RESEARCH ARTICLE OPEN ACCESS

Numerical Investigation of Forced Convection cooling of Electrical enclosure using CFD

Lakshminarasimha. N., IGBC AP
Assistant Professor, Department of Mechanical Engineering, MVJ College of Engineering, Bangalore, India

Abstract
Electrical enclosures consist of high heat generating electrical components, so removal of heat generated remains as our primary aim. To achieve this, cooling the electrical equipment is always an economical and optimum solution to keep the electrical components to their operating temperature limits. Placing the cooling components in the enclosure is another important parameter to be considered. This parameter can be judged using a simple CFD analysis.
Therefore in the present work CFD simulation has been carried out by considering a typical Aluminum Electrical enclosure of volume (300mm X 300mm X 300mm) with total internal heat dissipation of 150W. With those values into consideration the surface area of enclosure, enclosure temperature rise, air flow requirement in an enclosure is calculated and based on which the fan is selected.
Keywords- Enclosure, cooling, temperature rise, CFD

Terminology
CFD- Computational Fluid Dynamics
CFM- Cubic Feet per minute
ΔT- Temperature rise
3D- Three dimensional
TSA- Total surface area

I. Introduction

Any electrical system without an enclosure is incomplete. As enclosure protects the electrical components from environmental hazards and it helps to provide safe cooling mechanism for electrical components. Any Enclosures can be cooled through following cooling mechanisms: Natural convection, Forced convection and closed loop cooling.
Electrical enclosure cooling becomes a necessity because research has shown that enclosure temperature rise on every 10^0C rise above normal room temperature decreases life of electrical components and its reliability is cut by half [1]. Hence maintaining enclosure temperature rise becomes a preliminary criterion.
Present work deals with both flow and thermal analysis using FLUENT (CFD) on an electrical enclosure consisting heat generating source dissipating heat of 150 W. Study comprises on calculating surface area of enclosure, Internal temperature rise in an enclosure, air flow requirement, selecting a fan for an enclosure, determining maximum velocity and temperature through numerical simulation and graphical representation of the results.
CFD is a powerful tool for investigating complex internal flow problems and in predicting the flow and temperature in an enclosure and representing results through color post-script.

II. Literature Survey

Literature survey has been conducted based on available journals and Industrial data sheets. Summaries of few important surveyed literatures are as below:
Hoffman, Pentair Company, [1], [2003], this technical manual is a ready reckoner for designing an electronic enclosure. Also this manual is helpful for engineers in preliminary design stage of any electronic enclosure in evaluating any kind/type of design aspects.

MahendraWankhede, et al, [2], [2010], Paper deals with CFD analysis of Aluminium enclosure. Enclosure consists of 100W heat generating PCBs. Paper concludes that use of internal fans reduces enclosure internal air temperature by 20- 25% compared to enclosure with no fans.

Through the Literature survey, found that many literatures are available on cooling of Chip and PCBs. However from the available resource and data's, effort has been made to carry out this work.

III. 3. Methodology

ANSYS products have flexibility in modeling, meshing and analysis, all combined in single software. Present analysis work carried using ANSYS FLUENT.
CFD process involves following steps:
i. Preprocessing- involves Geometry, Meshing and Boundary conditions

ii. Solver- involves Discretization by substitution and solution for algebraic equation
iii. Post processing- involves color postscript

Similarly the present analysis on electrical enclosure has been carried out following above steps

3.1 Modeling

Figure 1, shows the 3D model of electrical enclosure consisting Inlet, Exhaust fan/outlet and Heat block. The enclosure is of volume 300mm X 300mm X 300mm. Inlet is provided at mid of the front wall and exhaust fan is at mid of rear wall. The enclosure material is aluminum.

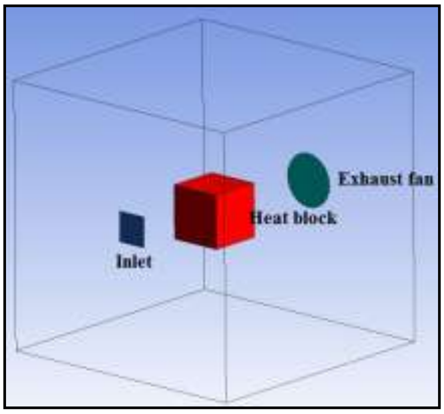

Figure 1: Electrical enclosure

3.2 Meshing

Figure 2 shows the hexa mapped mesh. Model has been meshed with 13512 elements and 14902 nodes. Figure 3 shows mesh convergence plot.

