International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 8, August - 2014

Numerical Investigation of Heat Transfer in Wavy Fin by Variying Geometry Parameters

M. Muthukumaran, K. Akilan, V. Dhanasekar
B.E. Department of Aeronautical Engineering, DSCET, Chennai, Tamilnadu, India

P. Srinivasan
Asst.Professor, Department of Aeronautical Engineering, DSCET, Chennai, Tamilnadu, India

***Abstract* - The study of wavy fin is to increase the Heat Transfer Rate from the heated surface to avoid the excess heat generation which will leads to damage of the mechanical parts is the key area of research in modern industry. With these aspects in mind we have considered the use of fins, an unexceptional field in the design of aircrafts, cars, others automobiles for our research. Our study focuses in particular, the increase in heat transfer rate in order to optimize the fin spacing and fin height by means of replacing the straight fin by wavy fin. This project shows a comparative study of the effect of fin spacing and height with the straight and wavy fins to attain maximum heat transfer rate.**

KEYWORDS: HEAT CONDUCTION, HEAT CONVECTION, WAVY FIN, HEAT TRANSFER, NUSSLET NUMBER

NOMENCLATURE:

RANS-Reynolds Average Navier-Stokes

RNG-Renormalization-Group

SST-Shear stress Transport

h- Fin spacing (mm)

s- Fin height (mm)

INTRODUCTION:

Energy is defined as the capacity of a substance to do work. It is a property of the substance and it can be transferred by interaction of a system and its surroundings. The student would have encountered these interactions during the study of Thermodynamics. However, Thermodynamics deals with the end states of the processes and provides no information on the physical mechanisms that caused the process to take place. Heat Transfer is an example of such a process. A convenient definition of heat transfer is energy in transition due to temperature differences. Heat transfer extends the Thermodynamic analysis by studying the fundamental processes and modes of heat transfer through the development of relations used to calculate its rate. The aim of this chapter is to console existing understanding and to familiarize the student with the standard of notation and terminology used in this book. It will also introduce the necessary units

The different types of heat transfer are usually referred to as 'modes of heat transfer'. There are three of these: conduction, convection and radiation.

Conduction: This occurs at molecular level when a temperature gradient exists in a medium, which can be solid or fluid. Heat is transferred along that temperature gradient by conduction. 2

Convection: Happens in fluids in one of two mechanisms: random molecular motion which is termed diffusion or the bulk motion of a fluid carries energy from place to place. Convection can be either forced through for example pushing the flow along the surface or natural as that which happens due to buoyancy forces.

Radiation: Occurs where heat energy is transferred by electromagnetic phenomenon, of which the sun is a particularly important source. It happens between surfaces at different temperatures even if there is no medium between them as long as they face each other.

In many practical problems, these three mechanisms combine to generate the total energy flow, but it is convenient to consider them separately at this introductory stage. We need to describe each process symbolically in an equation of reasonably simple form, which will provide the basis for subsequent calculations. We must also identify the properties of materials, and other system characteristics, that influence the transfer of heat.

CONDUCTION

The conductive transfer is of immediate interest through solid materials. However, conduction within fluids is also important as it is one of the mechanisms by which heat reaches and leaves the surface of a solid. Moreover, the tiny voids within some solid materials contain gases that conduct heat, albeit not very effectively unless they are replaced by liquids, an event which is not uncommon. Heat is transferred by conduction due to motion of free electrons in metals or atoms in non-metals Provided that a fluid is still or very slowly moving, the following analysis for solids is also applicable to conductive heat flow through a fluid.

CONVECTION

Convection heat transfer occurs both due to molecular motion and bulk fluid motion. Convective heat transfer may be categorized into two forms according to the nature of the flow: natural Convection and forced convection.

In natural of ‘free’ convection, the fluid motion is driven by density differences associated with temperature changes generated by heating or cooling. In other words, fluid flow is induced by buoyancy forces. Thus the heat transfer itself generates the flow which conveys energy away from the point at which the transfer occurs.

In forced convection, the fluid motion is driven by some external influence. Examples are the flows of air induced by a fan, by the wind, or by the motion of a vehicle, and the flows of water within heating, cooling, supply and drainage systems. In all of these processes the moving fluid conveys energy, whether by design or inadvertently.

The left of Figure 1.2 illustrates the process of natural convective heat transfer. Heat flows from the ‘radiator’ to the adjacent air, which then rises, being lighter than the general body of air in the room. This air is replaced by cooler, somewhat denser air drawn along the floor towards the radiator. The rising air flows along the ceiling, to which it can transfer heat, and then back to the lowerpart of the room to be recirculates through the buoyancy-driven ‘cell’ of natural convection.
The word ‘radiator’ has been written above in that way because the heat transfer from such devices is not predominantly through radiation; convection is important as well. In fact, in a typical central heating radiator approximately half the heat transfer is by (free) convection.
The right part of Figure 1.2 illustrates a process of forced convection. Air is forced by a fan carrying with it heat from the wall if the wall temperature is lower or giving heat to the wall if the wall temperature is lower than the air temperature.

