# Enhancement of Natural convection heat transfer coefficient by using V-fin array

Rameshwar B. Hagote, Sachin K. Dahake

Student of mechanical Engg. Department, MET's IOE, Adgaon, Nashik (Maharashtra,India).

Email. Id: Rameshwarhagote@gmail.com, Contact No. +919730342211

**Abstract**— Extended surfaces known as fins are, used to enhance convective heat transfer in a wide range of engineering applications, and offer an economical and trouble free solution in many situations demanding natural convection heat transfer. Fin arrays on horizontal, inclined and vertical surfaces are used in variety of engineering applications to dissipate heat to the surroundings. Studies of heat transfer and fluid flow associated with such arrays are therefore of considerable engineering significance. The main controlling variables generally available to the designer are the orientation and the geometry of the fin arrays. An experimental work on natural convection adjacent to a vertical heated plate with a multiple V- type partition plates (fins) in ambient air surrounding is already done. Boundary layer development makes vertical fins inefficient in the heat transfer enhancement. As compared to conventional vertical fins, this V-type partition plate works not only as extended surface but also as flow turbulator. This V-type partition plate is compact and hence highly economical. The numerical analysis of this technique is done using Computational Fluid Dynamics (CFD) software, Ansys CFX , for natural convection adjacent to a vertical heated plate in ambient air surrounding. In numerical analysis angle of V-fin is further optimized for maximum average heat transfer coefficient. Attempts are made to validate the results obtained by using CFD analysis by experimentation.

**Keywords**— h - Convective heat transfer coefficient (W/m2K) , As - Exposed surface area (m2),A - Cross section area (m2) , Q - Heat transfer rate (W), I - Current (A), V - Voltage (V) , Ti - Temperature of the respective thermocouple (°C), Ts - Temperature of surface (°C), T∞ & Ta - Temperature of atmosphere (°C) , Tf - Film temperature (°C) , ∆T - Temperature difference (°C or K)

## INTRODUCTION

### Convection

Convection is heat transfer by means of motion of the molecules in the fluid. Heat energy transfers between a solid and a fluid when there is a temperature difference between the fluid and the solid. Convection heat transfer cannot be neglected when there is a significant fluid motion around the solid. There are mainly two types of the convection heat transfer viz. Natural or Free Convection and Forced Convection.

### Natural Convection

The temperature of the solid due to an external field such as fluid buoyancy can induce a fluid motion. This is known as "natural convection" and it is a strong function of the temperature difference between the solid and the fluid. This type of convective heat transfer takes place due to only fluid buoyancy caused due to temperature difference between fluid layers. Natural convection in gases is usually accompanied by radiation of comparable magnitudes except for low emissivity surfaces. Thus natural convection cooling does not require external power.

### Forced Convection

Forcing air to blow over the solid by using external devices such as fans and pumps can also generate a fluid motion. This is known as "forced convection". Some external means for fluid motion is necessary in this type of convective heat transfer. Fluid mechanics plays a major role in determining convection heat transfer. For each kind of convection heat transfer, the fluid flow can be either laminar or turbulent. For laminar flow of fluid over solid surface, steady boundary layer formations takes place through which conductive heat transfer occur. This reduces convective heat transfer rate. Turbulent flow forms when the boundary layer is shedding or breaking due to higher velocities or rough geometries. This enhances the heat transfer. Heat transfer due to convection is described by Newton's Law of Cooling,

$$Q_{conv} = h \times A_S \times (T_S - T_\infty)$$

where,
Qconv = Rate of heat transferred to the surrounding fluid (W)
h = Convection heat transfer coefficient ($W/m^2K$)
AS = Area of solid in contact with the fluid ($m^2$)
Ts-T∞=Temperature difference between solid and surrounding fluid (K or $^0$C)

FINS

The fins are generally extended surfaces or projections of materials on the system. The fins are used to increase the heat transfer rate from the system to the surroundings by increasing the heat transfer area. Different fin geometries and heat sink used for natural convection are shown in figers.

