ISSN: 2278-0181
NCERAME - 2015 Conference Proceedings

# Thermal Management of Solar Power Pack using Computational Fluid Dynamics

Lakshminarasimha N [1]
[1]PG Student, Thermal Engg.
Department of Mechanical Engg.
Global Academy of Technology
Bangalore, India

Dr. M.S. Rajagopal [2]
[2]Professor & Head
Department of Mechanical Engg.
Global Academy of Technology
Bangalore, India

M. Vedavyasa [3]
[3]Associate Professor
Department of Mechanical Engg.
Global Academy of Technology
Bangalore, India

***Abstract*— Solar Power Pack (SPP) is an enclosure which houses heat generating electrical devices such as Battery Bank, Inverter and Controller. The present study involves cooling and ventilation analysis of the enclosure such that the electrical components operate at less than their limiting temperatures. Further, optimization studies are also carried out to determine fan location and locations of inlet and exit vents besides positioning of Baffles for uniform air motion. This study will also help in minimizing the pumping power cost by determining the pressure losses. Thermal Management of SPP has been carried out using CFD software ANSYS Workbench- FLUENT for both 2D and 3D geometry. Numerical results obtained from FLUENT agree well with analytical results.**

***Keywords*-- Thermal Management, Solar Power Pack, Enclosure, CFD, Optimization, FLUENT, Cooling, Ventilation**

***Terminology*--**

| | |
|---|---|
| L | = Length of SPP |
| W | = Width of SPP |
| H | = Height of SPP |
| $L_b$ | = Location of Battery Bank |
| $L_C$ | = Location of Controller |
| $L_i$ | = Location of Inverter |
| Lf | = Location of exhaust fan |
| $L_V$ | = Location of inlet vent |
| 2D | = Two-dimensional |
| 3D | = Three-dimensional |
| TSA | = Total surface area |
| CFM | = Cubic feet per minute |

## 1. INTRODUCTION

In today's climate of growing energy needs and increasing environmental concern, alternatives to use of non-renewable and polluting fossil fuels have to be invested. One such alternative is solar energy. Solar power is the conversion of sunlight into electricity directly using photovoltaics. The latest and the most cost effective method for integrating solar power to homes/offices/colleges/rural electrification etc. are through the use of Solar Power Pack. Present work is generic and can be used as preliminary literature for studying any different kinds of Solar Power Pack models.

Internal flow and thermal analysis in an enclosure containing heat generating equipments is always a challenge in the field of Heat transfer. Due to high expense of experimental study in recent years, Computational Fluid Dynamics (CFD) is gradually becoming the most efficient tool in thermal enclosure design. Study of Solar Power Pack comprises of; overall evaluation of design involving ventilation and cooling, study of velocity and temperature contours and optimization of design such as; fan size and location, location and size of inlet and exit vents and positioning of Baffles for uniform air motion. Also compute the many derived parameters along with the graphical representation of the interested regions. The main objective of the present work is to undertake a numerical investigation to evaluate the thermal design of SPP and further optimize the design to minimize the cost using simple 2D & 3D models.

## 2. LITERATURE SURVEY

In this section, literature survey has been conducted to understand the state of the art in thermal management of Solar Power Packs in particular and CFD in general to know the appropriate boundary conditions, modeling of heat sources and various CFD models. The summaries of most important papers are listed below.

Hoffman- A Pentair Company [1], [2003], released technical information data sheet on heat dissipation in an electrical enclosure. This technical data/cut sheet is majorly for electrical enclosures exposed to outdoor conditions. The technical information highlights majorly about, Enclosures temperature rise/ Determination of temperature rise for an enclosure, Evaluation of solar heat gain and Selection of fans as per cooling requirement for an enclosure. Hence this manual is highly helpful in preliminary design stage, in predicting the cooling requirements for electronic and electrical enclosures.

Bud Industries, Inc., [2], [2007], released technical data hand book on Enclosure design tips. This guide helps in selecting cabinets, enclosures and other packaging system for electronic products. Also this booklet summarizes some of the more important issues of packaging for electronic systems and products, with the goal of helping us evaluate our options quickly, and then selecting the optimum solution for our application. As per project concerned the booklet majorly highlights about Materials and finishes for an enclosure and Basics of cooling for cooling requirement in an enclosure.

Mahendra Wankhede, et al., [3], [2010], Paper comprises study on evaluation of cooling solutions for outdoor electronics. Both experimental and CFD simulation had been carried out for three different conFig.urations of enclosures including a solar radiation shield, double-wall enclosure and a air-to-air heat exchanger; using a typical Aluminium enclosure. Each enclosure was consisting 100W generating PCBs. The work showed that solar heat load can increase the internal air temperature by 20%, White coating of an enclosure reduces internal air temperature by around 25% and as the major part of the work highlighted that 50-55% can reduce ΔT due to the internal fans compared to a sealed enclosure with no fans and having radiation shield and double-walled enclosure with air circulation provided modest improvements of around 25%, where as air-to-air heat exchanger showed improvement by 75%.

Tom Kowalski and Amir Radmehr, [4], This Paper highlights the significance of using coupled FNM and CFD based analysis in cooling of electronic enclosure. The study demonstrates the application of coupled Fluid Network Method (FNM) and CFD in the analysis of flow behavior and Thermal behavior in a telecommunication cabinet. The cabinet is forced- flow air cooled consisting a stack of PCB's. The work showed detailed measurement of airflow velocities in various parts of the cabinet, the predicted variation of the flow rates through card passages and total flow through system were found to be within 10% of their experimentally measured values.

