NUMERICAL ANALYSIS OF HEAT PIPE HEAT EXCHANGER BY DELTA WING VORTEX GENERATOR

[1]Diksha Dilip Nadkarni, [2] Dr. R.R. Arakerimath
[1]ME Student, [2] Professor and Dean of Academics
[1, 2] Department of Mechanical Engineering
[1, 2] G.H.Raisoni College of Engineering and Management, Pune, India.

Abstract— Forced air convection heat pipe cooling systems play an essential role in managing the thermal performance of the electronic, power electronic, air conditioning, etc. When a large amount of heat is dissipated from the devices it becomes essential to cool the system to avoid the damage of the components. For this, an approach is proposed which is to enhance the heat transfer of the heat pipe heat exchanger using improved fin design consisting the vortex generator. The vortex generators can be placed on the existing setup with low cost and no negative impact on system. The modification implemented is studied numerically using CFD. The results show that addition of vortex generator enhances the system output by increasing the heat transfer. This finds application in daily appliances like laptops, PC's, air conditioner, etc.

Index Terms— Heat pipes, heat pipe heat exchanger, vortex generator, heat enhancement, CFD analysis

I. INTRODUCTION

Forced convection cooling of the extended surfaces such as heat sinks and heat pipe fin stack is a primary cooling technique used in microelectronics and telecommunication application. The increasing power densities in such areas have made the encouragement to enhance the research to improve heat enhancement methods. In the applications like telecommunication, air conditioning, microelectronics the cooling technique that is basically used is heat sink or heat pipe heat exchanger. The increasing heat losses in such areas due to the advancements have created a platform to improve the heat enhancement methods. Hence an arrangement of stacking fins on a heat pipe and placing vortex generators on fins is a solution to such problem. The vortex generators are a promising technique for enhancing heat transfer rate. The vortices are punched out of the fins or mounted on the fins to generate longitudinal vortices. The vortices as they move downstream causes bulk mixing of air, boundary layer modification, and convective heat transfer improvement. Advantage of this implementation is it is economic but the pressure penalty is faced. The vortex generator takes the fluid under the wake to swirl around upper side circling the periphery to center vortices. This results in destroying the thermal boundary layer and heat transfer is enhanced. Here we have carried out the numerical analysis using CFD and the aim of increasing the heat transfer rate has been achieved. The input parameters are speed of air, mass flow rate of air, angle of attack of vortex generator, fin material and vortex generator placing, temperature of heat pipe. The output parameters pressure loss, heat transfer rate, temperature distribution

In an early phase the vortex generators were used in a rectangular channel to determine the structure of flow and heat transfer characteristics numerically. The numerical solution gave the complete predicted results of Navier Stokes Equation and Energy equation. The heat transfer enhancement was determined between the channel and the flowing fluid. The skin friction factor and heat transfer characteristics are determined ((11) G.Biswas, et.al). The effect of longitudinal vortex generator array on turbulent boundary layer is studied. Heat transfer and fluid mechanics data are obtained from the turbulent boundary layer. Heat transfer is uplifted/ reduced when secondary flow is directed toward/ away from wall respectively. Multiple vortex generator pairs are arranged in an array to develop the stable patterns in vortices. ((10) W.R.Pauley). In a developing channel flow the impact on surface by the convection enhancement is experimentally studied. The air velocities from 0.9 to 2.5m/s, Reynolds number range 340-940 were used. The two pair V array deployed at 30^0 yields 12-36% augmentation for the channel flow, and is considered an appropriate design for heat exchanger ((3) Jing HE, et.al). The heat transfer and pressure drop for fin and tube exchanger with the use of wing type vortex generator was investigated. Local heat transfer was measured on the fin for the Reynolds number range of 600-2700. Flow losses are estimated and for inline tube arrangement the heat transfer rate increases by 55-65% and apparent friction factor increases by 20-45% ((7) M.Fiebig, et.al). Delta wings are placed on the leading edge of a flat plate, hence the stream-wise vortices are generated that modify the flow. By naphthalene sublimation the local and average convection coefficient are obtained and the structure of vortices is studied using flow visualization. The pressure drop is calculated. In regions where vortex induces a surface normal inflow, the local heat transfer coefficients are found to increase by 300% over the baseline flow, depending on the vortex strength and location relative to boundary layer ((2) M.C. Gentry, et.al).

