[Bhasker, 5(1): January, 2016]
ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

COMPUTATIONAL FLUID DYNAMIC ANALYSIS OF GROUND COMMUNICATION CABINET SYSTEM

B.Bhasker*, J.Nithin Kumar
*Asst.Professor,Department of mechanical engg, Geethanjali college of engg and tech., Hyderabad, India
Asst.Professor,Department of mech. engg, Geethanjali college of engg and tech., Hyderabad, India

ABSTRACT

Thermal design of a ground communication cabinet is taken up in this project with a view to keep the temperature below 100^0 C. Execution of this task needs information about heat transfer coefficient in forced convection environment in order to compute the maximum temperature against a specified heat load. Major complexity associated with heat transfer coefficient is not being a constant value like thermal conductivity. In fact heat transfer coefficient is governed by many factors like shape, orientation of the flow surface, etc. However empirical correlations are available for cabinets having regular shapes. But all practical cabinets will have irregular shapes for which only alternative left is to estimate the heat transfer coefficient through Computational Fluid Dynamics (CFD) software. Once again to make use of CFD software the designer should procure the software which will be a costly affair and further he should attain expertise. To get rid of this troublesome situation an analytical method will be established to estimate the heat transfer coefficient and the same will be validated with commercial CFD software package. Further maximum temperature will be estimated using the so evolved heat transfer coefficient As a part of the design process selection of fan will also be accomplished. The outcome of this project would be an analytical method which any thermal designer can make use of as a hand calculator to quickly arrive at the cooling scheme without depending on CFD software.

KEYWORDS: forced convection, thermal conductivity, CFD, temperature, PCB’s.

INTRODUCTION

1.1 General

During their course of application ground communication cabinets will be powered up to energize their internal subsystems viz. ICs positioned on PCBs. Though the primary objective of powering up ICs is to enable them to deliver their function, according to joules’ law of heating which states that when an electric current is passed through any resistor will in turn generate heat. The effect of this heat is raising the temperature of the component mounted on PCB. Each component will have allowable temperature specified by the manufacturer crossing which will result into either malfunctioning of the component or permanent failure of the component. As the total reliability of the ground communication cabinet is highly dictated by the component, a cooling mechanism is essentially required for an ground communication cabinet. Compared to other modes of heat transfer, forced convection means will provide faster cooling rates. This project deals with designing a cooling scheme for an ground communication cabinet in forced convection environment. During design the mass flow rate of cooling medium is required to be estimated in the initial stage. Later on heat transfer coefficient is required to be estimated using which the maximum surface temperature of the component will be calculated to ascertain that the temperature is with in limits with the designed mass flow rate of cooling medium. In the whole exercise estimation of heat transfer coefficient is a tedious task as standard literature mentions about standard geometries like plate, cylinder etc but not about this kind complex geometries like enclosures with PCBs. An analytical method will be established during this project in which an expression for heat transfer coefficient. This expression will be validated with that of evaluated using commercial CFD software package. The outcome of this project would be an analytical method which any thermal designer can make use of as a hand calculator to quickly arrive at the cooling scheme with out depending on CFD software.

1.2 *construction details*

The solid model of the ground communication cabinet in assembled configuration is shown in Figure 1.2.

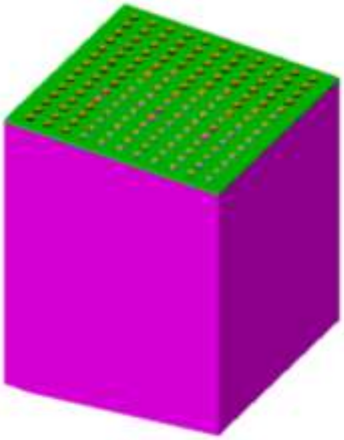


Figure 1.2.1Solid model of the ground communication cabinet

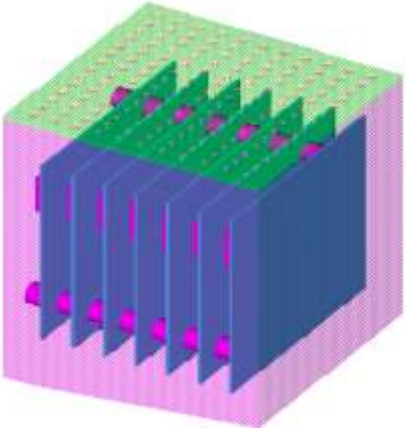
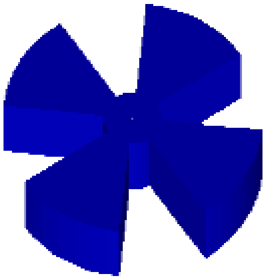


