CFD Analysis of Wind Loading In Solar Panels

Paturi Lakshmi Prasanna Kumar[1], K. Gowri Sankar[2]
[1]Dept Of Mechanical Engineering
[2]Assistant Professor, Dept Of Mechanical Engineering
[1, 2] DJR College of Engineering and Technology , Vijayawada , A.P

Abstract- *Solar energy is the area of renewable energy where lot of research is being carried out. In this paper an effort is made to explore the research in the field of solar energy at Kalaburagi. Karnataka. Effect of the wind on the solar panel is studied using the numerical tool ANSYS FLUENT.The solar panel array consisting of 12 solar panels with each 1.2m×0.6 m is placed with 3 rows and 4 columns, with panel inclination of 18 degrees, the analysis has been done for 0, 30, 45, 60, 135 and 180-degree angle of attack at both average and extreme wind velocities i.e., 5 and 25 m/s. The biggest suction pressure occurred at 0, 30 and 180degree in both 5 and 25 m/s. The maximum pressure in the case of 30-degree inclination is observed to increase by 30% compared to the 0degree angle of attack in both the cases. Results can be used to design the structural support for the panels and to prevent them from damage in the adverse wind loads and also to save material at the manufacturing level.*

Keywords*-* Solar energy, wind direction, wind speed, tilt angle, ANSYS

I. INTRODUCTION

To solve the ongoing energy crisis, lot of research has been carried out at the global and national level.Oil production is expected to decline, as the old sources are depleting fast and new discoveries are very slow. The Paris-based International Energy Agency estimates that the world would need to add the equivalent of six new Saudi Arabias by 2030 in order to meet declining production and growing demand [1]. Top universities of world and India are focusing on various sources for energy and also toincrease the efficiencies of the existing systems. Solar and wind are two such promising energy sources [2] and on which an extensive research is carried out. Conducting experimental research studies by the use of wind tunnel tests on ground mounted solar panels is a difficult task mostly due to physical limitations of research laboratories.

Instead of carrying out wind tunnel tests, it has also become popular to use CFD analysis tools to realize the solution of engineering problems. In this paper, ANSYS FLUENT [3] is used in order to model the effects of wind forces acting on solar panels. The solar panel array consists of 12 solar panels with each 1.2 m×0.6 m and placed with 3 rows and 4 columns. The thickness of the panels is 7 mm. In order to perform CFD analysis, a volume of fluid that will be air in this case is needed to be specified. This volume will provide the domain where the fluid will flow.

II. LITRATURE REVIEW

Georgeta et al.[4] studied the wind load effects on the solar panel. A solar array consisting of 12 panels is taken, angle of inclination is given to be 30 degrees as per the location of the Romania. Analysis is done using ANSYS CFX code. Angle of attack of wind is changed from 0 to 180 degrees and accordingly the effect i.e. the pressure contours and the velocity contours is drawn for each of the panel and also the pressure distribution on each panel is taken down. Bitsuamlak et al. [5] conducted a research on ground mounted stand-alone and arrayed solar panels by using Computational Fluid Dynamics (CFD) method in order to investigate the aerodynamic features of PV systems. In this study, it was shown that the pressure coefficient distribution obtained from numerical modeling is underestimated compared to full-scale experimental results. Moreover, when solar panels are placed in a row, sheltering effect from upwind solar panels decreases wind loads on adjacent panels. The test was done for an aluminium solar panel frame. Stathopoulos et al. [6] examined the wind loads on a stand-alone solar panel placed on ground, flat roof, and gable roof of a building. It was demonstrated that the wind angles of attack in the range of 105degree to 180degree cause the extreme pressure coefficient values with a maximum effect at 135degree. The authors also assessed various geometries and concluded that the effect of building height and panel location were not significant for the roof-mounted systems and the effect of panel inclination is significant only for the critical wind angle of attack.

