International Journal of Scientific Research in Science, Engineering and Technology (www.ijsrset.com)
© 2020 IJSRSET | Volume 7 | Issue 2 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
DOI : https://doi.org/10.32628/IJSRSET207219

Optimization study on Electrical Enclosure with Forced Convection cooling using Computational Fluid Dynamics

[1]Mohammed Nabeel Ahmed,[2] B. Meghanath,[3]Mayur Laxman Kesarkar, [4]Lakshminarasimha. N
[1,2,3]UG Scholar, Department of Mechanical Engineering, New Horizon College of Engineering, Bangalore, India
[4]Assistant Professor, Department of Mechanical Engineering, New Horizon College of Engineering, Bangalore, India

ABSTRACT

Any electronic walled in area comprises of heat creating electronic components, as heat produced by the electronic parts in a fenced in area decreases the life of electronic segments prompting serious harm or disappointment of the framework. Research shows that each 10°c rise above room temperature of the enclosure, the life of the electronic parts decreases. Thus for any electronic frameworks, cooling turns into a significant structure interest, practical and ideal answer for hold the electronic parts to its working limit. Therefore, in the present work CFD simulation has been carried out using ANSYS Fluent by considering a typical Aluminum Electrical enclosure of volume (150mm X 600mm X 250mm) with total internal heat dissipation of 84W. 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. Also optimization study has been carried out by changing the inlet opening position, exhaust fan location and providing baffle at inlet opening location. The results obtained from analysis are validated with analytical results.

Keywords : CFD, Electrical enclosure, air flow requirement, optimization, temperature rise, ANSYS Fluent, heat dissipation.

I. INTRODUCTION

Electrical walled in areas are typically produced using unbending plastics or metals, for example, steel, treated steel, or aluminum. Steel cupboards might be painted. For plastic fenced in areas abs is utilized for indoor applications not in mean situations. Polycarbonate, glass-strengthened, and fiber glass boxes are utilized where more grounded cupboards are required, and may also have a gasket to reject remains and damp. Metal cupboards may meet the conductivity necessities for electrical security holding and protecting of encased hardware from electromagnetic impedance. Non-metallic fenced in areas may require extra establishment steps to guarantee metallic channel frameworks are appropriately reinforced.

Electrical equipment is almost always specified by manufacturers for operation within an optimal temperature range in which the equipment's consistency, performance, efficiency, and physical integrity are protected. At higher temperatures, the efficiency of equipment decreases and component damage or even failure may result. A modern VFD (Variable Frequency Drive) typically has an efficiency of 93-97 percent and is designed for operation below 40 °C (104 °F), For every degree increase above the operating temperature, efficiency is decreases by 2 percent. In addition to an increased risk of equipment failure. Fortunately, this added cost, which can range from tens thousands to hundreds thousands of dollars, can be avoided with a little foresight by including a temperature control system in specifications.

The lifetime and disappointment free activity time of electrical device is firmly directed by natural conditions and the mode of activity. Every part or sort of electrical device has its own greatest operating temperature as indicated by the maker.

- High working temperature can expand the potential to device activity. A portion of the normal dangers related with high inside warmth are:
- Sudden Hardware Shutdown – A few makers program hardware segments to flip off at high temperatures.
- Weakened hardware execution– High temperature can cause undependable hardware execution, regularly heavy to inconvenient outcomes.

Research plainly shows that for each 18°F rise over the ordinary temperature of a room, the steady quality of electronic parts drops significantly. When an appropriately measured fenced in area for your equipment is chosen, you can choose the correct cooling framework for keeping up the correct temperature. A remarkably large cooling framework will set too every now and again, and will knock up working costs. Excessively little of a framework won't give the correct cooling required inside an enclosure and means you could be gambling decreased existence of segments or even a failure and has normal issues including:

Losing efficiency – Oversized cooling frameworks cool the space too quickly making the framework work all the more every now and again and consume power. Cycling on and off more often wears on the whole thing.

Shorter framework life – Undersized cooling frameworks that don't address the issue will enable heat to rise above peak working temperatures.

Controlling dampness –introducing an inappropriate size forced air system could enable the stickiness inside the nook to rise. Dampness prompts erosion in the enclosure and that will take out costly equipment. Furthermore, many climate control systems don't have the correct transfer structure when dampness is expelled.