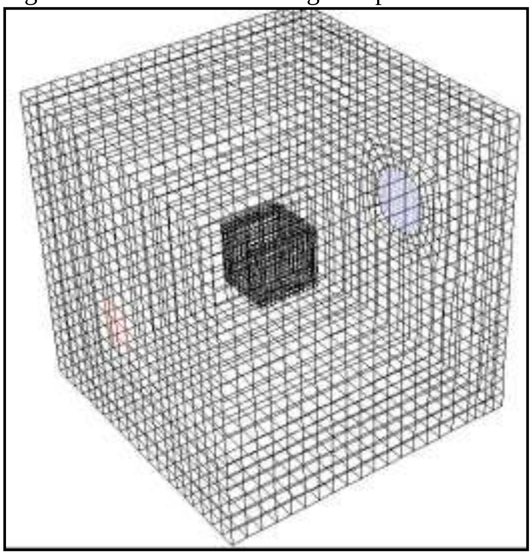
Figure 2: 3D Mesh model

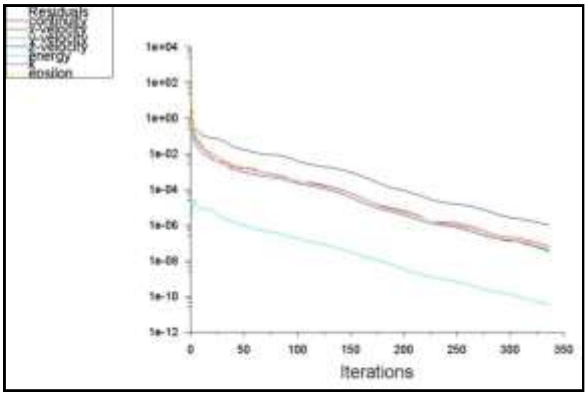

Figure 3: Mesh convergence plot

3.3 Boundary Conditions

Before applying Boundary conditions to the domain few parameters need to calculated and is followed below:

3.3.1 Calculating TSA of enclosure

Enclosure size: A x B x C= 300 mm X 300 mm X 300 mm
In feet,
TSA= 2(A x B) + 2(A x C) + 2(B x C)
TSA= 5.81 ft^2

3.3.2 Calculating air flow requirement in CFM and Fan selection

Enclosure internal heat dissipation is 150W. Hence from Figure 4, for 150 W heat dissipation: 4" fan is selected and enclosure ΔT is 6.5^{0}F (3^{0}C).

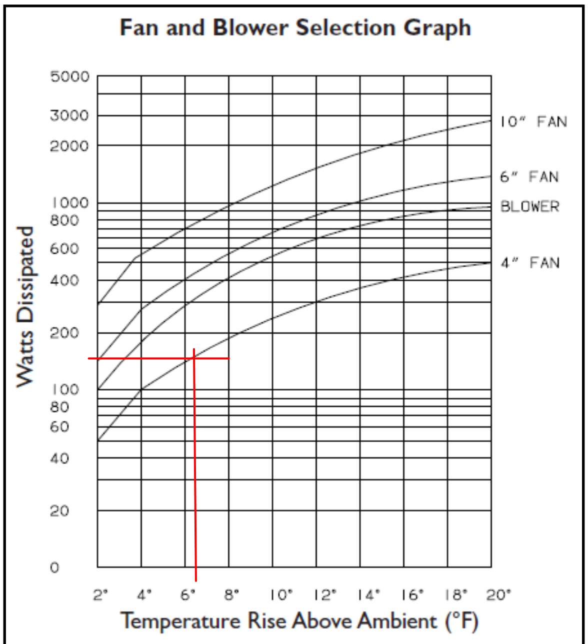

Figure 4: Fan selection chart [1]

Air flow requirement in CFM [1] = (3.16 x heat dissipated in Watts) / Temperature rise in ^{0}F

Therefore, Air flow required = (3.16 X 150)/ 6.5 = 72.92 CFM
Considering 25% safety margin [1],
= 1.25 X 72.92= 91.15 CFM= 92 CFM

Hence, CFM required for 4" fan is 92 CFM

Now Boundary conditions are applied based on above calculations:
Outlet: Exhaust fan with 92 CFM
Wall: Stationary and No slip condition
Block: Heat load of 150W
Thermal Condition: Coupled

3.4 Solution Controls

Flow is considered to be steady and turbulent. Turbulent model selected is standard K- epsilon model hence two equations: turbulent Kinetic energy and turbulent dissipation rate are solved. SIMPLE discretization scheme is used for solving momentum and turbulent equations.

3.5 Mathematical Models

Fluent code solves governing equations for each and every cell in the domain. The governing equations are:

i. Mass conservation equation
ii. Momentum conversation equation
iii. Energy conservation equation

Since flow is turbulent, another equation is solved that is K-epsilon model, it consists of two equations: turbulent Kinetic energy (eqn. a) and turbulent dissipation rate (eqn. b)

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

And

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

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 [5].