Forced air convection heat pipe cooling is considered. The excess heat dissipated is encountered by stacking fins on condenser section of heat pipe. Also the use of punched, adhered protrusions on the fin surface and the effect due to them are observed. The results from all the protrusion arrangement by using the CFD software are analyzed. The results matched the before results and both the results complement each other ((1) M.S.Aris, et.al). In commercial refrigeration system frost forms on the air side surface. On air side the performance is limited. Longitudinal vortex generation is prove and effective technique for thinning the thermal boundary layer and enhancing heat transfer but efficiency is not known. They have taken readings and found that heat transfer coefficient range from 33-53 Wm^2K^{-1} ((7) A.D.Sommers, et.al). Detail local Nusselt number distribution, in a channel for various vortex generator configurations is studied. Flow pattern and friction factors are measured by Doppler velocity meter and pressure transducer respectively. 12 single different vortex generators used to check heat transfer and friction factor. Also differences in fluid flow and heat transfer between single vortex generator and array are addressed. It was found that I and delta wing gave good thermal performance among all. Numerical analysis was done to study the thermal hydraulic characteristics of 3-d laminar inline and staggered plate fin and tube heat exchangers. VG span angle, VG transverse location investigated. Area reduction and the different internal factors effects was also investigated. A result showed that max area reduction for inline and staggered is 14.9-25.5% and 7.9-13.6% respectively ((4) Jiin Yuh Jang, et.al). Experimental results on the effectiveness of delta wing vortex generator applied to fin and tube heat exchanger. For the air flowing at Reynolds numbers from about 700 to 2300 Colburn and friction factor and data are obtained for a conventional refrigerator evaporator with and without a single row delta wing vortex generator placed at the inlet face of test heat exchanger. Colburn j factor enhancement up to 31% over the base line were obtained without any pressure drop penalties ((13) A.M. Jacobi, et.al).

EXPERIMENTAL SET-UP

Figure given below shows the schematic diagram of the wind tunnel used in the study. Air is used as working fluid. The main components of the systems were the air supply, instrumentations and data acquisition systems, test section. The wind tunnel system was designed to suck room air over the finned side of the heat exchangers by a 15 KW centrifugal fan. The speed of the fan could be adjusted by a frequency drive of motor. The tunnel has a rectangular duct 300 X 300 X 1000 mm in cross-section. To minimize heat loss to the surroundings, the tunnel surface was insulated with a 8 mm thick acrylic sheet. Being supported by stands of perforated steel plate, the tunnel system was kept 75 cm above the floor level of the laboratory.

The inlet and exit temperature across the air side of the test section is measured by two thermocouple meshes. The inlet measuring mesh consists of eight thermocouples while the --exit mesh contains sixteen thermocouples. These thermocouples were pre-calibrated which have an accuracy of 0.1 ^{0}C. The measuring points were located at positions as described in the ASHRAE standard. These data signals were individually recorded and then averaged. The air pressure drop across the heat exchangers and the nozzles were, respectively, measured by precision differential pressure transducers, whose accuracies were 0.4% and 0.25%.

Air was driven by a 3.0 HP frequency adjusted axial blower from a wind tunnel within a test section. The test section was constructed by using a commercial flexi-glass plate, 8 mm thick. The dimension of the test section was 300 mm (width), 300 mm (height) and 1000 mm (length). The parts that are on the wind tunnel are as follows: 1) Test section, 2) Blower Fan, 3) A.C. Motor, 4) Strain Gauge Balance, 5) Diffuser section, 6) Bell mouth with air straightener, 7) Anemometer, 8) Multi tube manometer, 9) Velocity profile measurement, 10) Control Panel.

The model is been placed in the test section of the wind tunnel. This has been utilized as a closed contour as the model is been tested in the CFD software. The different angle of attack, inlet heat pipe temperature and the velocity of air is given to the model and analyzed. The ultimate results obtained are discussed below.