Present work deals with both flow and thermal analysis using FLUENT (CFD) on an electrical enclosure consisting heat generating source dissipating heat of 84 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

R.Boukhanouf., et., al., [1], [2010]

This paper presents results of CFD study of an electronics cooling enclosure used as part of a larger telecommunication radar system. An original cooling enclosure was replicated using Flotherm which results were taken as the standard thermal performance. It was found that the operating temperature of one of the Radio Frequency (RF) apparatus will exceed the design temperature limit of the PCB. A solution involving a re-design of thermal spreading arrangements using a 3 mm thick copper shelf and a Vapor Chamber (VC) heat pipe was found to bring the operating temperatures of all RF components within the specified temperature limits. The use of a VC, in particular, reduced the 60 W RF component steady state temperatures by an average of 5.4 °C. The study also shows that increasing the finned heat exchanger cooling air flow rate can lower further the RF components temperature though at the expense of increasing energy consumption of the fan.

R. Mohan., et., al., [2], [2010]

The computational fluid dynamics is concentrated on the forced air cooling of the CPU. This paper uses CFD to recognize a cooling arrangement, which utilizes a 80 W CPU greatest while this number will be expanded in the scope of 70-120W in the expected work station frameworks. The structure can cool the undercarriage with one fan with air blowing over warmth sink connected to the CPU is sufficient to cool the entire framework and the power supply fan. The well converged, grid independent and well posed simulations are performed, and the results are compared with the experimental data.

Hoffman Pentair Company., [3], [2003]

This engineering technical data sheet helps in calculating the enclosure temperature rise, air flow requirements and also in choosing fan for an enclosure and it highlights about the effect of temperature on surface parameters of the enclosure.

Lakshminarasimha N., [4], [2015]

Study involves mathematical investigation on electrical enclosure with forced convection cooling consist 150W heat power source. Study has analytical calculations for airflow requirements and enclosure temperature rise.

III. METHODOLOGY

Ansys products have adaptability in geometry, meshing and calculations, all the features are combined in one software, present work is carried out in following steps using Ansys Fluent.

- Building a model using ANSYS workbench as per the dimensions
- Meshing the model
- Applying the boundary conditions
- Computational analysis and plotting convergence plot
- Plotting velocity vectors
- Theoretical analysis
- Visualization

3.1 Modelling

The 3D model of electrical enclosure consisting Inlet, Exhaust fan and heat source is as shown in Fig. 1. The enclosure is of volume 150mm X 600mm X 250mm. Inlet is provided at the center of the top wall and exhaust fan is provided at the bottom wall. The enclosure material is Aluminum.

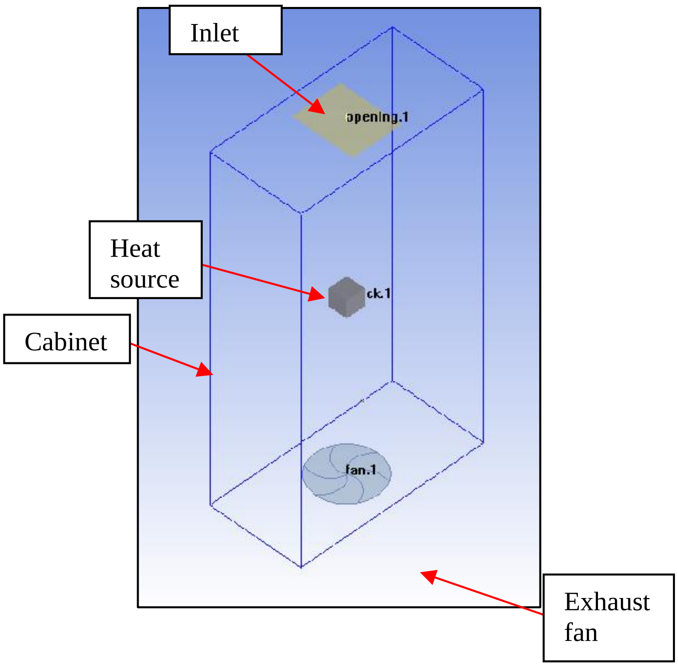

Fig 1 : Electrical Enclosure

3.2 Meshing

Mesh independence test was carried out by meshing the model with different Mesh element size and plotted for two important parameter i.e. velocity and temperature as shown in Fig 3(b) and Fig 3(c). From the plot it is clear that velocity and temperature becoming constant for 57744 mesh elements.