Figure: Heat Sink

Necessity of Fins

The heat that is generated in the system that conducts through the walls or boundaries are needed to be continuously dissipated to the surroundings or environment to keep the system in steady state condition. Large quantities of heat have to be dissipated from small area as heat transfer by convection between a surface and the fluid surroundings. It can be increased by attaching thin strips of metals called fins to the surface of the system. The fin is generally an extended surface on the system. Whenever the available surface is found to be inadequate to transfer the required quantity of heat with the available temperature drop and convective heat transfer coefficient, the surface area exposed to the surroundings is frequently increased by attachment to protrusions to the surfaces. These protrusions are called fins or spines. Thus, the fins increase the effective area of surface there by increasing the heat transfer by convection. Natural convection heat transfer between a surface (TS) and the fluid surrounding (T∞) is given by,

$$Q_{conv} = h \times AS \times (TS - T\infty)$$

where,
h= Heat transfer coefficient
$A_S$=Surface area of heat transfer
$T_S$= Surface temperature
T∞=Surrounding fluid temperature

V-FIN ANALYSIS USING ANSYS CFX

The required models for computational analysis are first made in PRO-E software and then imported in the CFX-Pre Processor. The aim of the project is to find the optimum included angle for V-fins. Therefore various models are created with included angles 0°, 30°, 60°, 90°, 120°, 180°. These models are then analyzed in CFX. The computational analysis in this project is carried out with the help of the software ANSYS 14.5. The vertical pitches for these models are first determined and then are used in PRO-E software.

CALCULATIONS FOR V FIN PARAMETERS

While selecting a heat sink a question that often arises is whether to select closely packed fins or widely spaced fins for a given base area. A heat sink with closely packed fins will have greater surface area for heat transfer but a smaller heat transfer coefficient because of the extra resistance the additional fins introduce to fluid flow through the inter-fin passages. A heat sink with widely spaced fins, on the other hand, will have a higher heat transfer coefficient but a smaller surface area. Therefore, there must be an optimum spacing that maximizes the natural convection heat transfer from the heat sink for a given base area WL, where W and L are the width and height of the base of the heat sink, respectively, as shown in Figure. S is the optimum fin spacing and t is the thickness of the fins.

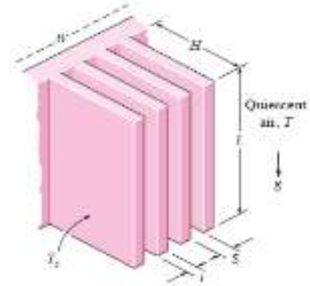



Figure: Finned Surface Oriented Vertically

To calculate the optimum fin spacing, let us consider the model taken by sable et al. for vertical fins as in Figure 4.2. The model considered is of base plate 250mm x 250mm with vertical fins having fin height 20mm and fin thickness 3mm. The heater input is 100W. The surface temperature is Ts = 115°and Temperature of air, Ta = 27°

Therefore the properties of air are calculated at the film temperature of,

$$T_f=(T_s+T_a)/2=((115+27)/2)=71°C$$

From the standard tables, properties of air at mean film temperature of 71°C and 1atm pressure are,

Thermal conductivity = 0.02888 W/m-K

Prandtl number = 0.71747

Kinematic Viscosity = 2.0868 x 10-5 m2/s

Table: Calculation for Vertical Pitch for Various Included Angles in V-Fins

| Total Included Angle $\theta i°$ | Half Angle $\theta h°$ | No. of fins ( n) $n = \frac{250 \text{ x } 13}{\frac{125}{\sin\theta_h} \text{ x } 2}$ | Vertical Pitch $p = \frac{250}{n}$(mm) |
|---|---|---|---|
| 0 | 0 | 13.0000 | 19.2308 |
| 30 | 15 | 3.3646 | 74.3020 |
| 60 | 30 | 6.5000 | 38.4615 |
| 90 | 45 | 9.1924 | 27.1964 |
| 120 | 60 | 11.2583 | 22.2058 |
| 180 | 90 | 13.0000 | 19.2308 |

V-FIN MODEL CREATIONS IN PRO/E

The various types of fins with proper orientation and spacing can be modeled on different CAD software available in the market. These softwares are compatible with the softwares for computational fluid dynamics. For the current modeling, Pro|ENGINEERWILDFIRE 5.0 has been used. The models so formed are along with the fluid domain where in the effects of the heat flow has to be analyzed. Pro|ENGINEER is a feature-based, parametric solid modeling system with many extended design and manufacturing applications. As a comprehensive CAD/CAE/CAM system, covering many aspects of mechanical design, analysis and manufacturing, Pro|ENGINEER represents the leading edge of CAD/CAE/CAM technology.

