles modeled the three panels (See Details pg.19). All three of the PV panels had a length of 20 ft, a thickness of 4 in, were tilted relative to the ground at 35°, had the lowest panel edge 9 ft from the ground, and were spaced 34 ft apart.

General Fluent Meshing
In Fluent Meshing, the All Triangle's Design was used for both of the meshes for the client's design. For both meshes, elements had a size of 0.025 ft around the panel. The left hand edge of the control volume was an inlet or an outlet, the top edge was an outlet, the right hand edge was an inlet or an outlet (opposite of the left hand edge), and the bottom edge was an impermeable surface (or "wall"). The edges of the PV panels were walls.

Single Panel Fluent Meshing
In the single panel design, the element size (See Definitions pg.19) for the wind area was 1 ft. There were 73,781 nodes and 80,576 elements (Figure 11 pg.18). All other meshing values and parameters were default.

Triple Panel Fluent Meshing
In the triple panel design, the team kept the default values for inflation (See Definitions pg.19). To save computation time, the team had an element size of 4 ft for the wind area. There were 36,697 nodes and 68,370 elements (Figure 12 pg.18). All other meshing values and parameters were default.

Running the Models
Precision was doubled in the Ansys Fluent Solution startup window as this will create better model results. System processors were kept at 1. The Reynolds Stress Viscous model was used for the solution. The fluid time scale was left with the default settings, and the fluid and wall material were left as default (air and aluminum respectively). The force vector of the wind was modeled as [1,0] (x,y) and was measured in Newtons. The inlet was modeled as a velocity-inlet and the outlet as a pressure-outlet. Four different scenarios were run for the client's single panel design (See Table 1 pg.6). All variables were kept constant within these scenarios except for the direction and wind speed. In the 70 MPH scenarios, 500 iterations were run, and in the 114 MPH scenarios, 800 iterations were run.

Table 1: Wind speed and panel scenarios. "Front" refers to the left hand side of the control volumes and "Back" refers to the right hand side.

Scenario	Wind Speed	Panel #	Wind Inlet	Wind Outlet
1	70 mph	1	Front	Back
2	70 mph	1	Back	Front
3	70 mph	3	Front	Back
4	70 mph	3	Back	Front
5	114 mph	1	Front	Back
6	114 mph	1	Back	Front
7	114 mph	3	Front	Back
8	114 mph	3	Back	Front

Results

Python - NASA Ames Analysis

The model equations for the force coefficient in the x, y, and z directions are below in Table 2 (pg.6) including the associated R^2 values. For the force coefficient in the y-direction the data needed to be split up into two for a viable R^2 value. An equation was given for the y-direction when the wind is coming at an angle from 0° to 180° and another equation for 180° to 360°. The plots of these equations can be found in Plots 1-4 (pg.12-13).

Table 2: Force coefficient equations and associated R^2 values.

Direction	Equation	R^2
x	$CFx =$ $0.149\,sin\left(t^{0.4}\right) - 0.101\,cos\left(t^{0.4}\right) + 0.252\,cos\left(t^{0.5}\right) + 0.466$	0.667
z	$CFz =$ $0.833\,sin\left(t^{0.4}\right) + 0.244\,cos\left(t^{0.4}\right) - 0.0625\,cos\left(t^{0.5}\right) + 0.0475$	0.775
y (0-180°)	$CFy =$ $-5.63\,cos\left(t^{1.13*10^{-9}}\right) + 5.71\,cos\left(t^{5.14*10^{-9}}\right) - 0.32\,sin\left(t^{0.4}\right) - 0.0591$	0.576
y (180-360°)	$CFy =$ $201\,sin\left(t^{-0.0128}\right) + 0.349\,cos\left(t^{0.608}\right) + 0.0622\,cos\left(t^{0.588}\right) - 161$	0.989

ASCE 7-16 Standards

The wind forces on a heliostat the size of the one from the NASA Ames Laboratory wind tunnel experiments with a wind angle of 180° and a tilt angle of 30° and 60°, respectively, were calculated using the ASCE 7-16 equations and are provided in Tables 3-4 (pg.7) below [2][3].

Table 3: Force results for tilt angle ≤ 45° (considered as open building with monoslope roof) (See Equation 1 pg.21) [3].

Equation	Force (lb/ft^2)	Location	Force (lbf) / ft^2 of surface area for a 200 mm^2 x 130 mm high panel
$p = q_h G C_N$	13.7	Windward and Leeward sides	5.9

Table 4: Force results for tilt angle > 45° (considered as solid sign) (See Equation 1 pg.21) [3].

Equation	Force (lb/ft^2)	Location	Force (lbf) / ft^2 of surface area for a 200 mm^2 x 130 mm high panel
$p = q_h G C_N$	10.2	Windward (extreme angle)	4.41

Ansys Fluent

The models made in Ansys Fluent represent wind forces acting on and around the PV panels. This resulted in plots that reported velocity or total pressure. Figures 2-3 (pg.7-8) below are a representation of a model output for the triple panel series with wind flowing from the front and back at 114 MPH and will be analyzed further in the Discussion section (pg.8). Figures 13-16 (pg.22-25) report all the model outputs for every scenario looked at on the single panel and triple panel RUTE SUNTRACKER model. For the triple panel model, the panels are numbered 1-3 going from left to right (See Details pg.19).
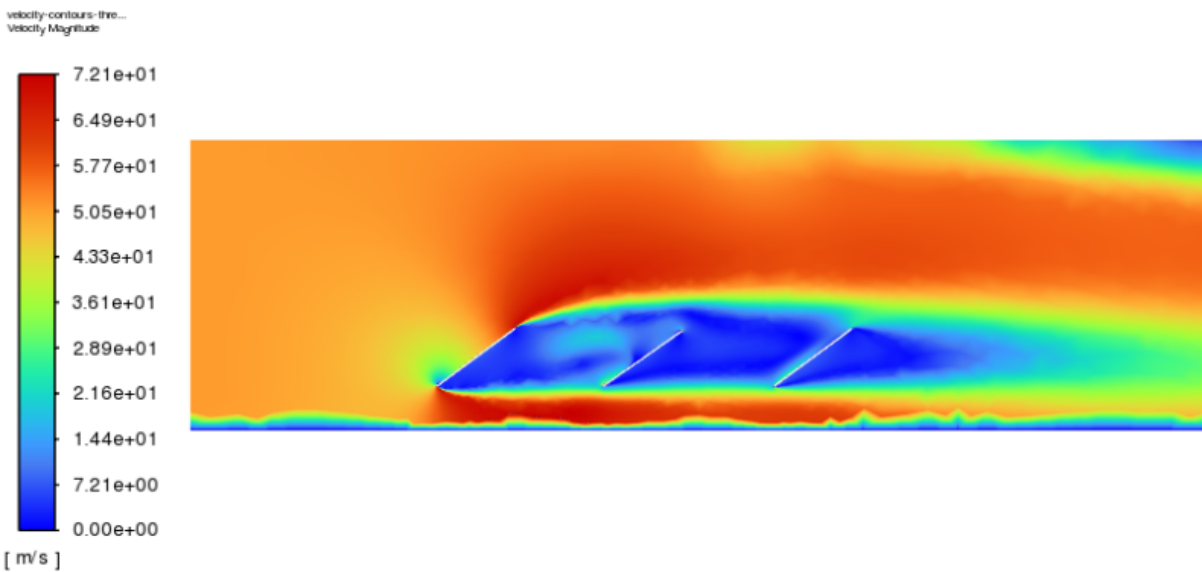

Figure 2: 114 mph wind on 3 panels with wind coming from the front (left or 0° angle).
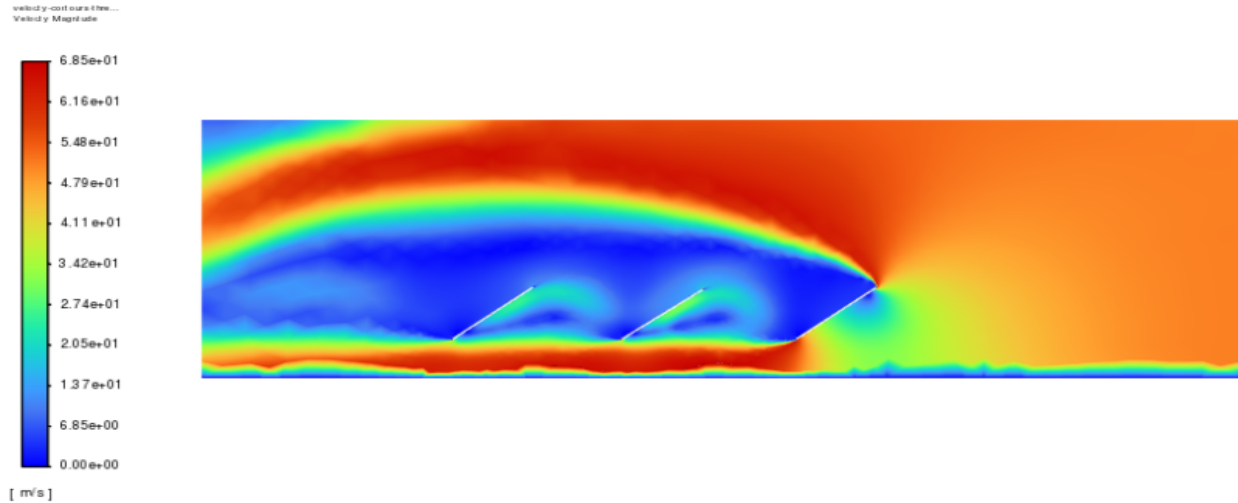
Figure 3: 114 MPH wind on 3 panels with wind coming from the back (right or 180° angle).