Ogedengbe et al. [7] investigated the pressure distribution on a scaled model of ground mounted solar panel with 25 degree and 40degree inclination and measured wind profiles at 0, 30, 150 and 180degree of wind angle of attack. The 25 inclination of the panel was found to produce larger loading in comparison with the 40 case. It was also noted that the gap between upper and lower panels plays an important role in pressure distribution and should be considered in structural design of the panel. Kopp et al. [8] studied wind loading on the ground mounted array to illustrate the effect of the building on pressure distribution of the roof mounted solar arrays. This study was performed at the Boundary Layer Wind Tunnel II (BLWT II) at the University of Western Ontario. It was found that there was a considerable difference in aerodynamics loading between ground mounted and roof mounted solar panel arrays as a result of the interaction of the flow with the building itself. Warsido et al. [9] conducted wind tunnel testing and investigated the effect of row spacing on wind loads for a solar panel mounted on both flat roof and ground. The testing was performed on 1:30 geometric scale model with inclination of 25 degree and wind angles of attack ranging from 0 to 180 degrees at 10 intervals. It was shown that the isolated panels experienced higher wind pressure compare to those placed in an array configuration.

Debdutta et al.[10] used the K-epsilon equations at IIT Delhi to see the result for the solar array placed over the rooftop of CSIR-CBRI which is having a solar power generation of 100 kW, it is concluded that the solar array is subject to significant suction under the wind loading which may cause lifting of the array if it is not properly designed. Jubayer et al. [11] performed 3D unsteady Reynolds-Averaged Navier-Stokes (RANS) to find the effect of wind load on the ground mounted stand-alone photovoltaic (PV) panel with 25 degree inclination. The authors illustrated that mean pressure coefficients of the solar panel surfaces are in a good agreement with the experimental results of wind tunnel test by Abiola-Ogedengbe [7]. It has also been concluded that the thickness of the panel does not have much effect on the load calculation for the range of 6mm-60 mm. Shademan et al. [12] conducted a research on ground mounted solar panels in both stand-alone and array configuration with a tilt angle of 45degree. In this study CFD simulations using 3D steady Reynolds-Averaged Navier-Stokes (RANS) were performed to investigate the effect of wind loading. Wind angles of attack ranging from 0 to 180 at 30 degrees intervals were employed and the influence of spacing between individual modules and ground clearance was studied. Shademan et al. [13] furthered the investigation and carried out work on the ground mounted solar panel using Detached eddy simulation (DES). It was reported that as the ground clearance increased, stronger vortex shedding and larger unsteady forces were observed. It is also concluded that the panels at the bottom experiences maximum coefficient of drag. J.Franke et al. [14] gives the thorough guidelines for considering the various parameters while simulating and solving a CFD in wind engineering problem. This study shows that the inlet, lateral and the top boundary conditions for a given computational domain should be 5H away from the selected body of height H to prevent the acceleration of the flow. Also the outflow should be 15H to allow for the fully developed flow.

Emanuelo et al. [15] proposes an algorithm to determine the optimum tilt angle, the regression equation and the mathematical model is used to find the tilt angles and also optimum tilt angle are computed by means of the algorithm using solar irradiation data acquired from 1985 to 1989 at the site of Stockholm. Using the algorithm and the study it is found that tilt angles were significantly related to the latitude value, the result allows estimating the optimum tilt angles of solar panels as a function of latitude. Kaveri et al. [16] used various simulation software like RETSCREEN, PVSYST and NREL SAM. The solar radiation energy is simulated at the horizontal surface using all these different software packages for 6 different cities namely Ahmadabad, Delhi, Kolkata, Bhopal, Chennai and Mumbai to cover different regions in central, north, south, east and west. The results are compared to come to a common conclusion. It is found that the annual optimum tilt angle is close to the value of the location latitude in most cases and for some locations the optimum tilt angles obtained was varied +1 to +3 from their latitudes. Saurav kumar et al. [17] used experimental setup consisting of power control unit, solar flux meter, and dial angle gauge. The voltage and current varies as the slope of the panel changes in the Jaipur, Rajasthan. The optimum angle of inclination has been proposed for the Jaipur as equal to the latitude of the location.

From the literature survey it is found that the wind loads on the solar panels can be analyzed at the different angle of attack, since the tilt angle is fixed for a given location, thus the panel inclination of 18 degrees equal to the latitude of the location is selected. Also using the K-ε turbulence model the array of the solar panel is analyzed.

III. METHODOLOGY

Domain dimensioning

Detailed documents with regards to the basic requirements on carrying on CFD analysis are available guideline for the CFD simulation of Flows in the Urban Environment J.Franke et al.[15].

