© August 2015 | IJIRT | Volume 2 Issue 3 | ISSN: 2349-6002

HEAT TRANSFER ANALYSIS OF TWO ADJACENT NARROW PLATES

Pallavarapu Hari[1], Neeraj Kumar[2]
[1]M.Tech student, Malla Reddy College of Enginnering,Telangana, India.
[2]Asst.Professor, Malla Reddy College of Enginnering, Telangana, India.

***Abstract*- Numerical and analytical studies of the interaction of the natural convective flows over two adjacent vertical and inclined narrow isothermal flat plates in the laminar flow region has been discussed. In the present work two cases are considered. In one case, the plates are horizontally adjacent to each other, the plates being horizontally separated while in the other case, one plate is symmetrically placed above the other plate the plates being vertically separated. Attention has been given to the effects of the inclination angle of the plates to the vertical, to the effects of the vertical or horizontal dimensionless gap between the heated plates, and to the effects of the dimensionless plate width on the mean heat transfer rates from the two heated plates for a wide range of Rayleigh numbers. Empirical equations for both the case of horizontally separated and vertically separated plates have been given. Thermal analysis and CFD analysis has carried in Ansys for above cases and 3D models are drawn in Pro/Engineer. Thermal analysis is done on the horizontally separated plates and vertically separated plates for two materials Aluminum and Copper. By observing the results, the heat transfer rate is more for horizontally separated plates than vertically separated and copper has high heat transfer rates. The Nusselt number is increasing for horizontally separated plates which mean that the heat transfer coefficient is more thereby more heat transfer rates.**

***Index Terms*—CFD, Heat Transfer, Fluid Flow, Natural Convection, Horizontal Enclosure**

I. INTRODUCTION

Natural convection phenomena in enclosures have become one of the major topics of interest in research due to its applications involved in various engineering applications. Buoyancy driven flows have many applications in thermal engineering since passive cooling of electronic components by natural convection is the least expensive, quietest and most reliable method of heat rejection alternatives. Among these applications involving enclosures are solar energy systems, nuclear reactors and electronic packages of computer components. Heat transfer and fluid flow characteristics in closed enclosures with differently heated walls mainly depends upon the enclosure orientation, horizontal or vertical. Under certain circumstances, electronic components are packaged within sealed enclosures, while one or more of the walls are cooled. The main source of heat within the medium is electronic components or boards situated in various configurations. In the design of electronic packages, there are strong incentives to mount as much electronic components as possible in a given enclosure. This leads to high power generation density and this may raise the temperature of the packages above the allowable limit [1]. Heat transfer rate from the packages must be maximized to overcome this problem. Using finned surfaces is the most common technique for maximizing heat transfer rate. Fins orientation and geometry of fins array are the main parameters which affects the enhancement ratio of heat transfer.

As shown in the Fig.1, the cold fluid rises along the plate surface, becoming heated in the process, and the momentum boundary layer grows in thickness with distance along the plate. A sample velocity profile in the momentum boundary layer is shown.

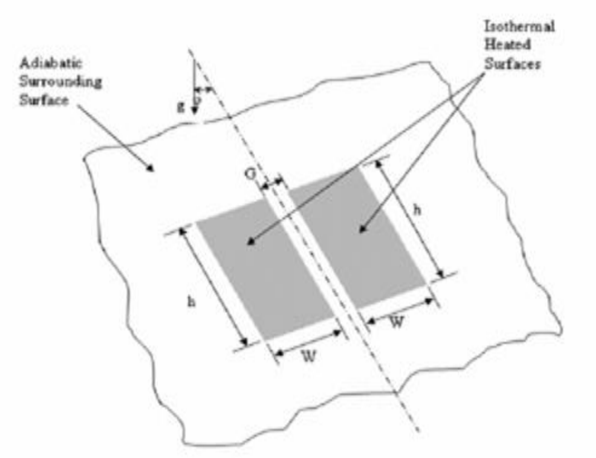


Fig.1: Horizontally adjacent plate flow situation considered.

Note that in this type of boundary layer, the velocity must be zero not only at the solid surface, but also at the edge of the boundary layer. Because the profile was sketched free-hand in PowerPoint, we are unable to show the smooth approach to zero velocity with a zero slope at the edge of the boundary layer properly, but that is how the correct velocity profile would appear. Compare this velocity profile with that in a momentum boundary layer that forms on a flat plate when fluid approaches it with a uniform velocity U∞ . We should try to make a sketch of the thermal boundary layer on the same plate when the fluid is air, for example, and also when it is a viscous liquid with a Prandtl number that is large compared with unity.