Empirical Data Calibration - From Heliostat Model
Our single heliostat model for the wind tunnel experiment had a dimensionless force coefficient in the x-direction of 2.24 and the NASA Ames Laboratory study had 0.69 [2]. Our force coefficient was calculated from the total force reported by the model on the heliostat which was 97.6 N. This was then entered into the drag force equation to solve for the force coefficient (Equation 2 pg.21).

Discussion

Python - NASA Ames Analysis
From the empirical data and the nonlinear regression, the team can see that the angle at which wind hits the PV panel can affect the force of wind on the PV panels. For a panel tilted at 35° to the ground, the wind force is typically the greatest when the panels are rotated around the z-axis (axis perpendicular to the ground) between 180° or 270° and typically the least between 50° and 120° (when 0° is when the face of the panel is directly facing the wind). The angle at which the wind hits the PV panel can make the magnitude of force vary by a factor of 10. This suggests that being able to rotate the PV panels so that the wind hits it from an angle range between 50° to 120° in lieu of 180° to 270° could reduce the force that the PV panels experience as they tilt to track the sun. With the wind force models provided the client can determine whether prioritizing solar exposure or wind impact will be more imperative in the design process.

Ansys Fluent Modeling
When considering the triple panel models, results show a significant reduction in force behind the first panel in the array. For most scenarios this reduction was greatest in the second panel. The third panel also received a significant reduction in force when compared to the first panel. These results imply that the convoy effect can take place behind the panels (See Figure 2-3 pg.7-8).

The greatest shear stress for the panels generally occurs in the top edge of the first panel, when the wind is coming from the front (left or 0° angle), or the bottom edge, when the wind is coming from the back (right or 180° angle). Shear stress generally varies by a factor of six, which means

that, during a large wind event, different edges of the panel may experience a shear stress six times greater than other edges. This implies that extra supports may need to be added to some edges of the panels.

A few interesting phenomena occurred during various modeling scenarios. Firstly, in the 114 MPH wind coming at a 0° angle scenario, the middle panel experiences a net force coming from the opposite direction of the wind. The team hypothesizes this is due to the deflection of the wind from the first and third panel creating a "back current" or circulation of air creating a vortex, this can be seen between the panels in Figure 2 and 3 (pg.7-8).

Secondly, the force on the first panel in the 114 MPH wind coming at a 0° angle scenario was roughly two times greater than the force on a single panel in the 114 MPH wind coming at a 0° angle scenario. As of yet, the team has not come up with a hypothesis for this phenomenon.

Thirdly, another phenomenon that needs to be researched further is the variance between scaled residuals. Scaled residuals (See Definitions pg.19) are a measurement of how well the models run. Convergence of residuals represent ideal conditions. Scaled residuals varied greatly between models. All residuals converged to other residuals within the residual graph, but at different values. One possible area of improvement within the model would be to adjust the timestep (See Definitions pg.20). At our current time step, some cells in the mesh would not have been solved for, which could have contributed to the discrepancy in the residuals.

Empirical Data Calibration - From Heliostat Model
Calibration is a necessary step for receiving accurate values from the ANSYS Fluent models. Raw experimental data from the NASA Ames Laboratory study [2] is a valuable resource that can be used to calibrate and validate our models. Due to time constraints, the engineering team did not get to calibrate our models using this data. Since we did not have time to adjust parameters on our heliostat model to minimize the difference between the modeled force coefficient and the empirical one, our outputs lack validation and could be off by an order of magnitude. It is important to note that it is undetermined if force coefficients can serve as an adequate form of empirical data for ANSYS model calibration.

Conclusion

The Electric Roots engineering team was able to analyze empirical data to learn about wind forces on PV arrays. Ansys Fluent was utilized to create 2D models of a single and triple PV panel RUTE SUNTRACKER design. Wind forces were represented through outputs of total pressure and velocity.

Further modeling work and understanding is needed. The team recommends modeling the panels and the wind area as 3D objects, since this more closely resembles real world conditions. Adding more panels in the modeled array would account for more complex phenomena. Research into the strength of materials that govern the PV panel supports should be conducted and joined with wind force and pressure data to see if the supports can withstand the 114 MPH wind scenario.

An array wide sensor system that, in the event of a large wind storm, each PV panel in the array can be turned to the angle that receives the least amount of wind force could be valuable to decreasing damages from wind forces on the SUNTRACKER system. We recommend further

testing in Ansys such as model validation and calibration of our current Ansys models. Modeling a scaled SUNTRACKER array in a wind tunnel or getting measurements from a test site would also help with calibration of the model, with the potential addition of force and pressure sensors from the field.

The implications of the presence of the convoy effect resulting in a reduction of forces on inner panels could mean a reduction in materials cost. Interior panels experiencing lower wind forces would transfer less of those forces to the support cables and outer pitch poles of the RUTE SUNTRACKER array. The materials for interior panels, support cables, and pitch poles could then be scaled down to reduce manufacturing costs.

Appendices

References

[1]   RUTE Foundations. (n.d.). *Rute SUNTRACKER*. RUTE. Retrieved October 24, 2022, from https://www.rutefoundations.com/suntracker

[2]   NASA Ames Fluid Mechanic Laboratory. (2011). *Heliostat Wind Tunnel experiments*. Retrieved February 3, 2023, from https://www.google.org/pdfs/google_heliostat_wind_tunnel.pdf

[3]   *Solar panel wind load calculation ASCE-7-16*. SkyCiv Cloud Structural Analysis Software | Cloud Structural Analysis Software and Calculators. (2022, November 2). Retrieved February 3, 2023, from https://skyciv.com/docs/tech-notes/loading/solar-panel-wind-load-calculation-asce-7-16/

[4]   *What is a mesh?: SimWiki documentation*. SimScale. (2023, February 17). Retrieved March 17, 2023, from https://www.simscale.com/docs/simwiki/preprocessing/what-is-a-mesh/

[5]   *What is the meaning of time steps in Ansys 13.0? - researchgate*. (n.d.). Retrieved April 13, 2023, from https://www.researchgate.net/post/What_is_the_meaning_of_time_steps_in_Ansys_130