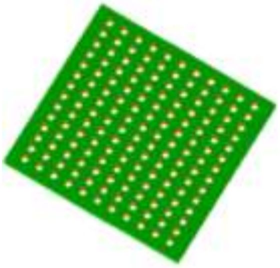
Figure 1.2.2 Solid model of the ground communication cabinet (Transparentview)

The solid model of the ground communication cabinet in transparent view (Housing and cover) showing internal sub systems is shown in Figure 1.2. The solid model of the ground communication cabinet in exploded view is shown in Figure 1.2.2.

These PCBs will be inserted in to the housing from topside through guides provided integral to the housing as shown below and then held in position by locking them.Cover encloses the housing and series of vent (Air outlet) holes are provided on cover as shown below.

Housing consists of a foot - print (Hole) for inserting the inlet fan, which is shown below.





1.3 *Printed Circuit Boards (PCBs)*

A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, tracks or traces etched from copper sheets laminated onto a non-conductive substrate. It is also referred to as printed wiring board (PWB) or etched wiring board. A PCB populated with electronic components is a printed circuit assembly (PCA), also known as a printed circuit board assembly (PCBA). PCBs are inexpensive, and can be highly reliable. They require much more layout effort and higher initial cost than either wire-wrapped or point-to-point constructed circuits, but are much cheaper and faster for high-volume production.

The inventor of the printed circuit was the Austrian engineer Paul Eisler (1907–1995) who, while working in England, made one circa 1936 as part of a radio set. Around 1943 the USA began to use the technology on a large scale to make rugged radios for use in World War II. After the war, in 1948, the USA released the invention for commercial use. Printed circuits did not become commonplace in consumer electronics until the mid-1950s, after the Auto-Sembly process was developed by the United States Army. Before printed circuits (and for a while after their invention), point-to-point construction was used. For prototypes, or small production runs, wire wrap or turret board can be more efficient. Predating the printed circuit invention, and similar in spirit, was John Sargrove's 1936-1947 Electronic Circuit Making Equipment (ECME) which sprayed metal onto a Bakelite plastic board. The ECME could produce 3 radios per minute.

Originally, every electronic component had wire leads, and the PCB had holes drilled for each wire of each component. The components' leads were then passed through the holes and soldered to the PCB trace. This method of assembly is called through-hole construction. In 1949, Moe Abramson and Stanislaus F. Danko of the United States Army Signal Corps developed the Auto-Sembly process in which component leads were inserted into a copper foil interconnection pattern and dip soldered. With the development of board lamination and etching techniques, this concept evolved into the standard printed circuit board fabrication process in use today. Soldering could be done automatically by passing the board over a ripple, or wave, of molten solder in a wave-soldering machine. However, the wires and holes are wasteful since drilling holes is expensive and the protruding wires are merely cut off.

In recent years, the use of surface mount parts has gained popularity as the demand for smaller electronics packaging and greater functionality has grown.Conducting layers are typically made of thin copper foil. Insulating layers dielectric are typically laminated together with epoxy resin prepreg. The board is typically coated with a solder mask that is green in color. Other colors that are normally available are blue and red. There are quite a few different dielectrics that can be chosen to provide different insulating values depending on the requirements of the circuit. Some of these dielectrics are polytetrafluoroethylene (Teflon), FR-4, FR-1, CEM-1 or CEM-3. Well known prepreg materials used in the PCB industry are FR-2 (Phenolic cotton paper), FR-3 (Cotton paper and epoxy), FR-4 (Woven glass and epoxy), FR-5 (Woven glass and epoxy), FR-6 (Matte glass and polyester), G-10 (Woven glass and epoxy), CEM-1 (Cotton paper and epoxy), CEM-2 (Cotton paper and epoxy), CEM-3 (Woven glass and epoxy), CEM-4 (Woven glass and epoxy), CEM-5 (Woven glass and polyester). Thermal expansion is an important consideration especially with BGA and naked die technologies, and glass fiber offers the best dimensional stability.