It is observed that very few literatures are available on thermal management of SPP using CFD though thermal analysis at PCB and chip level is available. However, industrial data sheets are available as ready reckoner to help designers to choose the right size of fan and number of fans. This study is aimed at optimizing the cooling and ventilation parameters to have better understanding of the system.

## 3. METHODOLOGY

The analysis of SPP is been carried out using Commercial CFD code ANSYS Workbench-FLUENT and Modeling and Meshing for 3D SPP enclosure were carried out in ANSYS ICEPAK.

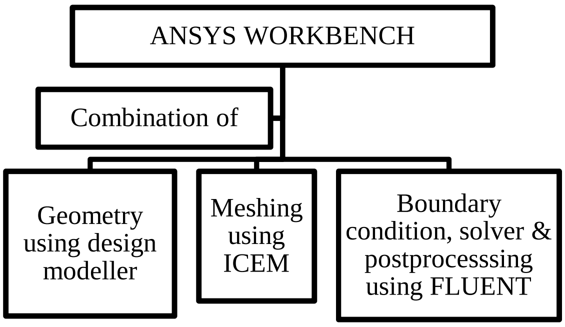


Fig. 1: Combined module of CFD code

ANSYS Workbench delivers innovative, dramatic simulation technology advances in every major physics discipline, along with improvements in computing speed and enhancement to enabling technologies such as **geometry handling**, **meshing** and **post-processing** (Fig. 1). These advancements alone represent a major step ahead on the path forward in Simulation driven Product Development.
ANSYS ICEPAK provides highly simplified way of modeling and meshing for cubical electronic enclosure problems.

### 3.1 GEOMETRY

The Geometry of SPP consists of Battery Bank, Inverter, Controller, and Inlet vent and outlet/exhaust fan. The most important thing to be noted is that the geometry is planar symmetry. The dimensions specified for the models below in 3.1.1 and 3.1.2 are elaborated in the above terminology.

### 3.1.1 2D Geometry model

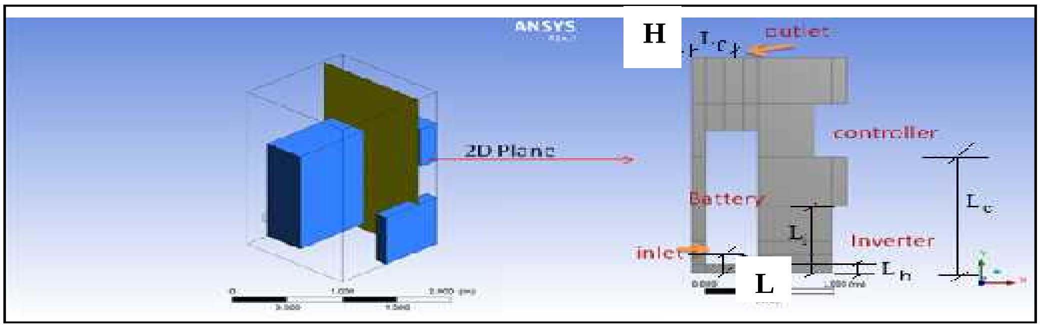


Fig. 2: 2D SPP Model

Fig. 2, shows the 2D cut plane (XY plane- right bottom corner of Fig. 2) taken from planar symmetry 3D model which has been considered for 2D analysis. A simplified 2D analysis is carried out at the mid-plane of SPP to get an insight of the physics of the problem though this model doesn't capture the overall objective of determination of accurate flow and temperature distributions. 2-D geometry does not accurately capture locations of inlet vent, fan location and heat generating devices.

### 3.1.2 3D Geometry model

The planar symmetry 3D SPP domain with dimensions as per above terminology is shown in Fig. 3. The 3D model captures the problem objective and fetches us real results and provides us the clear visualization of contours and vectors at our interested spots in the domain.

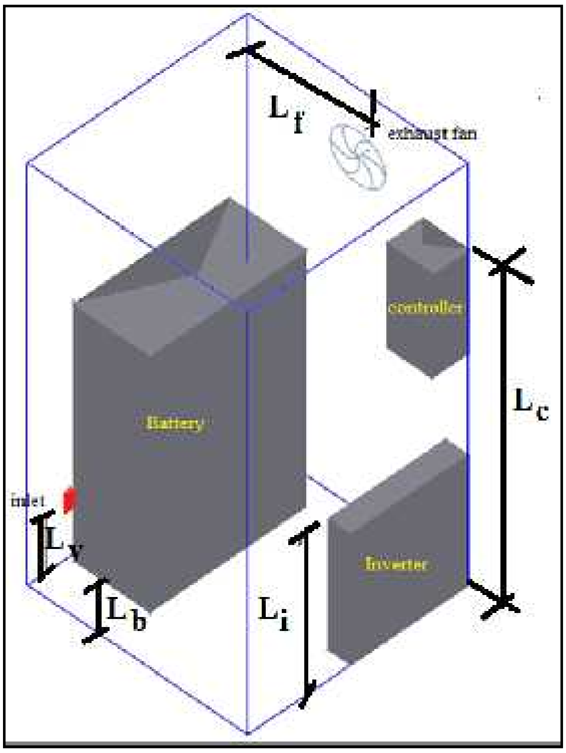


Fig. 3: 3D SPP Model

### 3.2 MESHING

Meshing is the important criteria as part of analysis considered. Maximum time spent in meshing is the time well spent. Mesh independence studies have been carried out for both 2D and 3D domain meshes. The details of meshing for 2D and 3D domain are as described below.