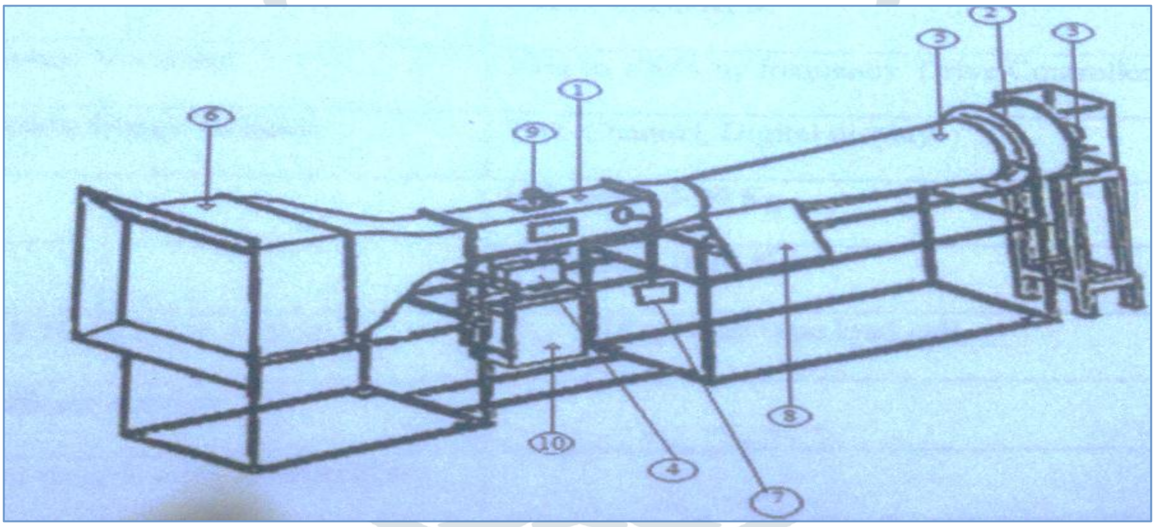

Figure 1- Schematic Diagram of Wind Tunnel

CFD ANALYSIS

Computational fluid dynamics is the science for the prediction of fluid flow, heat and mass transfer, chemical reactions. To predict these results the software makes use of the governing equations in its solver base. The equations like momentum conservation, energy conservation, mass conservation, etc. It is simulation software used to predict the results of flow analysis, compressible, incompressible, etc. CFD reduces the efforts and cost required for experimentation. CFD has its varied application in process industry, chemical industry, food processing industry and many more. It can be used for the system where multiphase or mixing of fluids occurs in a system, which is not the case for other solver. The CFD procedure works with 3 steps: 1) Pre-Processor, 2) Solver, 3) Post - Processor.

CFD MODELING

Following 3 models are modelled in CFD. The data required is found and put in the procedure and the results are obtained. The models are analyzed as per the procedure of CFD ICEM and FLUENT. This means the geometry is defined, then meshing is done, then the boundary conditions are given and solved using the equations, then the distributions are plotted.

GEOMETRY

The geometry is built in the CATIA with the proper dimensions as per the requirement for the work. The model made is imported into the ANSYS modeler in order to make an enclosure for it that would surround the model. This is essential since we are going to solve it for the forced convection. The schematic diagrams of the models are as shown in figures 1, 2, and 3. After the model is imported and body addition is done it is checked that the system is complete by giving the tolerance less than the minimum curve length size of model. This ensures that no free curves are available that would distort or hamper the meshing. The schematic diagram of the plain fin and heat pipe is shown in fig.3 which has 2 round pipes placed as shown. The heat pipe outside diameter D_c is 15mm, fin pitch is 3mm, fin thickness 1 mm. Air flow direction length is 43.3 mm and wave amplitude is 1.5 mm transverse pitch is 25 mm and longitudinal pitch is 21.7 mm. Material selection is most important parameter for fin and vortex generator. Copper and aluminum fins are commonly used. Generally fins are made from aluminum. Aluminum 1100 have strong corrosion resistance. They are sensitive to high temperature ranging between 200 - 250^0 C. also it have excellent forming characteristics. It contains 99% (min) aluminum and 0.12% copper. There will be protrusions on the fin of different sizes.

