International Journal of Advanced Information in Engineering Technology (IJAIET)
Vol.5, No.5, May 2018 ISSN: 2454-6933

COMPUTATIONAL STUDY OF THROUGH FLOW VARIATION IN RAYLEIGH-BENARD CONVECTION

Arjun G Nair, Sathil P.T, Prasidh E Prakash *Malabar College of Engineering and Technology, Kerala Technological University*

arjunagn911@gmail.com,ptsathil88@gmail.com,prasidhearath@gmail.com

***Abstract*— the thermal convection phenomena, which occur in the two dimensional fluid layer of definite length is studied. The layer is heated from below and simultaneously cooled at the top. As a result the particles of the fluid begin to move creating convectional rolls. These phenomena are known as Rayleigh-Benard convection.**

In this project CFD analysis is carried out to study the effect of through flow on large scale flows by varying the through flow velocity from 0 to 0.01cm/s in the Y-direction at an aspect ratio of 8. The simulation is carried out both in water and in air as medium of convection, and the results are compared. Convection due to density flow can also be due to temperature gradients in the fluid. In this study there are two plates placed one over the other with a small spacing, here the plate placed below is heated and the plate above is at lower temperature.

In near wall regions, line plumes are initiated. The line plumes have a finite thickness and a finite height, beyond which they breakup into mushroom type plumes. The line plumes elongate and merge with the adjacent sheets, a new sheet being initiated in the space vacated by the merger. The external shear due to the large scale circulation modifies these dynamics by aligning the sheets in the direction of the shear. However the plumes merged faster with increase in Ra and the mean of the spacing between the line plumes (h) at any instant decreased with increase in Ra. The lengths of the line plumes are easily and accurately measured compared to the normal spacing between them even if the plumes are oriented randomly. Since the line plumes carry the major part of the heat flux, characterizing the plume structure in terms of plume length could lead to unique relations between the length of the near-wall coherent structures.

***Index Terms*—- through flow, thermal diffusivity aspect ratio viscosity, Rayleigh number, Prandtl number.**

I. INTRODUCTION

Natural convection is a type of mass transport, in which the fluid is moving only by density differences in the fluid due to the temperature or concentration gradients. By the e effect of any of these gradients, density difference is generated inside the fluid. Thus there will be fluid with less and higher density. The sole driving potential for natural convection is buoyancy, a result of differences in fluid density caused by temperature or concentration differences. Since buoyancy is coming into the action, gravitational field has an important place for the entire process to occur.

There are two types of natural convection processes namely, direct and indirect. If the density gradients in a non vertical plane are created directly, it is a direct natural convection or else it is an indirect natural convection. An example of direct natural convection is a heated horizontal plate while convection over heated horizontal surface is an example of indirect natural convection. Some of the main types of indirect natural convection are Rayleigh-Benard convection, unsteady non-penetrative convection etc. In this project we study natural convection occurs due to the density difference produced by the concentration gradient.

In this type of flow, large thermal gradients exist near the wall(s) while near isothermal conditions prevail away from the wall(s). Experimental evidence suggests the convective heat transfer near the wall(s) is either from thermals (intermittent release of hot fluid) or essentially line plumes (continuous release of hot fluid from a line) which move randomly. Away from the wall(s) rapid mixing leads to near-isothermal conditions. The line plumes randomly move about on the surface, inclined forward in the direction of motion. Adjacent plumes merge with one another most of the time and rarely disappear.

The following non-dimensional parameters characterize turbulent natural convection. Rayleigh number, Ra=g CH^3/(D) is the ratio of the driving buoyancy forces to the restraining dissipative effects; The Prandtl number (Pr) or Prandtl group is a dimension-less number, named after the German physicist Ludwig Prandtl, defined as the ratio of momentum diffusivity to thermal diffusivity. Pr= /_, It gives the relative height of the viscous and species boundary layers in the problem. Aspect ratio, AR=L/H, is a geo-metric parameter. The governing equations in natural convection are continuity equation, momentum equation

Continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} = 0$$

Momentum equation

X-momentum equation

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=\frac{\partial p}{\partial x}+Sc\square\left(\frac{\partial\square u}{(\partial x\square)}\right)+\frac{(\partial\square u)\square}{(\partial y\square)}$$

Y-momentum equation

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=\frac{\partial p}{\partial x}+Sc\square\left(\frac{\partial\square v}{(\partial x\square)}\right)+\frac{(\partial\square v)\square}{(\partial y\square)}+(Ra.Sc.\Delta C)$$

Schmidt number (Sc) is a dimensionless number defined as the ratio of momentum diffusivity (viscosity) and mass diffusivity, and is used to characterize fluid flows in which there are simultaneous momentum and mass diffusion convection processes. This number is the counter part of Prandtl number in the thermal natural convection process in concentration based natural convection process; C is the change in the concentration of the fluid.

1. Near wall flow structures

From the visualizations done in different experiments it is clear that mass transport in natural convection due to unstable density difference happens due to different flow structures such as

a. Line Plumes: Initially a thin layer of less dense fluid is formed above the membrane. Due to the gravitational instability, eruption of this fluid layer occurs causing a continuous release of less dense fluid from a line. This flow structure is known as sheet plumes. These line plumes move randomly over the membrane inclined in the direction of motion, adjacent plumes move close to other and merge to become a single plume.

b. Large Scale Flow: Large scale flows or winds are the bulk circular motion of fluid observed in the two tanks which are driven by plume columns rising along the edges or the corners of the tank.

c. Through flow: When the denser fluid starts to move downwards due to the gravitational instability, the lighter fluid in the lower tank has to move upwards through the permeable membrane.

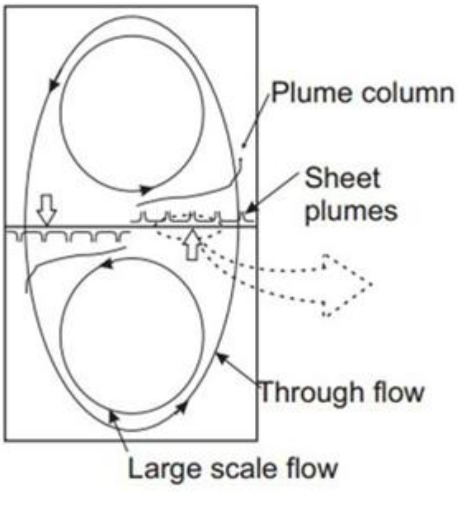

Fig. 1 near wall flow structures

II.METHODOLOGY

a. Description of the geometry

Figure below represents the mesh geometry of domain considered for the numerical analysis. Rectangular cross section is considered. For computational domain, which is, considered as a channel of height H = 1cm and L = 8cm.