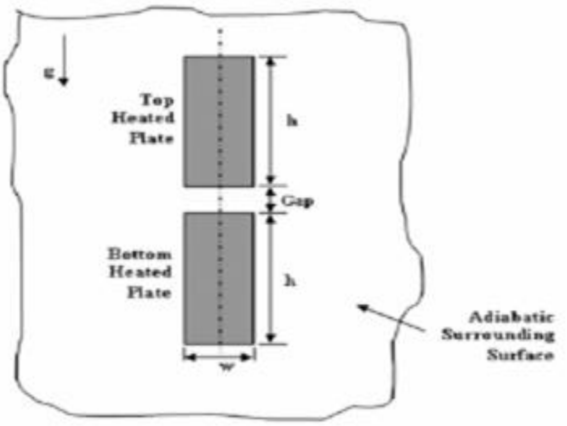


Fig. 2: Vertically adjacent plate flow situation considered.

The present work arises from the fact that when there are two adjacent narrow flat plates with a relatively small gap between the plates the flow near the adjacent plates alters the nature of the flow compared to that over a single narrow plate and this can lead to a significant change in the mean heat transfer rate compared to that from a single isolated plate under the same conditions as shown in Fig.2.

II. RELATED WORK

In literature, Numerical and/or experimental studies involving enclosures with fins or pins are very rare. Numerical experiments are conducted by Arquis and Rady [2] to investigate natural convection heat transfer and fluid flow characteristics from a horizontal fluid layer with finned bottom surface. For a limited low range of Rayleigh number, the effects of fin height, fin spacing and Rayleigh number on fin surface effectiveness have been studied. An experimental investigation of natural convection heat transfer and fluid flow in horizontal and vertical narrow enclosures with heated rectangular finned base plate for different fin spacing and fin lengths were studied by S. A. Nada [1]. He reported that Nusselt number and finned surface effectiveness increased with increasing the fin length. He also reported that the increasing the Ra for any fin geometry increases the Nusselt number it means the heat transfer is enhanced using finned base plates. 2-D Numerical analysis of natural convection in a differentially heated square cavity whose vertical walls have a finite conductivity and a thickness was studied by Raji et al. [3]. His numerical simulations of natural convection in a cavity having walls of finite width have shown the possibility to reduce significantly the heat transfer by using appropriate isolation techniques. The effects of spacing on cooling of heated electronic components and of the removal of heat input in one of the components were determined.

R. L. Frederick and S. G. Moraga [4] numerically investigated 3D natural convection in a cubical enclosure with a fin attached to the hot wall and air as fluid. The fin was horizontally attached to the hot wall, and the Rayleigh numbers ranged from 103 to 106 is studied. They concluded that for the above Rayleigh range a fin of partial width is more effective in promoting heat transfer than a fin of full width. A numerical simulation of conjugate turbulent natural convection air cooling of three heated ceramic components, mounted on a vertical adiabatic channel was studied by Bessaih and Kadja [5].

A numerical study has been carried out by E. Bilgen [6] in differentially heated square cavities, which are formed by horizontal adiabatic walls and vertical isothermal walls. He concluded that the heat transfer may be suppressed up to 38% by choosing appropriate thermal and geometrical fin parameters. An experimental investigation to study the effects of vertical fins on heat transfer rate in a horizontal fluid layer in a finite extent was carried out by Inada et al. [7]. For a single value of fin height and for limited range of Rayleigh number and fin spacing, the heat transfer rates have been reported by them.

A numerical investigation of steady state natural convection heat transfer in a longitudinally short rectangular fin array on a horizontal base was studied by Mobedi and H. Yüncü [8]. The study was limited to Rayleigh number rage ranging from 120 to 39000. The fin length and fin height were varied from 2 to 20 and 0.25 to 7-fin spacing, respectively. The mechanisms of the flows are discussed and flow patterns are plotted by them.

To the best of our knowledge, the detailed three dimensional computational analyses on heat transfer and fluid flow characteristics within a finned horizontal fluid layer is not available in the literature. In this paper the steady-state natural convection from heat sinks with parallel arrangement of rectangular cross section vertical plate fins on a vertical base are numerically investigated in order to obtain a validated model that is used for investigating inclined orientations of a heat sink. Taking a previous experimental study as a basis, aluminum heat sinks with two different practical lengths are modeled. The models and the simulation approach are validated by comparing the flat plate heat sink results with the available correlations, and by comparing the finned heat sink results with the experimental data. Natural convection and radiation heat transfer rates from the fronts of the heat sinks heated from the back with a heater are obtained from finite volume computational fluid dynamics simulations. The sensitivities of the heat transfer rates to the geometric parameters are determined. A set of dimensionless correlations for the convective heat transfer rate is suggested. The validated model is used for several upward and downward inclination angles by varying the direction of gravitational acceleration. At small inclinations, it is observed that convection heat transfer rate stays almost the same, even increases slightly for the downward inclinations.