Appendix A: Experimental Research and Setup

Plots
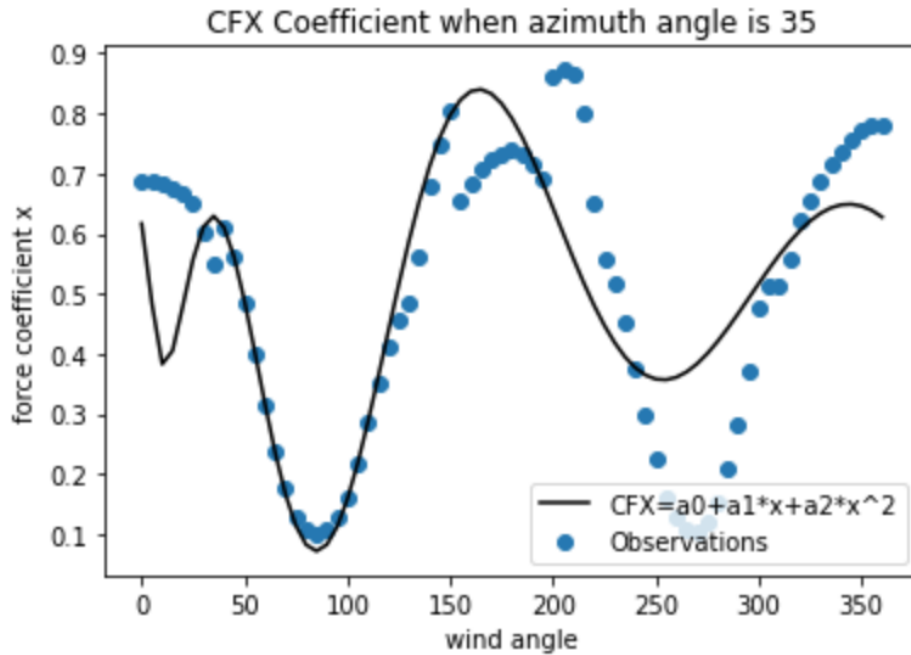
Plot 1: Force coefficient in the x-direction. Observed data points are from the NASA Ames Laboratory study of heliostats [2].
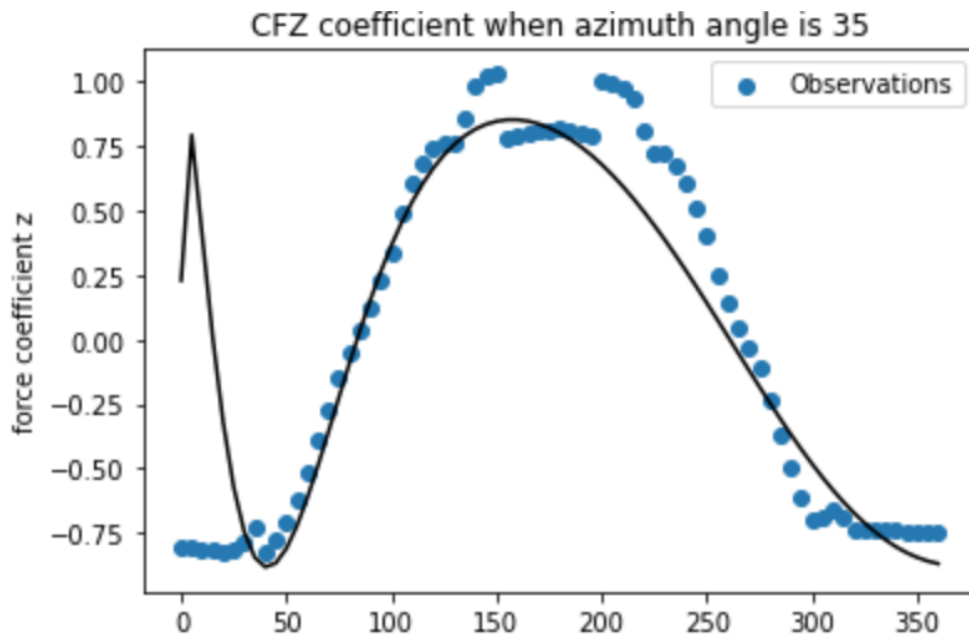
Plot 2: Force coefficient in the z-direction. Observed data points are from the NASA Ames Laboratory study of heliostats [2].
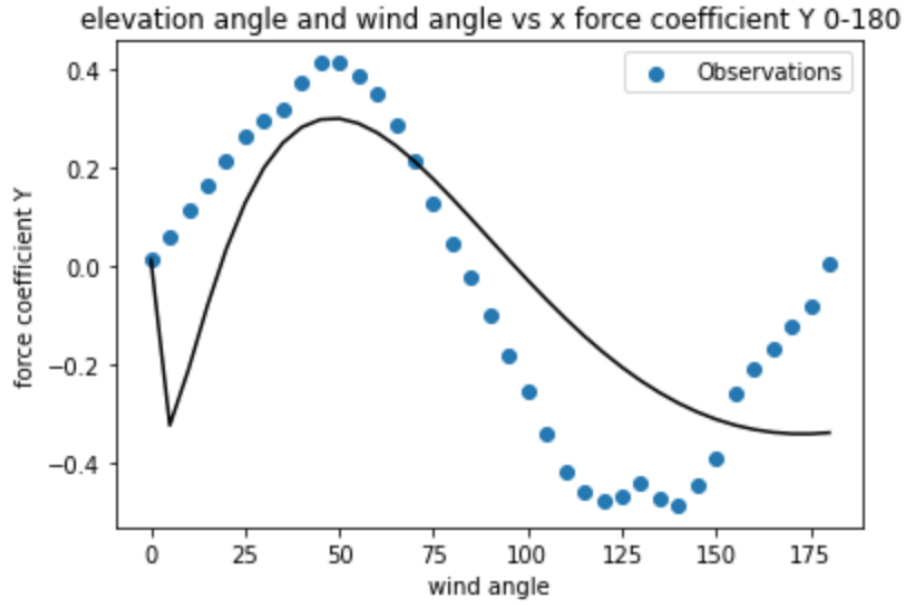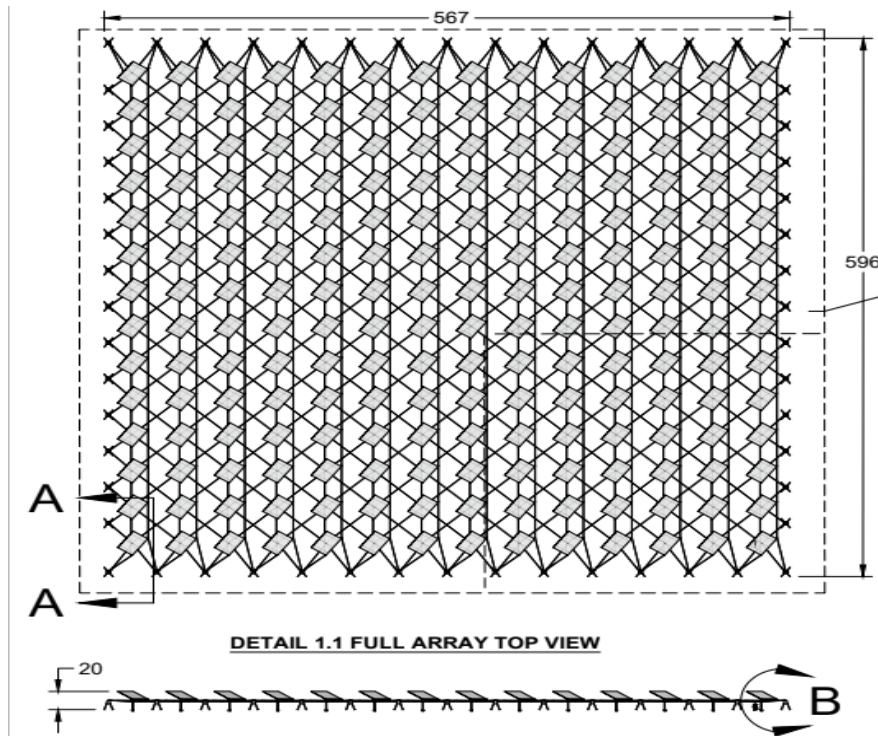
Plot 3: Force coefficient in the y-direction (0-180°). Observed data points are from the NASA Ames Laboratory study of heliostats [2].
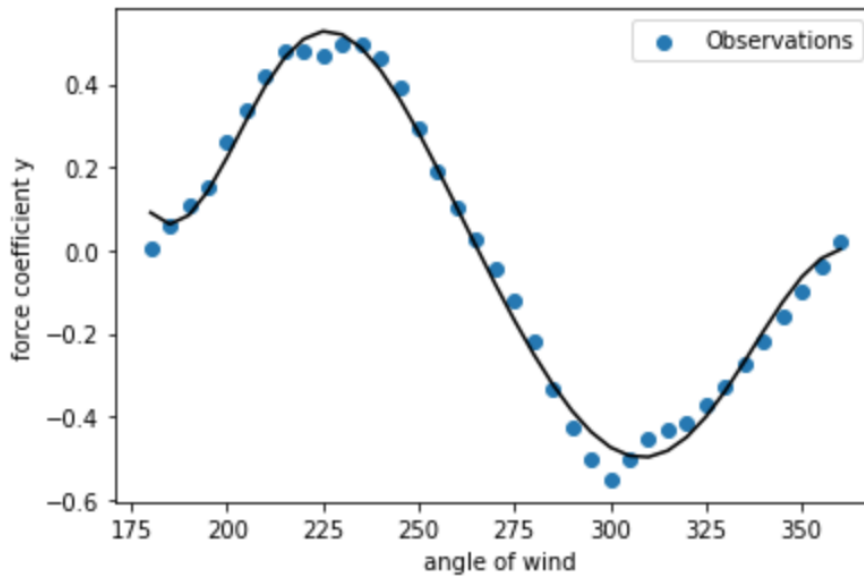
Plot 4: Force coefficient in the y-direction (180-360°). Observed data points are from the NASA Ames Laboratory study of heliostats [2].