Figure: V-Fins Model of 30o

CFD ANALYSIS FOR V-FIN

Ansys software is capable of performing stress analysis, thermal analysis, modal analysis, frequency response analysis, transient simulation. My requirement here is thermal analysis of the model. Finite element method of discretization is used. CFX is a commercial Computational Fluid Dynamics (CFD) program, used to simulate fluid flow in a variety of applications. The ANSYS CFX product allows testing systems in a virtual environment. The scalable program has been applied to the simulation of water flowing past ship hulls, fins, gas turbine engines, aircraft aerodynamics, pumps, and fans. For 60omodel analysis procedure and the computer requirement is mentioned below:

Meshing In ICEM CFD

Initially solid body and meshing for the geometrical modeling is done in ICEM which is explained below. At the start in ICEM new working directory is created. Then geometry is imported from pro-e software which was in the form of Step/iges file shown in Figure.

The file that opens, as above, is a surface model which does not have two surfaces in the front and back, as seen. Due to this, the fluid domain is still to be completely formed. The front and back faces are created by using create/modify surface as options. The next step is to create parts on the model as per convenience. The various parts are created for the fluid domain and aluminum domain.

In blocking, initially it is required to give the premesh parameters to define the mesh size and for this by selecting different edges of the model, number of nodes are given. For the outer domain 60 numbers of nodes are assigned. Once we define the mesh size, premesh is generated as shown in Figures.

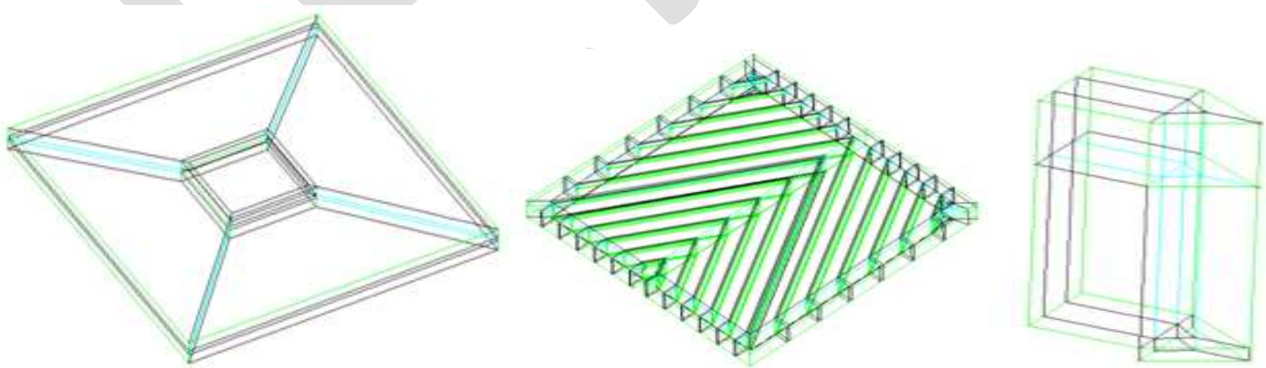


Figure: Final Blocking for outer domain, Figure: Final Blocking for half aluminum domain, Figure: Final Blocking for single V fin

The quality of the mesh can be checked by selecting mesh quality and keep the default parameters and select ok. A range of quality values appear on the screen, the minimum and lowest quality value being zero and the best, highest quality value being one. For this model minimum of quality is 0.66 and maximum is 1. The number of mesh element in each of the quality classes also appear in the dashboard below. The mesh quality is also verified by checking angle, aspect ratio etc. The value of angle should be more than $18^0$ and less than $90^0$. The angle for outer domain is around $40^0$. The volume above zero is acceptable.