RADIATION

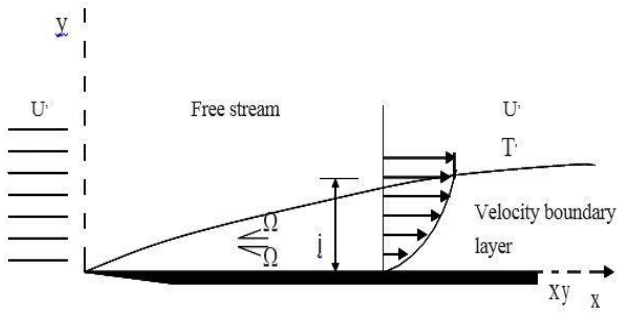
While both conductive and convective transfers involve the flow of energy through a solid or fluid substance, no medium is required to achieve radiate heat transfer. Indeed, electromagnetic radiation travels most efficiently through a vacuum, though it is able to pass quite effectively through many gases, liquids and through some solids, in particular, relatively thin layers of glass and transparent plastics.

Fins are often used to enhance the rate of heat transfer from heated surfaces to environment. They can be placed on plane surfaces, tubes, or other geometries. These surfaces have been used to augment heat transfer by adding additional surface area and encouraging mixing. When an array of fins is used to enhance heat transfer under mixed convection conditions, the optimum geometry of fins should be used, provided this is compatible with available space and financial limitations. Advantages in printed circuit boards have yielded increasing power dissipation from surfaces in a channel.

Rectangular fins are used extensively to increase the rates of convection heat transfer from systems, because such fins are simple and cheap, to manufacture. Providing adequate cooling of printed circuits boards has recently motivated experiments on the use of longitudinal fins to enhance heat transfer in rectangular channels. The heat transfer, to the fluid flowing through a channel by the heat dissipating surfaces can be obtained mainly by using the mechanisms of heat transfer by forced convection, natural convection and by radiate heat transfer.

Examples are seen on motorcycle engines, electric motor casings, gearbox casings, electronic heat sinks, transformer casings and fluid heat exchangers. Extended surfaces may also be an unintentional product of design. Look for example at a typical block of holiday apartments in a ski resort, each with a concrete balcony protruding from external the wall. This acts as a fin and draws heat from the inside of each apartment to the outside. The fin model may also be used as a first approximation to analyze heat transfer by conduction from say compressor and turbine blades.

THE VELOCITY BOUNDARY LAYER

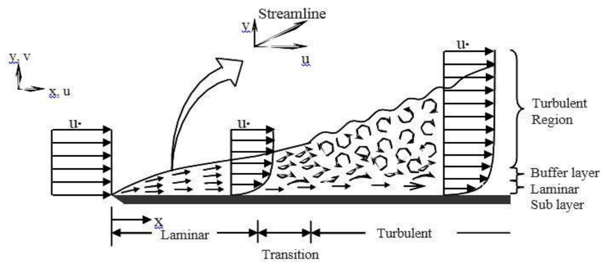


The velocity boundary layer on a flat plate

LAMINAR AND TURBULENT BOUNDARY LAYER

In convection problems it is essential to determine whether the boundary layer is laminar or turbulent. The convective coefficient h will depend strongly on which of these conditions exists. There are sharp differences between laminar and turbulent flow conditions. In laminar boundary layers the fluid motion is highly ordered. Fluid particles move along streamlines. In contrast, fluid motion in the turbulent boundary layer is highly irregular. The velocity fluctuations that exist in this regular form of fluid flow result in mixing of the flow and as a consequence enhance the convective coefficient significantly.

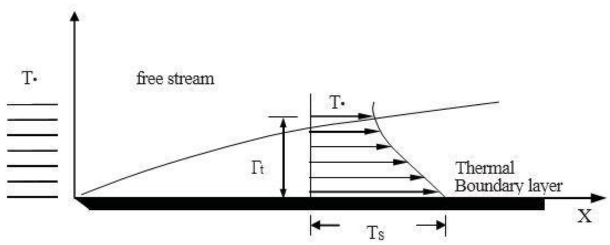
Figure shows the flow over a flat plate where the boundary layer is initially laminar. At some distance from the leading edge fluid fluctuations begin to develop. This is the transition region. Eventually with increasing distance from the leading edge complete transition to turbulence occurs. This is followed by a significant increase in the boundary layer thickness and the convective coefficient. Three different regions can be seen in the turbulent boundary layer. The laminar sub layer, the buffer layer and turbulent zone where mixing dominates. The location where transition to turbulence exists is determined by the value of the Reynolds's number which is a dimensionless grouping of variables.



Laminar and turbulent boundary layer on a flat plate

THE THERMAL BOUNDARY LAYER

Figure shows analogous development of a thermal boundary layer. A thermal boundary layer must develop similar to the velocity boundary layer if there is a difference between the fluid free stream temperature and the temperature of the plate. The fluid particles that come in contact with the plate achieve thermal equilibrium with the surface and exchange energy with the particles above them. A temperature gradient is therefore established.