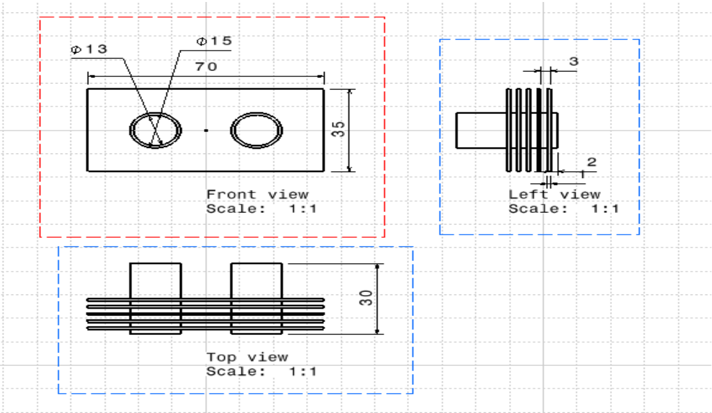

Figure -2 Catia Model

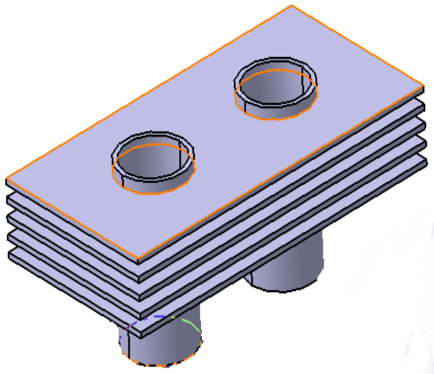

Figure -3 Model without protrusions

The protrusions that are made on 1 side and both side of the heat pipes are the small delta wings that are placed as an obstruction to the flow of air. They are placed on all the fins. The angle of attack is 20^0 and 22^0. The aspect ratio variations are from 2-4.

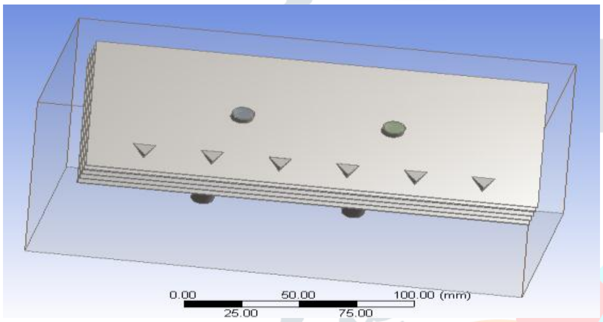

Figure-4 Model with 1 side protrusions

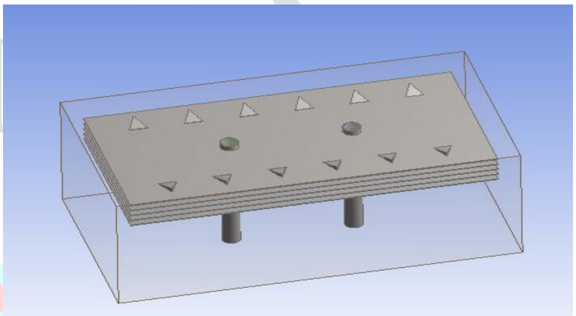

Figure-5 Model with both side protrusions

MESHING

The model after importing and adding body and adding the constraints is ready for meshing. The meshing parameters given to the models are fine, triangular, patch dependent, maximum size as 3mm. The number of nodes and elements for the following 3 cases are 75765, 113217, 75842, 114717, 75808, and 115252 respectively.