At larger angles, the phenomenon is investigated for the purpose of determining the flow structures forming around the heat sink. For the inclination angles of ±4°, ±10°, ±20°, ±30°, ±45°, ±60°, ±75°, +80°, ±85°, ±90° from the vertical, the extent of validity of the obtained vertical case correlation is investigated by modifying the Grashof number with the cosine of the inclination angle. It is observed that the correlation is valid in a very wide range, from −60° (upward) to +80° (downward). It is also observed that the flow separation inside the fin channels of the heat sink is an important phenomenon and determines the validity range of the modified correlation. It is further shown that the correlations are also applicable to all available inclined case data in the literature, verifying both our results and correlations. Since the investigated ranges of parameters are suitable for electronic device cooling, the suggested correlations have a practical use in electronics cooling applications.

III. PROPOSED MODEL

A. CFD ANALYSIS OF ADJACENT PLATES

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows.

The Navier–Stokes equations were the ultimate target of developers. Two-dimensional codes, such as NASA Ames' ARC2D code first emerged. A number of three-dimensional codes were developed (ARC3D, OVERFLOW, CFL3D are three successful NASA contributions), leading to numerous commercial packages.

METHODOLOGY

• In all of these approaches the same basic procedure is followed.

• During preprocessing

• The geometry (physical bounds) of the problem is defined.

• The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.

• The physical modeling is defined – for example, the equations of motion + enthalpy + radiation + species conservation

• Boundary conditions are defined. This involves specifying the fluid behavior and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.

• The simulation is started and the equations are solved iteratively as a steady-state or transient.

• Finally a postprocessor is used for the analysis and visualization of the resulting solution.

Discretization methods

The stability of the selected discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier–Stokes equations both admit shocks, and contact surfaces. Some of the discretization methods being used are:

Finite volume method

The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high number turbulent flows, and source term dominated flows (like combustion).

Finite element method

The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids. However, the FEM formulation requires special care to ensure a conservative solution. However, FEM can require more memory and has slower solution times than the FVM. In this method, a weighted residual equation is formed:

$$\frac{\partial}{\partial t}\iiint Q\,dV + \iint F\,d\mathbf{A} = 0,$$

where is the equation residual at an element vertex , is the conservation equation expressed on an element basis, is the weight factor, and is the volume of the element.

Finite difference method

The finite difference method (FDM) has historical importance and is simple to program. It is currently only used in few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids (with the solution interpolated across each grid).

$$R_i = \iiint W_i Q\,dV^e$$

where is the vector of conserved variables, and , , and are the fluxes in the , , and directions respectively.

Spectral element method

Spectral element method is a finite element type method. It requires the mathematical problem (the partial differential equation) to be cast in a weak formulation. This is typically done by multiplying the differential equation by an arbitrary test function and integrating over the whole domain.

Boundary element method

In the boundary element method, the boundary occupied by the fluid is divided into a surface mesh.

High-Resolution Discretization Schemes

High-resolution schemes are used where shocks or discontinuities are present. Capturing sharp changes in the solution requires the use of second or higher-order numerical schemes that do not introduce spurious oscillations. This usually necessitates the application of flux limiters to ensure that the solution is total variation diminishing.

Turbulence Models

For turbulent flows, the range of length scales and complexity of phenomena involved in turbulence make most modeling approaches prohibitively expensive; the resolution required to resolve all scales involved in turbulence is beyond what is computationally possible. The primary approach in such cases is to create numerical models to approximate unresolved phenomena. This section lists some commonly-used computational models for turbulent flows.

Turbulence models can be classified based on computational expense, which corresponds to the range of scales that are modeled versus resolved (the more turbulent scales that are resolved, the finer the resolution of the simulation, and therefore the higher the computational cost).

Reynolds-Averaged Navier–Stokes

Reynolds-averaged Navier-Stokes (RANS) equations are the oldest approach to turbulence modeling. An ensemble version of the governing equations is solved, which introduces new apparent stresses known as Reynolds stresses.

RANS models can be divided into two broad approaches:

Boussinesq Hypothesis

This method involves using an algebraic equation for the Reynolds stresses which include determining the turbulent viscosity, and depending on the level of sophistication of the model, solving transport equations for determining the turbulent kinetic energy and dissipation

Reynolds Stress Model (Rsm)

This approach attempts to actually solve transport equations for the Reynolds stresses. This means introduction of several transport equations for all the Reynolds stresses and hence this approach is much more costly in CPU effort.

IV. SIMULATION RESULTS

V.