There are fixed plates at the top and at the bottom and the temperature at the bottom is higher. The temperature difference of the plates is 10K. Hot plate has a temperature of 310K and cold plate has a temperature of 300K. Air is the fluid between the plates. Convection appears when the temperature gradient is big enough; consequently a small packet of fluid starts to move up into the colder region of higher density. If the buoyant force caused by difference of density is big enough, then the pocket moves upward so fast that the temperature cannot drop and the convective flow appears. There is also possible that the buoyant force is not strong enough, in such a situation the temperature of the pocket is able to drop before it can move up too much, and as a result fluid stays stable.

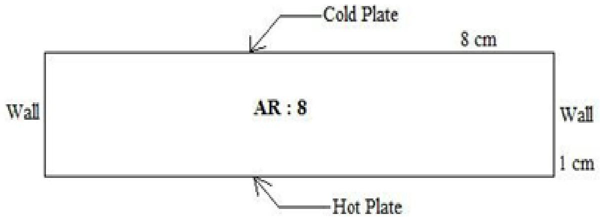

Fig. 2 Geometry of model

b. Boundary Conditions

The boundary conditions are as follows. The ends are considered as walls and no external velocities are give to the domain. Symmetric boundary condition is specified at both side-ends. A constant temperature condition was set at the walls. The bottom wall temperature was fixed at 310 K for all simulations. The top wall was set at 300 K. The side walls are adiabatic. The governing equation in Cartesian coordinates is: Continuity equation, Momentum equations, Energy equation. ANSYS 14.5 design modeler was used.

c. CFD program simulation

ANSYS Fluent is the most powerful computational fluid dynamics (CFD) software tool available, empowering to go further and faster as optimize product performance. Fluent includes well-validated physical modeling capabilities to deliver fast, accurate results across the widest range of CFD and multi physics applications. ANSYS Fluent software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from airflow over an aircraft wing to combustion in a furnace, from bubble columns to oil plat-forms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Special models that give the software the ability to model in-cylinder combustion, aero acoustics, turbo machinery, and multiphase systems have broadened its reach. ANSYS publishes engineering analysis software across a range of disciplines including finite element analysis, structural analysis, computational fluid dynamics, explicit and implicit methods, and heat transfer. The commercial computational fluid dynamics code FLUENT was used to solve the governing equations

d. Meshing of Geometry

Geometry was created with required dimensions. Meshing of the geometry was done by using ICEM CFD. Since the aim of the simulation was to study plumes and the large scale flows, the domain has to be meshed so that the solution can capture the large scale flows. Structured quadrilateral mesh is used with rectangular elements. Total number of elements in the mesh is 15000 and number of nodes is 15251. The case was set with a pressure based, segregated, steady solver with Green-Gauss Cell Based gradient treatment. Since the domain had no curved or irregular boundaries, the mesh had able to be made in perfect rectangular cell so as to avoid the quality issues related to the mesh characteristics

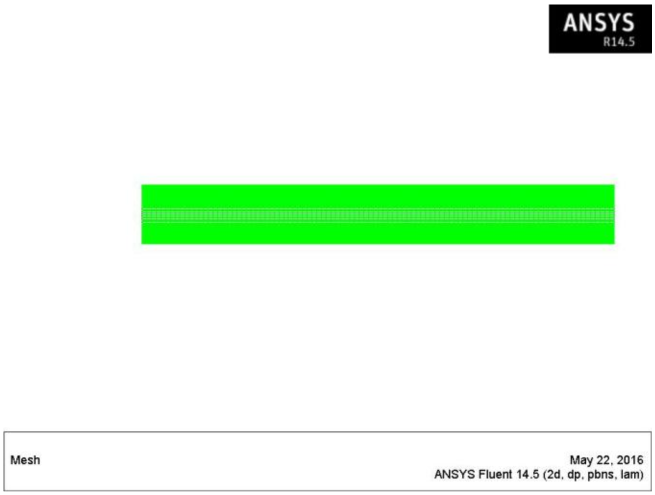

Fig.3 Geometry of Mesh

III. RESULTS AND DISCUSSIONS

The Computer fluid Dynamics program was used in order to create simulation of convection. Air and water properties were introduced as an input values, the temperature was set as: 310K at the bottom and 300K at the top of the fluid layer. Simulations were carried out using grids of 150100 cells with refinement towards the plates. Results are taken from CFD posts. From the posts of temperature, velocity and density the following results are obtained. The result of the simulation is as follows.

Convection appears when the temperature gradient is big enough; consequently fluid starts to move up from hotter region into the colder region of higher density. If the buoyant force caused by difference of density is big enough, then the fluid moves upward so fast that the temperature cannot drop and the convective flow appears.

1. Rayleigh-Benard convection of Air

The changes of temperature distribution in the fluid layer in air as medium for an aspect ratio of 8 for through flow velocities from 0 to 0.01 cm/s is shown below

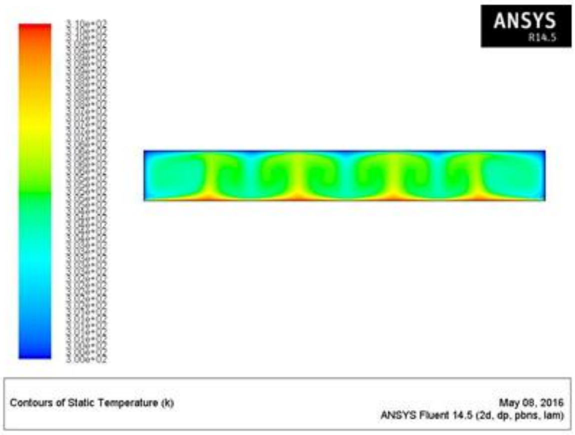

Fig.4 Temperature contour without through flow (AR: 8, Ra=4.14 $*10^5$)

The temperature contour of Rayleigh-Benard convection of air without through flow is shown above. Four plumes is generated when no through flow is given, when no through flow is given the lighter fluid moves up due to buoyancy force and denser fluid moves downward.