IV. Discussion of Results

The results obtained from the analysis are graphically represented through velocity and temperature contours plot which helps us to know maximum flow velocity and maximum temperature zones in an enclosure. The results are discussed below:

a) Velocity contour

Figure 5 shows the cut plane velocity contour. From the figure it is clear that the maximum velocity is found near inlet. The maximum velocity was found to be 36.51 m/s.

b) Temperature contour

Figure 6 shows the Temperature contour cut plane. Maximum temperature was found at surrounded surface of heated block and its value obtained is 82.06 ^{0}C.

c) Vector and Stream line plots

Figure 7 shows the vector plot and stream line plots of velocity and temperature. This signifies the distribution of air and direction of flow in the domain. The air flow is from left to right.

3.7 Results validation

Results obtained for the problem are converged results. However results obtained from analysis is compared with analytical calculations.

a) Temperature rise (ΔT):

Figure 8 shows the snap shot of FLUENT (CFD) result. It shows the temperature values at exhaust fan and inlet. The difference in those value gives ΔT and it is found to be 3 ^{0}C.

Area-Weighted Average Static Temperature	(k)
fan_1-side1	296.14371
inlet-side1	293.15002

Figure 8: Temperature rise- CFD result

From Figure 4, ΔT = 6.5 ^{0}F i.e. 3^{0}C. Hence CFD result is matching analytical approach for ΔT in an enclosure.

b) Checking for heat balance

Heat balance in here means heat input to the enclosure should be equal to heat lost from the enclosure. The total heat dissipated in an enclosure is 150 W, this is the amount of heat to be removed and heat balance to be maintained. Figure 9 shows the CFD result and it is evident that heat balance is maintained. Negative sign indicates that heat is removed.

Total Heat Transfer Rate	(w)
cabinet_default_side_minx-side1	0
fan_1-side1	100.48908
heat_block-side1	-150
heat_block-side1-1	150
inlet-side1	-250.48908
Net	-150

Figure 9: Total heat transfer rate- CFD result

c) Validating maximum velocity

Considered flow to be incompressible and steady
We know **92 CFM** is at Outlet,
Considering, Velocity inlet = Velocity outlet
= $[92 \times (0.305)^{3}] / 60 = 0.0435 m^3/s$
Now, 0.0435 / (0.03 x 0.03) = **48.33 m/s**

From Figure 5, CFD result for Maximum velocity was found to be 36. 51 m/s whereas analytically obtained maximum velocity at inlet is 48.33 m/s. The obtained result is validating and signifying that the CFD result is converging towards analytical result.

From all the above comparisons it is evident that the domain is maintained for both mass and heat balance and also for ΔT in an enclosure.

V. CONCLUSIONS

CFD analysis has been carried on Electrical enclosure consisting heat block, dissipating heat of 150 W. The heat block is cooled through forced convection by providing adequate amount of air flow in an enclosure. The following conclusions that can be drawn from the results obtained are:

i. CFD is the powerful and effective tool for Thermal management of Electronic & Electrical enclosures
ii. Work is a ready reckoner for engineers in helping them on how to: select a fan, calculate air flow requirement and temperature rise for an enclosure based on both analytical and CFD methods
iii. Temperature rise with fan placed at the optimum location was found to be 3 ^{0}C which is well below the threshold limit.

Present work is no means exhaustive; always there is a scope for further optimization.

REFERENCES

[1] Hoffman- A Pentair Company, "Heat dissipation in Electrical enclosure", "Technical information on Thermal Management of Electrical enclosures", ©2003 Hoffman Enclosures Inc.
[2] MahendraWankhede, VivekKhaire, Dr. AvijitGoswami 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
[3] Bud Industries, Inc., "Enclosure Design Tips Handbook", July 2007, © Bud Industries Inc.
[4] "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/
[5] ANSYS Fluent User Guide, Release 15.0, Nov. 2013 and ANSYS Fluent 12.0, Theory guide, April 2009
[6] H K Versteeg and W Malalasekara, "Computational Fluid Dynamics- Finite Volume Method", Text book of CFD, edition 1995
[7] Younus. A. Cengel, "Text book of Heat transfer- chapter 15- cooling of electronic equipments"