A. CFD ANALYSIS ON NATURAL CONVECTION HEAT TRANSFOR FOR TWO ADJACENT NARROW PLATES SEPERATED HORIZONTALLY

VELOCITY 7.5264 m/s

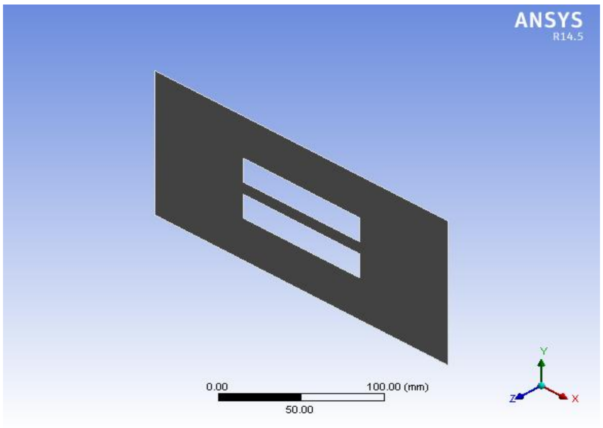
Ansys → workbench→ select analysis system → fluid flow fluent → double click

→→Select geometry → right click → import geometry → select browse →open part → ok

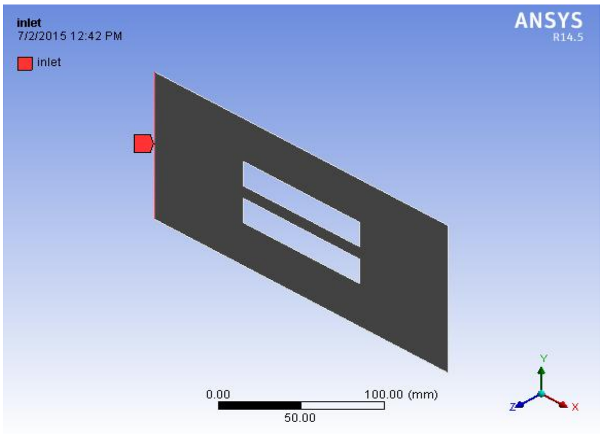
→→ Select mesh on work bench → right click →edit → select mesh on left side part tree → right click → generate mesh →

GEOMETRY

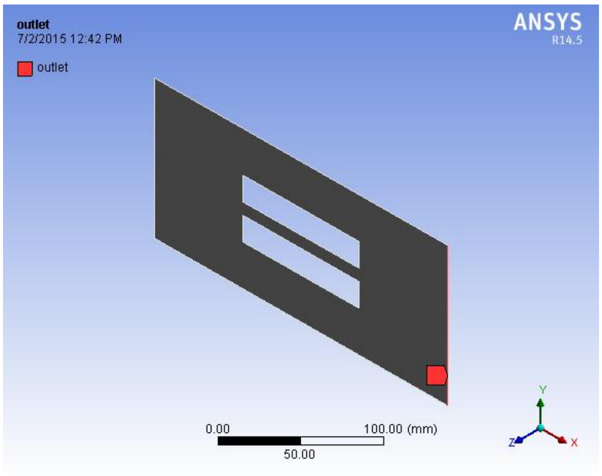
The following figures gives the idea about CFD analysis of natural convection heat transfer.



Select faces → right click → create named section → enter name → air inlet



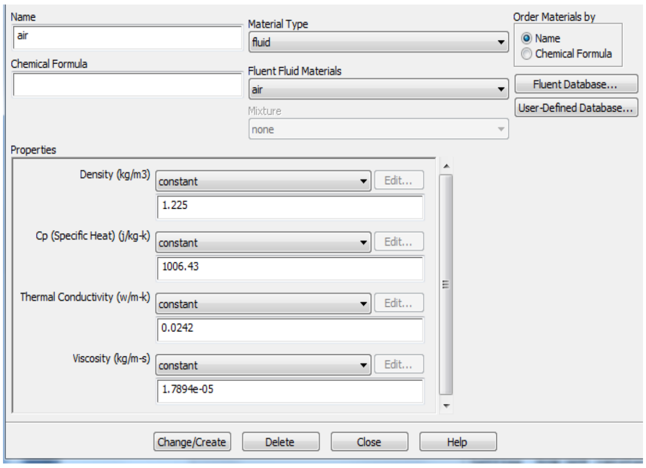
Select faces → right click → create named section → enter name → air outlet