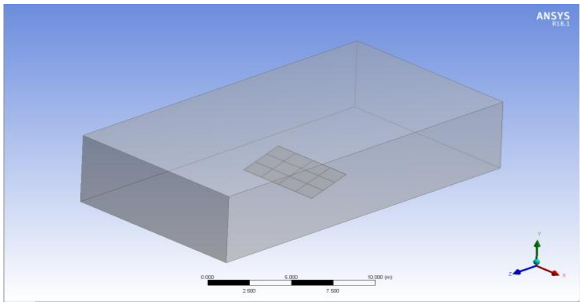

Figure1 CFD model using solid work

The recommendations for the description of volume of air domain that needs to be specified are as follows in this document: the domain top should be at least 5*H* above the top of the object that is of interest, where *H* is stated as the height of the object. In this paper, *H* is taken as the height from ground to the highest point on the panel. Therefore, the total height of the domain becomes 6*H*. With regards to the flow of wind from inlet, at least 5*H* distance of air domain should be provided before the wind first hits the object. After air flows around the object, it is also necessary to provide at least 15*H* distance of air to flow up to the outlet to permit flow to be redeveloped in the wake region. With regards to the 3D modeling, it is also necessary to provide 5*H* distance of air to flow in the lateral directions past the object of interest, as well.The representative domain of air and the panel are shown in Figure1.

Domain meshing and boundaries

Meshing of the domain of air and panel needs to be done carefully after creating the geometry of the parts. Mesh generator of ANSYS is used for this purpose to generate the grids for meshing. Tetrahedrons are selected with patch conforming method. Face size meshing is also added to the mesh generator in order to increase the accuracy of CFD analysis. There are 6 faces of solar panel used for the face sizing, where 2 of these faces are the top and bottom faces of the panels and the rest of the faces are resulting due to the thickness of the panel on the edges.

The top boundary and side boundaries of the air domain are specified as symmetry to enlarge the flow space near these boundaries, where this boundary specification also eliminates the need for mesh refinement near these boundaries. The bottom part of the air domain is defined as rough wall boundary condition that needs the specification of roughness height and constant. It is also important to specify the boundary condition between the air and the solar panel and the supporting structural system of the panel (if exists), where this boundary is selected as smooth wall in this paper. Pictorial views of the meshing of the air domain as well as the panel are demonstrated in Figures 2 to 5. In these Figures, the meshing for the cases of panel without supporting structures is considered.