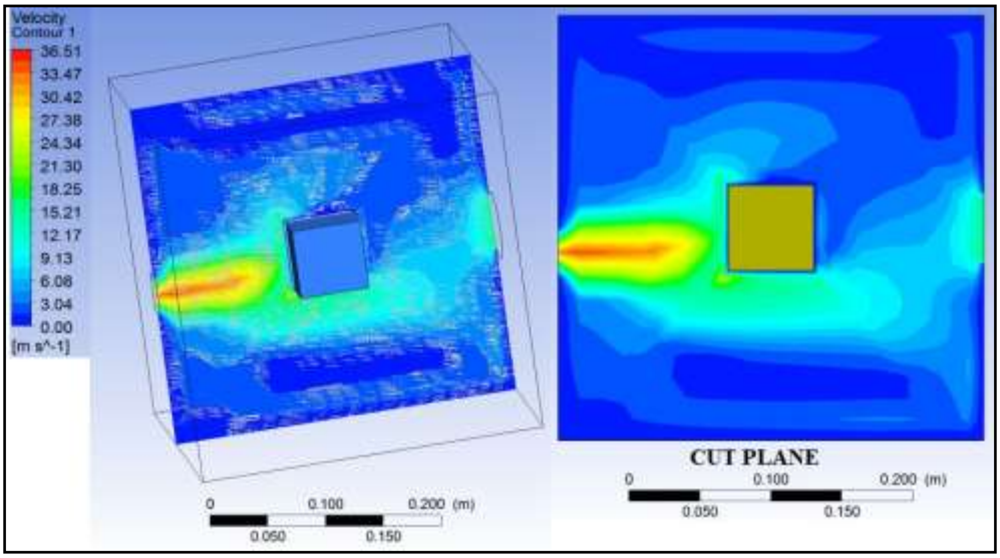

Figure 5: Velocity contour plot

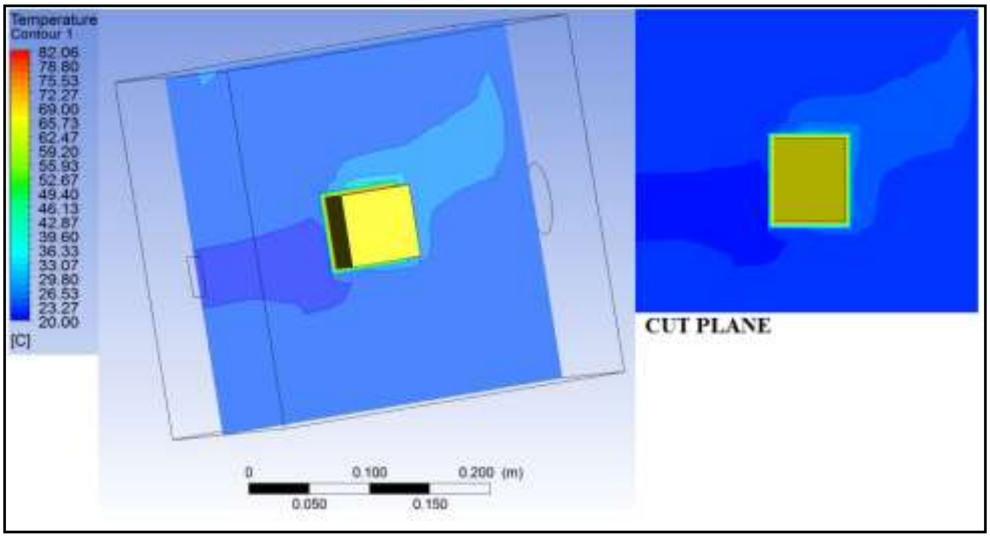

Figure 6: Temperature contour plot

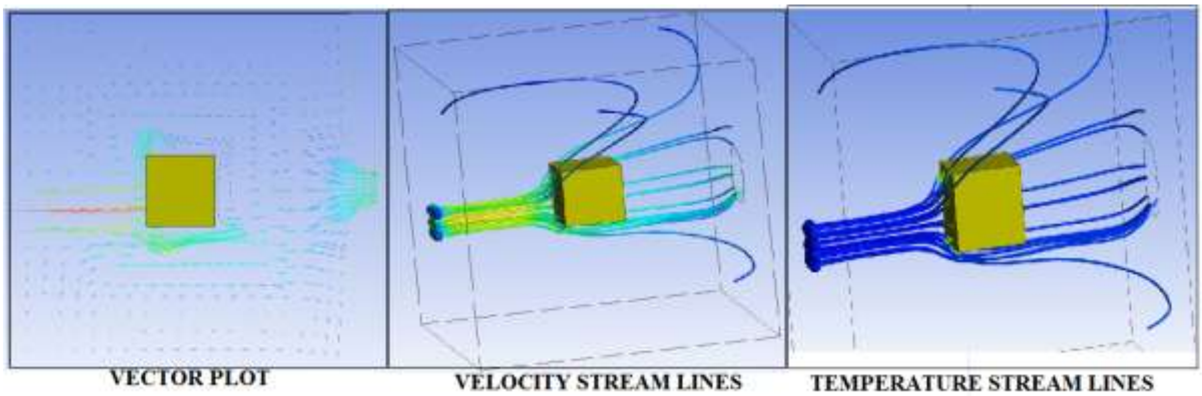

Figure 7: Vector and stream line plots