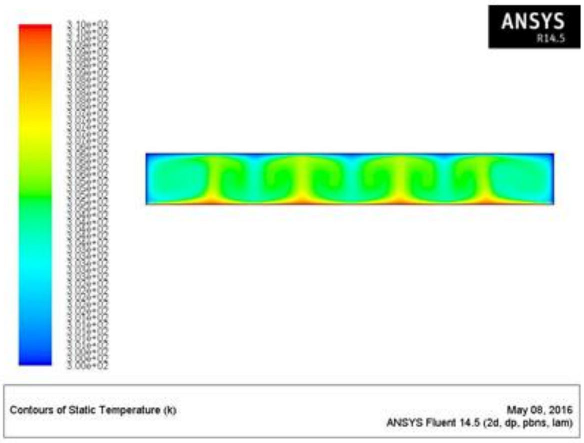

Fig.5 Temperature contour for through flow of 0.002cm/s (AR: 8, Ra=4.14 $*10^5$)

The temperature contour of Rayleigh-Benard convection of air with through flow of 0.002cm/s is shown above. Four plume are also generated at 0.002cm/s through flow velocity, but the spacing between the plumes when measured was found to decrease

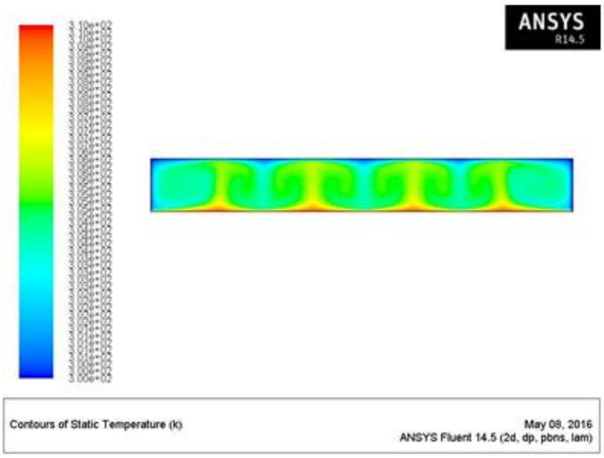

Fig.6. Temperature contour for through flow of 0.004cm/s (AR: 8, Ra=4.14 $*10^5$)

The temperature contour of Rayleigh-Benard convection of air with through flow of 0.004cm/s is shown above. Fluid near the hot plate is less dense compared to the fluid near cold plate. As the through flow velocity is increased from 0.002 to 0.004 cm/s the plume spacing was decreased.

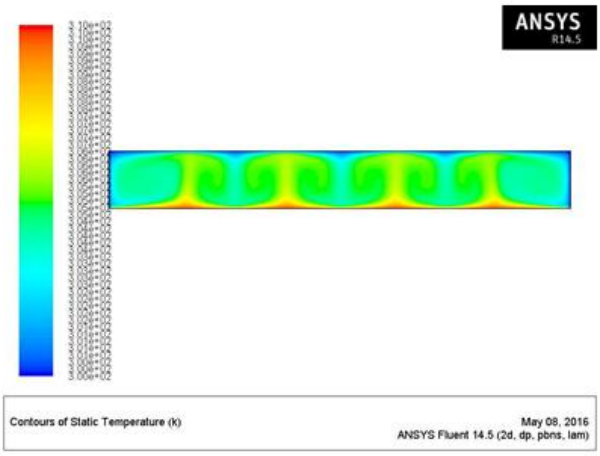

Fig.7 Temperature contour for through flow of 0.006cm/s (AR: 8, Ra=4.14 $*10^5$)

The temperature contour of Rayleigh-Benard convection of air with through flow of 0.006cm/s is shown above. Due to buoyancy force lighter fluid moves upward denser fluid moves downward. The mixing of denser and lighter fluid increases as the through flow is increased. As the through flow velocity is increased from 0.004 to 0.006 cm/s the plume spacing was decreased.

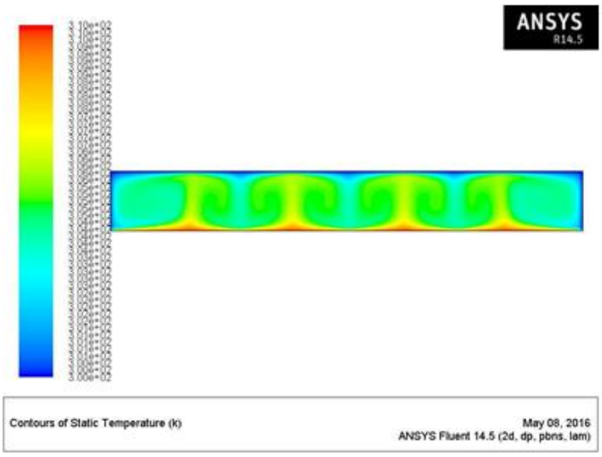

Fig.8 Temperature contour for through flow of 0.008cm/s (AR: 8, Ra=4.14 $*10^5$)

The temperature contour of Rayleigh-Benard convection of air with through flow of 0.008cm/s is shown above. The mixing of denser and lighter fluid further is increased as the through flow is increased. In this four plumes were also visible but plume spacing decreased with increase in the through flow.

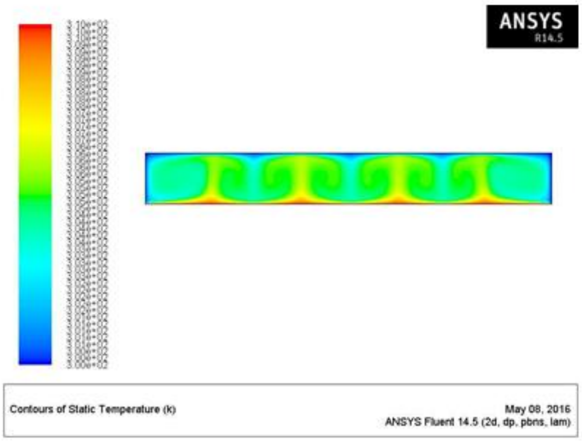

Fig.9 Temperature contour for through flow of 0.01cm/s (AR: 8, Ra=4.14* 10^5)

The figures show the temperature contour of Rayleigh Benard convection by CFD simulation with through flow velocities varying from 0 to 0.01cm/s. Fluid near the hot plate is less dense compared to the fluid near cold plate. Due to buoyancy force lighter fluid moves upward denser fluid moves downward. Plume is generated at the edges of the walls and it coagulates and flows upward. These two flows merge together and form a large scale flow. The mixing of denser and lighter fluid increases as the through flow is increased. In this four plumes are clearly visible and plume spacing decreases with increase in the through flow. The changes of velocity distribution in the fluid layer in air as medium for an aspect ratio of 8 for through flow velocities from 0 to 0.01 cm/s is shown below.