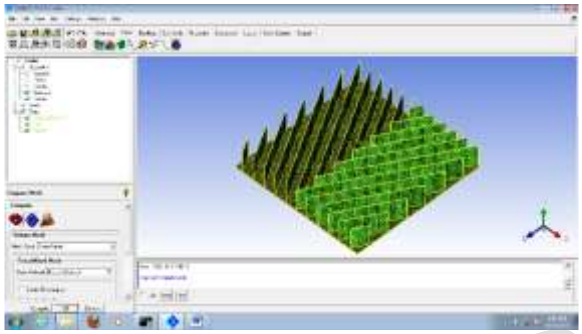


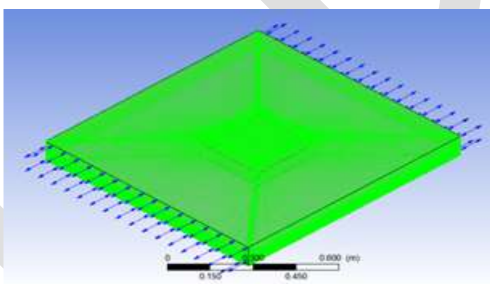
Figure: Meshed Solid Domains

To improve the mesh quality it is required to obtain the above mentioned values within the range. If the quality is still not good then, we have to improve the split, projections .This ICEM model is now complete and ready for pre-processing in CFX. Select the 'output' option in the tool chest and write input finally the save file.

Pre –Processing

Both the mesh solid and mesh fluid domains are now ready for pre-processing in CFX. Each of the models is individually called in the CFX pre file. Since the axes of both the models match, they do not have to be separately aligned in the CFX file.

The page has a tree flow on the right hand side which helps to follow a definite sequence of preprocessing. The main objective of pre-processing is to set up the necessary boundary conditions. These conditions are actually the constraints under which the solver has to determine the flow conditions. The actual ambience and experimentation can be carried out under a whole variety of different conditions and combinations. Therefore, defining boundary conditions is extremely critical in pre-processing. Even one minor change can lead to a set of totally different results than expected.



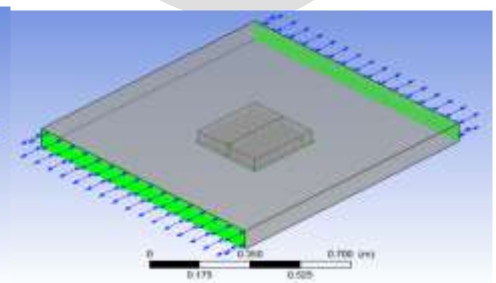
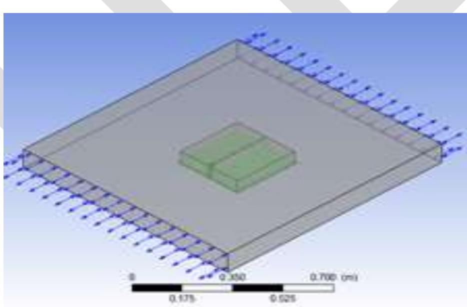


Figure: Fluid domain setting, Figure: opening Boundary Condition



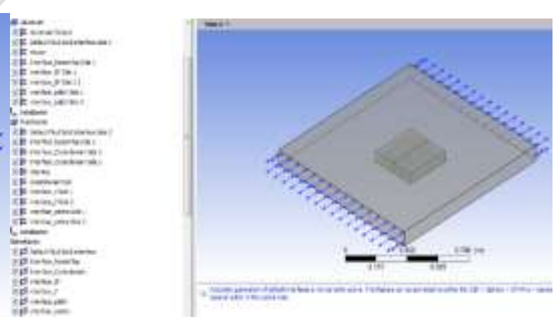


Figure: Solid Domain setting, Figure: Model with all Boundaries Defined

Right click solver control the number of iterations are changed according to requirement the number of iterations given are 1000. But the required iteration for this model is around 120.Under the convergence criteria residual target is kept 100. This is the number of digits after decimal, up to which the accuracy of convergence is required. If the order is increased to either -5 more, the accuracy of convergence improves, although the number of iterations increases by a significant value. For the current model it is seen that order of -5 provides the required accuracy. The last option in the solver is output control. This is not required to be handled in pre processing as these will be taken care of during the actual processing of the model .All these settings are shown in Figure. Finally save caseand define run.