### 3.2.1 2D Meshing

The 2D mapped fine mesh is shown in Fig. 4. Mesh independence study graphs for the 2D mesh are shown in Fig. 5 and Fig. 6. Based on this study, further analyses are carried out for Quad mesh with 46280 elements and 47204 nodes. The convergence criteria for the continuity, x-velocity, y-velocity, energy, k and epsilon are shown in Fig. 7.

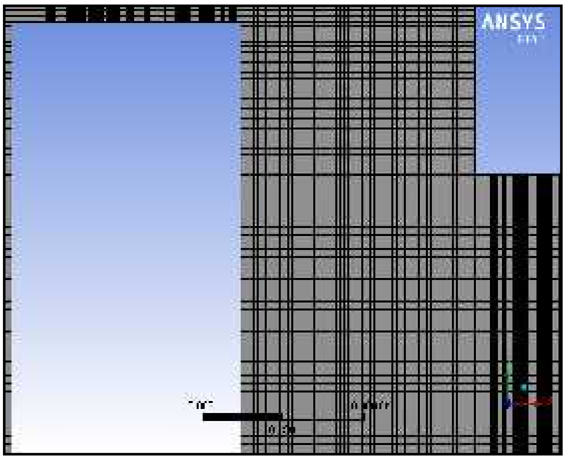


Fig. 4: 2D Mesh

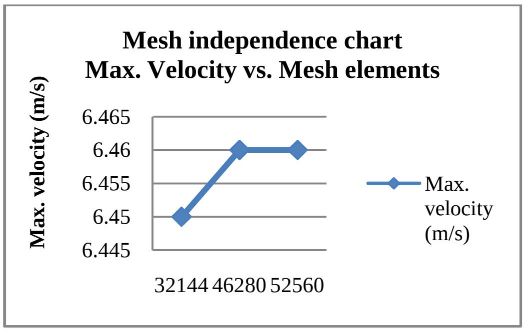


Fig. 5: 2D Mesh independence plot for Maximum Velocity vs. Mesh elements

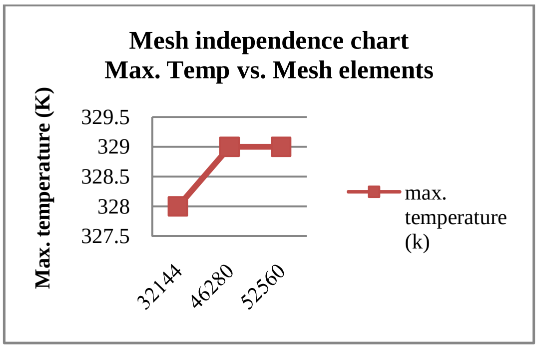


Fig. 6: 2D Mesh independence plot for Maximum Temperature vs. Mesh elements

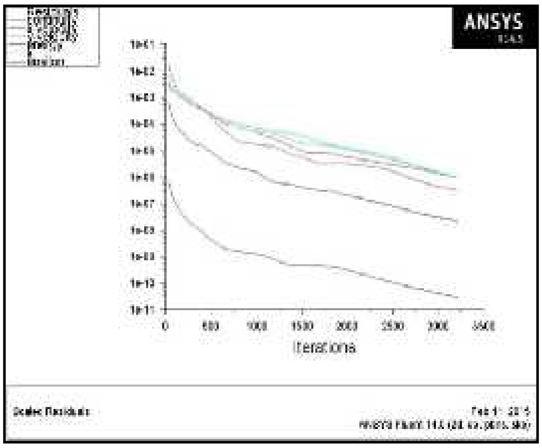


Fig. 7: 2D Mesh Convergence criteria

### 3.2.2 3D MESHING

Creating a 3D mapped hexahedral mesh is a big challenge; however Icepak provides an easy platform to create mapped hexahedral mesh for cubical geometries. Icepak also facilitates fluid and solid surface extractions which can be named appropriately. Icepak is used for modeling & meshing process & the mesh has been transferred to fluent for further analysis.

Mesh independence study graphs for the 3D mesh are shown in Fig. 9 & Fig. 10. The study shows that mesh with 272786 elements and 284868 nodes is the optimum mesh which is shown in Fig. 8. The convergence criteria trends for the continuity, x-velocity, y-velocity, z-velocity energy, k and epsilon are shown in Fig. 11.