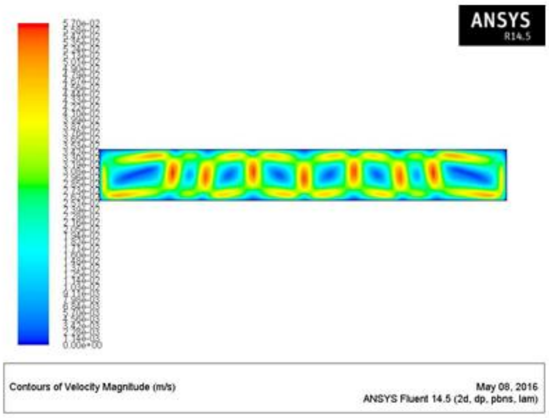

Fig.10 Velocity contour without through flow (AR: 8, Ra=4.14 $*10^5$)

The velocity contour of Rayleigh-Benard convection of air without through flow is shown above. The velocity distribution in the fluid layer (aspect ratio 8) shows that the velocity increases, while moving to the centre, compared to the walls.

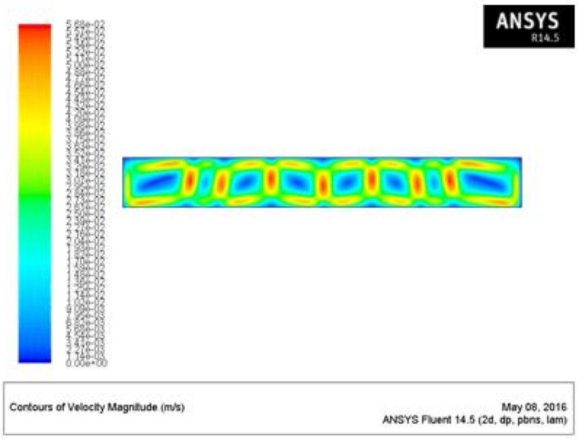

Fig.11 Velocity contour for through low of 0.002cm/s (AR: 8, Ra=4.14$*10^5$)

The velocity contour of Rayleigh-Benard convection of air with through flow velocity of 0.002cm/s is shown above. It is clear from the figure that when through flow is given the mixing of denser and lighter fluids becomes faster and the velocity in the mixing area is more.

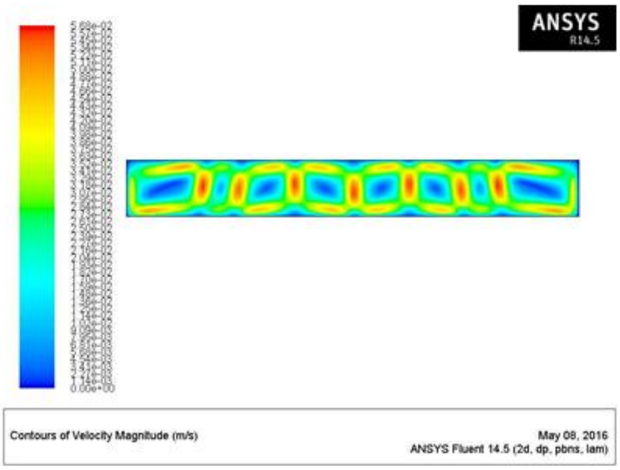

Fig.12 Velocity contour for through flow of 0.004cm/s (AR: 8, Ra=4.14$*10^5$)

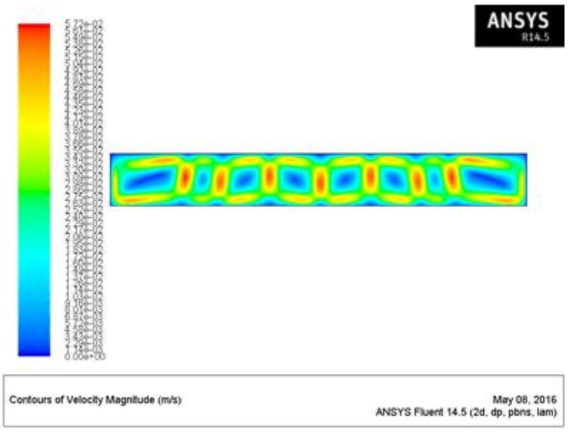

Fig.13 Velocity contour for through flow of 0.006cm/s (AR: 8, Ra=4.14$*10^5$)

The velocity contour of Rayleigh-Benard convection of air with through flow velocity of 0.006cm/s is shown above. The velocity was found to be more compared to 0.004cm/s due to the increased through flow velocity. As the through flow velocity is increased more mixing of denser and lighter fluids takes place and the velocity in the mixing area is more

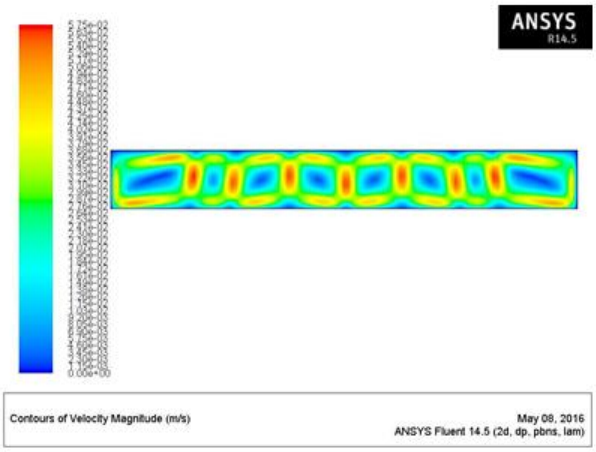

Fig.14 Velocity contour for through flow of 0.008cm/s (AR: 8, Ra=4.14$*10^5$)

It is clear from the figure that when through flow is increased from 0.006 to 0.008cm/s the mixing of denser and lighter fluids becomes faster and the velocity in the mixing area is more.