Processing In Solver Manager

After the pre-processing is complete and the run is defined, and the solver starts processing in the CFX solver manager. The number of iterations that are carried out depends on the convergence criteria. If the solution is converged before the total number of iterations, as specified, is complete, the solver stops and saves the processed data. If the solution does not get converged within the given number of iterations, we can give additional iterations to the solver till convergence is reached. When the iterations begin, the screen shows the variations in velocity, momentum etc as the iterations proceed. These graphs give a general idea of how the system is stabilized and convergence is approached.

## EXPERIMENTATION

CFD analysis indicates that, V-fin with included angle 60° gives the minimum $\Delta T$ and hence the maximum average heat transfer coefficient amongst all the considered models. Therefore it is decided to fabricate the V-fin model for 60°. The experiments are to be carried out under natural convection conditions. For the experimental evaluation of results temperature measurements at various locations of the model and also of the surrounding are required. The heater input values when measured will give total heat flow rate from the model under study.

The base plate is made from Aluminum over which fins are to be joined. Initially the fins of desired length (as per the dimensions in Pro-E) are cut by shearing. Then fins are joined on the base plate by using gas welding process. Another side of base plate, asbestos and wooden insulator are attached. In between the base plate and insulator, square plate type heater is sandwiched. Due to square plate type heater there is a uniform temperature distribution over the base plate. The dimensions of base plate, insulator, and heater are 250mm x 250mm x 3mm. The rated power output of heater is 1000 W.

The fins are manufactured as per the dimensions of model of V-fin with 60° included angle. The fins are manufactured by cold working operation. As per the model created in PRO-E, markings are made on the base plate, and the fins are joined on it by gas welding. The entre-line length of fin over the base plate is 3250mm.

Figure: Experimental Set-up

## RESULTS AND DISCUSSION

From the results of post processing the temperature variations on base plate due to various fin angles were found. The angle at which the temperature difference was minimum i.e. convective heat transfer coefficient was maximum, is the angle of interest for actual experimentation. Thus the angle of V-fin can be optimized. The optimized V-fin with base plate was tested for different heat input and different orientations. The results that are obtained after the processing can be viewed in the CFX post processor. In this, various parameters can be graphically seen. The streamlines, velocity vectors, pressures gradients, temperature contours are some of the major parameters that can be viewed. The temperature difference in the $60^{o}$ V-fin model was observed to be 20-30 within the surface and fins. The graphical values of variations of temperature on V-fin base plate V-fin model is as follows,

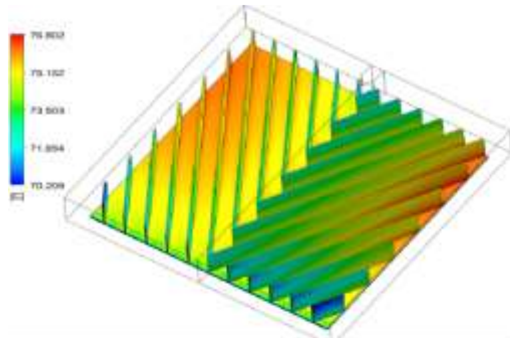


Figure: Temperature Variations on $60^0$ V-Fin Base Plate

The above temperatures contours show the color variations correspond to different temperatures and their locations on the base plate. The dark red color is the maximum temperature, which goes on decreasing as the shades get lighter. The blue colour indicates lower temperature. Figure shows that the temperature is high at the top sides and low in the bottom portion of the plate. Figure shows that the temperature is high at the top sides and low in the bottom portion of the plate. Figure shows that the temperature is high at the middle sides of base plate and low in the top and bottom portion of the plate. Figure shows that the temperature is high at the top sides and low in the bottom portion of the plate. Figure shows that the temperature is high at the middle and centre side of the plate and low at outer periphery of the base plate. Figure shows that the temperature difference on the $60^0$ V-fin base plate is minimum means average heat transfer coefficient is maximum. The temperature difference all other base plate is maximum it means average heat transfer coefficient is minimum. These temperature variations are seen due to buoyancy effect on different orientations of plate. The velocity contour and stream line of flow further clarifies the flow direction of fluid on $60^0$ V-fin model.