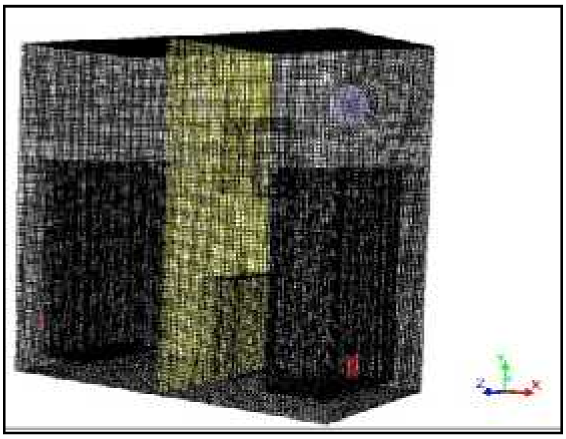

Fig. 8: 3D Mesh

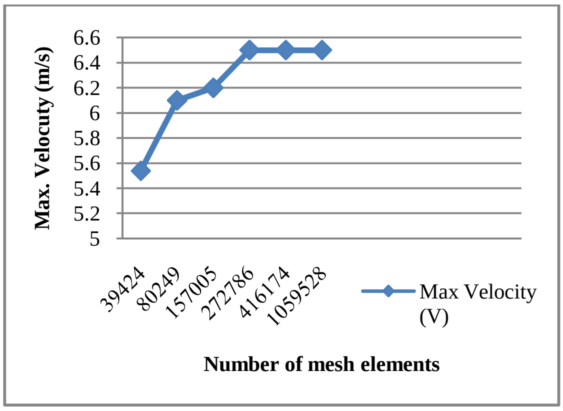


Fig. 9: 3D Mesh independence plot for Maximum Velocity vs. Mesh elements

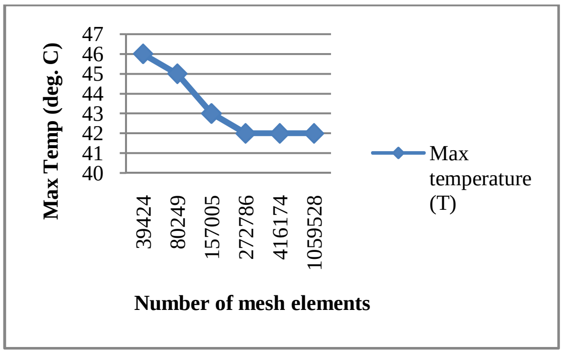


Fig. 10: 3D Mesh independence plot for Maximum Temperature vs. Mesh elements

ISSN: 2278-0181


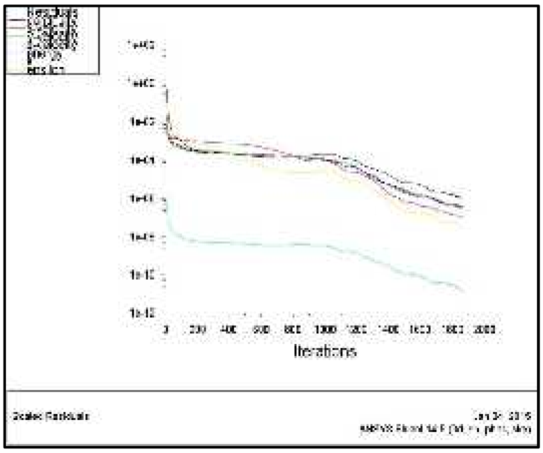


Fig. 11: 3D Mesh Convergence criteria

## 3.3 SOLUTION METHODOLOGY

In the present investigation, the enclosure exhaust fan of 285 CFM has been selected through analytical calculation [1]. Flow is considered to be incompressible and steady. The internal flow is governed by Reynolds Average Navier-Stokes equations (RANS). Standard K-epsilon model is used to solve the flow analysis. The heat generating sources wall are at heat flux and no slip condition. Standard K-epsilon model comes under two equation model of turbulence kinetic energy k and its dissipation rate $\epsilon$ & hence these two equations are solved for obtaining the result. The incoming air through inlet vent is due to negative pressure occurred & removing heat through exhaust fan. Transport equation for momentum and turbulence parameter is solved with SIMPLE discretization scheme.

## 3.4 MATHEMATICAL MODELS

Conservation equations of mass and momentum for all flows are solved in ANSYS FLUENT and an additional equation for energy is solved for flows involving heat transfer. Flow inside a Solar Power Pack involves both fluid flow and fluid flow with heat transfer, hence governing equations [8] that are solved in FLUENT are as listed below:

Mass conservation equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \vec{v}) = S_m$$

Momentum conservation equation:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla.(\rho \vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\tau}) + \rho \vec{g} + F$$

*Where, the stress tensor, τ is given by*

$$\bar{\tau} = \mu \left[ (\nabla \vec{v} + \nabla \vec{v}^{T}) - \frac{2}{3} \nabla.\vec{v} I \right]$$

Energy conservation equation:

$$\frac{\partial}{\partial t}(\rho E) + \nabla.\left(\vec{v}(\rho E + p)\right) = -\nabla.\left(\textstyle\sum_j h J_j\right) + S_h$$

The above equations are a general form of governing equations [8] and are valid for both compressible and incompressible flows. The governing equations with no time derivative term states steady flow.

Also additional transport equations as shown below, need to be solved since flow is turbulent. The turbulence model used for this analysis is ***Standard k- ∈ Turbulence model.***

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k$$

And

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

Where, turbulent or eddy viscosity, $\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$ and $G_k$ & $G_b$ represents the generation of turbulence kinetic energy due to mean velocity gradients and buoyancy. $Y_M$ represents the contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rate.

Here, the model constants $C_{1\epsilon}, C_{2\epsilon}, C_\mu, \sigma_k$, and $\sigma_\epsilon$ have the following default values as below:

$C_{1\epsilon} = 1.44$ , $C_{2\epsilon} = 1.92$ , $C_\mu = 0.09$, $\sigma_k = 1.0$ , $\sigma_\epsilon = 1.3$

All the above mathematical models cannot be solved by analytical method for complex flows. Hence all these equations are solved using FLUENT. Further in-depth details regarding mathematical models can be referred in FLUENT-Theory guide [8].

## 3.5 RESULTS AND DISCUSSION

### 3.5.1 2D RESULTS

It is stated in section 3.1.1, that the 2D analysis does not capture the overall problem objective but the main purpose of 2D analysis is to study different solver options, different Boundary conditions and effect of critical geometric parameters and understanding the velocity, pressure and temperature contours.

Figures shown below are the velocity (Fig. 12) and temperature contours (Fig. 13) for the SPP enclosure.