1.3.1 thermal design in previous ground Communication cabinets

In previous ground communication cabinets heat transfer coefficient used to be estimated using commercial CFD software packages. Procurement of CFD software packages is a costly affair which is not essentially required just to find out heat transfer coefficient.

Thermal design with proposed scenario

The proposed analytical method replaces the need of using CFD software for the present requirement.

METHODOLGY

2.1 convection heat transfer

Convection is the mode of heat transfer between a surface and a fluid moving over it. The energy transfer in convection is predominantly due to the bulk motion of the fluid particles, though the molecular conduction within the fluid itself also contributes to some extent. If this motion is mainly due to the density variations associated with temperature gradients within the fluid, the mode of heat transfer is said to be due to free or natural convection. On the other hand if this fluid motion is principally produced by some superimposed velocity field (like a fan, a blower or a pump), the energy transport is said to be due forced convection.As the motion of the fluid plays an important role in convective heat transfer, knowledge of the dynamics of fluid flow is essential for determination of the temperature fluid flow. The types of fluid flow relevant to convective heat transfer are the flow of viscous fluids over objects (external flows) and that of fluids through pipes or tubes (internal flows). Velocity distribution in a fluid flow and the pressure drop experienced by fluid as it flows through a certain length would be of primary interest. The requirement of the conservation of mass, momentum and energy forms the basis of the analysis for fluid flow.

2.1.1 *convective heat transfer coefficient*

The local heat flux, q, for an arbitrary shaped surface of area A at temperature T_s over which flows a fluid of velocity V and of temperature T_α is given by

$$q = h (T_s - T_\alpha)$$

where h is the local heat transfer coefficient

This equation is referred to as Newton's Law of cooling. The simplicity of the above equation is misleading because the convective heat transfer coefficient is actually a complicated function of the nature of fluid flow, thermal properties of the fluid and the configuration of the system. Due to the variation of flow conditions from point to point, the values of q and h along the surface also vary and that is why the adjective local will be applied to them. The total heat transfer rate may be obtained by integrating the above equation over the entire surface, assuming a uniform value of T_s

$$Q = \int_A q\, dA = (T_s - T_\alpha) \int_A h\, dA$$

Defining h* as the average or total heat transfer coefficient for the entire surface, above equation can be rewritten as

$$q = h^* A (T_s - T_\alpha)$$

Comparing above two equations, the average and local convection coefficient are related by

$$h_A^* = \frac{1}{A} \int_A h\, dA$$

2.2 *The boundary-layer concept*

The concept of a boundary layer as produced by Prandtl forms the starting point for the simplification of the equations of motion and energy. In this concept, the flow field over a body is divided into two regions:

1. A thin region near the body, called the boundary layer, where the velocity and temperature gradients are large.
2. The region outside the boundary layer where velocity and temperature gradients are very nearly equal to their free stream values.

The thickness of the boundary layer has been arbitrary defined as the distance from the surface at which the local velocity (or temperature) reaches 99% of the external velocity (or temperature). In general, both the velocity boundary layer and thermal boundary layer will exist simultaneously.

2.3 features of computational fluid dynamics(cfd)

The following features, which are required to meet the present requirement, are explored.

Sl. No.	Feature	Purpose
1.	Thermal boundary condition	To specify the heat load
2.	Surface property	To specify the surface roughness of PCB
3.	Flow surface	To specify surfaces which convect heat to fluid
4.	Fan	To specify the cooling medium parameters
5.	Vent	To specify air flow outlet
6.	Ambient conditions	To specify ambient temperature of air

2.4 Supported Geometry and Elements

The following geometry and element types support Thermal Boundary Conditions:

1. Surfaces (usually meshed with 2-D shell elements) support all boundary condition types (material properties must be non-fluid). Note that Heat Generationboundary condition shell elements must have non-zero thickness.
2. Edges (usually meshed with 1-D beam elements) support all Thermal Boundary Condition types (material properties must be non-fluid)
3. Volumes meshed with 3-D solid elements support Temperature, Heat Load , Heat Generation, Total Current, Current/element and Voltage boundary condition types.
4. Lumped Mass elements support all boundary condition types. Boundary conditions created on Lumped Mass elements are non-associative.
5. Surfaces meshed with Axisymmetric Solid elements support Temperature, Heat Load , Heat Generation, Total Current, Current/element and Voltageboundary condition types. (See Axisymmetric Modeling for details.)
6. Edges meshed with Axisymmetric Shell elements support all element types
7. 1-D Beam and 2-D shell elements having fluid material properties do not support any Thermal Boundary Condition type.