Hence model with 57744 elements has been chosen for all the studies to obtain the required result. Fig.2(a) and Fig.2(b) shows the 3D Hexa mesh model and Fig.3(a) shows mesh convergence plot for the continuity, x-velocity, y-velocity, z-velocity, k and epsilon.

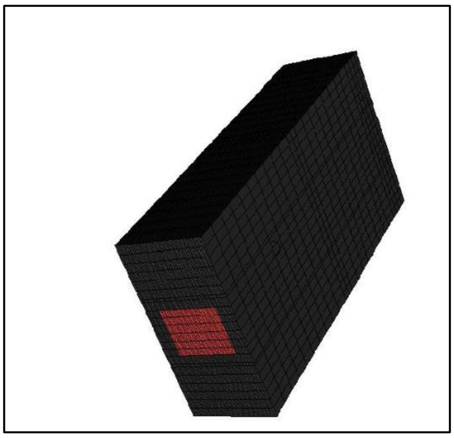

Fig 2(a) : 3D Hexa mesh model- view-1

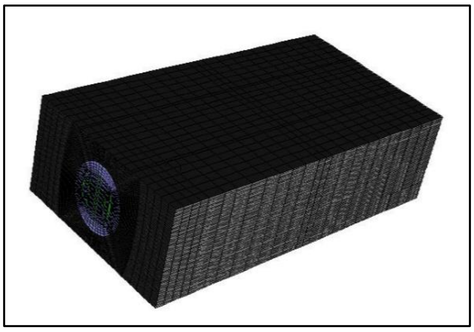

Fig 2(b) : 3D Hexa mesh model-view-2

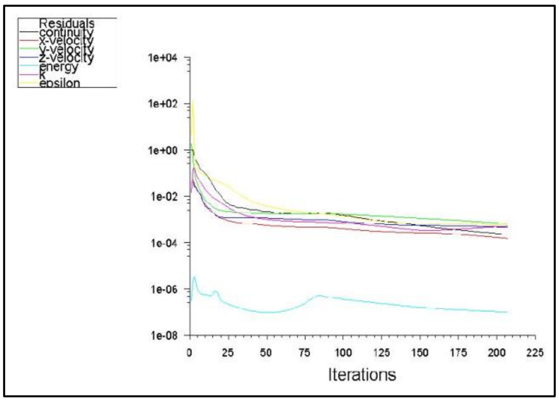

Fig 3(a) : Mesh convergence plot

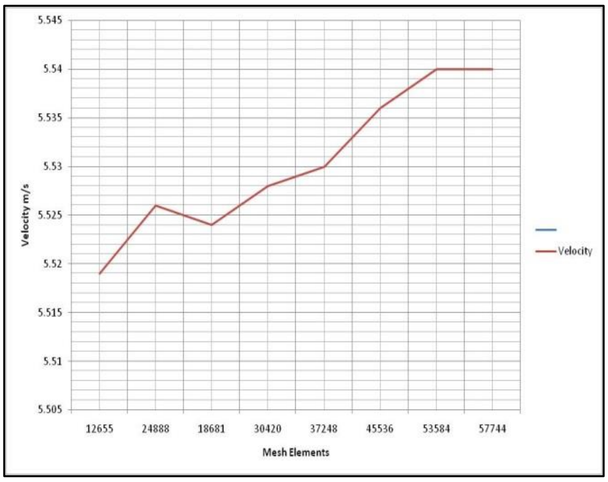

Fig 3(b): Mesh independence plot (Velocity vs no. of mesh element)