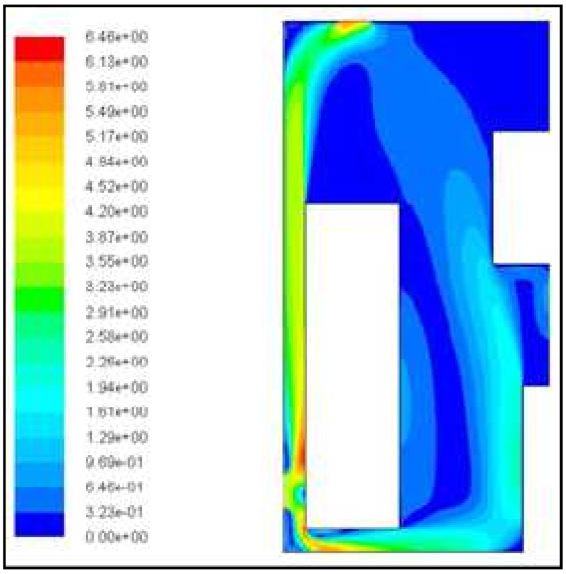

Fig. 12: Velocity contour for 2D domain

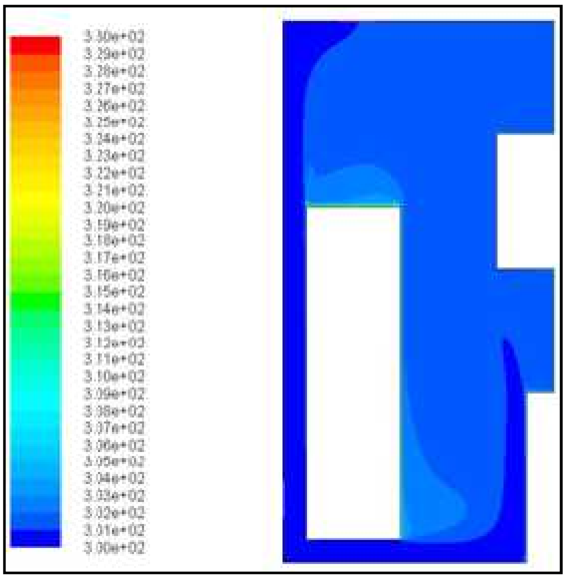

Fig. 13: Temperature contour for 2D domain

| | |
|---|---|
| Maximum Velocity (m/s) | 6.46 |
| Maximum Temperature (°C) | 57 |
| Enclosure fluid Temperature rise (ΔT) in (°C) | 0.809 |

Table 1: 2D analysis results

Table-1 shows the results obtained from 2D analysis. Maximum temperature is found at the top surface of the Battery Bank and maximum velocity obviously near inlet and outlet. The locations of Battery Bank, Inverter and Controller are actual in 3D, where as in 2D their locations cannot be captured accurately, since 2D mid x-y plane is considered. Further, locations of fan inlet and outlet vents are also approximate. Thus the results are expectedly not accurate, but certainly provide an insight of complex coupled flow and heat transfer problem.

### 3.5.2 3D RESULTS

#### 3.5.2a. Velocity contours and Temperature contours

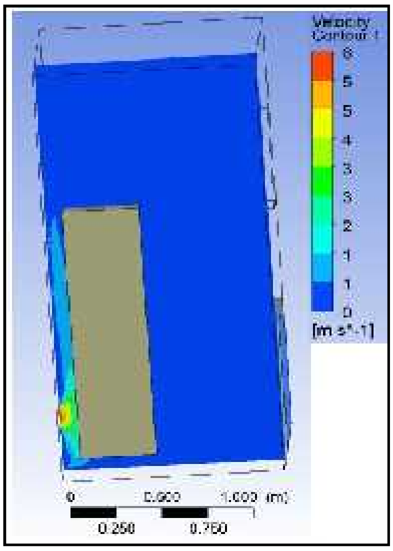

Fig. 14: Velocity contour for 3D domain

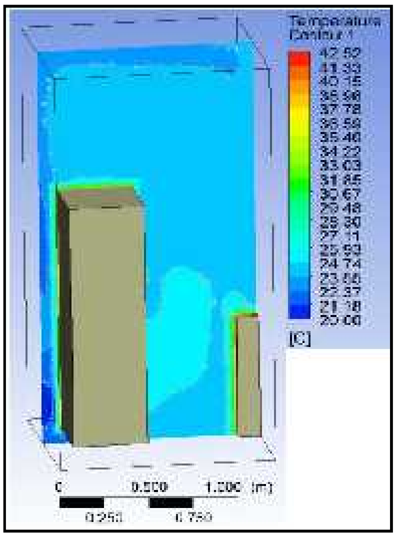

Fig. 15: Temperature contour for 3D domain

| | |
|---|---|
| Maximum Velocity (m/s) | 6.0 |
| Maximum Temperature (°C) | 42.5 |
| Enclosure fluid Temperature rise (ΔT) in (°C) | 4 |

Table 2: 3D analysis results

Figures 14 & 15 show the velocity and temperature contours in 3D domain. The temperature contours near the Battery is shown in Fig. 16 and near the Inverter and Controller in Fig. 17. The maximum temperatures at the Inverter, Battery and Controller are shown in table-3. Though the overall velocity distribution is relatively uniform, the flows are relatively lesser at the Inverter resulting in higher temperature. Hence flows need to be optimized such that the temperatures of the devices are uniform.

