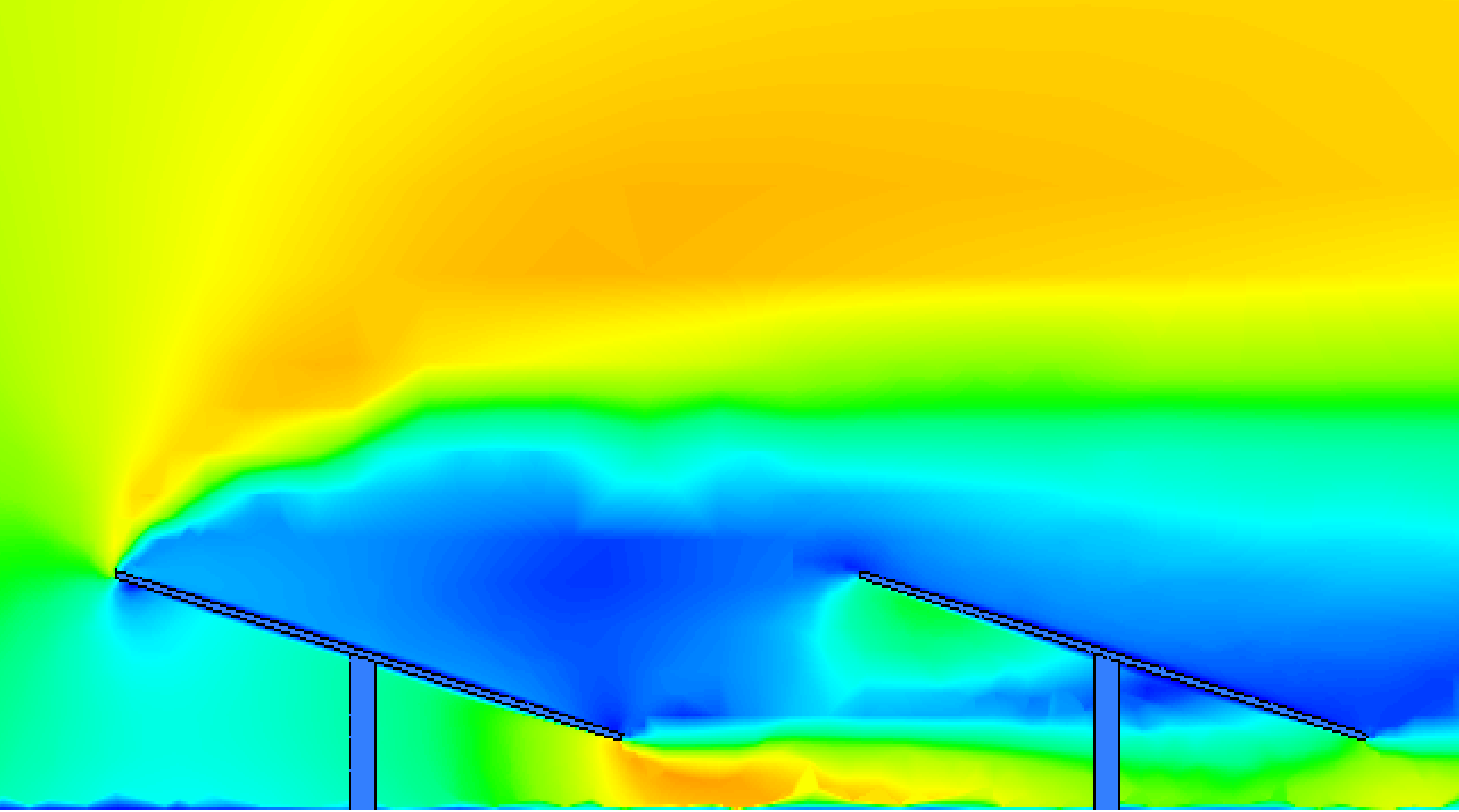
Optimization of Solar PV Module Mounting Structures using Computational Fluid Dynamics (CFD)

Rahul Chande

Niraj Mehta

Saddiq Peerzada



1 Abstract

The study aims to analyse the feasibility & concept of internal and external PV Solar Module Mounting Structures based on the wind effect using **computational fluid dynamic (CFD) analysis**.