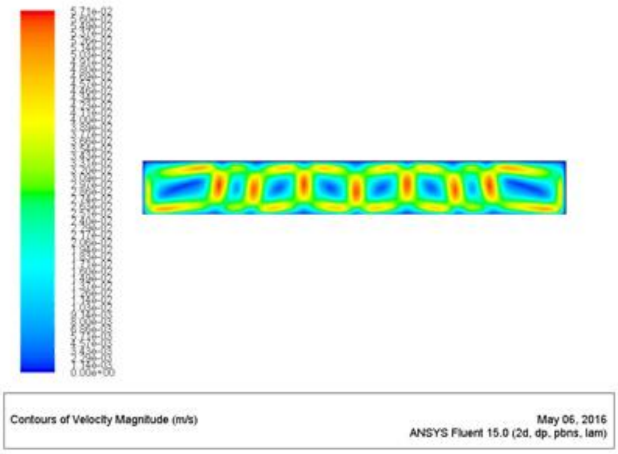

Fig.15 Velocity contour for through flow of 0.01cm/s (AR: 8, Ra=4.14* 10^5)

The velocity distribution in the fluid layer (Fig 5.7 to 5.12) shows that the velocity around the upper and lower convection cells increases when through flow is increased. It is clear from the figure that when through flow increases the mixing of denser and lighter fluids becomes faster and the velocity in the mixing area is more.

2. Rayleigh-Benard convection of Water

The changes of temperature distribution in the fluid layer in water as medium for an aspect ratio of 8 for through flow velocities from 0 to 0.01 cm/s is shown below.

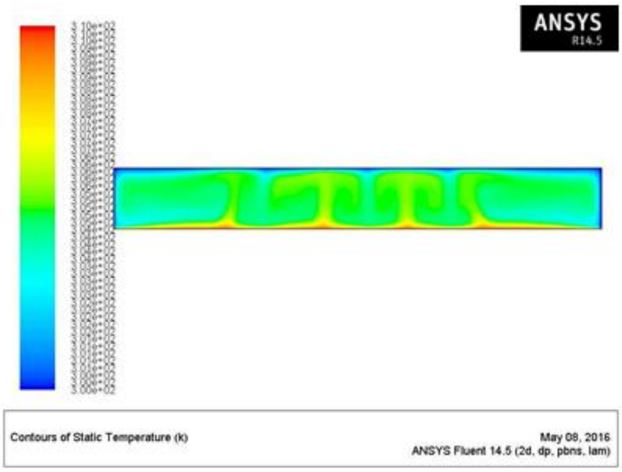

Fig.16 Temperature contour without through flow (AR: 8, Ra=13.18* 10^8)

The temperature contour of Rayleigh-Benard convection of water without through flow is shown above. Four plumes is generated when no through flow is given, when no through flow is given the lighter fluid moves up due to buoyancy force and denser fluid moves downward. By comparing with air the spacing was found decreased.

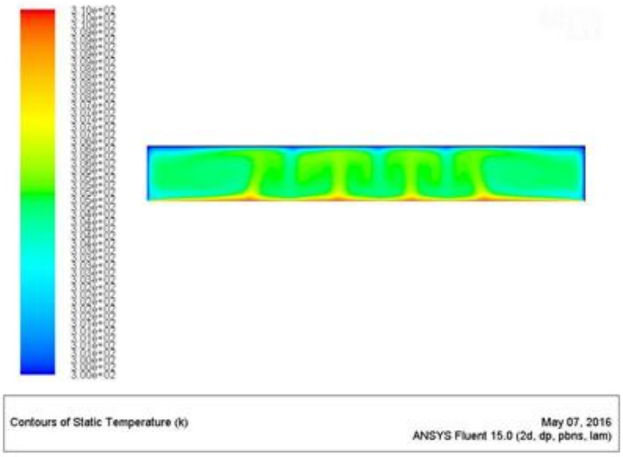

Fig.17 Temperature contour for through flow of 0.002cm/s (AR: 8, Ra=13.18 *10^8)

The temperature contour of Rayleigh-Benard convection of water with through flow of 0.002cm/s is shown above. Four plumes are also generated at 0.002cm/s through flow velocity, but the spacing between the plumes when measured was found to decrease with respect to the above.

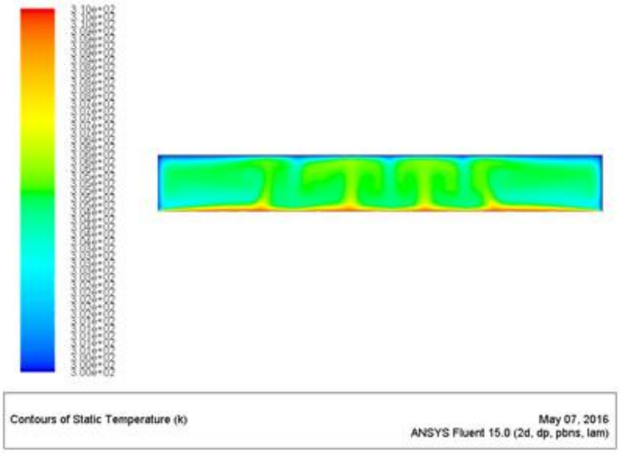

Fig.18 Temperature contour for through flow of 0.004cm/s (AR: 8, Ra=13.18* 10^8)

The temperature contour of Rayleigh-Benard convection of water with through flow of 0.004cm/s is shown above. Fluid near the hot plate is less dense compared to the fluid near cold plate. As the through flow velocity is increased from 0.002 to 0.004 cm/s the plume spacing was decreased.

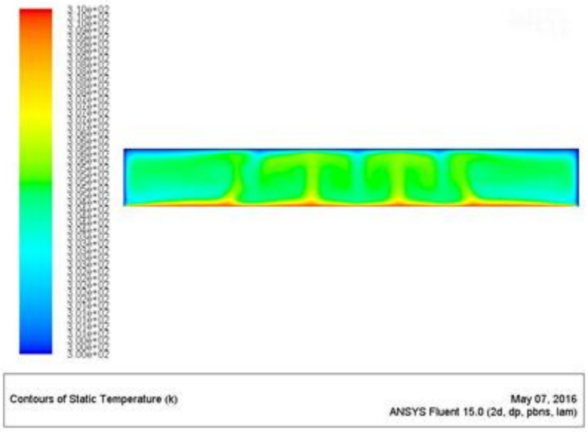

Fig.19 Temperature contour for through flow of 0.006cm/s (AR: 8, Ra=13.18* 10^8)