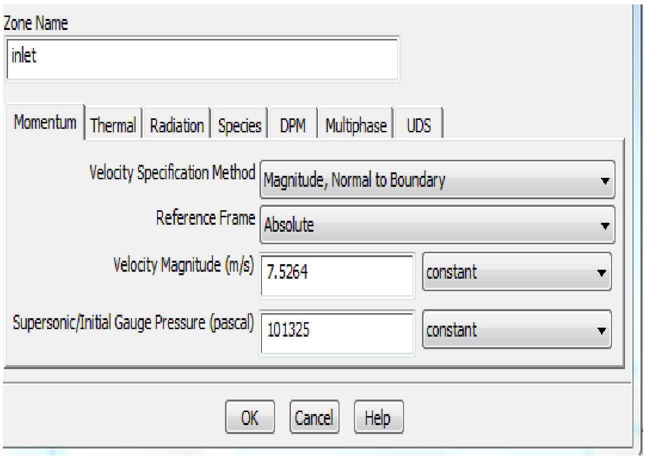
Update project>setup>edit>model>select>energy equation (on)>ok

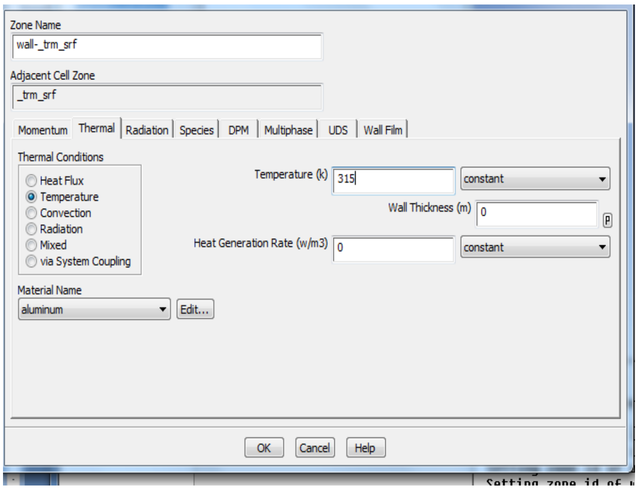
Materials> Materials > new >create or edit >specify fluid material or specify properties > ok

Select air



Boundary conditions>inlet>enter required inlet values



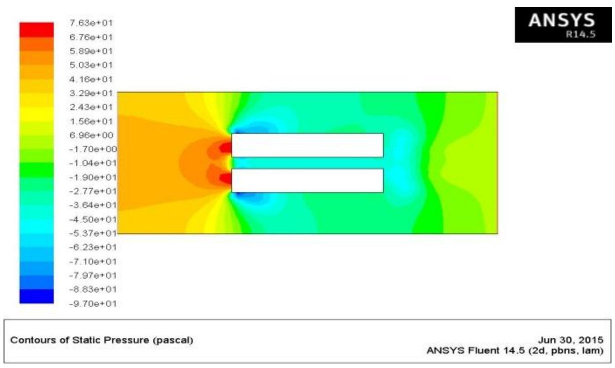


Solution > Solution Initialization > Hybrid Initialization >done

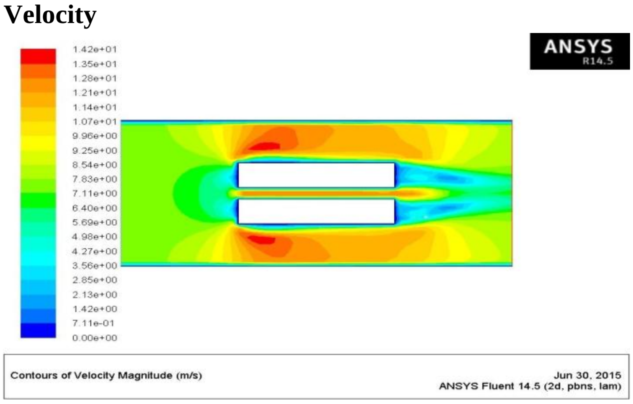
Run calculations > no of iterations = 10> calculate > calculation complete>ok

Results>edit>select contours>ok>select location (inlet, outlet, wall.etc)>select pressure>apply