With reducing energy tariffs in solar, structural design & optimization of PV solar Module Mounting Structures (MMS) is gaining more attention. Following traditional analysis methods and codal provisions, optimizing MMS has always been challenging, however, this can be achieved using advanced engineering tools like Finite Element Analysis (FEA).

Overall cost of MMS for any large utility-scale solar PV plant can be optimized for cost, durability and sustainability by providing project-specific design coefficients essentially including local wind, terrain and other geographical conditions. Our past experience following this approach has adequately optimized the structures leading to the concept of exterior, interior and far interior structures for solar PV module mounting.

The concept of the exterior-interior table is based on the reduction in wind-induced pressure observed in the interior/far interior table rows due to the shielding effects from exterior table rows. CFD solves equations for the conservation of mass, momentum, and energy and provides detailed information on the fluid flow behaviour, distribution of pressure, velocity, temperature, etc. Ansys CFD tool is used to analyse multiple rows of tables, simulate wind pressure reduction on interior MMS rows, and identify the shielding effect due to exterior MMS tables.

With its extensive experience evaluating over 66GWs of MMS reviews, SgurrEnergy has established a tried-and-true methodology for design optimization using computational fluid dynamics (CFD), finite element analysis (FEA), and fatigue analysis, leading to high confidence in the structural reliability over the project life.

2 Introduction

Following the traditional design approach of building and other structural codes, for MMS, may not produce solutions that are optimally representational of the project location, geographical and climatic conditions. This could weaken the purpose of design optimization, leading to higher project cost or the possibility of a durability risk during the course of the project.

SgurrEnergy has experienced traditional industry design tools have not been consistent over the period leading to huge uncertainties being observed in various updates and software releases. With such updates and changes in the tool on which the MMS is designed, the durability risk further increases. Given the fact that, there are no specific codes for designing solar PV mounting structure, simulating the local climatic conditions becomes an essence for optimizing the MMS.

This technical article aims to understand the behavior of wind on structure and to find out the effect of wind load on structural tables leading to a design which is expected to be a cost effective and a durable approach. Study examines the wind effects on structure by performing CFD simulation using industry leading FEA tool. Simulations are performed for eight wind directions, right through North, North-East, East and clockwise for every 45 degree. The study assumes a table with two rows of 28 modules in each; typical PV module dimension used is 2279mm x 1134mm orientated in portrait. The structure assumed is a 15° fixed tilt structure facing south.

Figure 2-1: Typical PV Solar MMS

3 Why Computational Fluid Dynamics (CFD)?

Effect of wind on structures can be assessed either by a wind tunnel test or simulated using computational fluid dynamics. While both have their respective pros and cons, it may be important to know that wind-tunnel test essentially requires a scaled down model for testing; this is understood to be a complex and an expensive approach. It is understood that, the probes used in wind tunnel test disrupts the wind flow. Hence, it is difficult to capture the precise behavior of the structure. Whereas, CFD can run full scale model and since there are no physical probes, precise wind evaluation can be done to understand the accurate wind flow pattern over the PV Modules.

In addition, CFD reduces time and cost required to perform the identical study by using advanced techniques and simulations; this also has flexibility to assess multiple permutations and combinations leading to an effective optimization.

CFD provides added insights into the design process with high resolution and “mapping” of results visually, which is highly informative.

4 CFD Flow chart

Given the fact that the technical article focuses only on CFD for design optimization, Figure 4-1 essentially presents a high level flow process being used for design. Although there are multiple established tools for CFD simulations in the industry, the article typically uses Ansys Fluent for the study.