With ESC's finite difference conduction method, conduction is solved on elements, not nodes. This has important implications for meshing. Since you cannot apply Boundary Conditions to nodes, you must create elements of the correct order of dimension for the geometry on which you are defining the Boundary Condition. Create beam elements for edges and curves, shell elements for surfaces and 3-D solid elements for volumes.

Most Thermal Boundary Conditions are applied to surfaces or edges. These surfaces or edges are normally meshed with 2-D shell elements or 1-D beam elements respectively. In most cases these elements are assigned Physical Properties and Material Properties that enable them to model conduction. For example, you might specify a measured thickness (Physical Property) and a conductivity (Material Property) for 2-D shell elements modeling a PC Board.

2.4.1 Using a Bulk Convection Coefficient

You may wish to specify a bulk convection coefficient rather than calculating actual values for individual elements. Values for bulk heat transfer coefficients can be obtained from engineering handbooks or experimental data. This coefficient uses the Reference Temperature For Bulk Heat Transfer Coefficients, specified on the Ambient Conditions form (see Defining Ambient Conditions). The Reference Temperature defaults to the setting Ambient.

The BULK heat transfer refers to the heat transfer coefficient with respect to a single reference temperature. On the Ambient Conditions form, you will see aReference temp for BULK heat trans coeff. The equation for convection heat transfer is:

$Q = h*A*(T_{wall}-T_{fluid})$

The T_{fluid} could be the local fluid temperature, or the ambient (or infinite) fluid temperature.

Thus, the BULK refers to the H's based on a single T_{fluid}. The other heat transfer coefficient uses the local fluid temperature, that is, the temperature of the fluid element right in front of the wall surface element.

2.5 Flow surface

Flow Surfaces model convection on the thermal model surfaces to or from the fluid flow domain, as well as surface drag on the fluid. Electronic System Cooling solves the thermal (conduction) model and the fluid flow model individually. The solution is thermally coupled (by convection) at the solid / fluid interface defined by Flow Surfaces. Therefore, at least one Flow Surface is required for a coupled flow / thermal model.The roughness and convection characteristics for the convecting surface are defined using Surface Properties. A Flow Surface that forms a boundary the modeling domain, or is embedded in the fluid (with fluid volumes on both sides) must be meshed with 2-D thin shell elements. These 2-D elements can be created before or after the 3-D fluid mesh, and before or after the Flow Surface entity is defined. The 2-D elements must be created before solving the model.

A Flow Surface that is shared by two volumes, one meshed with fluid elements and the other with 3-D thermal solid elements, need not be meshed. You can define the Flow Surface directly on the surface and the solver will use the adjacent free faces of the thermal and fluid elements to model convection. Use Flow Surfaces to model surfaces that constrain or direct the fluid flow (such as PC boards, baffles, ducts or chambers) with or without convection. During model simulation, heat paths or conductances are established from the surfaces and obstructions to adjacent 3-D fluid flow elements. The ESC flow solver splits the fluid mesh at the embedded Flow Surface creating a 3-D obstruction to fluid flow.

2.5.1 Procedure for Modeling Convection

The basic steps for modeling convection with Flow Surfaces are:

1. Creating surfaces for modeling convection
2 Meshing the surfaces with 2-D elements of appropriate characteristics

2.5.2 Creating Surface Properties for convection

Specifying convection by either:

- Selecting and specifying individual Flow Surfaces (and optionally, Surface Obstructions) as described below, or

- Specifying Auto-Convection for the entire model.