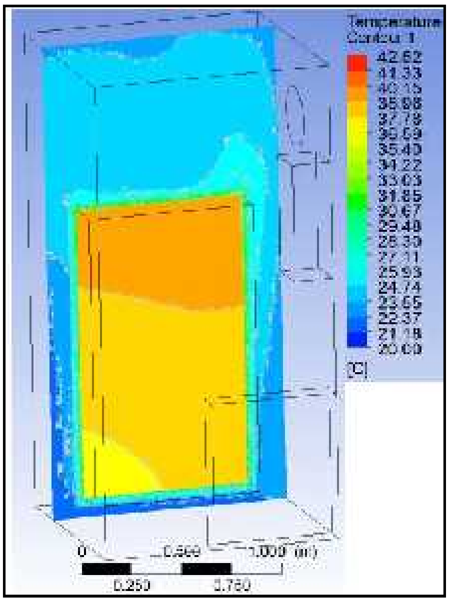


Fig. 16: Temperature in Battery

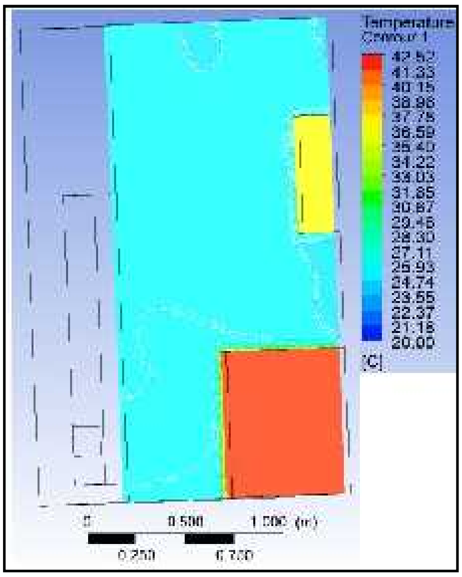


Fig. 17: Temperature in Inverter and Controller

| Heat generating devices | Temperature obtained in ℃ |
|---|---|
| Battery | 39 |
| Inverter | 42.5 |
| Controller | 34.61 |

Table 3: Temperature of Heat generating devices

Table-2 shows the maximum velocity and temperatures in 3D analysis. Comparison of Table-1 and Table-2, show that there is considerable difference in the values of maximum temperature, maximum velocity and temperature difference of fluid temperature. This is expected since 2-D and 3-D geometries are different.

The velocity stream line plot is as shown in Fig. 18.

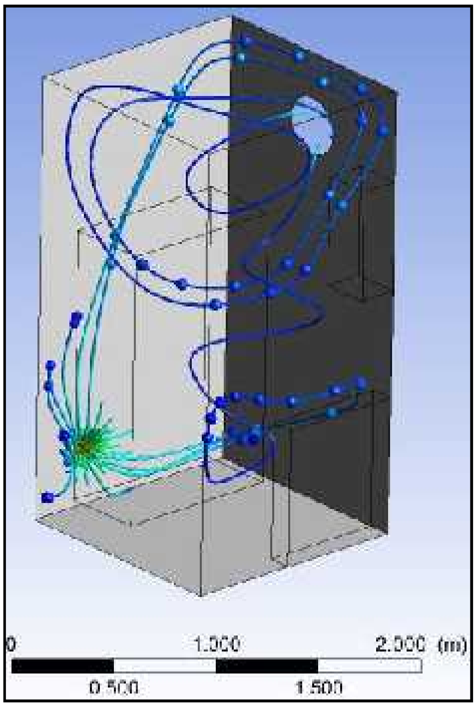


Fig. 18: Velocity stream line plot

From the stream line plot it is quite clear that the flow is channeled in all four directions (Top, right, left & bottom) and the most predominant flow being in the top direction. Considering this there is scope to optimize the flow and a suitable Baffle at the right position can be provided to direct more air towards the Inverter. These are discussed in section 3.5.2c. Further the locations of inlet and outlet also are studied in section 3.5.2b to understand its effect on temperature distribution.

### 3.5.2b. Optimization of the location of inlet-vent and exhaust fan

Two different locations for the inlet vent have been studied (Fig. 19) and the results are shown in Table-4

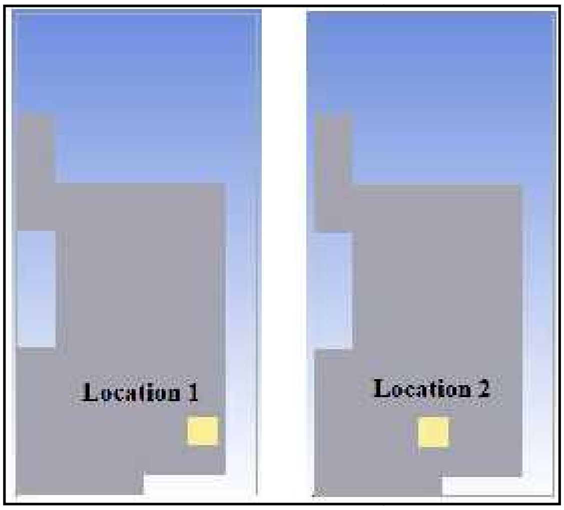


Fig. 19: Inlet-vent optimization

| Devices | Temperature in °C for inlet location-1 | Temperature in °C for inlet location-2 |
|---|---|---|
| Battery | 39 | 37.64 |
| Inverter | 42.5 | 47 |
| Controller | 34.61 | 38.2 |

Table 4: Results obtained by optimizing the location of inlet vent

Changing the location of inlet vent from location-1 to location-2, the Battery temperature is reduced but the temperatures of Inverter and Controller have increased. Hence location-1 is optimum.
Similarly, effect of two different outlet locations on temperatures has also been carried out (Fig. 20). It is found that there was no considerable change in the temperatures of heat devices. For location-2, the distance between inlet to exhaust fan is lesser compared to location-1 and considerable amount of air escapes without cooling the heating devices. And hence is not preferred.