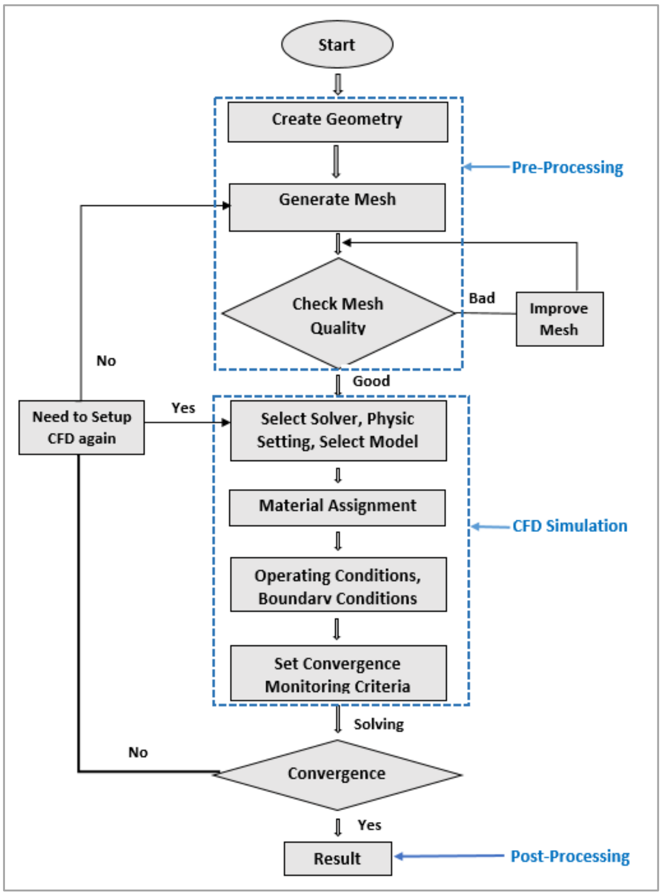


Figure 4-1: CFD Flowchart

5 Pre-Processing

Pre-processing is an initial aspect of Ansys fluent work flow. This involves the initial setup of the model which mainly includes geometry creation, meshing, and setting up the CFD domain.

5.1 Assumptions and simplifications

Following are some assumptions and simplifications made for CFD analysis:

a. Small geometrical details such as gaps, member flanges, supports, etc. are neglected. However, the wind pressure from CFD can be mapped in the mechanical tool and the structural design of precise geometry can be captured in mechanical analysis.

b. The CFD is done considering a simplified negative volume in the Fluent domain.

c. CFD is performed only till ground level.

d. Area topography and ground undulations are not modelled in the CFD domain.

5.2 Modelling and Meshing

A Three Dimensional (3D) CAD model of MMS for a tilt of 15° is created using the Ansys SpaceClaim tool. The MMS tables are arranged in multiple rows as shown in Figure 5-1. Ansys fluent Mosaic technique was used for meshing.

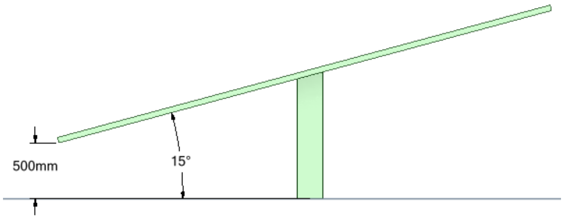


Figure 5-1: Single table side view

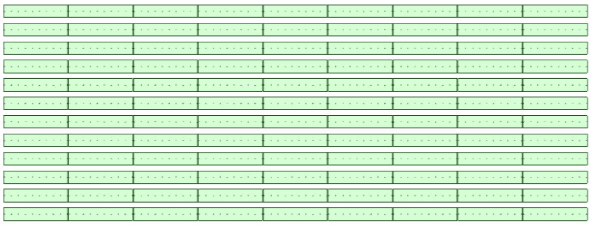


Figure 5-2: PV table array

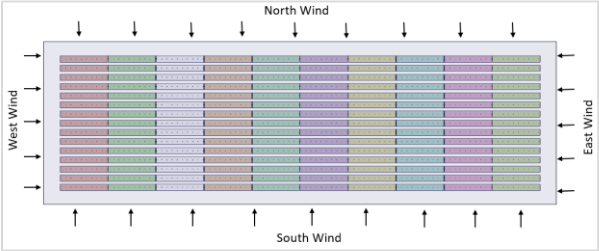


Figure 5-3: Model for wind analysis

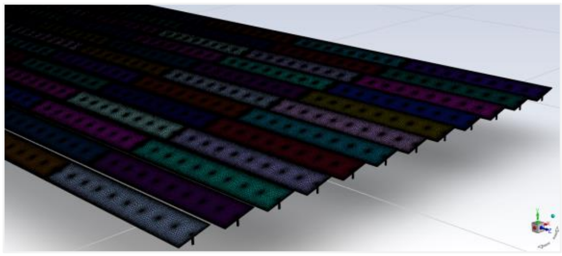


Figure 5-4: Meshed Model

Poly-Hexcore meshing is used to fill the bulk region with octree hexes while retaining the high quality layered poly-prism grid in the boundary layer. Face sizing control is applied in order to improve the mesh quality. Minimum and maximum cell lengths of 50 mm and 6400 mm respectively are applied. Fine mesh is generated near the area of interest and course mesh is formed away from it without compromising the solution accuracy. Inflation layers are added on fluid-structure interaction (FSI) surfaces to capture the boundary layer effect accurately.

