International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 3, No. 2, April, 2014

Research Paper

DESIGN AND CFD THERMAL ANALYSIS OF SHIP's RADAR FRAME

Dharmendra Kumar Madhukar[1]*, Jitendra Nath Mahto[2] and S C Roy[3]

Corresponding Author: **Dharmendra Kumar Madhukar, ✉ madhukardharmendra@gmail.com*

As the electronic equipment is powered, heat is being developed and in increase of temperature in components. Overheating can shorten the life expectancy of costly electrical components or lead to catastrophic failure. For safe functioning of component, the allowable temperature must be provided and if the temperature exceeds these limits, the component fails. In this paper, thermal design of an enclosure for ship's radar frame and its assembly is taken up and ensuring component temperature within the limits. For CFD thermal analysis purpose ANSYS 11 has been used.

Keywords: ANSYS, Design, Homodyne, Radar Frame, Temperature, etc

INTRODUCTION

Ship's radar frame include radar antenna, transmitter, receiver and other electronic enclosure like homodyne, etc. which are usually packaged as a single unit, enclosed in either a radome or in housing with a rotating, bar-like antenna mounted on its top surface. As the electronic equipment is powered, heat is being developed and the effect of heat developed is raising the temperature of the components mounted on radar frame. If the radar frame assembly crossing the allowable temperature specified by manufacturer will cause either malfunctioning of the components or permanent failure (R Parales Jr. 1993); (R Ridley, M Reynell and S Kern, 1993), i.e. in other word the accumulation of heat in an enclosure is potentially damaging to electrical and electronic devices. Overheating can shorten the life expectancy of costly electrical components or lead to catastrophic failure. For thermal analysis and modeling computational fluid dynamics (CFD) is accurate approach for calculating heat transfer coefficient based on air flow condition and also, simulate and analyzed the component in the actual operating condition (G Groove, 1996). The maximum junction temperature of an electronic component position on NRF structure limited to 80°C with air inlet temperature of 55°C i.e. the maximum allowable temperature raise from air inlet to component hot spot temperature = 80-55= 25°C. The design of frame component is taken up in this work with a view to keep the temperature below 80°C.

[1] M. Tech Scholar (Machine Design), Mechanical Engineering Deppt., BIT Sindri, Dhanbad, Jharkhand, India.
[2] Assitant Professor, Mechanical Engineering Deppt., BIT Sindri, Dhanbad, Jharkhand, India.
[3] Professor, Mechanical Engineering Deppt., BIT Sindri, Dhanbad, Jharkhand, India.

CFD THERMAL ANALYSIS OF SHIP's RADAR FRAME

Finite Element Modeling

The geometry of 3D radar frame enclosure is built up in Unigraphics-NX as per provided by user in 2D. All the subsystem like homodyne, card cage and other component are considered for radar frame's Assembly modeling and saved as parasolid. This parasolid is imported in ANSYS to perform CFD Analysis to find out the temperature distribution due to power dissipation of LBH, HBH, etc. The solid model of radar frame and exploded view is shown in Figure 1 & 2 respectively.

Figure 1: Solid Model of Radar Frame

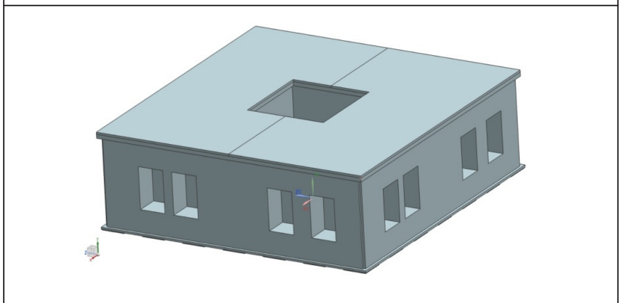
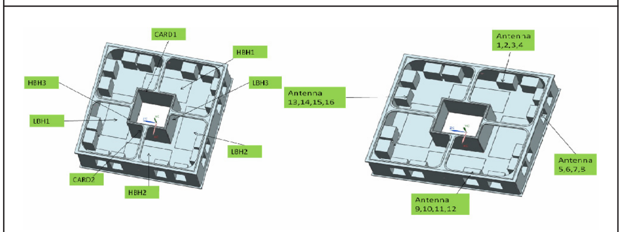


Figure 2: Solid Model of Radar Frame Enclosure in Exploded View with LBH, HBH, Card and Antenna



Material Properties

The radar frame and its enclosure are made using Aluminum HE 30 material. The analysis is done for air inlet temperature of 55°C without the effect of radiation. For carrying out the steady state CFD thermal analysis material property as assigned as table no 1.