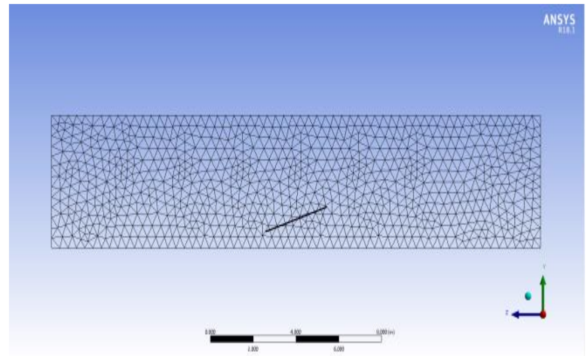

Figure2 Side view of CFD mesh model

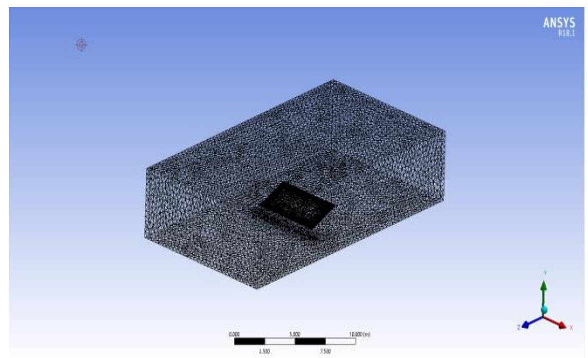

Figure3 Meshing ofpanel and the domain

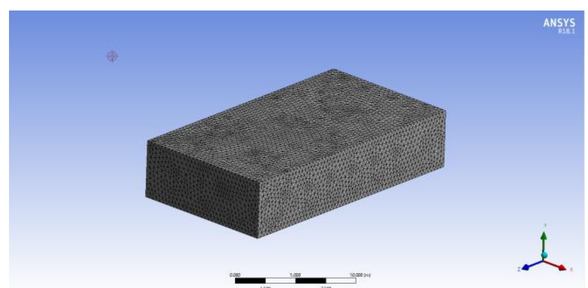

Figure4 Solid view of mesh at solar panel and domain

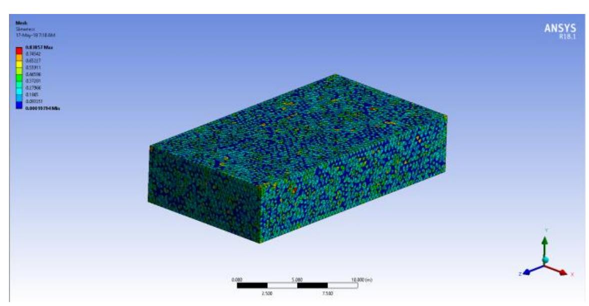

Figure5 Skewness of the domain

Figure5 represents the quality of the mesh where skewness is measured in the values ranging from 0 to 1 where 0 is the higher quality element and the 1 is the lower quality element. The Figure5 shows the colored representation of the elements where blue elements represent higher quality mesh and red colored represents lower quality, we can observe that most of the elements are in the blue region.

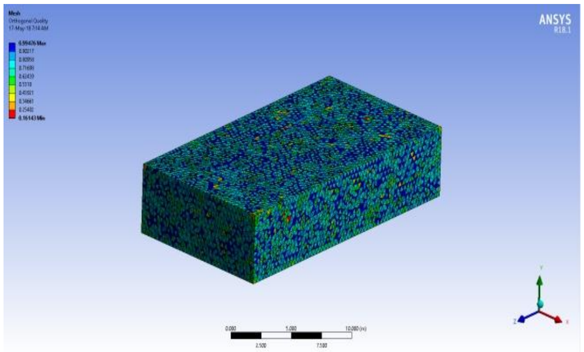

Figure6 Orthogonal quality of the domain

Figure6 represents the quality of the mesh where orthogonal quality is measure in the values ranging from 0 to 1 where 0 is the lower quality element and the 1 is the higher quality element, the Figure 6 shows the colored representation of the elements where blue elements represent higher quality mesh and red colored represents lower quality, we can observe that most of the elements are in blue region.

Specification of boundary conditions

Initial gauge pressure is set to zero in ANSYS FLUENT Solver, since same air pressure is considered to be present at every point. For the velocity inlet, z direction of velocity (m/s) is specified, i.e. the direction normal to the inlet face. For the turbulence part, intensity and viscosity ratio method is chosen. For external flows, turbulent viscosity ratio is typically selected between the ranges of 1% to 5%. Since the flow is calm at the inlet part, the turbulent intensity is set to 1% in this study. Turbulent viscosity ratio is fixed to be 10.