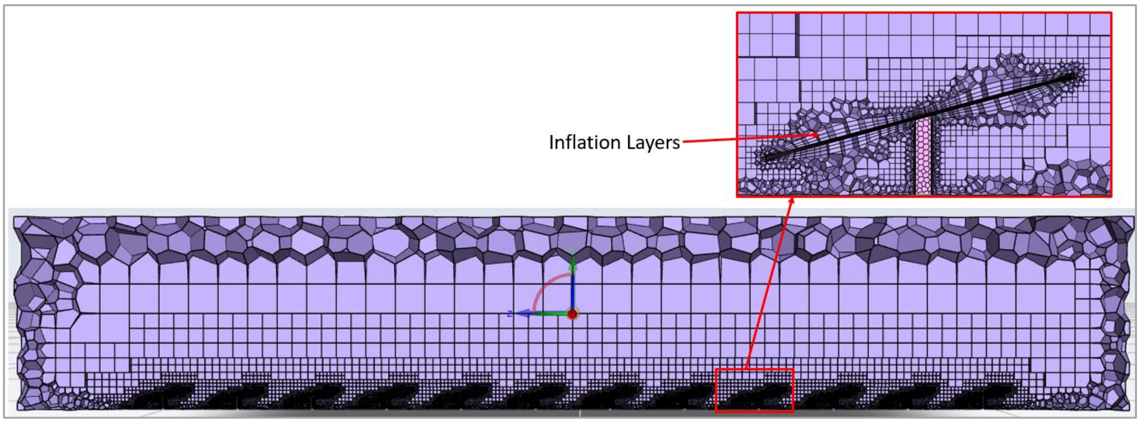


Figure 5-5: Meshed Model Cross-section

6 CFD Analysis

The current CFD simulation adopts Shear Stress Transport (SST) K- omega (k–ω) turbulence model. As flow is turbulent, the k–ω turbulence model is well suited for simulating the flow in the viscous sub-layer. Also, the SST model exhibits less sensitivity to free stream conditions (flow outside the boundary layer) than many other turbulence models. Air was selected as a fluid material for CFD analysis. Air velocity at inlet 47 m/s was applied at atmospheric conditions of pressure and temperature. With all these input parameters, the discretized conservation equations were solved iteratively until convergence.

Table 6-1: CFD Details

CFD Inputs and Boundary Conditions	
Parameters	**Value / Specifications**
Domain physics	Cell Type
Solver Type	Pressure Based
CFD Model	Viscus - SST k-omega
Fluid Material	Air
Boundary conditions at the inlet	Air Velocity = 47 m/s
Boundary conditions at the outlet	Pressure outlet
Boundary conditions at FSI	Wall
Gravitational acceleration	9.81 m/s^2
Operating pressure	101325 Pa (ATM)
Turbulent Intensity	5 %
Operating Temperature	300 K

7 Results and Discussion

Wind flow from eight directions are considered for the study and the results are illustrated in the below table.

Table 7-1: Number of analyses for different eight AOA

Analysis No.	Angle of wind attack	Wind Directions	Tilt angle	Velocity Input (m/s^2)
1	0°	North	15°	47
2	45°	North-East	15°	47
3	90°	East	15°	47
4	135°	South-East	15°	47
5	180°	South	15°	47
6	225°	South-West	15°	47
7	270°	West	15°	47
8	315°	North-West	15°	47

CFD post-processing is conducted to extract the results. Velocity streamlines are lines that smoothly connects velocity vectors at an instance in time and are plotted on sectional planes over the MMS tables. Pressure contour is extracted to know the pressure variation along the rows of the tables.

7.1 Velocity Distribution

Velocity streamlines are extracted for eight wind directions. These streamlines help to understand the behavior of air over the structures.