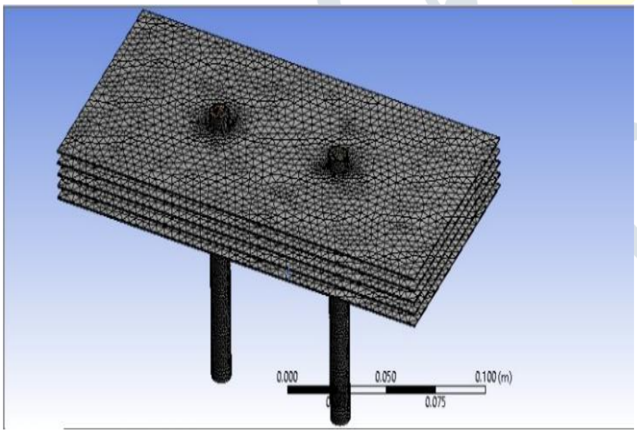

Figure-6 Mesh with no protrusions

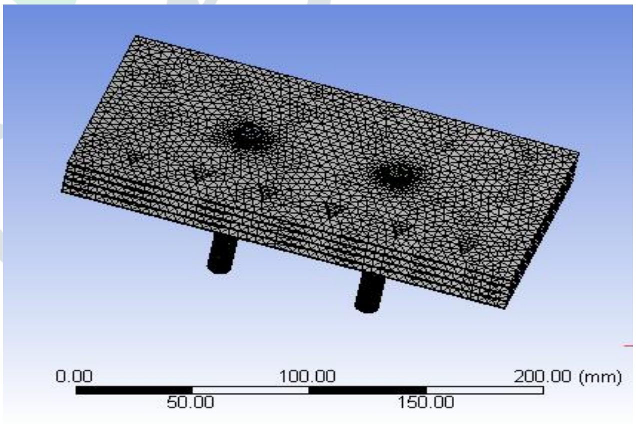

Figure-7 Mesh with 1 array protrusions

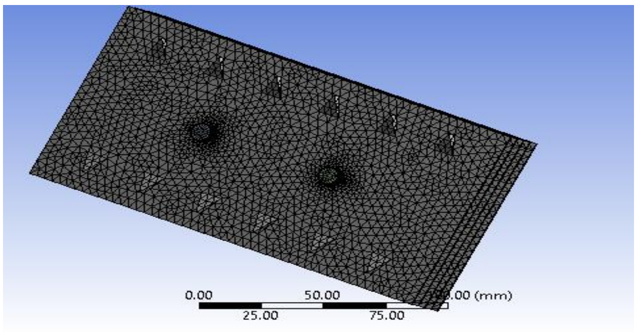

Figure-8 Mesh with 2 array protrusions

BOUNDARY CONDITIONS

The boundary conditions are essential to be set before the system is put to solve the conditions by equations for the required output. The inlet boundary condition is the temperature that is applied at heat pipe inlet as 50^0, 60^0, 70^0. The solution controls are turbulence viscosity=1, turbulence kinetic energy=0.75, turbulence dissipation rate=0.75, energy=0.75, momentum=pressure=0.5. The method selected to solve the models is a coupled pressure-velocity coupling with the second order upwind for the pressure, momentum, turbulent kinetic energy, turbulent dissipation rate, energy. The second order is given so that the accuracy increases. On giving all the above conditions then the system is given a hybrid initialization and on its completion, the calculation is operated with iteration as 500.

MATERIALS AND CELL ZONE CONDITIONS

The defaults are air and aluminum. It has been changed to air, water and copper. The materials are copied from the database and the solver equation like energy and k-episilon are activated. 500 iterations are given and the convergence is obtained and then results are plotted on the contours or graph.

RESULTS & DISCUSSION:

Heat flux contour:

The results show that on addition of vortex generators the heat flux increases and thus the cooling of the system occurs at a fast rate. The result for case 1with no protrusion is maximum heat flux is 41222 Weber per meter square and minimum heat flux is 29.288 Weber per meter square. The results for case with 1 side protrusion is 42429 Weber per meter square and minimum is 262.68 Weber per meter square, for both sides is Weber per meter square, 246.06 Weber per meter square. From the analysis it is confirmed that the heat transfer on adding protrusions to the fins increases. The results are obtained when the inlet temperatures and the convective coefficient is given. Areas with red show high heat rate and reduces towards the end of fin that occurs in blue.