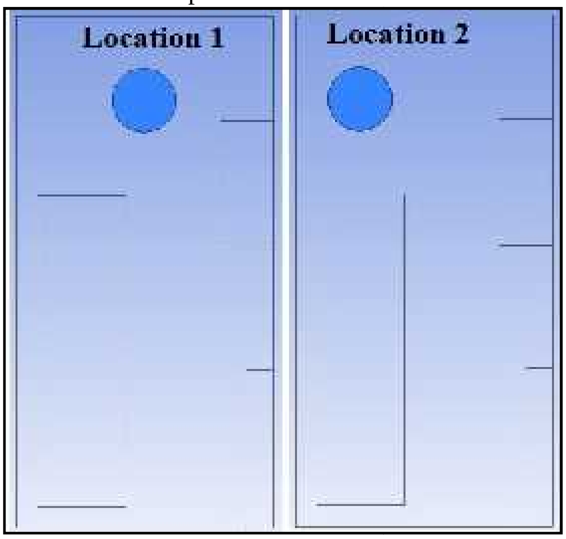


Fig. 20: Optimization of exhaust fan

### 3.5.2c. Positioning of Baffle

As stated in section 3.5.2a, the flow is predominant in top direction, hence decided to provide Baffle at position-1 and position-2.
In the first analysis, Baffles without perforation are provided near the inlet to direct more air towards the Inverter (Fig. 21). Fig.22 gives the comparison of temperatures of heat devices. It is observed that position-2 is better since the maximum temperature in all devices is limited to 38°C.

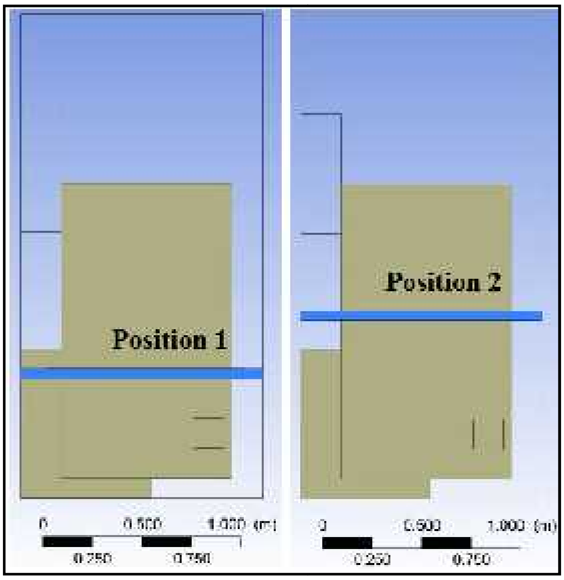


Fig. 21: Positioning of Baffle with no perforated vents

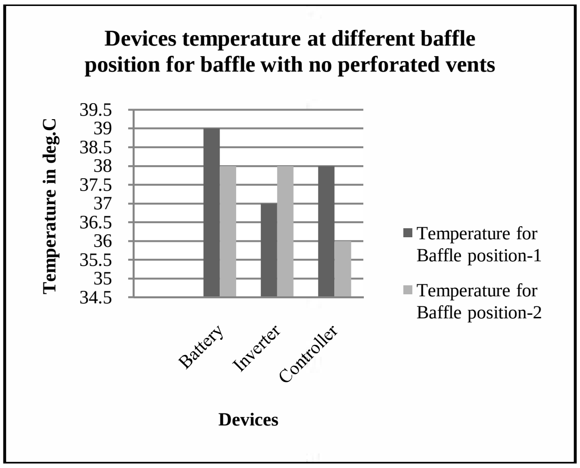


Fig. 22: Devices temperature at different Baffle position for Baffle with no perforated vents

In the next analysis, perforated Baffles with rectangular holes are provided at position 2 as shown in Fig. 23. The openings provided on the Baffle are 50% of the total area of the Baffle.

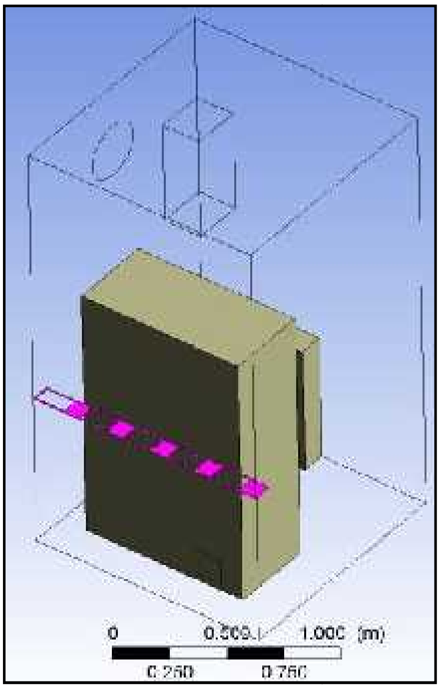


Fig. 23: SPP with Baffle with perforated rectangular vents

Fig. 24 shows the comparison results for three different cases viz. i. With no Baffle ii. Baffle with no vents iii. Baffle with rectangular vents. It is evident that results are better with Baffle compared to the case without Baffles. In the cases with Baffles, Baffle without vents gives reduced temperatures but with increased pressure drops.