2.5.3 Creating Surfaces for Modeling Convection

The ESC flow solver requires that a Flow Surface be adjacent to elements of the 3-D fluid mesh, and co-planar with the faces of these 3-D fluid elements. The 3-D elements must therefore be created in volumes bounded by these surfaces. Sometimes these surfaces already exist when the fluid volume is defined. In other cases, additional surfaces must be created specifically to delimit Flow Surfaces. Since the surfaces define the location and extent of the convecting surface, the surfaces must be created and constrained carefully. A Flow Surface defined on a surface is fully associative with that surface. If the dimensions of a surface are modified, a part update will automatically modify the Flow Surface to reflect the change.

2.5.4 Creating Elements for Flow Surfaces

Flow Surfaces are meshed with 2-D thin shell elements. The Flow Surface elements characterize the primary convecting surface with their material and physical properties. These elements convect from both sides if the two sides are in contact with the fluid flow domain. If a single side is in contact with the fluid flow domain then only this side will convect. When modeling turbulent flow (air flow in electronic enclosures is almost always turbulent) it is best if the fluid element size next to a convecting Flow Surface is not smaller than the depth of the laminar sub-layer. This is important since wall functions (and not a fine mesh) are used to accurately model the details of turbulent flow and the laminar sub-layer. Best results are achieved using two or three elements between PC boards that have a standard spacing between them.

The Flow Surface element nodes do not need to align with the fluid flow nodes. You can use a finer or coarser mesh for the fluid flow domain than for the thermal conduction model. However, if possible, ensure that the elements of the flow surface are coplanar with internal faces in the fluid flow mesh. Alignment between the periphery of the surface elements and the internal boundaries of the fluid elements results in accurate modeling of convecting surfaces (see Aligning a Shell Mesh with the Fluid Mesh). If it is not possible to align the shell mesh with the fluid mesh, see the article Tips and Tricks for Using Flow Surfaces.

2.5.5 Specifying Convection

There are several ways to specify convection:

Most convection is modeled by simply selecting one or more meshed surfaces (or the shell elements on them) and filling out the options on the Flow Surface form.

An additional method, Auto-Convection, allows you to define default Flow Surface characteristics to apply all surfaces matching certain criteria. You can also use a combination of the first two methods. First define Auto-Convection with characteristics appropriate to most surfaces. Then create Flow Surfaces only for those surfaces that differ from the default. You can also define convection by specifying the Convection option when creating a Flow Blockage.

2.5.6 Positive and Negative Sides of a Flow Surface

Different surface properties can be applied to the positive or negative side of a flow surface. If the Flow Surface contacts the fluid only on one side, any defined positive Surface Property is applied to that side. If, however, the Flow Surface contacts the fluid on both sides, you must determine which side is positive and which side is negative. The positive/negative sides of a Flow Surface shell element are determined by the shell element Z-axis. Element axes can be displayed by selecting the Display Options icon, then the Element Triad option. To reverse Z-axis polarity, select Element Reverse Connectivity in the ESC menu (use Control+M to turn on the menus).

Avoid shell normal or Z-axis inconstancy on a single Flow Surface. Use Element Reverse Connectivity to adjust inconsistent or incorrect element connectivity.

CFD ANALYSYS

3.1 introduction

The maximum junction temperature of the electronic component obtained using analytical method is validated with that of obtained by carrying out computational fluid dynamics (CFD) Analysis. The dimensional model of ground communication cabinet is considered for analysis. A volume element is chozen for discretizing the ground communication cabinet and its CFD model is obtained. Same heat load and the mass flow rate used for analytical method are also considered as inputs to the CFD model. Then the CFD model is solved using Electronics System

Cooling (ESC) solver to obtain maximum junction temperature of the electronic component. Steady state conditions are assumed while carrying out analysis. This chapter brings out the details of CFD analysis and subsequently the results.

3.2 CFD modeling

Geometry of the electronic ground communication cabinet is built up in 3-D CAD software. All the subsystems like 5 PCBs and components are considered for modeling. Geometric model of the electronic ground communication cabinet in exploded view is shown in Figure3.2.1.