Figures

567

596

A

A

DETAIL 1.1 FULL ARRAY TOP VIEW

20

B

Figure 4: RUTE SUNTRACKER array design [1].

CFz,
CMz

CFx,
CMx

CFy,
CMy

Hinge
Axis

Mirror
Centroid

Wind
direction

Coordinate system used, heliostat shown here at α = 45°, β = 45°
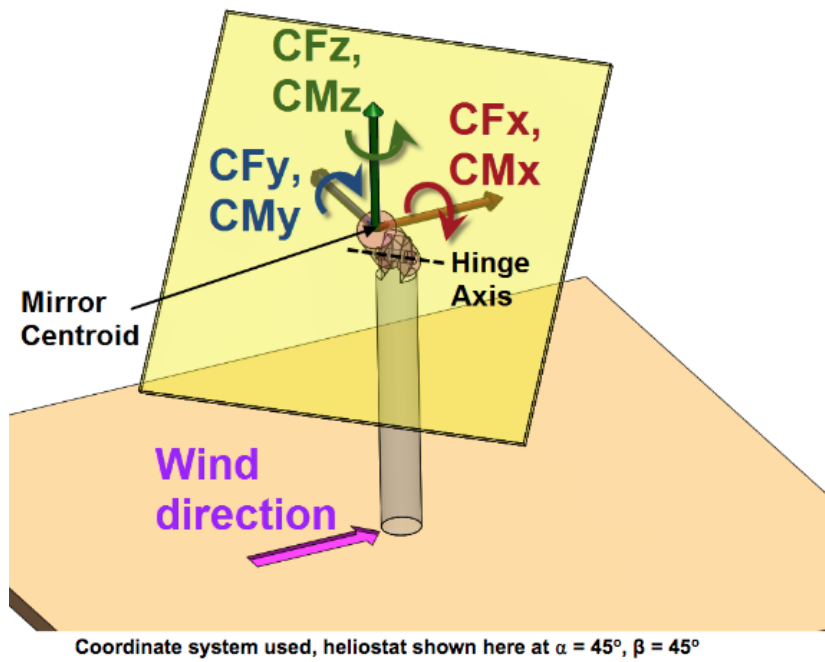
Figure 5: Heliostat coordinate system [2].
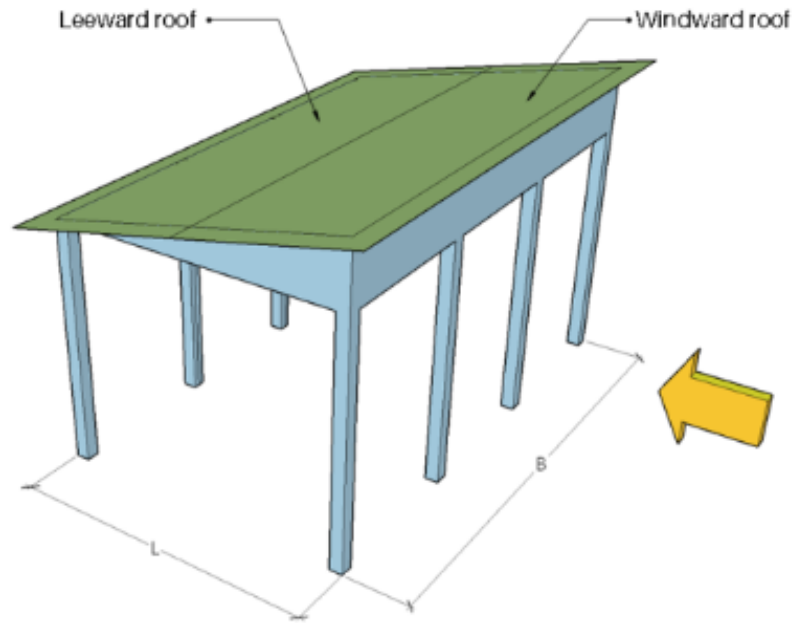
Figure 6: Windward and leeward zones for a direction angle of 0° [3].
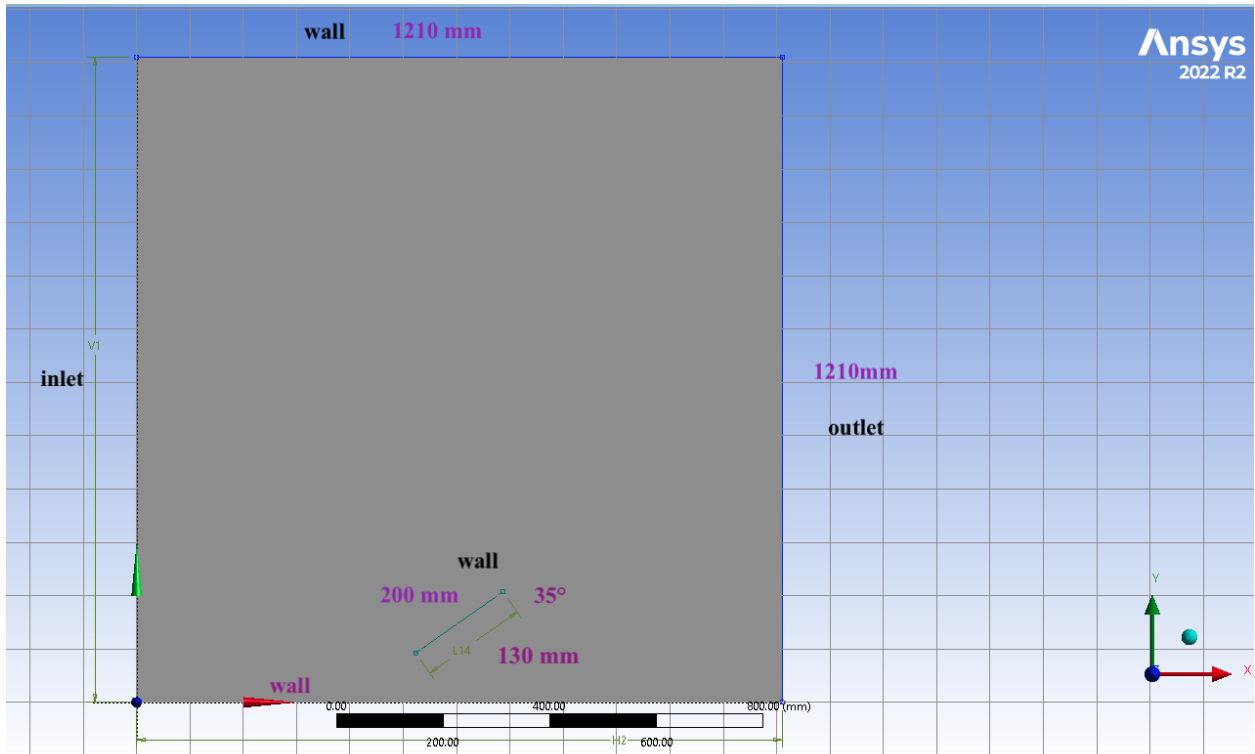
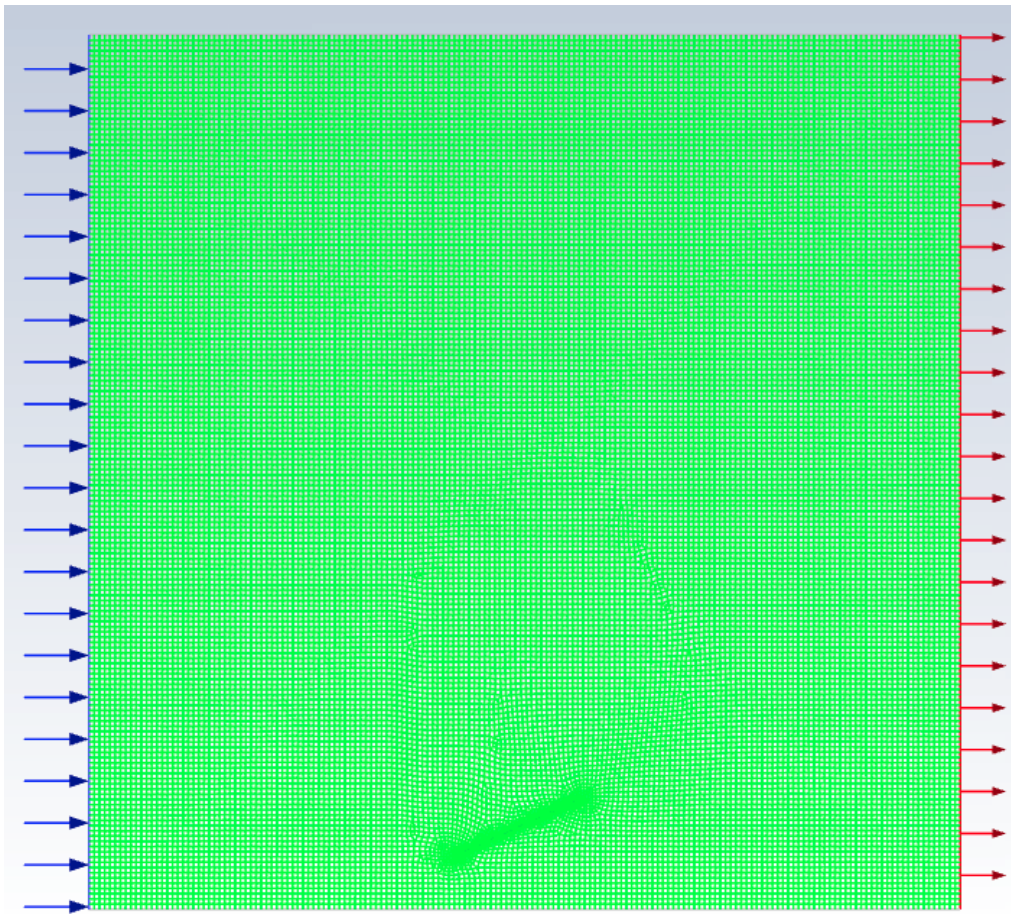Figure 7: Geometry and dimensions of NASA Ames Laboratory wind tunnel single heliostat mesh Ansys model.