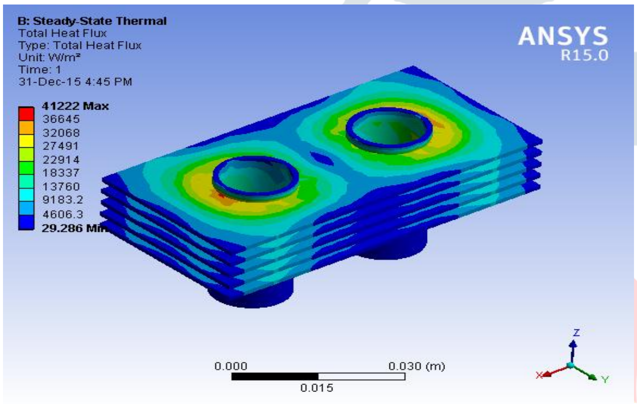

Figure-9 Heat flux contour of no protrusion

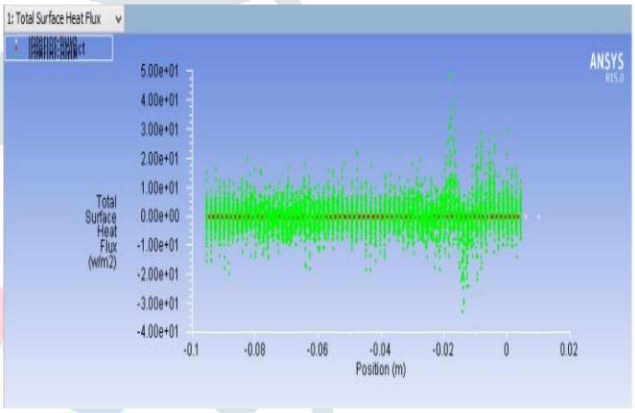

Figure-10 Heat flux graph of no protrusion

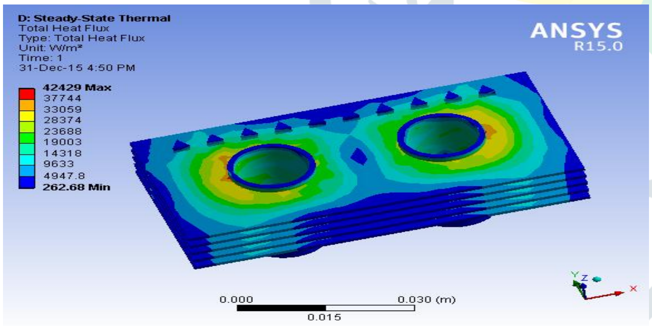

Figure-11 Heat flux contour of 1 side protrusion

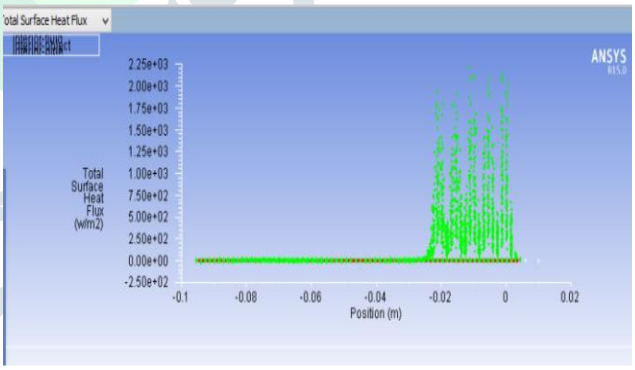

Figure-12 Heat flux graph of 1 side protrusion

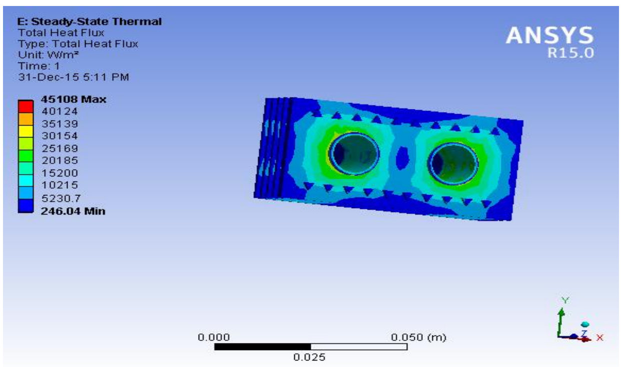

Figure-13 Heat flux contour of both side protrusion

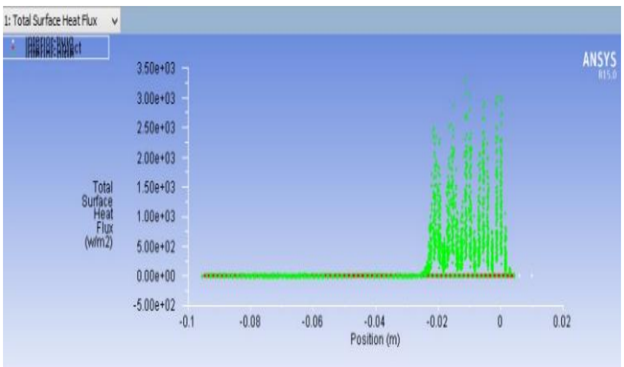

Figure-14 Heat flux graph of both side protrusion

Temperature contour and pressure graph

The pressure graphs are of static and dynamic pressure. It shows that the air pressure decreases initially to a limit and then it increases. The static curve takes the shape of a semicircle upside and dynamic downside. The temperature plot shows that the maximum temperature of 343K is reached for 2 side arrays. Areas with red are hot and that with blue are cold. The changes that occur in temperature plots are due to the addition of vortex generator. This addition improves the thermal boundary layer. The temperature increases since as the velocity is low there is more time for fluid to heat up.