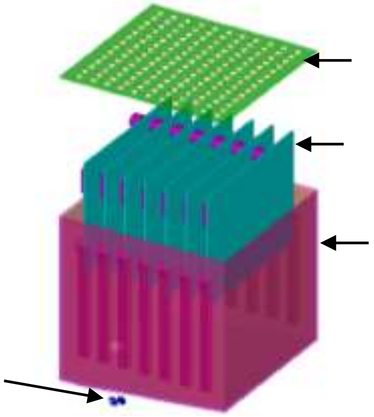


Figure 3.2.1 Geometric model of the electronic ground communication cabinet

Geometry model is converted into CFD model by discretizing the electronic ground communication cabinet, PCBs using linear three nodded triangular shell elements. Where as air surrounding the PCBs with in the ground communication cabinet are discretized using 4 nodded solid linear tetrahedron elements. CFD model of the electronic ground communication cabinet is shown in Figure 3.2.2.

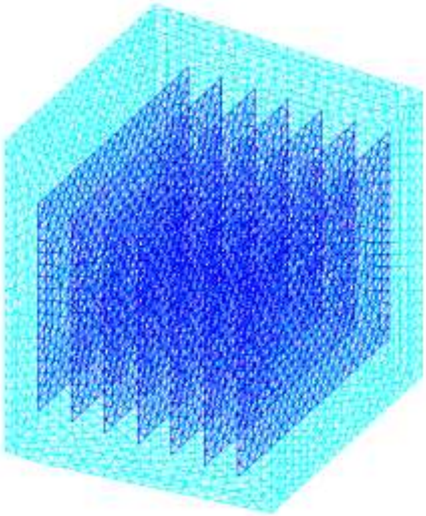


Figure 3.2.2 CFD model of the electronic ground communication cabinet

3.3 Physical properties

The wall thicknesses of walls of electronic ground communication cabinet and PCBs given in Table 3.1are defined as physical properties to the CFD model.

Sl. No.	**Part**	**Thickness**
1.	Walls of electronic ground communication cabinet	3 mm
2.	PCBs	1.6 mm

Table 3.1 Physical properties

3.4 *Material properties*

For carrying out steady state CFD analysis thermal conductivity is required as material property. Rest of other material properties will be in built in the software. Thermal conductivity values of materials of all subsystems are given in Table 3.2

Sl. No.	**Sub system**	**Material**	**Thermal Conductivity**
1.	Walls	Aluminium	200 W/mK
2.	PCB	Epoxy	18 W/m K
3.	Air		0.0263 W/m K

Table 3.2 Material properties

3.5 *Boundary conditions and loads*

1. Heat load of 40 W is applied on each PCB.
2. An ambient temperature of 45^0 C is applied.
3. An inlet fan is created in the FE mode with mass flow rate of 10 gm/Sec.
4. Vent (Opening for air exit to ambient) is defined on perforated holes provided on the cover.
5. Flow surfaces are defined on the front and rear faces of PCBs.

3.6 *analysis*

The CFD model is solved for temperature distribution. Maximum component temperature is observed to be 92.2^0 C. The temperature distribution plot is shown in Figure 3.6.1

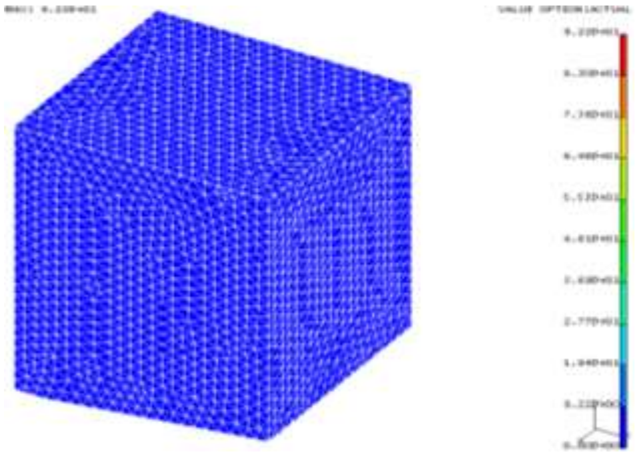


Figure 3.6.1Temperature distribution of electronic ground communication cabinet

In order to show that maximum temperature is developed on component on PCB the above temperature distribution plot-discarding housing is shown in Figure 3.6.2