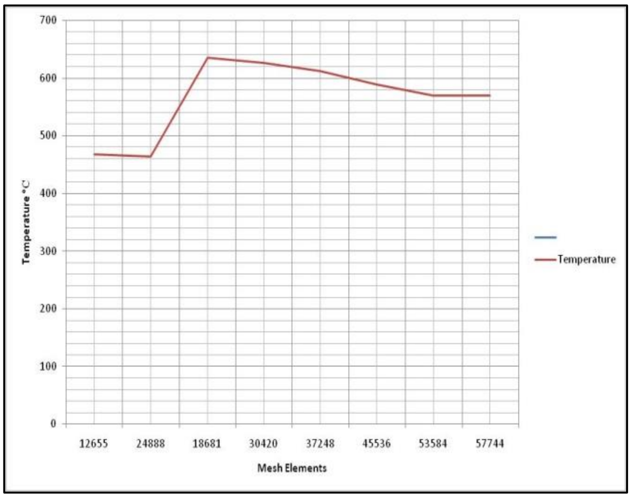

Fig 3(c) : Mesh independence plot (Temperature vs no of mesh element)

3.3 Boundary Conditions

The Boundary applied for the problem is discussed in this section.

3.3.1 Calculating air flow requirement in CFM and fan selection

Flow is considered to be incompressible, steady and turbulent flow. Standard k-epsilon model is used to solve the flow equation.

Total internal heat dissipation in enclosure is 84Watts. From Fig.4, for 84Watts heat dissipation 4” fan is selected and enclosure ΔT is 3.5˚F.

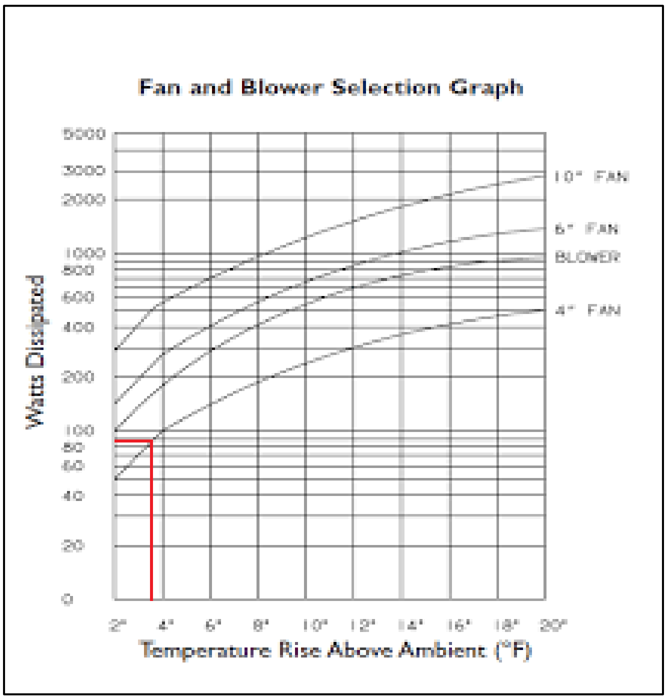

Fig4 : Fan selection chart [3]

Air flow requirement in CFM [3] = (3.16*Heat dissipated in Watt)/Temp rise in ˚F

From graph, Temp rise in ˚F= 3.5˚F or 1.556˚C and Heat dissipated in Watt= 84Watt

= (3.16∗84)/3.5 =75.84 CFM

From literature, considering safety margin of 25% [3]

Therefore, 75.84*1.25= 94.8 CFM

Hence, CFM required for 4" fan is 95 CFM.

Based on above calculations, boundary conditions are applied:

Outlet: Exhaust Fan of 95 CFM

Block: Heat load of 84 Watts

Wall: stationary and no slip condition.

3.4 Mathematical models

Fluent solves governing equations for each and every mesh element in the domain. The governing equations are

1. Mass conservation equation
2. Momentum conservation equation.
3. Energy conservation equation.

IV. DISCUSSION OF RESULTS

The outcomes acquired from the investigations are graphically represented through velocity and temperature plots which help to know most extreme flow velocity and temperature in an enclosure. The results are as discussed below in this section.

a) Velocity Streamline plot

The velocity streamline plot is as shown Fig.5. It was found that maximum velocity found near opening and is 5.5401m/s.