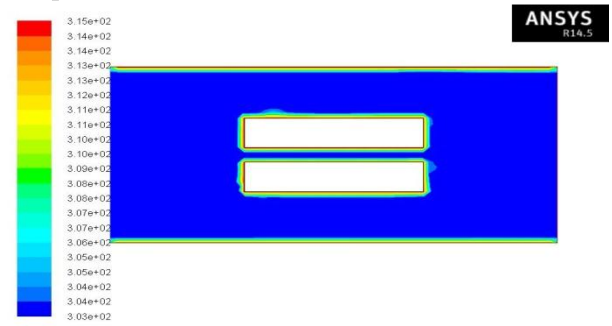
Pressure



Velocity

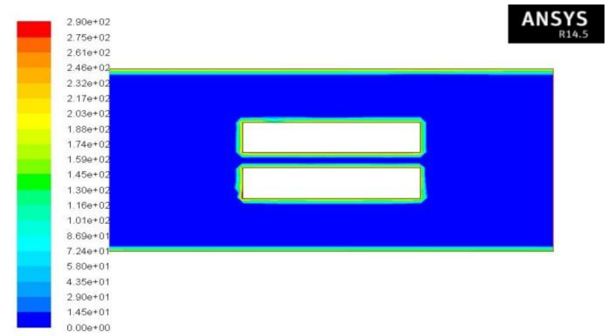


Temperature



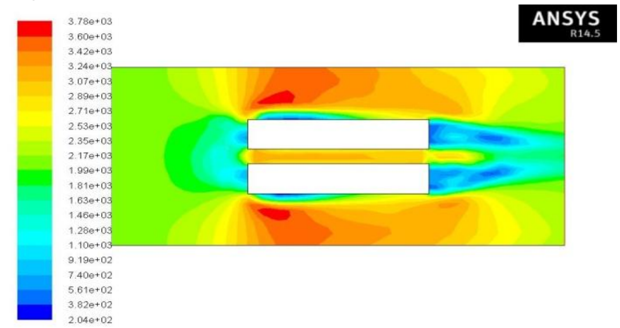
Contours of Static Temperature (K) Jun 30, 2015 ANSYS Fluent 14.5 (2d, pbns, lam)

Nusselt number



Contours of Surface Nusselt Number Jun 30, 2015 ANSYS Fluent 14.5 (2d, pbns, lam)

Reynolds number

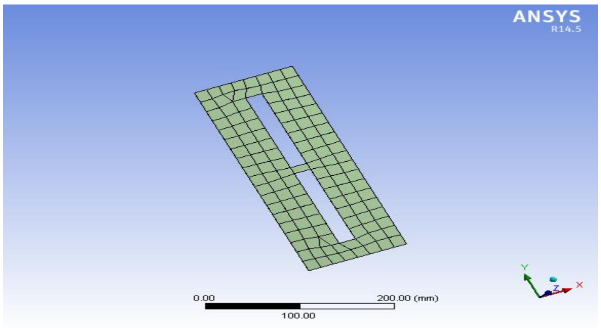


Contours of Cell Reynolds Number Jun 30, 2015 ANSYS Fluent 14.5 (2d, pbns, lam)

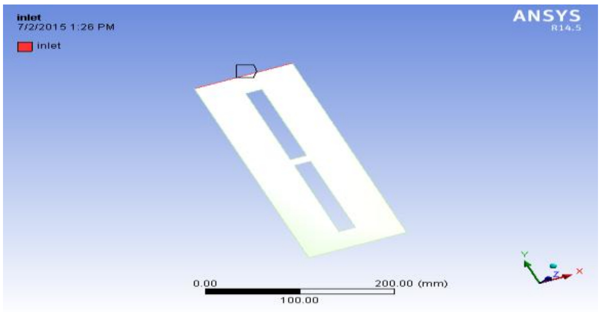
B. CFD ANALYSIS ON NATURAL CONVECTION HEAT TRANSFER FOR TWO ADJACENT NARROW PLATES SEPERATED VERTICALY VELOCITY 7.5264 m/s : Ansys → workbench→ select analysis system → fluid flow fluent → double click →→Select geometry → right click → import geometry → select browse →open part → ok

→→ Select mesh on work bench → right click →edit → select mesh on left side part tree → right click → generate mesh →

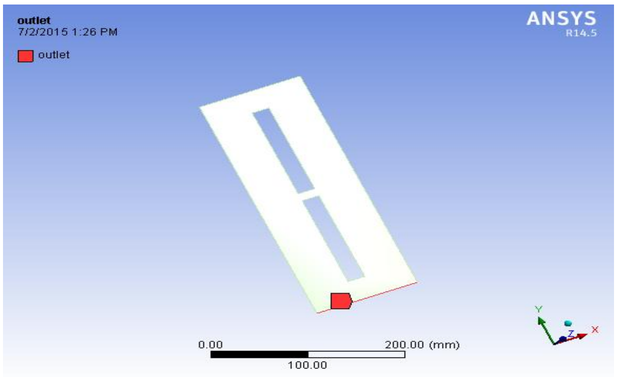
GEOMETRY



Select faces → right click → create named section → enter name → air inlet



Select faces → right click → create named section → enter name → air outlet



Update project>setup>edit>model>select>energy equation (on)>ok

Materials> Materials > new >create or edit >specify fluid material or specify properties > ok