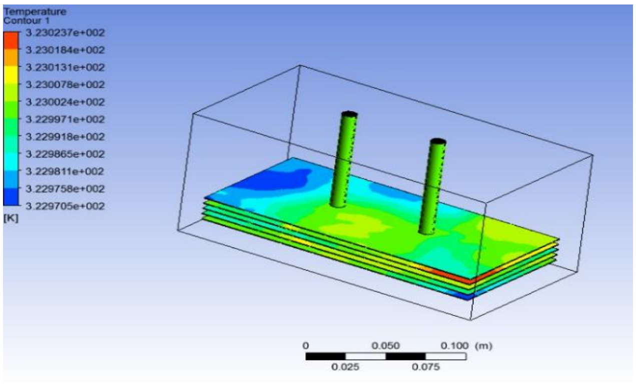

Figure-15 Temperature contour of no protrusion

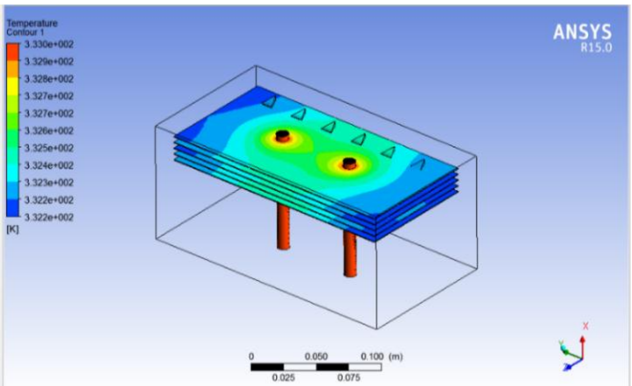

Figure-16 Temperature contour of 1 side protrusion

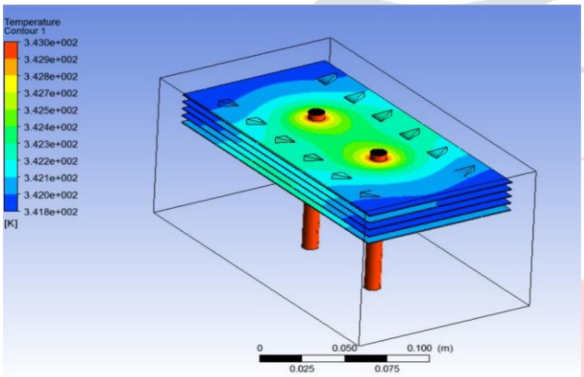

Figure-17 Temperature contour of both side protrusion

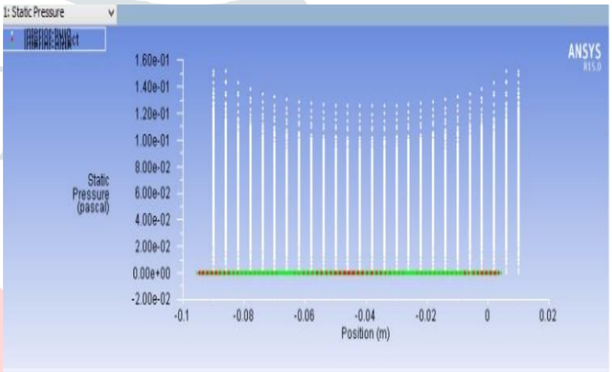

Figure-18 Pressure graph of no protrusion

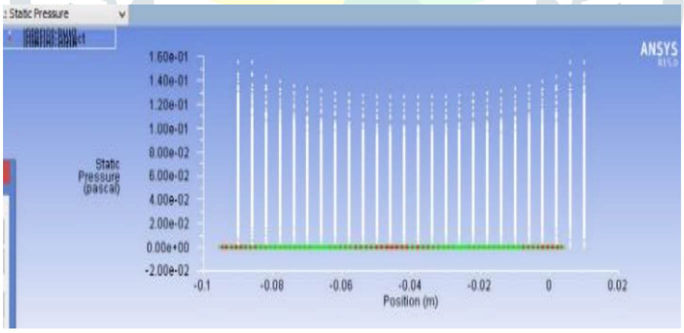

Figure-19 Pressure graph of 1 side protrusions

CONCLUSION

From the analysis that has been done the following can be concluded:

1. A flat fin placed on heat pipe with the delta type wing protrusions gives the better heat transfer rate.
2. The maximum temperature is 343 Kelvin and minimum is 341Kelvin when two arrays of vortex generators are placed.
3. The heat pipe performance increases.
4. From the analysis done for heat flux it is proved that the heat transfer increases. For system without protrusion it is 41222 Weber per square meter, with protrusion on 1 side it is 42429 Weber per square meter and with protrusion on both sides is 45108 Weber per square meter.
5. The results are maximum when the vortex generators angle of attack is 22^0 and aspect ratio 3.5. Hence as the angle of attack increases the rate increases.
6. The only drawback is that temperature above 100^0 cannot be given.

REFRENCES

[1] M.S.Aris, R.McGlen, I.Owen, C.J.Sutcliffe, An experimental investigation into the deployment of 3-D, finned wing and shape memory alloy vortex generators in a forced air convection heat pipe fin stack, (2011), Applied Thermal Engineering, Vol. 31 pp. 2230-2240.

[2] M.C.Gentry, A.M.Jacobi, Heat Transfer Enhancement by delta wing generated tip vortices in flat plate and developing channel flows, (2002), ASME, Vol. 124, pp. 1158-1168.

[3] Jing HE, Anthony M.Jacobi, Air side heat transfer enhancement by a v formation delta winglet array in a developing channel flow, (2010), International Refrigeration and Air conditioning conference at Purdue.