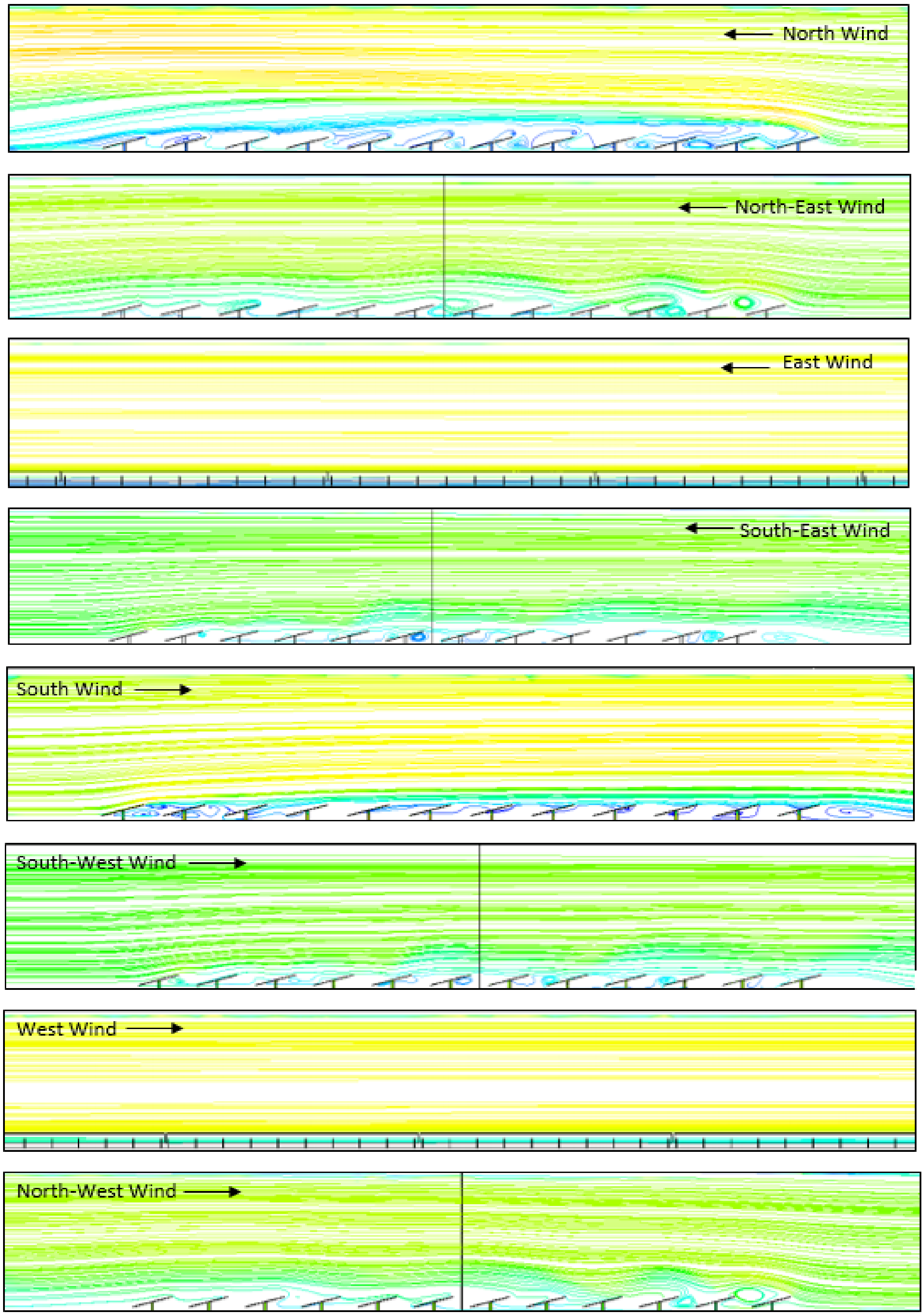


Figure 7-1: Velocity streamlines from eight wind directions

Velocity streamlines flow patterns for eight wind directions are shown in Figure 7-1. Streamline patterns for North and South wind directions change as the MMS tables obstruct airflow and induce turbulence near and behind the tables. As air flows parallel along the length of tables in the east and west wind directions, streamline flow patterns are almost parallel throughout the structure.

7.2 Pressure Distribution

The pressure plot on the structure shows the generated pressure due to applied wind load. These pressure plots help understand the variation of pressure at the FSI (fluid-structure interaction) surface. Also, it shows the change in pressure if there is a change in wind direction.