Select air

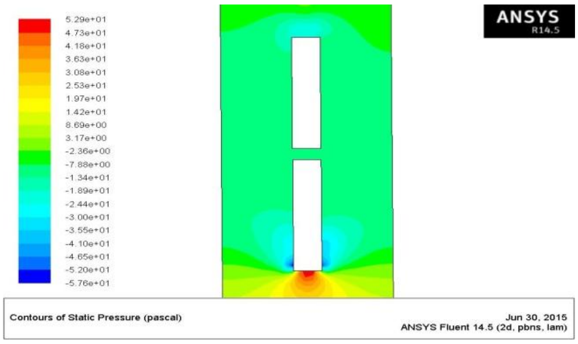
Boundary conditions>inlet>enter required inlet values

Solution > Solution Initialization > Hybrid Initialization >done

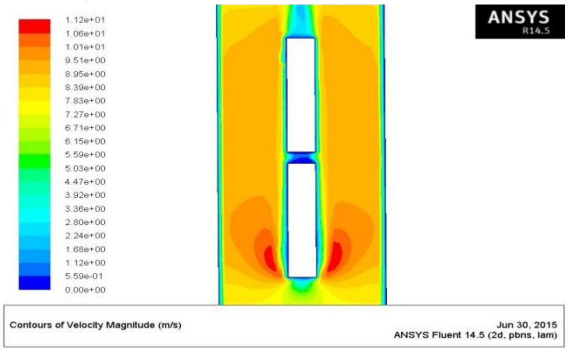
Run calculations > no of iterations = 10> calculate > calculation complete>ok

Results>edit>select contours>ok>select location (inlet, outlet, wall.etc)>select pressure>apply

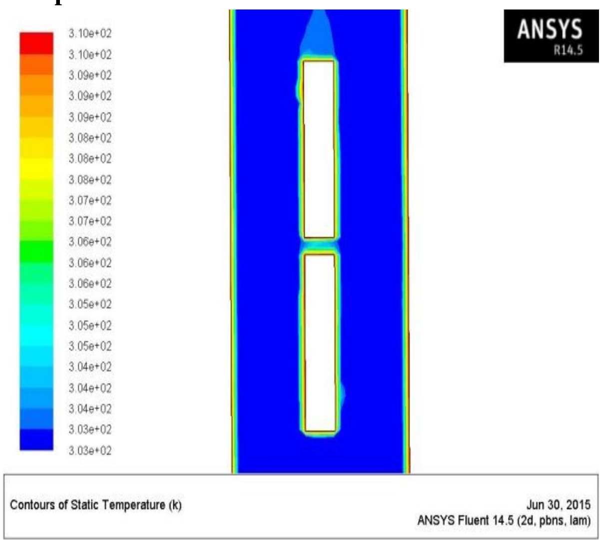
Pressure



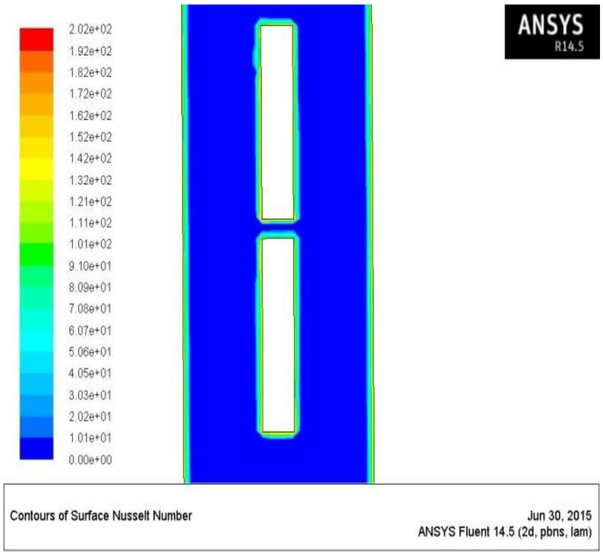
Velocity



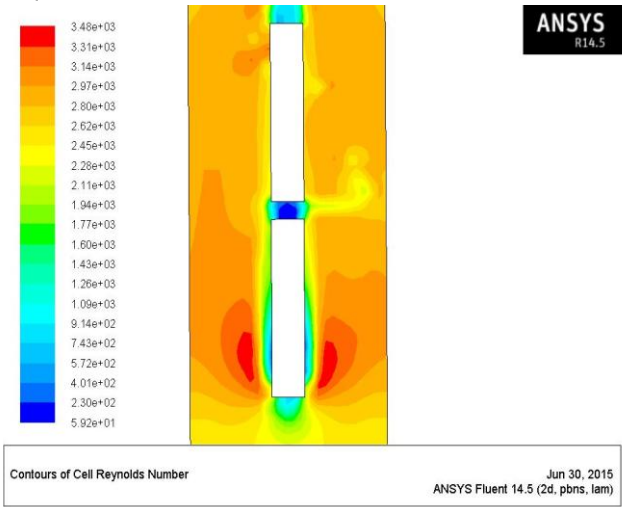
Temperature



Nusselt number



Reynolds number



VI. RESULTS THERMAL ANALYSIS RESULTS

VERTICALLY SEPERATED

MATERIAL	TEMPERATURE (K)	THERMALGRADIENT (K/mm)	THERMAL FLUX (w/mm²)
ALUMINUM	313.06	6.37931	1.33966
COPPER	312.692	5.30097	2.04087