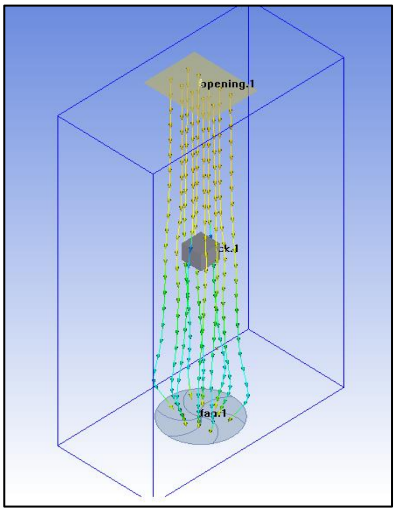

Fig.5 : Velocity Streamline plot

b) Temperature contour

The temperature contour cut plane shows maximum temperature around the heat source is found to be 568.685˚C as shown in Fig.6.

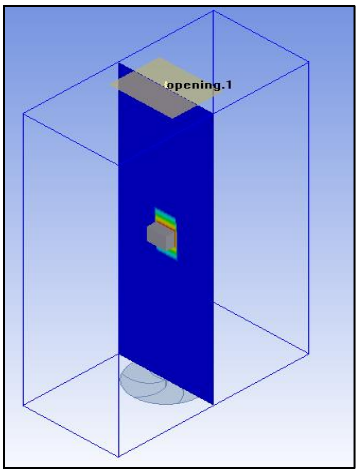

Fig.6 : Temperature contour cut plane

4.1 Results validation

Results obtained for the problem are converged results. However, results obtained from analysis are compared with analytical calculation.

a) Temperature rise

The temperature rise is the difference of exhaust fan and opening and is as shown in snapshot below.

Area-Weighted Average Static Temperature	(k)
opening_1-minz	293.14999

Area-Weighted Average Static Temperature	(k)
fan_1-miny	294.75796

Therefore, temperature at fan is equal to 294.7515K and opening temp is equal to 293. 149K. Therefore ΔT is equal to 1. 5815˚C. Now from theoretical calculation ΔT is equal to 1.556˚C *(See Section 3.3.1).* Hence CFD results are matching for ΔT of an enclosure.

b) Validating maximum velocity

Consider flow to be incompressible and steady flow, we know that, 95 CFM at the outlet *(See Section 3.3.1)*
Considering, velocity inlet equal to velocity outlet
Since $A_1=A_2$

$V_1=V_2=\frac{94.8*(0.305)^3}{60}= 0.0448\ m^3/s$

Therefore$V_1=V_2= \frac{Q}{A}$,the opening area 100mm*80mm

$V_1= V_2= \frac{0.0448}{0.1*0.08} = \mathbf{5.6}\boldsymbol{m/s}$

Therefore, from analysis, velocity found to be equal to 5.546 m/s as shown in snapshot below. Hence the analysis and analytical calculation is validated.

Area-Weighted Average Velocity Magnitude	(m/s)
opening_1-minz	5.5465607

c) Mass flow rate

From theoretical calculation

m= ρav

=1.2 *(0.1*0.08) *5.6

= **0.0537Kg/s**

From analysis, as shown in snapshot below the mass flow rate is found to be equal to **0.05196 Kg/s.**

Mass Flow Rate	(kg/s)
opening_1-minz	0.051961768
Net	0.051961768

From all the above comparisons it is evident that the analysis and analytical calculations are balanced.

4.2 Optimization study

Optimization study has been carried out by changing the inlet opening position, exhaust fan location and providing baffle at inlet opening location. The results obtained for the same is as discussed below in this section.

Case 1: Position of inlet

a) Left position: The position of inlet is changed to the left face of the model as shown in Fig.7.