Figure 8: NASA Ames wind tunnel single heliostat mesh using the quadrangle method.
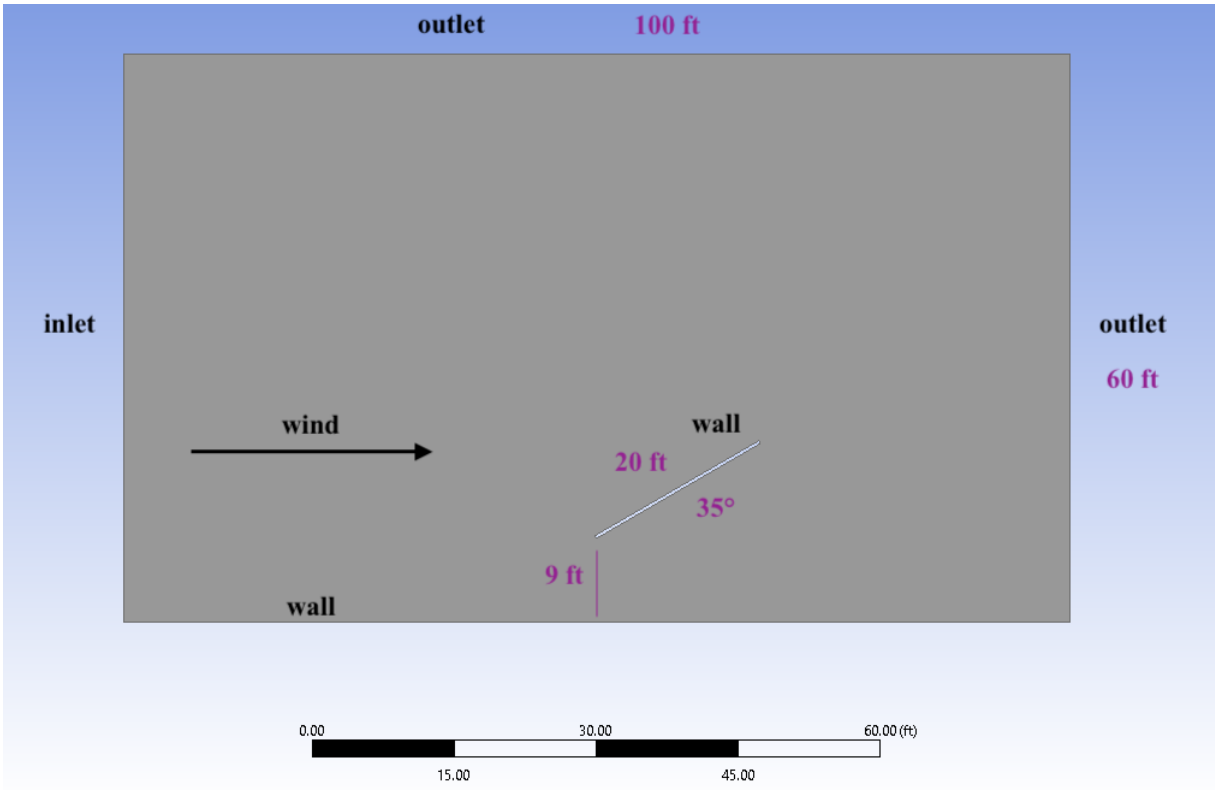
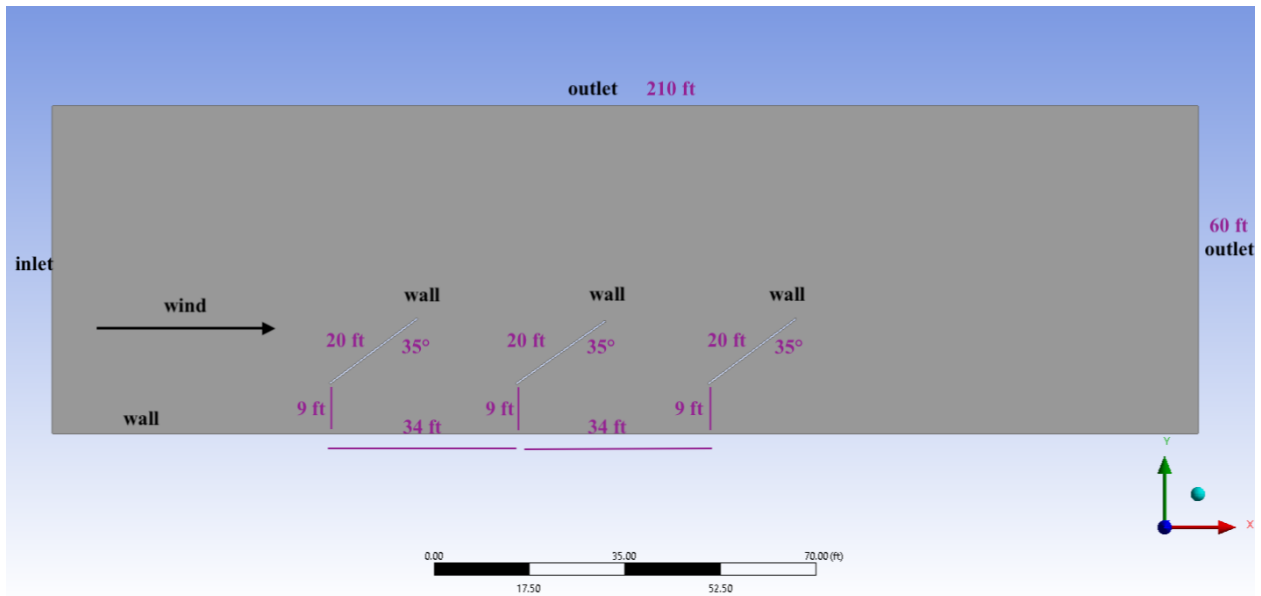Figure 9: Geometry and dimensions of single panel mesh Ansys model.