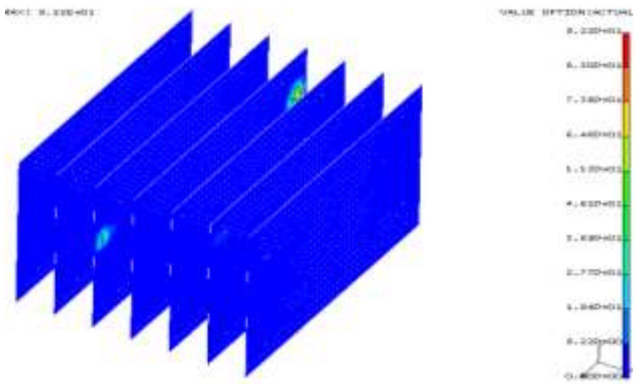


Figure3.6.2 Temperature distribution of electronic ground communication cabinet (PCBs)

RESULTS AND DISCUSSION

4.1 summary of thermal design using analytical method

The outcome of thermal design carried out using analytical method is summarized in Table 4.1.

Sl. No.	Design Parameter	Value
1.	Mass flow rate	10 g/sec
2.	Hydraulic diameter	0.005
3.	Weight velocity flow	2.857 Kg/secm^2
4.	Reynold's number	726
5.	Colburn factor	0.0094
6.	Heat transfer coefficient	34.33 W/m^{2}K
7.	Component hot spot temperature	90.91^0 C

Table 4.1 Summary of thermal design carried out using analytical method

4.2 summary of cfd analysis

The outcome of CFD anaysis is given below.

- The maximum junction temperature of component on PCB is observed to be 92.2^0 C.

4.3 comparison of results

The maximum component temperature estimated using analytical method is compared with that of obtained from CFD in Table 4.2.

	Analytical method	**CFD**
Maximum component temperature on PCB	90.9^0 C	92.2^0 C

Table 4.2 Comparison of results

CONCLUSIONS

1. A cooling scheme has been designed for a ground communication cabinet in forced convection environment.
2. An analytical method has been established during this project in which an expression for heat transfer coefficient has been brought out.
3. This expression has been validated with that of evaluated using commercial CFD software package.
4. The maximum junction temperature of component on PCB is observed to be 92.2^0 C.
5. As component hot spot surface temperature < 100^0 C and hence design is satisfactory.
7. Maximum junction temperature estimated using analytical method is matching well with that of obtained using CFD.
8. The degree of closeness established indicates the confidence of the analytical method.
9. Based on this validation study this analytical method can be comfortably used as a digital calculator to quickly accomplish the thermal design task of electronic ground communication cabinets.

ACKNOWLEDGEMENT

We would like to express my gratitude and appreciation to all those who gave me the possibility to complete this report. A special thanks to our HOD Dr. T.Siva Prasad garu whose stimulating suggestions and encouragement, helped us to coordinate my project especially in writing this report. I would also like to acknowledge with much appreciation the crucial role of the staff of Mechanical Laboratory, who gave the permission to use all required Softwares. Last but not least, many thanks go to the Principal Dr,S. Uday Kumar Garu in achieving the goal as well as his encouragement to maintain our progress in track.

REFERENCES

[1] Cong Qi, Numerical simulation of natural convection in a square enclosure filled with nanofluid using the two-phase Lattice Boltzmann method, Nanoscale Research Letters, 8:56, A Springer open jounal, 2013.
[2] T. F. Smith, Combined Conduction, Natural Convection and Radiation andHeat Transfer in an Electronic Chassis, 382 / Vol. 113, DECEMBER 1991, Trans. ASME 1991.
[3] Onyango, Enhancement of Natural Convection Heat Transfer in a Square Enclosure with Localized Heating from Below, Volume 2, International Journal of Science and Research (IJSR), Issue 10, October 2013.
[4] S. Parvin, Analysis of the flow and heat transfer characteristicsfor MHD free convection in an enclosure with a heated obstacle, Nonlinear Analysis: Modelling and Control, 2011, Vol. 16, No. 1, 89–99.
[5] Emre, CFD modeling of forced cooling of computer chassis, Journal of Engineering applications of CFD, Vol.1, No.4, pp: 304-313, 2007.

AUTHOR BIBLIOGRAPHY

	**B.Bhasker** Designation: Asst.Professor college:geethanjali college of engg. And tech. experience: industry:6 years teaching:3 years
	**J.Nithin kumar** Designation: Asst.Professor college:geethanjali college of engg. And tech. experience: industry: 2years teaching: 4 years