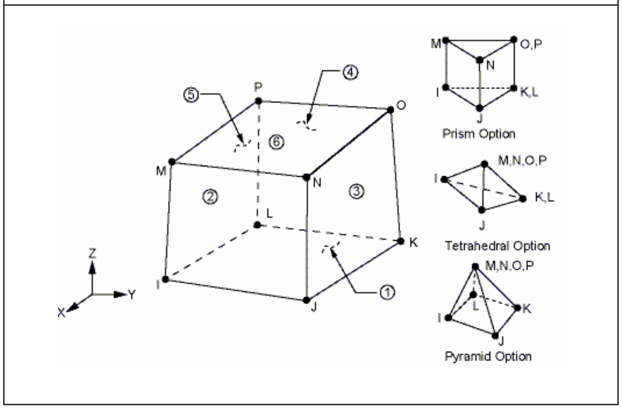
Table 1: Material Properties Used

Material Properties		
1	Young's Modulus (N/mm2)	7.0E+04
2	Poisson's Ratio	0.3
3	Density (kg/mm3)	2700
4	Thermal Conductivity (in w/m2k)	235

To capture the thermal conductivity from homodyne to the bottom plate Solid 70 element is used. Description of Solid 70 element used for thermal model of radar frame is given below.

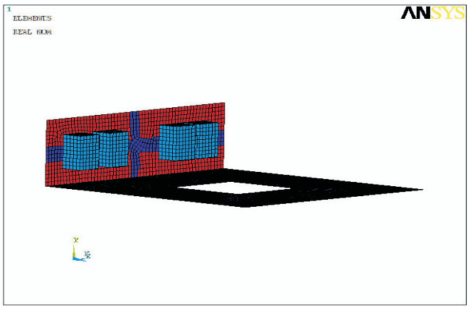
SOLID70 Element has a 3-D thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis. The element also can compensate for mass transport heat flow from a constant velocity field. If the model containing the conducting solid element is also to be analyzed structurally, the element should be replaced by an equivalent structural element. The solid 70 geometry is shown as in Figure 3.

Figure 3: The Solid 70 Geometry



A detailed Finite Element model was built with shell and mass elements to idealize all the components of the Radar Frame. The FE model of radar frame is shown as in Figure 4.

Figure 4: Finite Element Model of the Radar Frame



Design Inputs

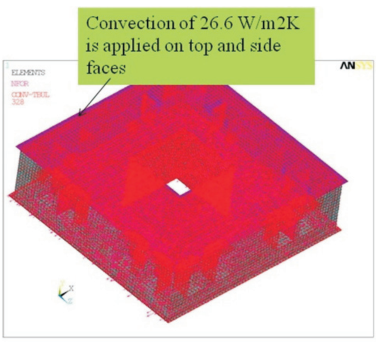
The thermal boundary condition applied on radar frame for CFD analysis. LBH, HBH and Card received a total heat generation load of 42000 w/m^3, 45000 w/m^3 and 38000 w/m^3 respectively. Next, there is heat transfer co-efficient boundary condition of 26.60 w/m^{2}K is applied is shown in the Figure 5 Following are the equation use to determine the heat transfer coefficient of the fluid flow.

$$h_{plate} = \frac{K}{L} Nu_L$$

$$Re = \frac{pVL}{\mu} = \frac{VL}{v}$$

$$Nu_L = 0.664 * (Pr)^{\frac{1}{3}} \sqrt{Re_L}$$

Figure 5: Thermal Boundary Conditions Applied for Thermal Analysis



RESULTS AND DISCUSSION FOR CFD THERMAL ANALYSIS

The CFD model is solved for the temperature distribution in order to show maximum temperature developed on radar frame enclosure; more specifically on the homodyne temperature. The overall temperature distribution of entire assembly is shown in figures 6, 7 & 8. The red and blue in color in fig. shows the maximum and minimum temperature in the frame assembly.