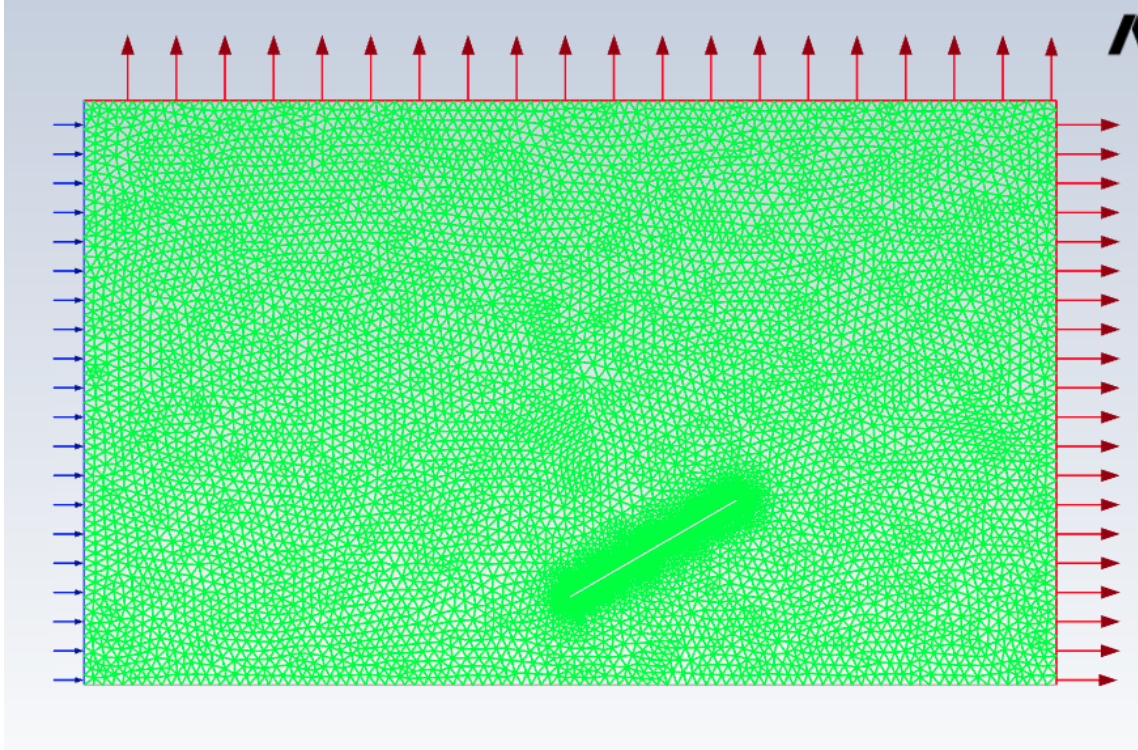Figure 10: Geometry and dimensions of three panel mesh Ansys model.

17

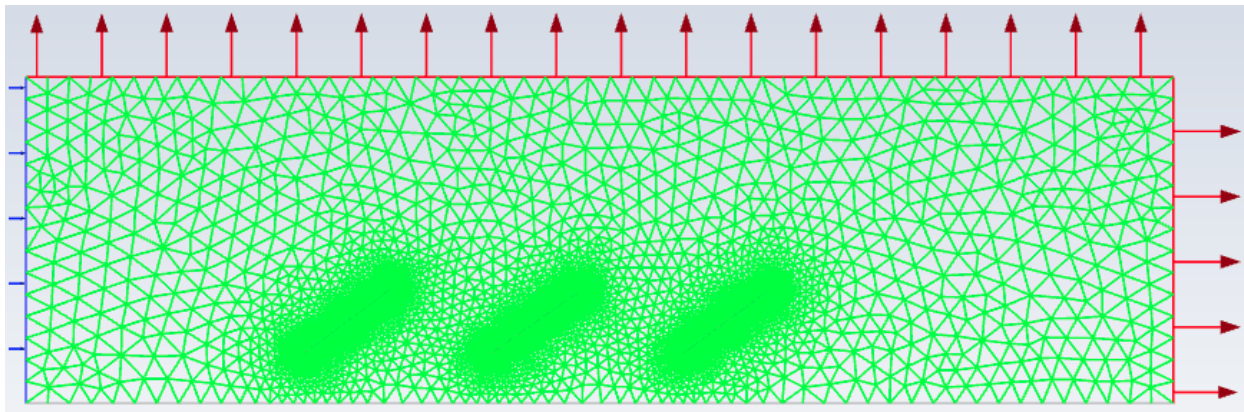Figure 11: Ansys mesh for the single panel RUTE SUNTRACKER.

Figure 12: Ansys mesh for the three panel RUTE SUNTRACKER series.

Appendix B - Details and Definitions

Details

Page 5a: The geometry uses cartesian coordinates, with +x heading right in relation to the computer monitor, and +y heading up in relation to the computer monitor. The first rectangle modeled the area of the wind. It had a length of 100 ft, and a height of 60 ft and started at the origin (0,0). The second rectangle modeled the PV panel. The origin of the rectangle was at (50, 9) and the second point of the rectangle was at (66.4, 20.5).

Page 5b: The origin of the first rectangle was at (50,9) and the second point of the rectangle was at (66.4, 20.5). The second rectangle represented the second panel in the series. It had an origin of (84,9) and a second point of (100.4, 20.5). The third rectangle represented the third panel in the series. It had an origin of (118, 9), and a second point of (134.4, 20.5).

Page 7: The first panel refers to the rectangle at origin (50,9), the second panel refers to the rectangle at origin (84,9), and the third panel refers to the rectangle at origin (118,9). Origin here refers to the lowest edge of the panels.

Definitions

Control Volume: The model space that contains the area of interest. In this case, it contains the area that wind can move through and the PV panel/panels.

Convoy Effect: A decrease in pressure forces behind a solid object that deflects wind forces in front of it. When multiple objects are placed in rows, the first in the line up would take the most pressure force and would decrease the pressure forces to the objects behind.

Elements: Elements are the cells within a mesh. If a mesh has more elements, it will produce more accurate results, but take longer to solve. Each element is solved separately, and the combined results of the elements produce the solution.

Force Coefficients: A dimensionless measure of force that describes the drag/resistance of an object in a fluid (in this case air). Represented by CFx, CFy, and CFz for the different directions x, y, and z (See Equation 2 and Figure 3).

Geometry: Geometry, in Ansys, is the shape of the systems that are being analyzed. It can be either a fluid or a solid.

Inflation: Inflation is the process of adding more elements to areas of the mesh that need more accurate results. If there is a greater inflation rate, more elements will be added to the area of the mesh where inflation is applied.

Mesh: A mesh is a "network formed of cells and points [nodes]" [4] that is used for finite element analysis simulations [4]. Meshing is generating a grid of these cells (elements) and nodes to solve the simulations that are being modeled [4].

Residuals: A residual, in a CFD, is synonymous with the imbalance of an equation. Since Fluent uses a finite pressure and force equation, the left side of the equation (the inputs) and the right side of the equation (the outputs) need to equal each other. The residuals are the differences between the left and right side of the equation. Therefore, a residual needs to converge to zero (or

close to zero) to be a valid model, since that would mean that energy is conserved. The lower the residual, the more accurate the model.

Time Step: A Time Step is " the incremental change in time for which the governing equations are being solved" [5]. Essentially, it is how frequently a time based model is solved. The team's simulation was time dependent since it utilized a velocity inlet for the inlet wall, therefore a time step needed to be set. Each simulation had a starting length and an ending length, and a constant velocity at which the wind flowed, therefore it was time dependent. Typically time steps are set using the equation of cell size divided by the velocity. This ensures that the simulation does not skip on cells in the mesh.

Appendix C - Equations

Equation 1: Wind Pressure Equation (Solar panel wind load calculation ASCE 7-16) [3].

In calculating wind load on solar panels, we will be using the ASCE 7-16 Chapter 27 – Wind Load – Directional Procedure. We will consider the ground-mounted solar panel as an open building with monoslope roof when the tilt angle is less than or equal to 45° and as a solid sign for tilt angle greater than 45°.