The temperature contour of Rayleigh-Benard convection of water with through flow of 0.006cm/s is shown above. Due to buoyancy force lighter fluid moves upward denser fluid moves downward. The mixing of denser and lighter fluid increases as the through flow is increased. As the through flow velocity is increased from 0.004 to 0.006 cm/s the plume spacing was decreased

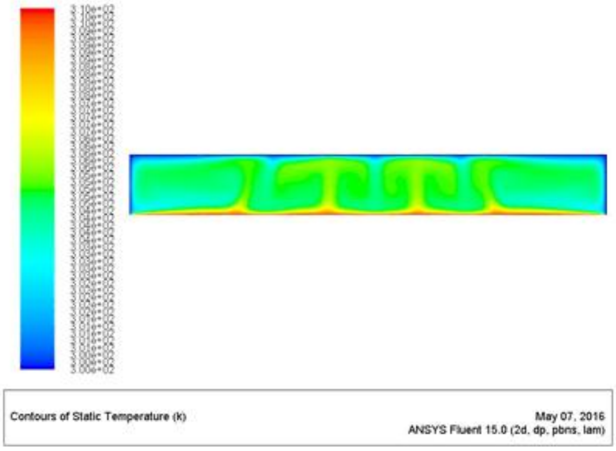

Fig.20 Temperature contour for through flow of 0.008cm/s (AR: 8, Ra=13.18* 10^8)

The temperature contour of Rayleigh-Benard convection of water with through flow of 0.008cm/s is shown above. The mixing of denser and lighter fluid further is increased as the through flow is increased. In this four plumes were also visible but plume spacing decreased with increase in the through flow.

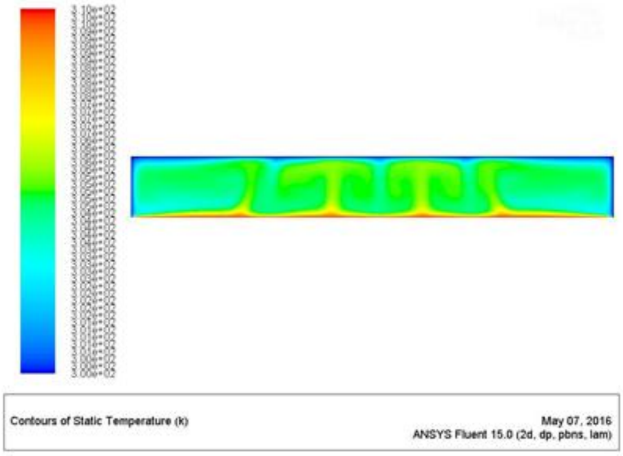

Fig.21 Temperature contour for through flow 0.01cm/s (AR: 8, Ra=13.18* 10^8)

The figures show the temperature contour of Rayleigh Benard convection by CFD simulation with through flow velocities varying from 0 to 0.01cm/s. Fluid near the hot plate is less dense compared to the fluid near cold plate. Due to buoyancy force lighter fluid moves upward denser fluid moves downward. Plume is generated at the edges of the walls and it coagulates and flows upward with the velocity from the through flow. These two flows merge together and form a large scale flow. The mixing of denser and lighter fluid increases as the through flow is increased.

In this four plumes are clearly visible and plume spacing decreases with increase in the through flow. As the medium is water, due to the closely packed arrangement of molecules less the spacing of plumes where found.

The changes of velocity distribution in the fluid layer in water as medium for an aspect ratio of 8 for through flow velocities from 0 to 0.01 cm/s is shown below

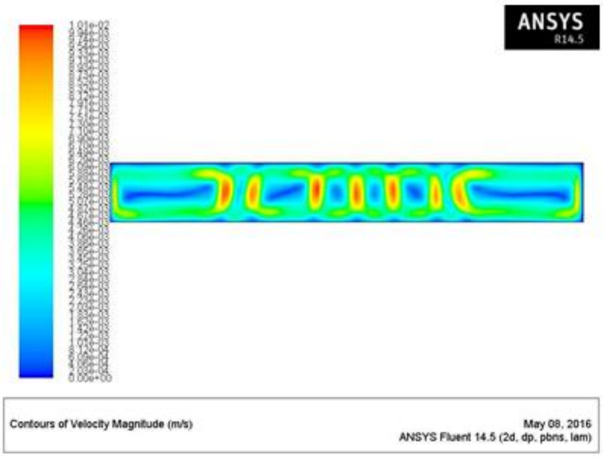

Fig.22 Velocity contour without through flow (AR: 8, Ra=13.18 *10^8)

The velocity contour of Rayleigh-Benard convection of water without through flow is shown above. The velocity distribution in the fluid layer (aspect ratio 8) shows that the velocity increases, while moving to the centre, compared to the walls.

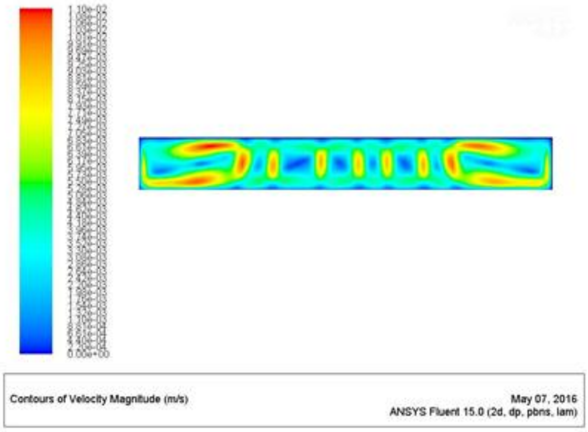

Fig.23 Velocity contour for through flow of 0.002cm/s (AR: 8, Ra=13.18* 10^8)

The velocity contour of Rayleigh-Benard convection of water with through flow velocity of 0.002cm/s is shown above. It is clear from the figure that when through flow is given the mixing of denser and lighter fluids becomes faster and the velocity in the mixing area is more.