Thermal boundary layer development on a flat plate

Governing Equations

The governing equations of flow i.e. conservation of mass, momentum and energy can be represented as follows:

Conservation of Mass:

$$\frac{\partial \rho}{\partial t} + \Delta(\rho v) = 0$$

Conservation of Momentum:

Similar equations can be written for Y and Z direction

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{zx}}{\partial x} + \rho f_x$$

Similar equations can be written for Y and Z direction.

Conservation of Energy:

$$\rho \frac{D}{Dt}\left(e + \frac{v^2}{2}\right) = \rho q + \frac{\partial}{\partial x}\left(\kappa \frac{dT}{dx}\right) + \frac{\partial}{\partial y}\left(\kappa \frac{dT}{dy}\right) + \frac{\partial}{\partial z}\left(\kappa \frac{dT}{dz}\right)$$

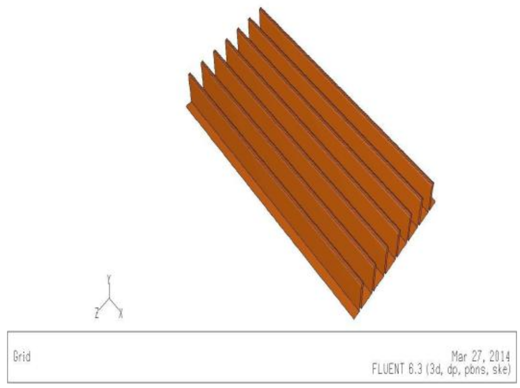
$$-\frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z} + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x}$$

$$+\frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(v\tau_{xz})}{\partial x} + \frac{\partial(v\tau_{yz})}{\partial y} + \frac{\partial(v\tau_{zz})}{\partial z} + \rho f.V$$

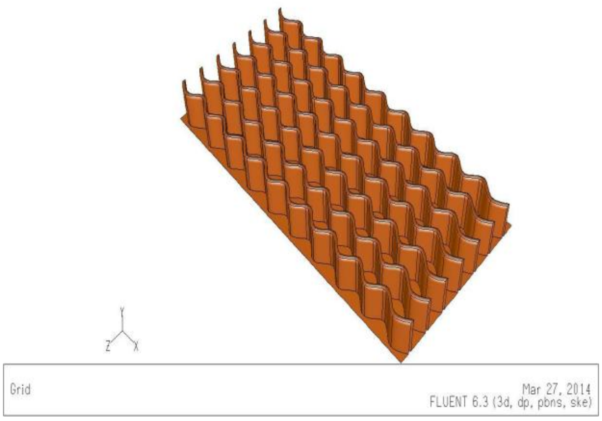
GEOMETRICAL DESCRIPTION OF MODELS:

3.4.1 Model Description:

Base Plate: Copper with dimension 600X100mm Fins: Aluminium with dimension 600 X 1 mm



ISSN: 2278-0181
Vol. 3 Issue 8, August - 2014



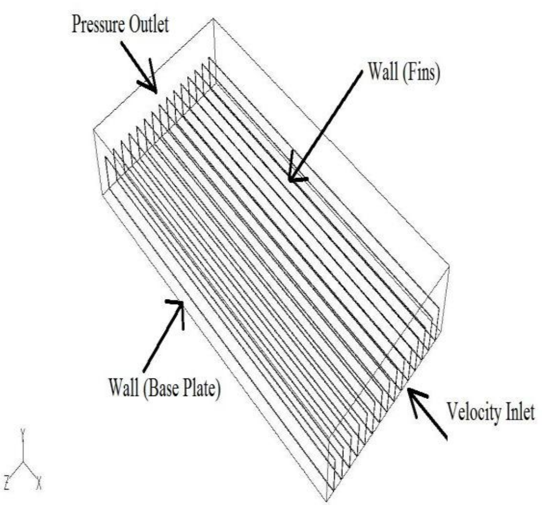
Dimension of computational model

Base Plate: 100 X 50X 1 mm
Fins: 100 X 1 mm
Domain: 100 X 50 X 25 mm

DOMAIN:

The domain selection and domain size place a vital role in convergence criteria to obtain acceptable results. The domain size must be suitable for the model length and height. Choosing a proper domain will reduce the complexity of meshing the entire domain. Therefore, Rectangular domain is chosen and the domain size is arbitrarily taken as suitable for the model size.

Boundary Conditions



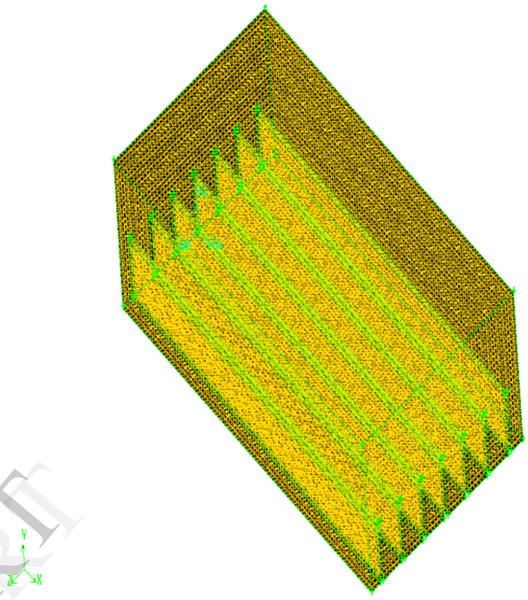
Fluent Boundary Conditions:

Entry: Velocity Inlet = 0.155 m/s
Exit: Pressure Outlet =1 atm
BasePlate: Wall =400 K
Fins:Wall =300K (atmospheric temperature)

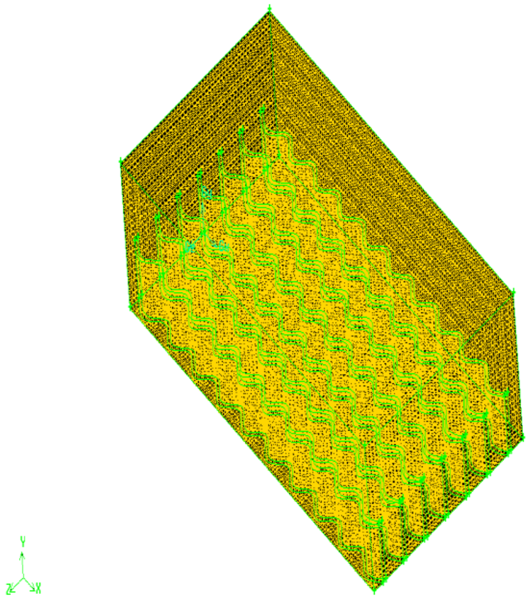
GRID GENERATION

Mesh Details:

Mesh Element : Hex
Mesh Type : Map
Element Size : 1
Skewness : less than 0.1
Number of nodes: 133926 (Straight) & 139240 (Wavy)



Computational Grid of Straight Fin

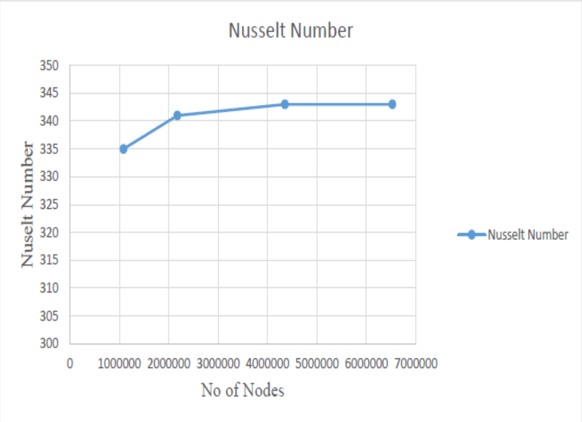


Computational Grid of Wavy Fin

GRID INDEPENDENT STUDY:

Mesh:
Type : Structured Mesh
Element: Hex

Scheme : Map
Skewness: 0.2



TURBULENCE MODELS:
FLUENT provides the following choices of turbulence models:

1. Spalart-Allmaras model
2. k-Ɛ models

   Standard k-Ɛ model

   Renormalization-group (RNG) k- Ɛ model

   Realizable k- Ɛ model

1. k-ω models

standard k-ω model

Shear stress transport (SST) k-ω model

Reynolds-averaged Navier-Stokes (RANS):

The Reynolds-averaged Navier-Stokes (RANS) equations govern the transport of the averaged flow quantities, with the whole range of the scales ofturbulence being modeled. The RANS-based modeling approach therefore greatly reduces the required computational effort and resources, and is widely adopted for practical engineering applications. An entire hierarchy of closure models is available in FLUENT including Spart-Allmaras, k-Ɛ and its variants, k-ω and its variants, and the RSM. The RANS equations are often used to compute time dependent flows, whose unsteadiness may be externally imposed (e.g., time-dependent boundary conditions or sources) or self-sustained (e.g., vortex shedding, flow instabilities).

The time-averaging is defined as

$$\bar{f} = \lim_{T\to\infty} \frac{1}{T}\int_0^T f(x_i,t)dt$$

The instantaneous field is defined as the sum of the mean and the fluctuating component, such as
$p=\bar{p}+p'$
$ui=\bar{ui}+ui'$
by averaging the Navier-Stokes equations,

$$\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0$$

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right]$$
$$+\frac{\partial}{\partial x_j}(-\overline{\rho u_i' \rho u_j'})$$

The above equations are called Reynolds-averaged Navier-Stokes (RANS) equations. They have the same general form as the instantaneous Navier-Stokesequations, with the velocities and other solution variables now representing ensemble-averaged (or time-averaged) values.

Standard k-Ɛ model:

This is the simplest model of turbulence of two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined.

It is a semi empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism. As the strengths and weaknesses of the standard k-Ɛ model become known, improvements have been made to the model to improve its performance. Two of these variants are available in FLUENT the RNG k-Ɛ model and the realizable k-Ɛ model.