Figure 7-2, shows the pressure plots due to eight wind directions, right through North, North-East, East and clockwise for every 45 degree wind flow, the flat table surface is normal to the direction of wind flow, generates higher pressure as compared to East and West wind direction. The first few tables on the windward side are the most affecting tables as they show more pressure values than the rear tables. Therefore, by considering all eight wind directions, external tables exposed to wind are the most critical tables for wind load.

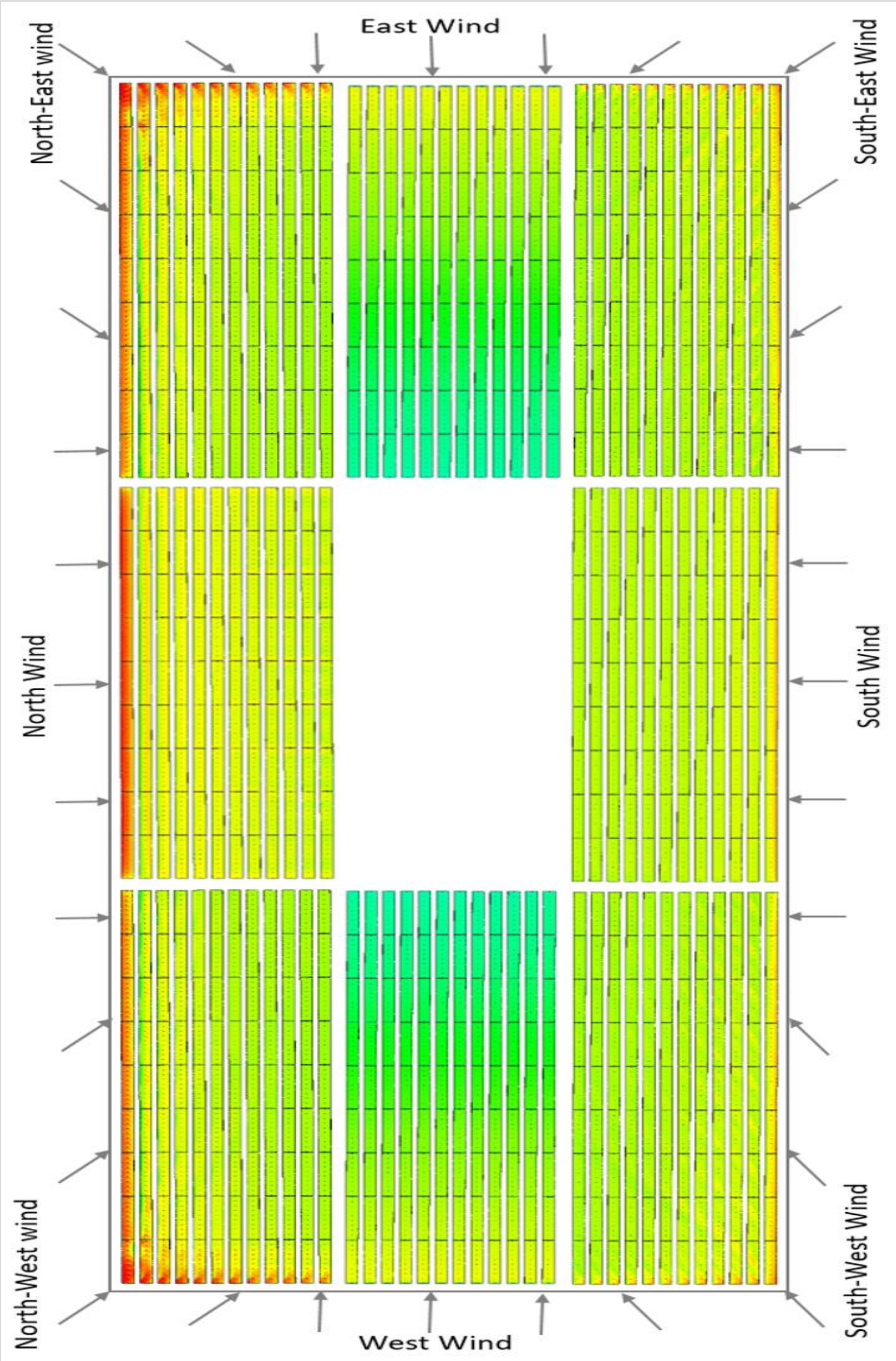


Figure 7-2: Pressure contour for eight wind direction

7.3 Concept of Exterior and Interior tables

The Exterior-Interior table concept is based on the reduction in wind effects observed in the internal table rows due to the shielding effects of the external table rows. As the steady wind flow passes over the first few external tables, the wind gets diverted and wind flow gets disturbed leading to dissipation of its energy causing a reduction in wind effects on the subsequent internal rows of tables. Thus, wind loads on internal tables are lesser as compared to external tables indicating scope of optimization in the overall tonnage for MMS for internal tables.