## CONCLUSIONS

Based on the present work for V-shaped fin with various included angle, it is concluded that The maximum convective average heat transfer coefficient is obtained for 600 V-fin array. CFD and experimental results for base plate with V-fin showed the similar trend. As the included angle of the V-fins increases, the convective heat transfer coefficient increases. It reaches maximum at 600 included angle and thereafter, the heat transfer coefficient decreases. It was also observed that low pressure suction region is created in the nose region of each V-fin which eventually admits the low temperature ambient fluid easily from surrounding areas. It increases the heat transfer rate. conformity of style throughout a conference proceedings. Margins, column widths, line spacing, and type styles are built-in; examples of the type styles are provided throughout this document and are identified in italic type, within parentheses, following the example. Some components, such as multi-leveled equations, graphics, and tables are not prescribed, although the various table text styles are provided. The formatter will need to create these components, incorporating the applicable criteria that follow.

## REFERENCES:

[1] Wankhede S. (2008), Experimental investigation of heat transfer from inverted notch fin arrays (INFA) under natural and forced convections, IOSR journal of mechanical and civil engineering (IOSR-JMCE)ISSN(e) : 2278-1684, ISSN (p): 2320–334X, PP: 14-22.

[2] Barhatte S. (2012), Experimental and computational analysis and optimization for heat transfer through fins with triangular notch, International Journal of Emerging Technology and Advanced Engineering , Volume 2, Issue 7 ,ISSN 2250-2459.

[3] Kharche S., Farkade H. (2012), Heat transfer analysis through fin array by using natural convection, International Journal of Emerging Technology and Advanced Engineering (ISSN 2250-2459, Volume 2, Issue 4).

[4] Toshiyuki M., Kenzo K. (1990), Natural convection heat transfer from a vertical heated plate. Heat transfer –Japanese Research, Vol-19 No.1 pp-57.

[5] Edlabadkar R., Sane N., and Parishwad G. (2008), Computational analysis of natural convection with single V-type partition plate, 5th European Thermal-Sciences Conference, Netherland.

[6] Sable M., Jagtap S., Patil P., Baviskar P., Barve S. (2010), Enhancement of natural convection heat transfer on vertical heated plate by multiple V-fin array, IJRRAS, 5 (2).

[7] Aberra T., Armfield S., Behnia M., McBain G. (2012), Boundary layer instability of the natural convection flow on a uniformly heated vertical plate, International Journal of Heat and Mass Transfer 55, 6097–6108.

[8] Tsuji T., Kajitani T., Nishino T. (2007), Heat transfer enhancement in a turbulent natural convection boundary layer along a vertical flat plate, International Journal of Heat and Fluid Flow 28, 1472–1483.

[9] Mobedi M., Sunden B. (2006), Natural convection heat transfer from a thermal heat source located in a vertical plate fin, International Communications in Heat and Mass Transfer 33,943–950.

[10] Sane S., Sane N., Parishwad G. (2008), Computational analysis of horizontal rectangular notched fin arrays dissipating heat by natural convection, 5th European Thermal-Sciences Conference, Netherland.

[11] Hussein W., Razzaq A. (2011), Enhancement of natural convection heat transfer from the rectangular fins by circular perforations, (IJAME), ISSN: 2180-1606 (Online); Volume 4, pp. 428-436.

[12] Alessa A., Odat M. (2009), Enhancement of natural convection heat transfer from a fin by triangular perforations of bases parallel and toward its base, The Arabian Journal For Science and Engineering, Volume 34