The formula in determining the design wind pressures are as follows:

For tilt angle ≤ 45° (considered as open building with monoslope roof):

$$p = q_h G C_N \text{ (1)}$$

For tilt angle > 45° (considered as solid sign):

$$p = q_h G C_f \text{ (2)}$$

Where:

G = gust effect factor
C_N = net force coefficient for open monoslope roof
C_f = net force coefficient for solid signs
q_h = velocity pressure at reference height, h, in psf, given by the formula:

$$q_h = 0.00256 K_z K_{zt} K_d K_e V^2 \text{ (3)}$$

K_z = velocity pressure coefficient
K_{zt} = topographic factor
K_d = wind directionality factor
K_e = ground elevation factor
V = basic wind speed in mph

Equation 2 : Equations from the NASA Ames Laboratory study that are used to calculate force and moment coefficients. EQN 1 is the drag force equation [2].

$$F = C_f * \frac{1}{2} * \rho * V^2 * A \qquad \text{(EQN 1)}$$

$$M = C_m * \frac{1}{2} * \rho * V^2 * A * L_{ref} \qquad \text{(EQN 2)}$$

Where:
- $F \ (or \ M) =$ Force [N] or Moment [N-m]
- $C_f \ (or \ C_m) =$ Force or Moment coefficient (respectively)
- $\rho =$ Air density at given air temperature [kg/m₃]
- $V =$ Air velocity (wind) [m/s]
- $A =$ Frontal Area [m₂]
- $L_{ref} =$ Reference Length [m]
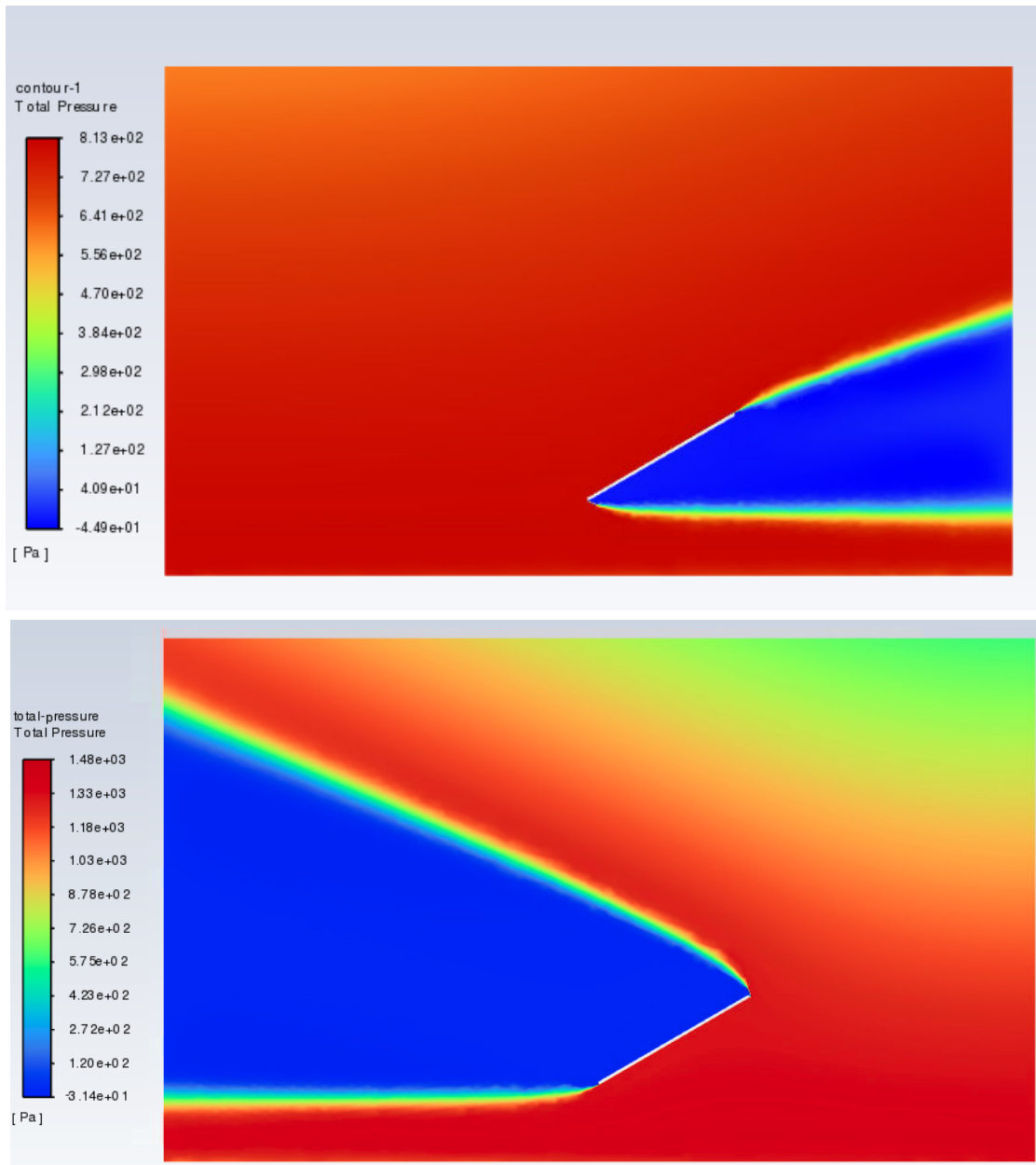
Appendix D: Results

Figures

Figure 13: 70 MPH wind on single panel. The upper image has the wind coming from the front (left or 0°) and the lower image has the wind coming from the back (right or 180°).
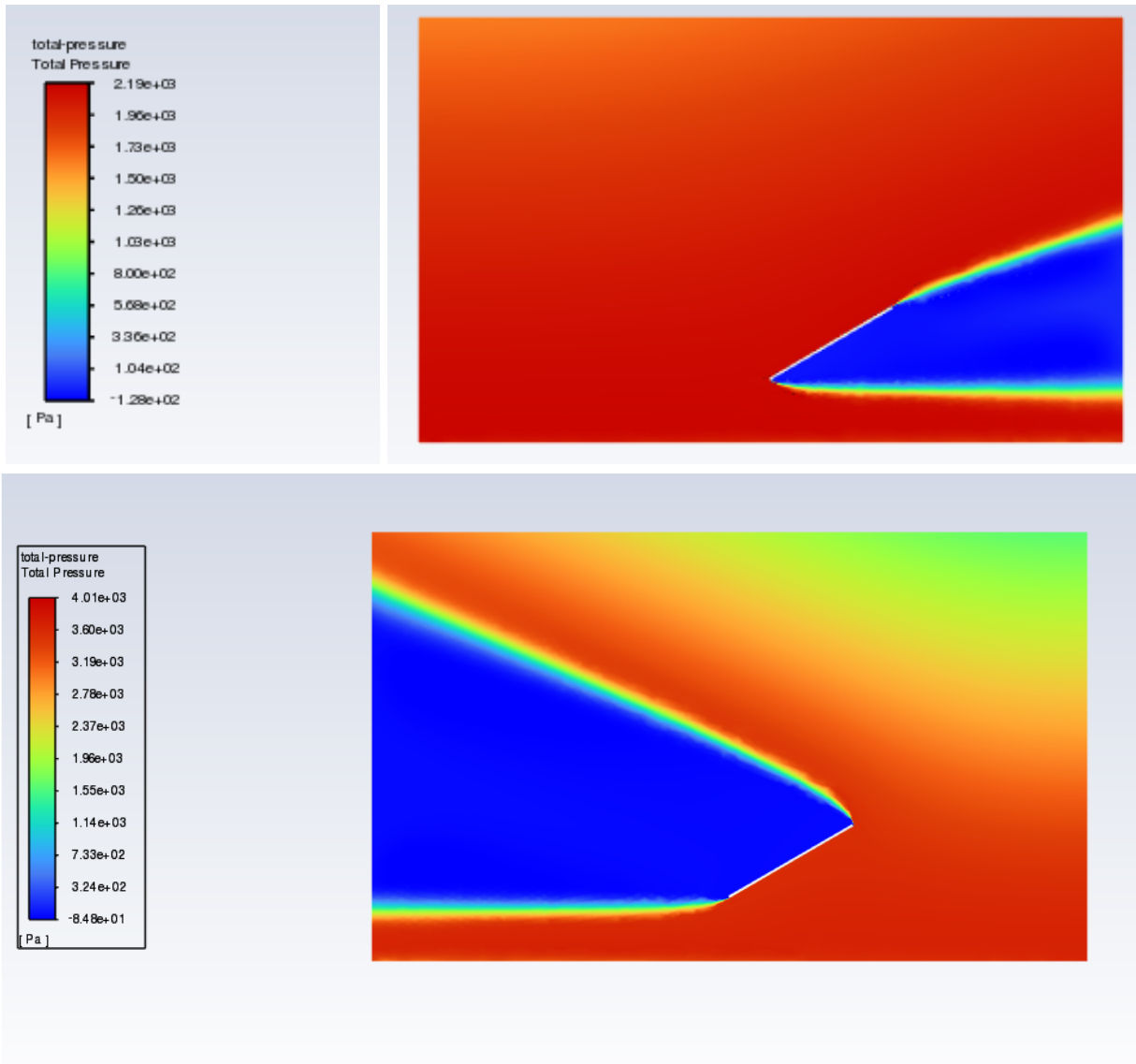
Figure 14: 114 MPH wind on single panel. The upper image has the wind coming from the front (left or 0°) and the lower image has the wind coming from the back (right or 180°).
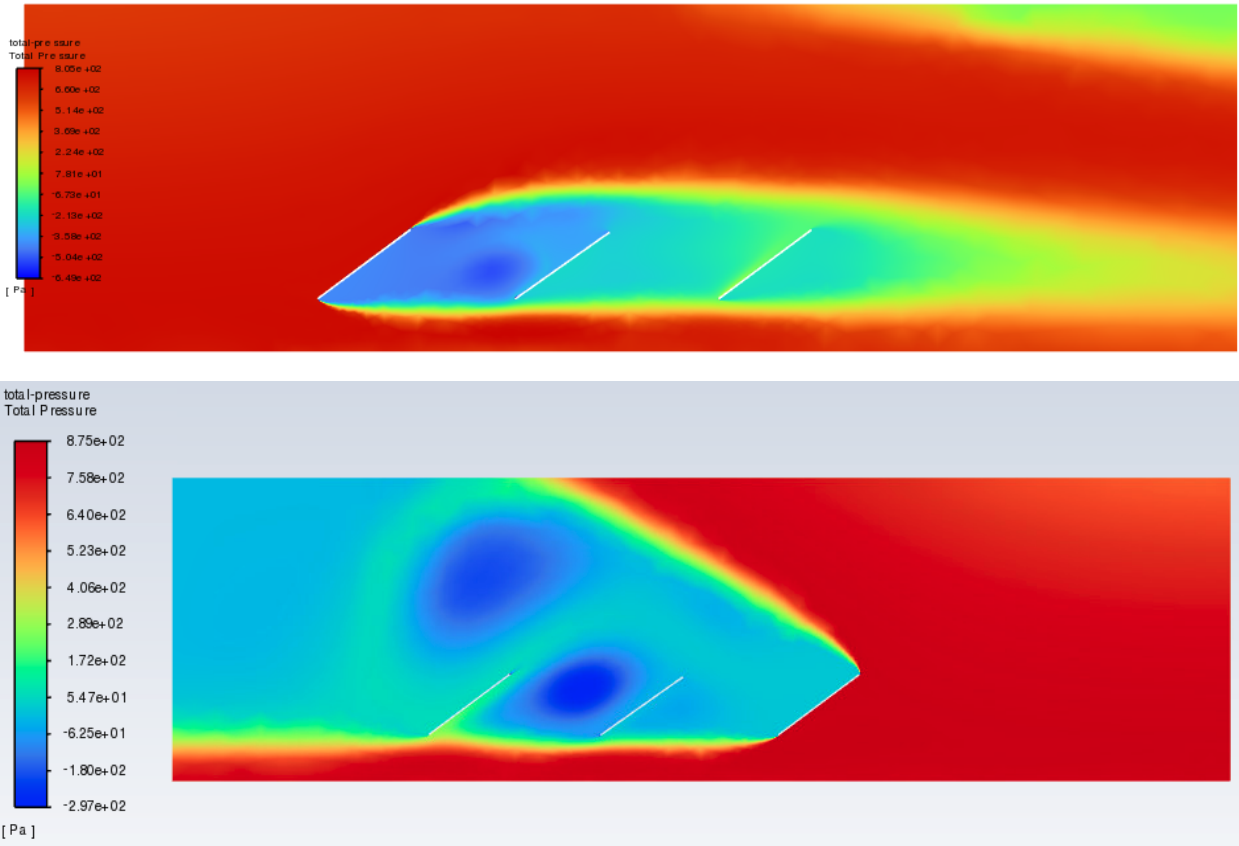
Figure 15: 70 MPH wind on the 3 panel series. The upper image has the wind coming from the front (left or 0°) and the lower image has the wind coming from the back (right or 180°).