6.1.2. HORIZONTALLY SEPERATED

MATERIAL	TEMPERATURE (K)	THERMALGRADIENT (K/mm)	THERMAL FLUX (w/mm^2)
ALUMINUM	313.009	6.1206	1.28533
COPPER	303.565	0.56953	21.927

CFD ANALYSIS RESULTS

	HORIZONTAL PLATES	VERTICAL PLATES
PRESSURE (Pa)	$7.63e^{+01}$	$5.29e^{+01}$
VELOCITY (m/s)	$1.42e^{+01}$	$1.12e^{+01}$
TEMPERATURE (K)	$3.15e^{+02}$	$3.10e^{+02}$
NUSSELT NUMBER	$2.90e^{+02}$	$2.02e^{+02}$
REYNOLDS NUMBER	$3.78e^{+03}$	$3.48e^{+03}$

VII. CONCLUSON

In this thesis, heat transfer and fluid flow characteristics are investigated under natural convection for two cases. In one case, the plates are horizontally adjacent to each other, the plates being horizontally separated while in the other case, one plate is symmetrically placed above the other plate the plates being vertically separated. 3D models are done in Pro/Engineer. Thermal analysis is done on the horizontally separated plates and vertically separated plates for two materials Aluminum and Copper. By observing the results, the heat transfer rate is more for horizontally separated plates than vertically separated and copper has high heat transfer rates.

The change in the separation has effect on the Nusselt number and Reynolds number which can be observed from the CFD results. The Nusselt number is increasing for horizontally separated plates which mean that the heat transfer coefficient is more thereby more heat transfer rates.

REFERENCES

[1] S.A. Nada,"Natural convection heat transfer in horizontal and vertical closed narrow enclosures with heated rectangular finned base plate", International Journal of Heat and Mass Transfer 50, 2007, pp. 667–679.

[2] E. Arquis, M. Rady,"Study of natural convection heat transfer in a finned horizontal fluid layer", Int. J. Therm. Sci. 44 2005, pp. 43–52.

[3] A. Raji, M. Hasnaoui, S. Goujon-Durand, P. Vasseur, "Numerical study of the effect of the insulation of the walls on natural convection in a differentially heated cavity", International Scientific Journal for Alternative Energy and Ecology, 62 (6), 2008, 122-130.

[4] R. L. Frederick, S.G. Moraga,"Three-dimensional natural convection in finned cubical enclosures", International Journal of Heat and Fluid Flow 28, 2007, pp. 289-298.

[5] R. Bessaih, M. Kadja,"Turbulent natural convection cooling of electronic components mounted on a vertical channel", Appl. Therm. Eng. 20, 2000, pp. 141–154.

[6] E. Bilgen,"Natural convection in cavities with a thin fin on the hot wall", International Journal of Heat and Mass Transfer 48, 2005, pp. 3493-3505.

[7] S. Inada, T. Taguchi, W.J. Yang,"Effects of vertical fins on local heat transfer performance in a horizontal fluid layer", Int. J. Heat Mass Transfer 42, 1999, pp. 2897–2903.

[8] M. Mobedi, H. Yüncü,"A three dimensional numerical study on natural convection heat transfer from short horizontal rectangular fin array", Heat and Mass Transfer 39, 2003, pp. 267-275.

[9]. Natural convection from inclined plates to gases and liquids when both sides are uniformly heated at the same temperature by Massimo Corcione, Emanuele Habib, Antonio Campo, International Journal of Thermal Sciences 50 (2011) 1405-1416

[10]. Natural convection heat transfer from inclined plate-fin heat sinks by Ilker Tari, Mehdi Mehrtash, International Journal of Heat and Mass Transfer, Volume 56, Issues 1–2, 1 January 2013, Pages 574–593

BIODATA

AUTHOR 1

Pallavarapu Hari , currently pursing his M.Tech in Thermal Engineering from Malla Reddy College of Enginnering,Telangana, India .

AUTHOR 2

Neeraj Kumar working as Asst. Professor in Malla Reddy College of Enginnering,Telangana, India .