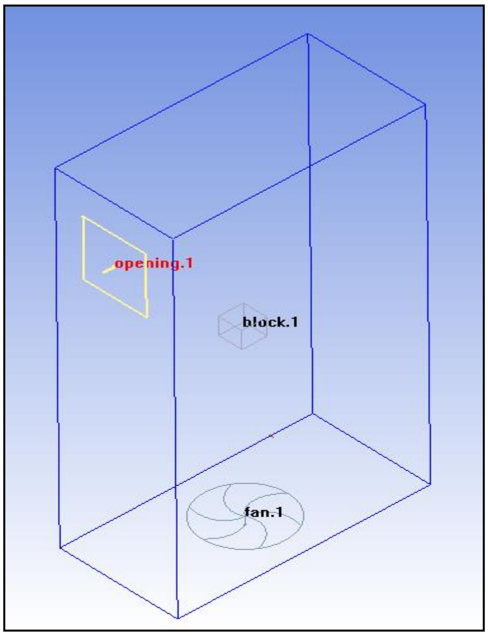

Fig.7 : Left inlet position

The results obtained for the inlet position-Left are:
Max. Temp =565˚C

Mass Flow Rate	(kg/s)
opening_1-minz	0.051961768
Net	0.051961768

Area-Weighted Average Velocity Magnitude	(m/s)
opening_1-minz	5.5465607

Area-Weighted Average Static Temperature	(k)
opening_1-minz	293.14999

Area-Weighted Average Static Temperature	(k)
fan_1-miny	294.75796

b) Right Position: The position of inlet is changed to the right face of the model as shown in Fig.8.

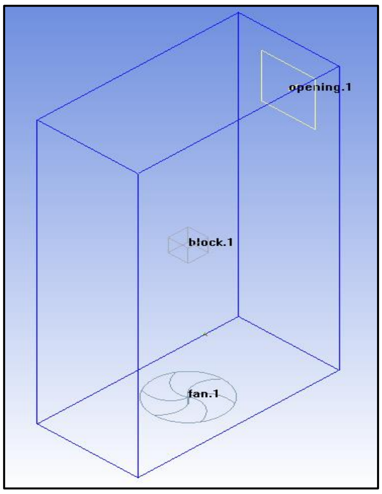

Fig.8 : Right inlet position

The results obtained for the inlet position-Right are:
Max. Temp =567.649˚C

Mass Flow Rate	(kg/s)
opening_1-minz	0.051961396
Net	0.051961396

Area-Weighted Average Velocity Magnitude	(m/s)
opening_1-minz	5.5465329

Area-Weighted Average Static Temperature	(k)
opening_1-minz	293.14999

Area-Weighted Average Static Temperature	(k)
fan_1-miny	294.75759

Case 2: Exhaust fan position

Exhaust fan position is being changed to the left face as shown in Fig.9.

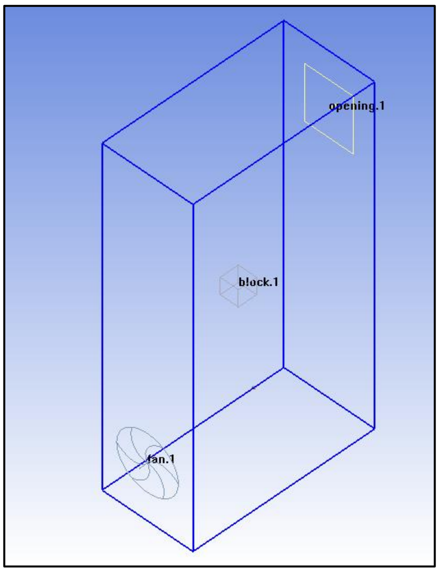

Fig.9 : Exhaust fan position

The results obtained for the exhaust fan position-Left are:
Max. Temp=570 ˚C

Mass Flow Rate	(kg/s)
opening_1-minz	0.051961698
Net	0.051961698

Area-Weighted Average Velocity Magnitude	(m/s)
opening_1-minz	5.5464109

Area-Weighted Average Static Temperature	(k)
opening_1-minz	293.14999

Area-Weighted Average Static Temperature	(k)
fan_1-minz	294.76111

Case 3: Providing Baffle

Baffles are provided at the inlet location as shown in Fig.10.

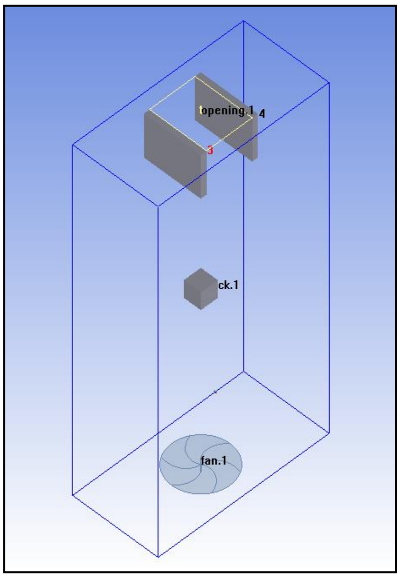

Fig.10 : Baffle position

The results obtained for the Baffle provided at the inlet are:

Max Temp=576.2 ˚C

```
                    Mass Flow Rate                  (kg/s)
-------------------------------- --------------------
                    opening_1-miny          0.051959231
-------------------------------- --------------------
                               Net          0.051959231

              Area-Weighted Average
                 Velocity Magnitude                   (m/s)
-------------------------------- --------------------
                    opening_1-miny            5.5483295
              Area-Weighted Average
                 Static Temperature                     (k)
-------------------------------- --------------------
                    opening_1-miny               293.15

              Area-Weighted Average
                 Static Temperature                     (k)
-------------------------------- --------------------
                         fan_1-miny            294.76195
```

The consolidated results of optimization study are as shown in Table. 1

Table.1 : Results of optimization study

CASES	ΔT(˚C)	V(m/s)	m(Kg/s)	T_{max}(˚C)
Without Baffle	1.5815	5.5401	0.051961	568.685
Inlet Position 1	1.6079	5.5456	0.051961	565
Inlet Position 2	1.6076	5.5465	0.051961	567.649
Exhaust Fan	1.6112	5.5464	0.051961	570
With Baffle	1.6119	5.5483	0.051959	576.2

V. CONCLUSION

CFD analysis has been carried on electrical enclosures consisting of heat source which dissipate 84Watts. The heat block is cooled through forced convection. The following conclusions that can be drawn from the results obtained are

3. CFD is very powerful tool to determine the maximum velocities, temperature difference, mass flow rate, maximum temperature.
4. Optimization study have been carried out to understand the effect of following parameters.
   - Location of opening.
   - Location of exhaust fan.
   - Providing baffle.
5. It can be concluded that after optimization study, inlet position-1 i.e. left position is recommended and the temperature rise limit is below threshold the limit.
6. 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 to make decisions faster. Further, they will aid in preparing a ready reckoner for cooling and ventilation design.

VI. ACKNOWLEDGEMENT

We take this opportunity to express our gratitude to the people who have been instrumental in the successful completion of this project.

We would like to thank our Dean and HOD, **Dr.M.S.Ganesha Prasad** for guiding us in every step. The guidance and support received from all teaching and non-teaching staff of Mechanical Engineering department, **New Horizon college of Engineering** who contributed to this project. We are grateful for their constant support and help.

We are also grateful to our project guide **Prof.Lakshminarasimha.N**, Assistant Professor, Mechanical Engineering Department, New Horizon college of Engineering, for his tremendous support and help. Without his encouragement and guidance this project would not have materialized.

VII. REFERENCES

[1]. R. Boukhanouf and A. Haddad., "A CFD Analysis of an electronics cooling enclosure for application in telecommunication systems", "Applied Thermal Engineering-Elsevier", volume 30, Issue 16, Nov. 2010, pp. 2426-2434, ISSN: 1359-4311.

[2]. R. Mohan and Dr. P. Govindarajan., "Thermal Analysis of CPU with variable Heat Sink Base Plate Thickness using CFD", "International Journal of the Computer, the Internet and Management", Vol. 18, issue 1, Jan. 2010, pp. 27-36, ISSN: 0858-7027.

[3]. Hoffman- A Pentair Company., "Heat dissipation in Electrical enclosure", "Technical information on Thermal Management of Electrical enclosures", 2003.

[4]. Lakshminarasimha N., "Numerical Investigation of forced convection cooling of Electrical enclosure using CFD", "Journal of Engineering Research and Applications", Vol. 5, Issue 11, (Part - 5) Nov. 2015, pp.62-66, ISSN: 2248-9622

Cite this article as :

Mohammed Nabeel Ahmed, B. Meghanath,Mayur Laxman Kesarkar, Lakshminarasimha. N, "Optimization study on Electrical Enclosure with Forced Convection cooling using Computational Fluid Dynamics", International Journal of Scientific Research in Science, Engineering and Technology (IJSRSET), Online ISSN : 2394-4099, Print ISSN : 2395-1990, Volume 7 Issue 2, pp. 34-42, March-April 2020. Available at doi : https://doi.org/10.32628/IJSRSET207219

Journal URL : http://ijsrset.com/IJSRSET207219