Figure 6: Temperature Distribution on the Radar Frame

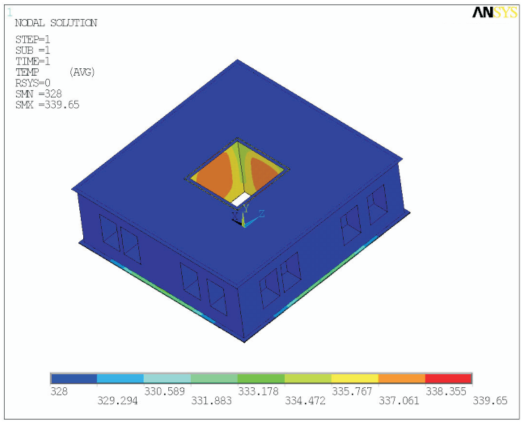


Figure 7: Temperature Distribution on the Base Plate of Radar Frame

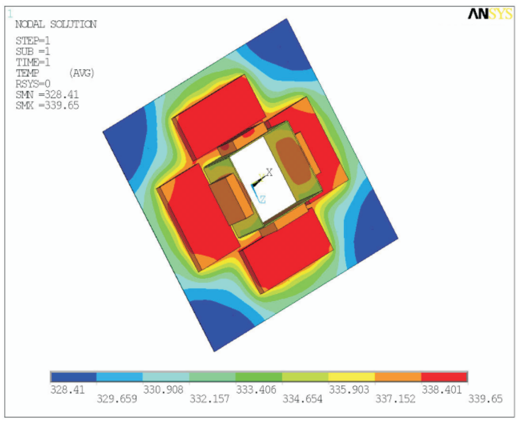
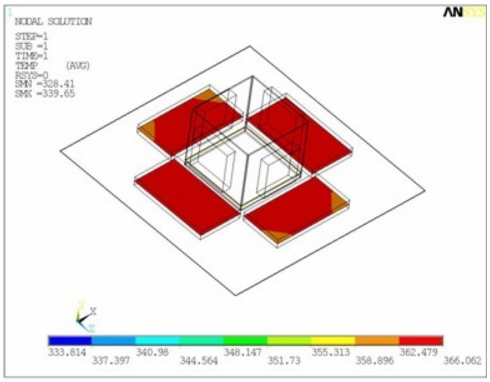


Figure 8: Temperature Distribution on the Base Plate of Radar Frame



From the above CFD thermal analysis it is observed that maximum temperature of 63°C is observed on the homodynes which is well below the maximum allowable temperature of homodyne.

CONCLUSION

In the present paper a Ship's of radar frame has been designed and studied for temperature.

From Figure 6, Figure 7 and Figure 8, it was found that the temperatures are well below from the maximum allowable temperature of homodynes i.e. 80°C.

Therefore it concluded that the radar frame is safe under the given operating conditions.

ACKNOWLEDGEMENT

I would like to thank Mr. H. Pradeep Reddy, Head Project Engineer at RAMTECH Engineering Service, Hyderabad. I am very grateful to Prof. (Dr.) B N Prasad, NIT Jamshedpur, for being incredibly supportive throughout the work and his exhaustive guidance for his immense support and encouragement. I own my greatest debt to my parents specially to my father Sri Mahadev Modi, mother Smt. Shanti Devi, Mr. Upendra Kumar and Shruti Kumari for their constant support, encouragement and their moral support; without them I would not be what I am right now.

References

1. ANSYS 11 Help manuals.
2. G Groove (1996), "Thermal Analysis & optimization of small, high density de power system by FEA" *proc IEEE Int. Telecommunication Energy conference*, pp. 718-722.
3. R Parales Jr (1993), "Solving the thermal concerns of low voltages power supply using CFD modeling", *Proceeding of flowtherm user's conference*.
4. R Ridley, M Reynell and S Kern (1993), "Thermal consideration of distributed power converters system" *Proceeding of IEEE applied power electronic conference*, pp. 866-872.