For this concept, the initial wind is assumed to have a steady laminar flow before it acts on the first external row. The assumption of laminar steady flow is applicable for ideal cases where the swirling wind is not formed. This type of wind flow is valid until the formation of swirling winds.

7.4 Limitations of Modelling Swirling Wind in CFD

One of the key downside of performing simulation or wind tunnel test is their inability to simulate swirling wind phenomena, that is occasionally witness on ground.

Swirling winds are the conical-shaped tornadoes formed during cyclonic events and usually travel along with the cyclone path as inverted-cone-shaped swirls. They may have damaging effects on the structures along their path. The occurrence of the swirling wind along with the damage caused is unpredictable. Similar to wind tunnel tests wherein the swirling winds are not captured, the present CFD study is also performed considering laminar wind flow. However, the requirement of studying turbulent wind effects is captured, which are caused by the obstructions created due to the structures along the wind flow path. Thus, the swirling wind is not given as an input in the present CFD analysis.

It is difficult to capture the effects of swirling wind while designing MMS. Furthermore, the current CFD evaluation, similar to wind tunnel study, also does not capture this effect. However, based on the results of the CFD analysis, a CAPEX optimization can be performed specific to the project. The risk associated with possible generation loss due to the damage caused by swirling wind phenomena would be project specific; this can be adequately captured within a contingency in the cost of structure design.

Assuming that the probability of swirling winds along with maximum gusts is low, and these phenomena can be considered a force-majeure event.

8 Conclusion

Computational fluid dynamic (CFD) analysis is performed for studying the airflow over the solar PV module mounting structure (MMS). The analysis is performed for eight wind directions, right through North, North-East, East and clockwise for every 45 degrees. Simulations are performed to extract the results for air velocity streamlines and pressure distributions. The study concludes with the observations enlisted below:

- The concept of exterior, interior and far interior tables may be suitable for any project; however, this would essentially depend on project specifics like location, wind prevailing direction, land boundary, orientation & terrain.

- Wind pressure weakens upon exposure to peripheral structure casting a wind shielding effect on the subsequent structures. CFD simulations essentially provides precise pressure values on each structural rows for the mechanical analysis of MMS.
- The industry currently adopts a random approach for designing exterior and interior structures with no sufficient rationale. CFD analysis aids in providing adequate justification for the variation in wind pressure on exterior and interior tables. This in turn helps in optimizing the overall MMS costing with proper rationale.
- Given that the current technical article primarily focuses on south-facing PV plant as a case study, tables get affected by mostly South and North wind flow as flat surfaces of tables are facing the wind flow direction. As West wind and East wind flow parallel throughout the length of tables, these two wind directions have a lesser impact. However, his would change over prevailing wind direction.
- This approach of computational fluid dynamics (CFD) may eventually lead to overall project cost optimization and increases the investors' confidence on the project bankability.

CFD simulation performed within this technical article is exhaustive and assumes eight wind directions. Independent study would essentially be required to be performed for every project as per site-specific requirements and layout. CFD is understood to be an instrumental approach in optimizing the overall tonnage of MMS and can be utilized for site and project-specific conditions.

9 SgurrEnergy Approach

SgurrEnergy has extensive experience optimising module mounting structures for large-scale solar PV projects. The CFD expert at SgurrEnergy is actively engaged in ongoing research to accurately estimate wind loads by simulating project-specific conditions. The CFD results can further be used in mechanical finite element analysis (FEA) for the optimized design of structures.

By applying CFD and mechanical FEA to optimise the module mounting structure, there has been a significant value addition to lower the overall cost in recent projects. The analytical team supports the use of FEA in the design of mounting structures and also provides comprehensive services to clients related to its methodology and application to achieve optimization in solar projects.