### 3.5.2d. Pressure losses and Pumping power cost

Pressure drop is directly proportional to the Pumping power. Among the various cases studied, pressure losses are minimum for the case without Baffle and case with Baffle having rectangular vents though the later is having slightly lesser pressure drop (Fig. 24). It is obvious that Baffles result is higher drops at the entry, but pressure losses further downstream are also important. In general when the ventilation design is void of constrictions, pressure losses can be reduced which is also evident from CFD results. However, CFD plays a major role in determining the exact pressure drops which will be very useful to the designer.

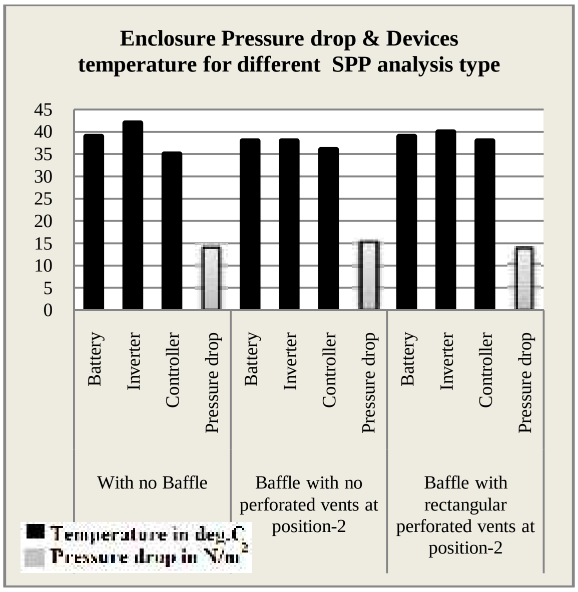


Fig. 24: Comparison graph for Temperature and Pressure drop for SPP with Baffle & with no Baffle

## 3.6 COMPARISON OF RESULTS WITH ANALYTICAL RESULTS

Considering flow to be incompressible and steady, Applying simple mass balance i.e. flow at inlet = flow at outlet. Since same areas provided at both inlet and outlet, the velocity at both inlet and outlet should be 6.0 m/s as per manual calculations. It is observed to be same from CFD as well. This is applicable for both 2-D & 3-D cases.
Similarly, considering heat balance, the heat dissipated by the heat devices should be equal to heat carried away by the fluid. This is verified in both 2-D & 3-D cases.
2-D case:
Fluid temperature rise by analytical method: 0.88℃
Fluid temperature rise by CFD: 0.809℃
3-D case:
Fluid temperature rise by analytical method: 4.0℃
Fluid temperature rise by CFD: 4.0℃
Thus it is evident that the CFD results are justified as both mass and heat balances are maintained.

## 3.7 CONCLUSIONS

The following conclusions can be drawn based on CFD analysis carried out on Solar Power Pack (SPP) enclosure consisting Battery Bank, Inverter and Controller.
1, CFD is very powerful tool to determine the exact pressure drops, maximum velocities and temperatures.
2. CFD also provides insight of the cooling and ventilation problem, by determining velocity and temperature distributions. This will help the designer.

3. Parametric studies have been carried out to understand the effect of following parameters:
3.1 Location of Inlet vent
3.2 Location of exhaust fan (outlet vent)
3.3 Provision of Baffles
3.4 Effect of Baffles with and without vents
4. It can be concluded that for minimizing temperatures, Baffles without vents are recommended though there is a higher pressure drop.
5. Considering both temperature and pressure drop, Baffle with rectangular holes is recommended.

It must be concluded that the present study is by no means exhaustive and there is further scope for optimization. However, the CFD results are very useful to the designer and these results will help him to make decisions faster. Further, they will aid in preparing a ready reckoner for cooling and ventilation design.

## ACKNOWLEDGMENT

We would like to thank the Global Academy of Technology college management and Department of Mechanical Engineering, (VTU Research Centre) for providing us the facility to successfully complete this project and encouraging us in publishing paper. We also thank M/s. SunEdison for providing data at critical junctures and their overall support.

## REFERENCES

[1] Hoffman- A Pentair Company, “Heat dissipation in Electrical enclosure”, “Technical information on Thermal Management of Electrical enclosures”, ©2003 Hoffman Enclosures Inc.

[2] Bud Industries, Inc., “Enclosure Design Tips Handbook”, July 2007,

© Bud Industries Inc.

[3] Mahendra Wankhede, Vivek Khaire, Dr. Avijit Goswami and Prof. S. D. Mahajan, “Evaluation of cooling solutions for outdoor electronics”, Journal on electronics cooling from electronics-cooling.com, Volume 16, No. 3, Fall 2010

[4] Tom Kowalski and Amir Radmehr, “Thermal Analysis of an Electronics Enclosure: Coupling Flow Network Modeling (FNM) and Computational Fluid Dynamics (CFD)”, no details

[5] Süddeutscher Verlag onpact (Rittal GmbH & Co. KG), “Project Planning Manual: Enclosure Heat Dissipation”, © 2009 by Süddeutscher Verlag onpact

[6] Anetis Stylianos, “The process of heat transfer and fluid flow in CFD problems”, American Journal of Science and Technology, 2014, 1(1): 36-49

[7] “A practical formula for air-cooled boards in ventilated enclosures”, from link: http://www.electronics-cooling.com/1997/09/a-practical-formula-for-air-cooled-boards- in-ventilated-enclosures/

[8] ANSYS Fluent User Guide, Release 15.0, Nov. 2013 and ANSYS Fluent 12.0, Theory guide, April 2009

[9] H K Versteeg and W Malalasekara, “Computational Fluid Dynamics- Finite Volume Method”, Text book of CFD, edition 1995

[10] Younus. A. Cengel, “Text book of Heat transfer- chapter 15- cooling of electronic equipments”