Transport Equations for the Standard k- ϵModel:

The turbulence kinetic energy, k, and its rate of dissipation, ϵ, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\epsilon-Y_M+S_k$$

$$\frac{\partial}{\partial t}(\rho\epsilon)+\frac{\partial}{\partial x_i}(\rho\epsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_j}\right]+C_{1\epsilon}\frac{\epsilon}{k}(G_k+C_{3\epsilon}G_b)$$
$$-C_{2\epsilon}\rho\frac{\epsilon^2}{k}+S_\epsilon$$

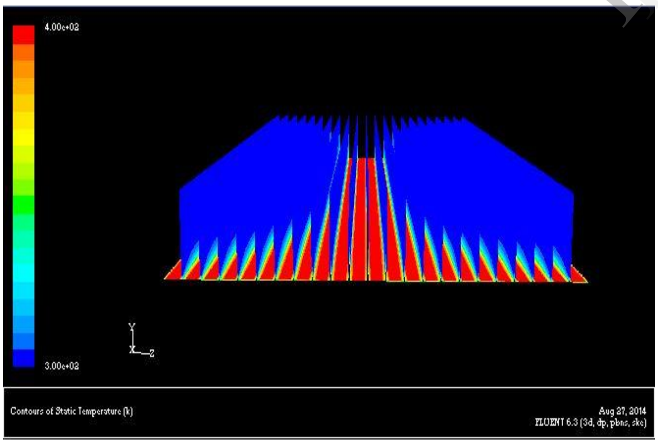
In these equations, Gk represents the generation of turbulence kinetic energy due to the mean velocity gradients. $Gb$is the generation of turbulence kinetic energy due to buoyancy.YM represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate.$C1\epsilon$,$2\epsilon$and $C3\epsilon$are constants.$\sigma k$and$\sigma\epsilon$are the turbulent prandal numbers for k and, ϵ respectively.Sk and $S\epsilon$ are user-defined source terms.

Computational method:

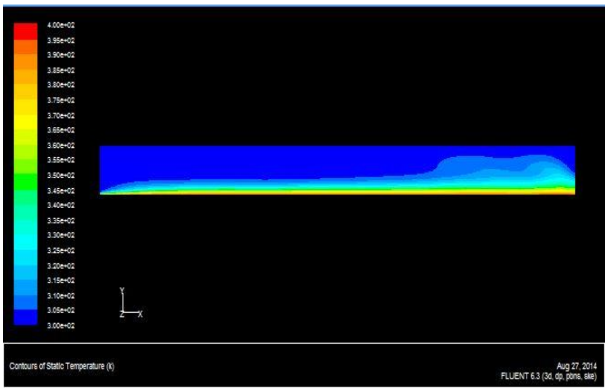
The overall solving process is monitored properly and the results is obtained and verified from the CFD post processor and the comparison of plots is done using Microsoft offices excel –graphs.

By replacing the straight fins by wavy fins, the surface area of heat transfer increases will leads to increase in heat transfer rate. Flow velocity between the fins also a major component for maximum heat transfer. How long the cold flow stays inside the passage will observe more heat from the source. In straight fins passage the cold fluid leaves without any disturbance and it will not observe that much heat from the source.But in wavy fins, the cold flow stays longer than in the straight fins. This results in observing the heat from the source much better than in straight fins. Also in wavy fins the flow is recirculate between the fins and tends to observe more heat and leaves the passage. From the velocity comparison plot we can clearly came to know that flow velocity is delayed in wavy fins which means it let the cold flow to stay longer than in straight fins. The velocity difference will also leads to the maximum heat transfer.

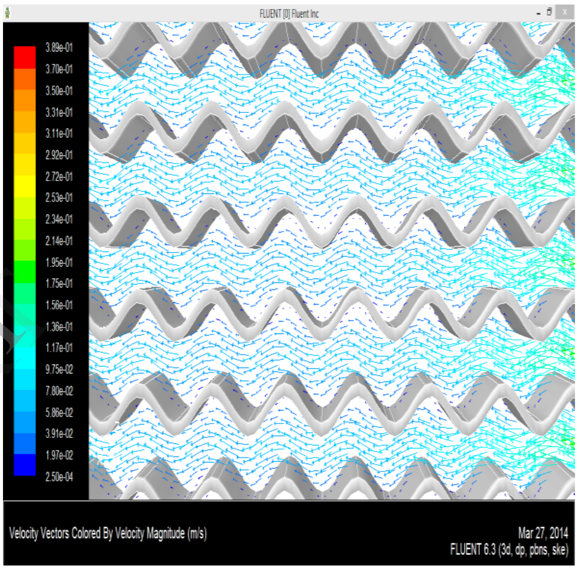
FLUENT RESULTS



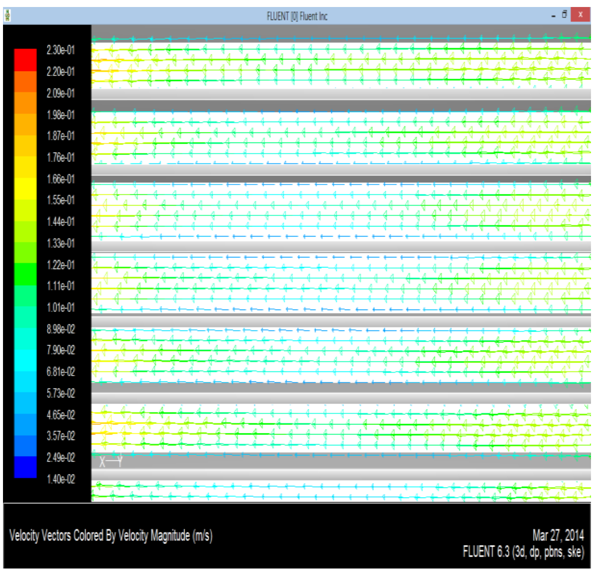
Temperature contour for the Front view of the base plate and fins