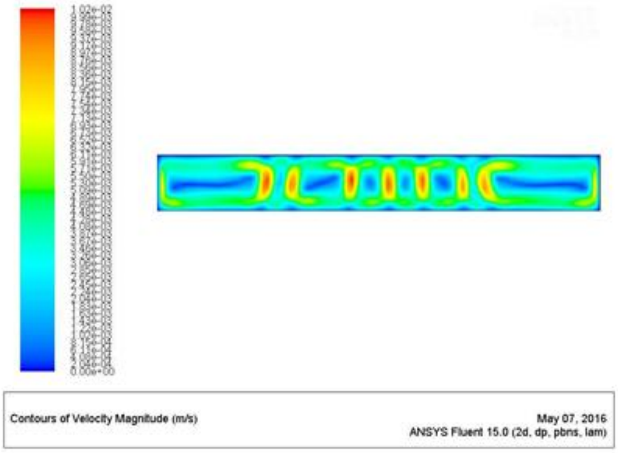

Fig.24 Velocity contour for through flow of 0.004cm/s (AR: 8, Ra=13.18* 10^8)

The velocity contour of Rayleigh-Benard convection of water with through flow velocity of 0.004cm/s is shown above. The velocity was found to be in the mixing region compared to the walls. The peek velocity was found to be less as that of the above case.

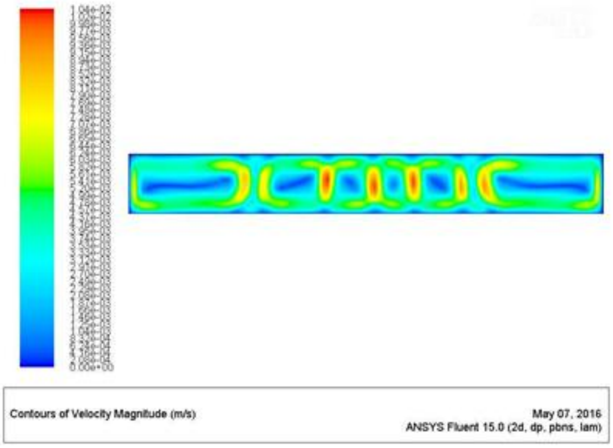

Fig.25 Velocity contour for through flow of 0.006cm/s (AR: 8, Ra=13.18* 10^8)

The velocity contour of Rayleigh-Benard convection of water with through flow velocity of 0.006cm/s is shown above. The velocity was found to be more compared to 0.004cm/s due to the increased through flow velocity. As the through flow velocity is increased more mixing of denser and lighter fluids takes place and the velocity in the mixing area is more.

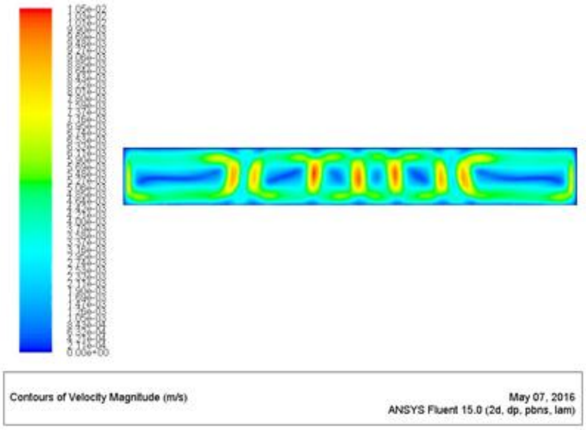

Fig.26 Velocity contour for through flow of 0.008cm/s (AR: 8, Ra=13.18* 10^8)

The velocity contour of Rayleigh-Benard convection of water with through flow velocity of 0.008cm/s is shown above. It is clear from the figure that when through flow is increased from 0.006 to 0.008cm/s the mixing of denser and lighter fluids becomes faster and the velocity in the mixing area is more.

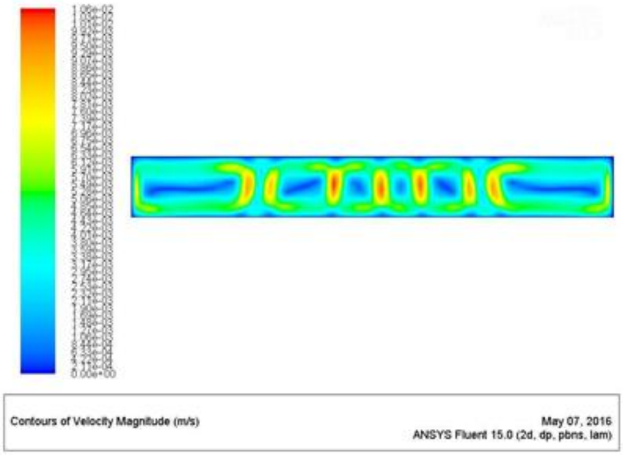

Fig.27 Velocity contour for through flow of 0.01cm/s (AR: 8, Ra=13.18* 10^8)

The velocity distribution in the fluid layer (Fig 5.19 to 5.24) shows that the velocity around the upper and lower convection cells increases when through flow is increased. It is clear from the figure that when through flow increases the mixing of denser and lighter fluids becomes faster and the velocity in the mixing area is more.

3. Comparison of Air and Water

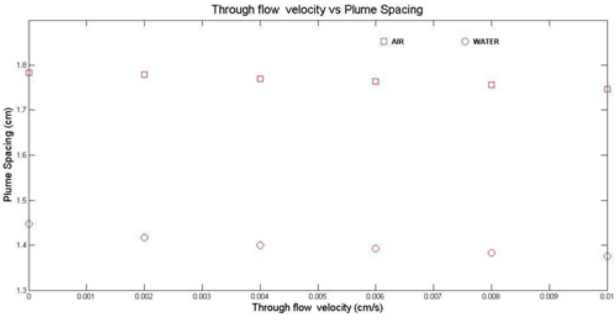

Fig.28 through Flow Velocity vs. Plume Spacing

With water as the working fluid at Ra in the range of 10^8 to $3\ 10^{10}$, in cells of aspect ratio above one, reveal different types of structures. Erupting warm plumes break of from the lower boundary layer and traverse the cell to the upper boundary layer where they excite waves. The waves can lead to the formation of cold plumes which then descend to the lower boundary layer, where they again excite waves. At high Ra, the fluid layer divides into bulk and near-wall regions which respectively have predominantly turbulent and diffusive transport. The near-wall region, being diffusive, offers the predominant resistance to the transport of heat. In these near wall regions, line plumes are initiated. The line plumes have a definite thickness and a definite height, beyond which they breakup into mushroom type plumes. The line plumes elongate and merge with the adjacent sheets, a new sheet being initiated in the space vacated by the merger. The external shear due to the large scale circulation modifies these dynamics by aligning the sheets in the direction of the shear. However the plumes merged faster with increase in Ra and the mean of the spacing between the lines plumes (h) at any instant decreased with increase in Ra. The lengths of the line plumes are easily and accurately measured compared to the normal spacing between them even if the plumes are oriented randomly. Since the line plumes carry the major part of the heat flux, characterizing the plume structure in terms of plume length could lead to unique relations between the lengths of the near-wall coherent structures. At large enough Ra the flow near the surface consists of (i) a laminar boundary layer near the edge followed by (ii) longitudinal rolls perpendicular to the edges and (iii) in the central region a flow structure very similar to what is observed in turbulent Rayleigh Benard convection randomly moving line plumes. In region (iii) the local heat flux is observed to be independent of distance from the leading edge.