Validation of the results

Case 1: Solar panel with 30degree inclination with 0 degree angle of attack

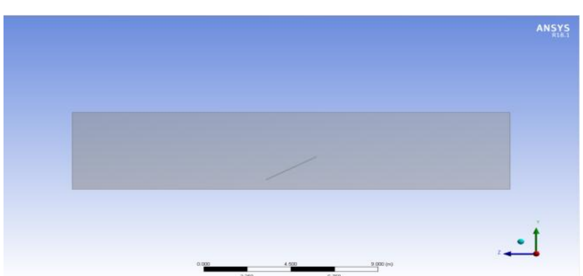

Figure7 Geometry of GeorgetaBaetu

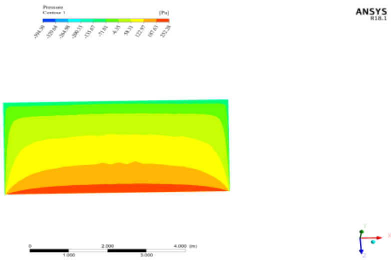

Figure8 Pressure distribution on solar panel

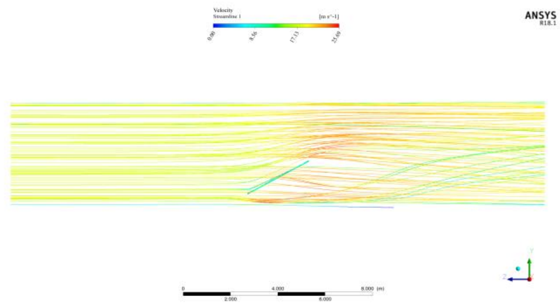

Figure9 Stream line of flow at AOA =0 degree

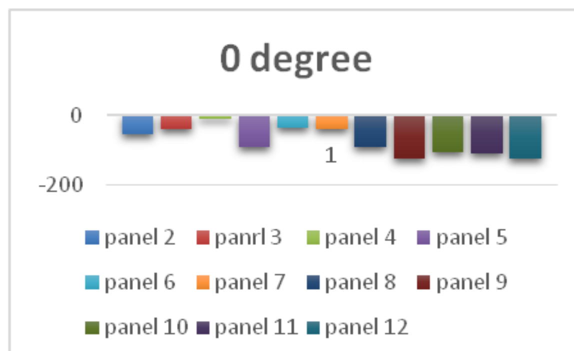

Figure10 Mean pressure values on solar panels at AOA=0 degree

Figure7 shows the model which is used in the base paper and the same is taken for validation with the wind speed of 18 m/s, panel inclination of 30 degrees and angle of attack equal to 0 degrees. Figure8 shows the pressure distribution over the panels and Figure9 shows the flow over the panel and it is observed that the pattern of both pressure distribution and flow is same as obtained by GeorgetaBăetu. Figure10 shows the mean pressure values on solar panels and the values are also comparable.

IV. RESULTS

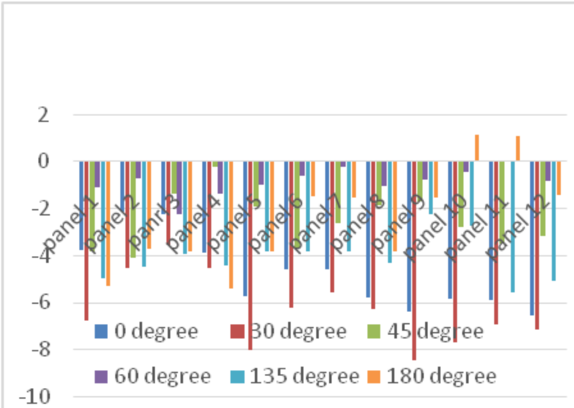

Figure 11 Mean pressure distribution on the solar panels for different angle of attacks.

From the Figure11 it can be seen that the pressure is more for the angle of attack of 0 and 30 degrees and the maximum values of -5 to -8.5 Pa are observed on the panels. Also the pressure acting for the angle of attack of 45, 60 and 135 degrees is less, this is because the pressure acts almost same on both side of the panel.

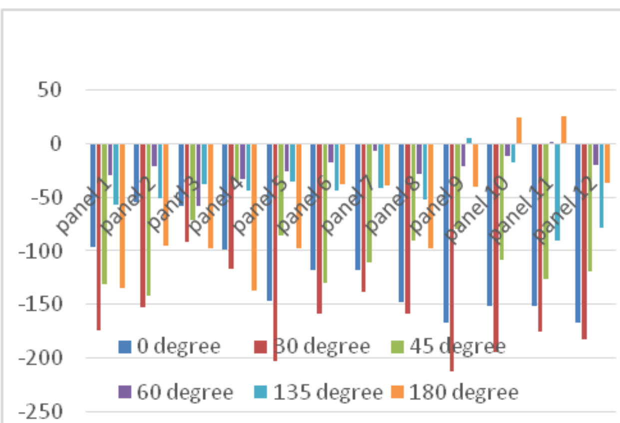

Figure 12 Mean pressure distribution on the solar panels for different angle of attack

From the Figure12 it can be seen that the pressure is more for the angle of attack of 0 and 30 degrees and the maximum values of -150 to -220 Pa are observed on the panels. Also the pressure acting for the angle of attack of 45, 60 and 135 degrees is less, this is because the pressure acts almost same on both side of the panel and most of the values ranges from 0.1 to -100 Pa only. For the angle of attack of 180 degrees the pressure values ranges from 20 Pa to -140 Pa.