Side view of temperature contours for a single fin section



Top view of Wavy fin velocity vector contour

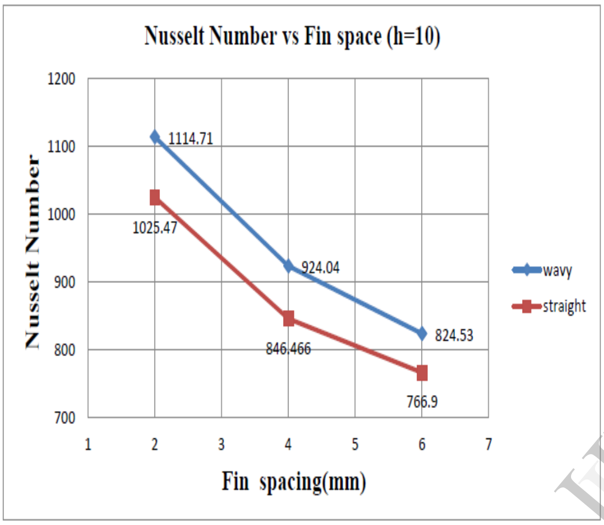


Top view of Straight fin velocity vector contour

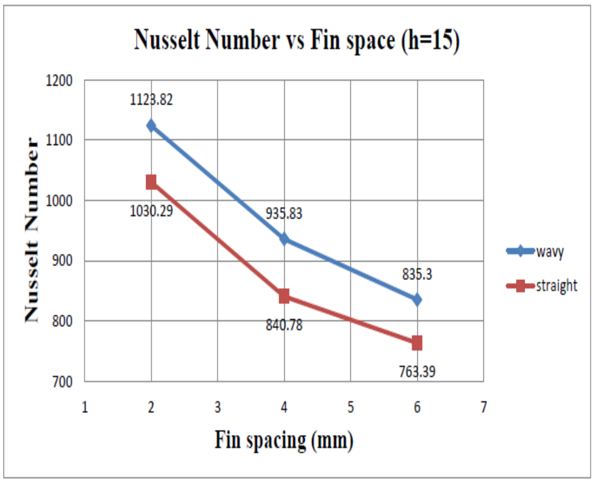
RESULT AND PLOT COMPARISON

Numerical simulation was performed to obtain Nusselt Number of the base plate. Computations were carried out using commercial available software FLUENT 6.3. A validation test was performed by using the details available in the literature. The details of the results obtained from computations are discussed in this chapter.