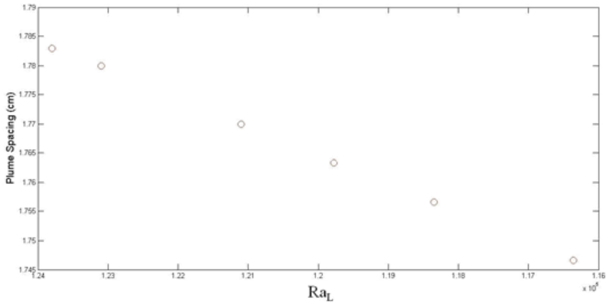

Fig.29 Rayleigh Number Vs. Plume Spacing for Air

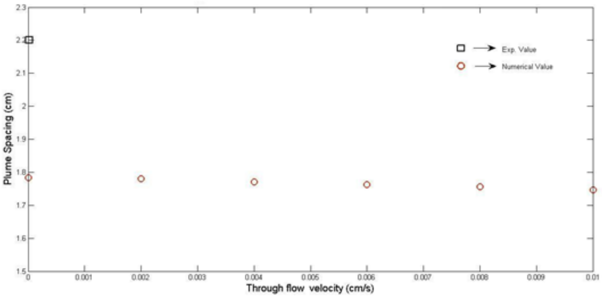

Fig.30 Comparison with the experimental value

Rayleigh number (Ra) is calculated as the product of Grashof number (Gr) and Prandtl number (Pr). In the Grashof number the domain length L is taken as the distance between the first and last plume as there are multiple plumes formed. From this analysis for different through flow velocities, different domain lengths were identified and Rayleigh number also changed correspondingly.

The calculated Rayleigh number and corresponding plume spacing were graphed. From Baburaj et.al [4] journal the plume spacing without through flow was calculated and com-pared with the result from simulation (Figure 5.27). The results shows slight change in the values got from experiment and simulation due to the change in the Rayleigh number.

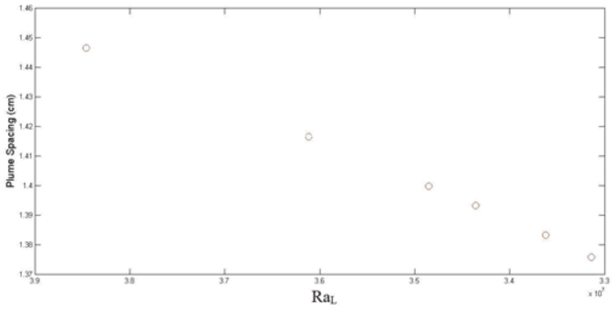

Fig.31 Rayleigh Number Vs Plume Spacing for Water

For water as medium the Prandtl number (Pr) is higher and correspondingly more Rayleigh number is obtained. From this analysis for different through flow velocities, different domain lengths were identified and Rayleigh number also changed correspondingly. The calculated Rayleigh number and corresponding plume spacing were graphed. At high Ra, the fluid layer divides into bulk and near-wall regions which respectively have predominantly turbulent and diffusive transport. The near-wall region, being diffusive, offers the predominant resistance to the transport of heat. In these near wall regions, line plumes are initiated. The line plumes have a finite thickness and a finite height, beyond which they breakup into mushroom type plumes. The line plumes elongate and merge with the adjacent sheets, a new sheet being initiated in the space vacated by the merger. The external shear due to the large scale circulation modifies these dynamics by aligning the sheets in the direction of the shear. However the plumes merged faster with increase in Ra and the mean of the spacing between the lines plumes (h) at any instant decreased with increase in Ra.

IV.CONCLUSIONS

In present work, by varying the through flow velocity from 0 to 0.01cm/s in the Y-direction at an aspect ratio of 8, the problem is carried out to study Rayleigh-Benard convection. The effect of through flow on natural convection was studied. The numerical study involved two-dimensional geometry.

Simulation of Rayleigh Benard convection where done with different through flow velocities. A large scale flow is clearly visible in the figure. Fluid near the hot plate is less dense compared to the fluid near cold plate. Plume is generated and it coagulate and flow upward, and same process occur in the top and another flow is generated towards down .This two flows merge together and form a large scale flow. Multiple flow cells are created. As the through flow increases plume spacing was found to decrease. At higher velocities the vertical pressure gradient drives the less dense fluid up and creating plumes. The plume spacing was more in Air compared to water due to the density differences.

REFERENCES

[1] Baburaj A. Puthenveettil and Jaywant H. Arakeri. Plume structure in high Rayleigh number convection. J. fluid Mech., 542:217{249, 2005.
[2] S. Ananda Theerthan and Jaywant H. Arakeri. A model for near-wall dynamics in turbulent Rayleigh Benard convection. J. fluid Mech., 373:221{254, 1998.
[3] G. Vijaya Rama Reddy and B. A. Puthenveettil. High Schmidt number natural convection boundary layers with blowing. International Conference on Advances in Mechanical Engineering. 2008.
[4] Baburaj A. Puthenveettil, G. S. Gunasegarane, Yogesh K.Agrawal, Daniel Schmeling, Johannes Bosbach and Jaywant H Arakeri. Length of near wall plumes in turbulent convection. J. fluid Mech., 2011.
[5] D.A Neild. Throughflow effects in the Rayleigh benard convective instability problem. J. fluid Mech., 185:353{360, 1987.

ARJUN G NAIR: received B.Tech degree in Mechanical Engineering from Jyothi Engineering college, in 2013 and is currently pursuing M.Tech in Thermal Engineering from Malabar College of Engineering and Technology.

SATHIL P T: Assistant Professor, Malabar College of Engineering and Technology, Desamangalam, Trissur.

PRASIDH E PRAKASH: Assistant Professor, Malabar College of Engineering and Technology, Desamangalam, Trissur.