V. CONCLUSION

From the flow pattern seen in the simulation we came to know that the angle of attack plays a major role to obtain the maximum mean pressure values where the panel is placed at ground, with inclination of 18 degrees and the analysis have been done for 0, 30, 45, 60, 135 and 180 degree angle of attack at both minimum and extreme velocities i.e., 5 and 25 m/s. The values of suction pressure are compared in the Figure 11 and Figure12 at different angle of attack on each solar panel. From the results the following observations are made:

The biggest suction pressure occurred at 0, 30 and 180 degree in both 5 and 25 m/s cases and the values are as high as -8.25 and -220 Pa in the cases of 5 and 25 m/s respectively.Also the pressure acting for the angle of attack of 45, 60 and 135 degrees is less, this is because the pressure acts almost same on both side of the panel and most of the values ranges from 0.1 to -100 Pa only.The maximum pressure in the case of 30 degree inclination is observed to increase by 30% in both the cases.The pressure distribution is also observed to get shifted towards the left side of the panel that is on panel numbers 1, 5 and 9 as the wind direction is changed to 30 and 45 degrees.

REFERENCES

[1] World energy outlook, International Energy Agency, ISBN: 978-92-64-04560-6, 2008

[2] Loraima Jaramillo-Nieves, and Pablo del Rio, “Contribution of renewable energy sources to the sustainable development of islands: An overview of the literature and a research agenda”, Sustainability, Vol. 2, pp. 783-811, 2010.

[3] ANSYS® Fluent, Acadamic Research Mechanical, Release 18.1.

[4] GeorgetaVasies, Elena Axinte, and Elena- Carmen Teleman, and Victoria-Elena Rosca “Numerical simulation of wind action on solar panel array for different wind directions”, UniversitateaTechnica Gheorghe asachi, Vol. 1, 2012

[5] Girma T. Bitsuamlak, Agerneh K. Dagnew, and James Erwin, “Evaluation of wind loads on solar panel modules using CFD”. The fifth International Symposium on Computational Wind Engineering, 2010.

[6] Stathopolous T, Zisis I, and Xypnitou, “Local and overall wind pressure and force coefficients for solar panels”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 125, pp. 195-206, 2014

[7] Ayodeji Abiola- Ogedengbe, “Experimental investigation of wind effect on solar panels”, Electronic thesis and Dissertation Repository, 1177, 2013

[8] Kopp G A, Surry D, and Chen K, “Wind loads on a solar array”, Wind and Structures an International Journal, Vol. 5, pp 393-405, 2002.

[9] Warsidoworkamaw, Bitsuamlak G, Batarajohann,

Chowdhury Arindam, “Influence of spacing parameters on the wind loading of solar array”, Journal of Fluids and Structures, Vol. 48, 2014.

[10] Debdutta Ghosh, Siddharth Behera, A.K Mittal, “Assessment of wind flow around rooftop solar array- A case study”, Indian Society for Wind Engineering, pp. 213-230, 2014.

[11] Jubayerchowdhury, HanganHoria, Siddiqui Kamran, “CFD analysis of convective heat transfer from ground mounted solar panels”. Solar energy, Vol. 133, pp 556, 2016.

[12] Mehrdad Shademan, HoriaHangan, “Wind Loading on Solar Panels at Different Inclination Angles”, 11th Americas conferences on Wind Engineering, 2009.

[13] Mehrdad Shademan, Ron Barron, Ram Balachandar, HoriaHangan, “Numerical simulation of wind loading on ground- mounted solar panels at different flow configuration, Canadian Journal of Civil Engineering, 2014

[14] J.Franke, C. Hirisch, A.G Jensen, H.W Krus, M. Schatzmann, P.S Westbury, J.A. Wisse, N.G. Wright, “Recommendations on the use of CFD in Wind Engineering”.

[15] Emanuele Calabro, “An algorithm to determine the optimum tilt angle of a solar panel from global horizontal solar radiation”, Journal of Renewable Energy, Vol. 13, 2013

[16] Kaveri Markam, K. Sudhakar, “Estimation of optimum tilt angle for solar photovoltaics installations in India”, International Reasearch Journal of Engineering and Technology, Vol. 03, 2016

[17] Sourav Kumar, P.B.L Chaurasia, Hari Kumar Singh, ”Experimental Study of optimum tilt angle for solar PV panel in Jaipur”, International Journal of Science and Research, ISSN 2319-7064, 2012.