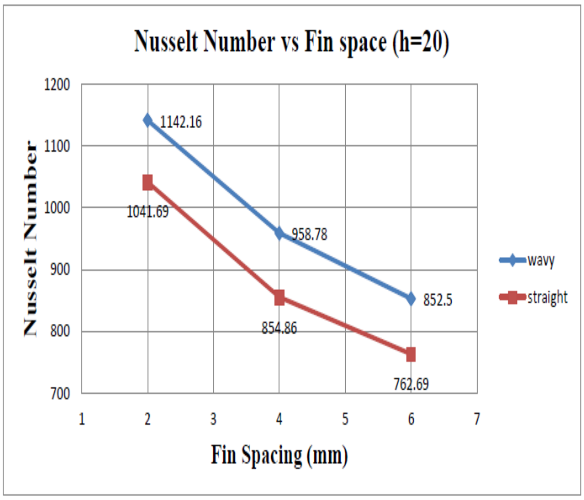
COMPARISON OF STRAIGHT FIN AND WAVY FINS FOR VARIOUS FIN HEIGHTS



Comparison plot for straight vs wavy fin for fin height 10 mm

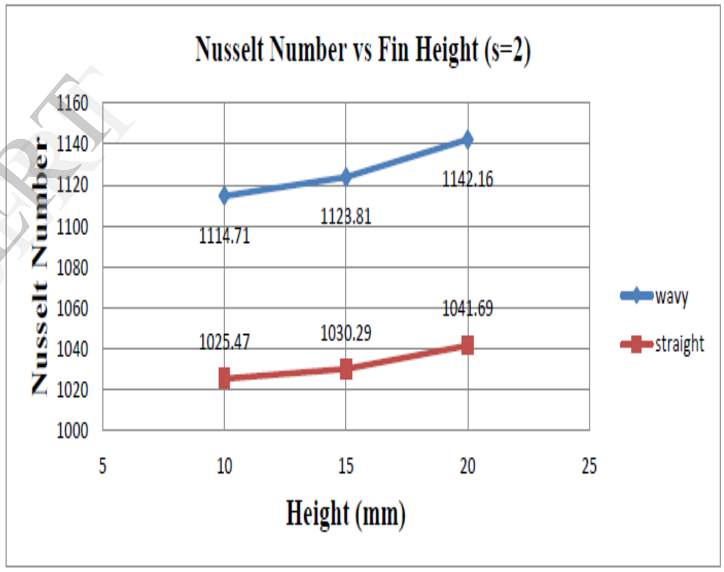


Comparison plot for Straight Vs Wavy Fin for fin height 15 mm

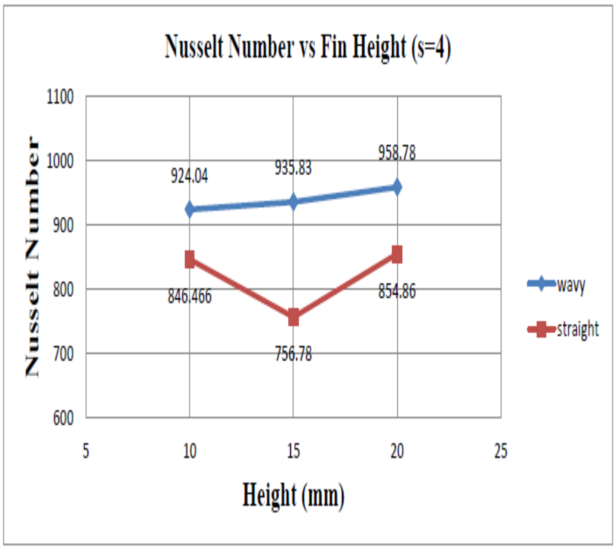


Comparison plot for Straight Vs Wavy Fin for fin height 20 mm

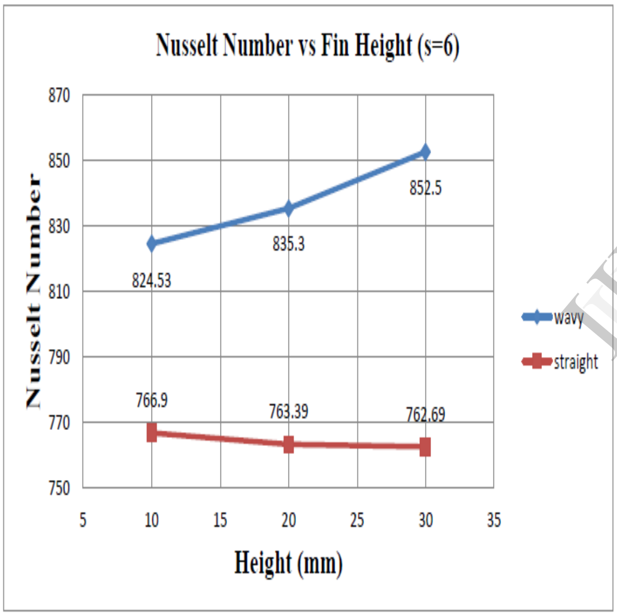
COMPARISON OF STRAIGHT FIN AND WAVY FINS FOR VARIOUS FIN SPACING



Comparison plot for Straight Vs Wavy Fin for fin spacing 2 mm

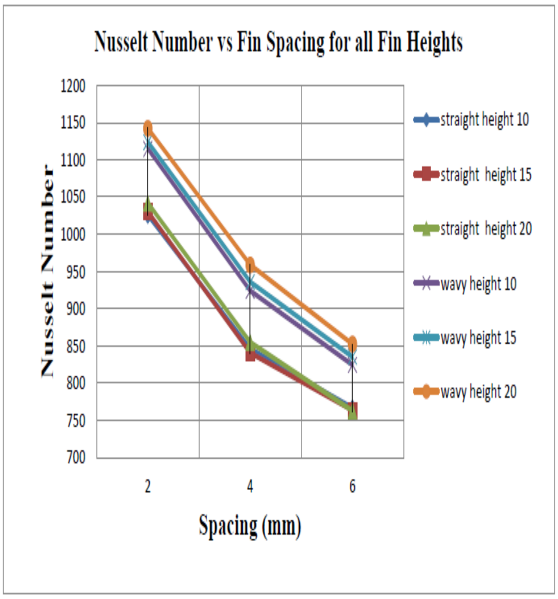


Comparison plot for Straight Vs Wavy Fin for fin spacing 4 mm

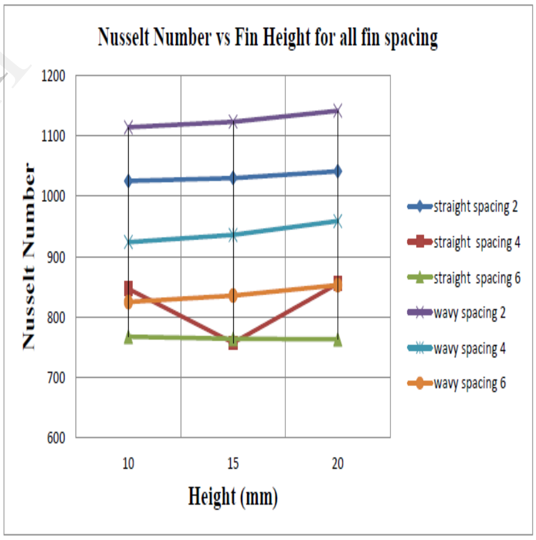


Comparison plot for Straight Vs Wavy Fin for fin spacing 6 mm

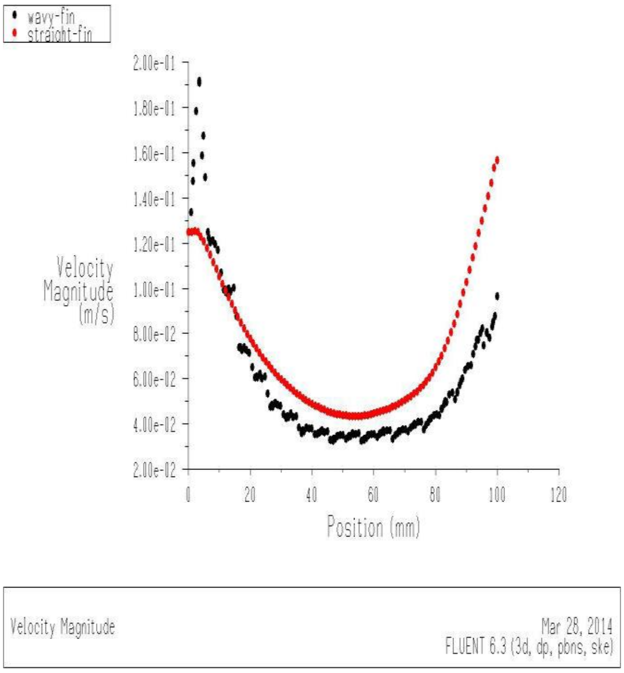
COMPARISON OF STRAIGHT FIN AND WAVY FINS FOR ALL FIN HEIGHT AND FIN SPACING



Comparison plot for Straight Vs Wavy fin at all Fin Heights



Comparison plot for Straight Vs Wavy fin at all Fin Spacing



Velocity plot for straight and wavy fins

CONCLUSION

Computational studies have been carried out to estimate the Nusselt Number of base plate with straight and wavy fins having flow velocity of 0.125m/s. Three dimensional simulations using FLUENT 6.3 have been performed. Results obtained through computations are discussed in detail in the previous chapters. Fins are the extended to increase the heat transfer rate, instead of using straight fins, wavy fins gives comparatively high heat transfer rate. It gives the gain for the design and also efficiency to the system. Fins height and spacing is a major component of increasing the heat transfer rate. The optimum spacing and height of fin is obtained by the results and its around 2mm spacing with 20mm of fin height will give the maximum heat transfer. By increasing the fin spacing beyond the optimized value, number of fins will be less so the surface area of heat transfer decreases, and if spacing is decreased further the optimized value by adding more number of fins, boundary layer interaction occurs. This