velocity-contours-thre...
Velocity Magnitude

7.21e+01
6.49e+01
5.77e+01
5.05e+01
4.33e+01
3.61e+01
2.89e+01
2.16e+01
1.44e+01
7.21e+00
0.00e+00
[m/s]

velocity-contours-thre...
Velocity Magnitude

6.85e+01
6.16e+01
5.48e+01
4.79e+01
4.11e+01
3.42e+01
2.74e+01
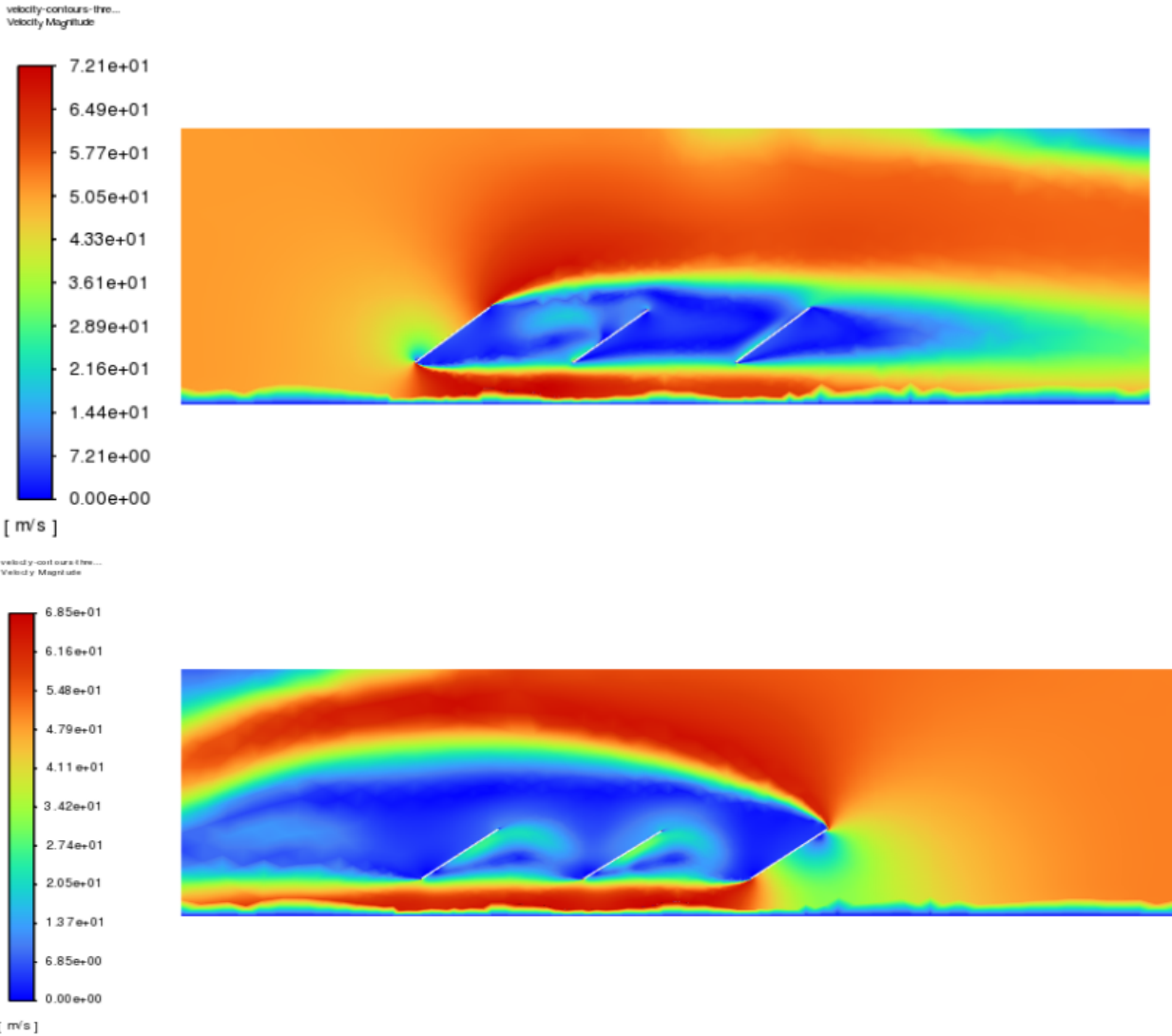2.05e+01
1.37e+01
6.85e+00
0.00e+00
[m/s]

Figure 16: 114 MPH wind on the 3 panel series. The upper image has the wind coming from the front (left or 0°) and the lower image has the wind coming from the back (right or 180°).

Tables

Table 5: Force values for 114 MPH 3 panel wind scenario*

Panel	Force (0°)	Force (180°)
1st Panel	7020 Newtons	-535 Newtons
Second Panel	-1705 Newtons	-189 Newtons
Third Panel	1305 Newtons	-8118 Newtons

*These results were created from a model that was not calibrated or validated therefore we can not say with confidence that these findings are significant or accurate.