[4] Jiin Yuh Jang, Ling Fang Hsu, Jin Sheng Leu, Optimization of the span angle and location of vortex generator in a plate-fin and tube heat exchanger, (2013), International journal of heat and mass transfer, Vol. 67, pp. 432-444.
[5] Hrishikesh S Kulkarni, Sandeep S. Kore, Experimental Investigation of Heat Transfer Enhancement by Longitudinal Rectangular Vortex Generator, International Journal of Innovative Research in Advanced Engineering, Volume 1, Issue 8.
[6] A.D.Sommers, A.M. Jacobi, Air side heat transfer enhancement of a refrigerator evaporator using vortex generation, (2005), International journal of refrigeration, Vol. 28, pp. 1006-1017.
[7] M Fiebig, A.Valencia, N.K.Mitra, Wing type vortex generators for fin tube heat exchangers, (1993), Experimental thermal and science, 7, 287-295.
[8] Martin Fiebig, Peter Kallweit, Nimai Mitra Stefan Tiggelbeck, Heat transfer Enhancement and Drag by longitudinal vortex generator in channel flow, (1991), Experimental thermal and science, 4, 103-114.
[9] W.R.Pauley, J.K.Eaton, The effect of Embedded Longitudinal Vortex Arrays on Turbulent Boundary Layer Heat Transfer, (1994), ASME, Vol.116 (871-879).
[10] G.Biswas, H.Chattopadhyay, Heat transfer in a channel with built in wing type vortex generators, (1992), International Journal of Heat and Mass Transfer, Vol.35, No. 4, pp.803-814.
[11] A.I.Elsherbini, A.M.Jacobi, The thermal- hydraulic impact of delta wing vortex generator on the performance of a plain fin and tube heat exchanger, (2002), HVAC&R, Vol. 8 No.4.
[12] Jacobi A.M., Shah R.K., Air side flow and heat transfer in compact heat exchanger: A discussion of Enhancement Mechanisms, Heat Transfer Eng., 19, pp. 29-41
[13] Fiebig, Vortices, Generators and Heat transfer, Chem. Eng. Res. Des., 76(A2), pp. 108-123