affects the cold flow getting inside the fins. If the spacing is increased or decreased than the optimized value, the heat transfer rate will be decreases. If the fin height is decreased below the optimized value the flow cannot able to pass through the fin passages. This will decreases the heat transfer rate. By replacing the straight fins by wavy fins the heat transfer rate increases due to increase in surface area of heat transfer and the flow velocity is delayed between the fins which gets enough time to observe more heat from the source and it leaves the passage. Thus the optimized value of fin spacing and height is calculated numerically for both straight and wavy fins.

SCOPE OF FUTURE WORK:

Computational work is done for the both straight and wavy fins and the results are compared. The effects of fin spacing and height have been investigated. After obtaining the results it is suggested that further studies can be carried out experimentally to get in depth knowledge in this area. Some of the suggested work is outlined below.

- Effect of fin spacing and height in wavy fins
- Effect of heat transfer by introducing dimples in the wavy fins
- Effect of heat transfer by varying the curvature of the wavy fins.

REFERENCES:

[1] Experimental investigation of mixed convection heat transfer from longitudinal fins in a horizontal rectangular channel .M. Dogan, M. Sivrioglu,

[2] Analysis of heat transfer through Bi-convection fins. A.-R.A. Khaled

[3] Experimental determination of natural convection heat transfer coefficients in an attic shaped enclosure .T.N. Anderson, M. Duke, J.K. Carson

[4] Natural Convection in a Horizontal Wavy Enclosure. Slimani Abdelkader, Rebhi Mebrouk, Belkacem Abdellah, Bouhadef Khadidja.

[5] F. Harahap, H.N. McManus, Natural convection heat transfer from rectangular fin arrays, J. Heat Trans. Trans. ASME Ser. C 89 (1967) 32–38.

[6] Charles D. Jones, Lester F. Smith, Optimum arrangement of rectangular fins on horizontal surfaces for free convection heat transfer, J. Heat Trans. Trans. ASME Ser. C 92 (1970) 6–10.