OPEN ACCESS

International Journal Of Modern Engineering Research (IJMER)

Effect of Inlet Port Geometry on Uniformity of Flow Distribution in the Channels of Heat Sink Using CFD

Abhishek Basappa[1], Vijaya Raghu B[2]
*(Scholar, Mechanical Engineering, MIT Mysore, India)
** (Professor, Mechanical Engineering, MIT Mysore, India)

ABSTRACT: *Heat sinks with rectangular channels are commonly used to dissipate heat from electronic components. To ensure uniform heat dissipation of heat, coolant fluid has to be uniformly distributed among the channels of heat sink. Space constraint forces the designers to keep divergent inlet port shorter with larger diverging angles. Due to flow separation in the divergent portion, flow rates will be higher in the channels nearer to the center line and lower in the channels placed at the extremes. In this work, fluid flow is simulated numerically using commercial CFD software through the heat sink with diverters placed in the inlet port. Main objective of this work is to study the effect of geometry of the diverter on the uniformity of flow distribution among the channels. Length and inclination of the diverter pivoted at the center of the divergent portion are varied and flow rates in each of the channels are observed. It is observed that longer diverters with smaller angles and shorter diverters with larger angles result in better distribution of flow. It is also observed that inlet velocity has very little effect on the flow distribution.*

Keywords: *CFD, Diverter, Inlet Port, Heat Sink, Uniformity*

I. INTRODUCTION

A heat sink employed in any industry, usually, a metallic device, is used with electrical and electronic components that attain higher temperatures during their working. Main objective of using heat sinks is to keep temperature of components under control by uniform removal or dissipation of heat. Usually a blower is used to blow cold air through the channels of the heat sink to remove the heat. The air flowing through the heat sink is distributed through the channels in the sink.

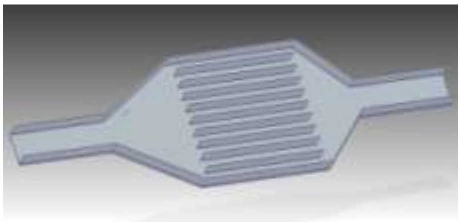


Fig 1.1: Heat sink with rectangular channels

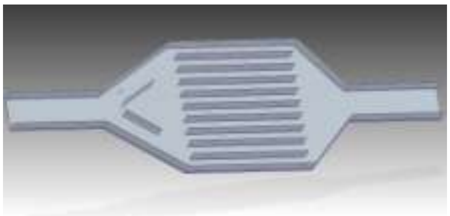
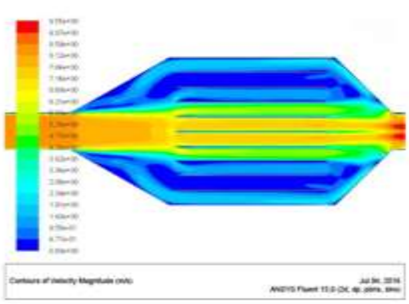


Fig 1.2: Heat sink with rectangular channels and diverter

As shown in the velocity contour fig 1.3, the tendency of the fluid flow is to attain high velocities in the channels at the center and flow rate will be lesser in the channels at the extremities. Heat dissipation and heat carrying capacity will be naturally more in the channels where flow rate is higher, but the heat sink has to be designed for uniform removal of heat. As related to heat sinks, a diverter is a plate introduced in the 2D flow region to disturb and divert the flow with the intension of achieving uniform distribution through all the channels. In the 3D flows diverters in the form hallow frustum of cone and square pyramid are used. Diverters are so placed in the flow field that they divert the flow from the region where flow rates are high towards the region where flow rates are low. In case of heat sinks with square/rectangular channels, as shown in the figure 1.2, these plates divert flow away from channels at the centre towards the channels placed at the ends. Fig 1.4 shows the velocity contour from post processing of commercial software Ansys. It is clear from the contour that introduction of the diverter in the inlet port of the heat sink has improved the flow distribution in the channels.



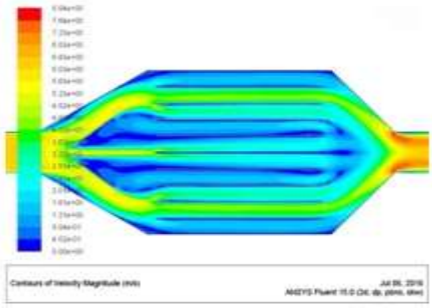


Fig 1.3: Velocity contour without diverter **Fig 1.4:** Velocity contour with diverter

1.1 Objectives of the present study

The major objectives of the present study is to design inlet port of heat sinks to obtain nearest to uniform fluid flow velocity through all of the channels, at the same time keeping the pressure drop within the limits. With this objective, it is proposed to use CFD to analyse the flow in the heat sinks with various inlet port geometries. In this work, heat sink with square section channels is considered for the analysis and, deviators and flaps are introduced in the inlet port to improve the performance. Extensive parametric study has been carried out to understand the effect of the various parameters, such as, length and inclination of the deviators and flaps, arrangements and inlet flow velocities. Majorly, it is to quantify the uniformity of flow rate through the channels with various inlet port geometrical configurations. For the analysis, parameters like Average velocity in the channels, Pressure drop, Standard Deviation and Coefficient of Variance are to be extracted from the post processing.

1.2 Validation of CFD Methodology

The process of validation involves the determination of various factors like extent of spatial discretization, the choice of turbulence model, criteria of convergence etc. for the cases taken up in the present study. In the present study the methodology has been validated by analyzing the pressure drop of the fluid flowing through a pipe both in laminar and turbulent modes. Analytical solutions are available for the problem of flow through pipes and the numerically computed results are compared with them. Comparison so done leads in the selection of most suitable turbulent model, grid size, growth rate etc. On validating numerical results for flow through pipe with analytical solution, k-ω turbulence model proved to be better.

II. SOLUTION DOMAIN AND BOUNDARY CONDITIONS

Study involves two dimensional numerical analysis of forced flow of cooling air through channels of heat sink with diverters. Flow of air through the channels is simulated with 4 inlet velocities (2m/s, 4m/s, 6m/s and 8m/s) and initial computations are made for the channels without any diverters. Subsequently, different configurations of diverters are incorporated into the flow domain. The flow is assumed to be two dimensional, turbulent, incompressible and steady so that heat transfer through buoyancy and radiation effects is not considered.

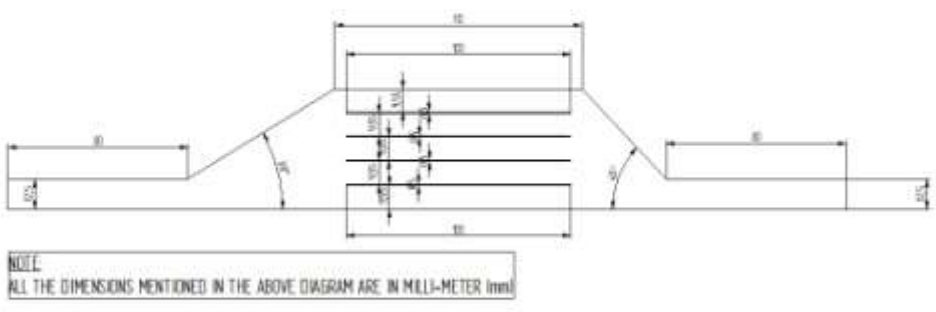


Fig 2.1: 2D Geometry of the Heat Sink Domain without Diverter.

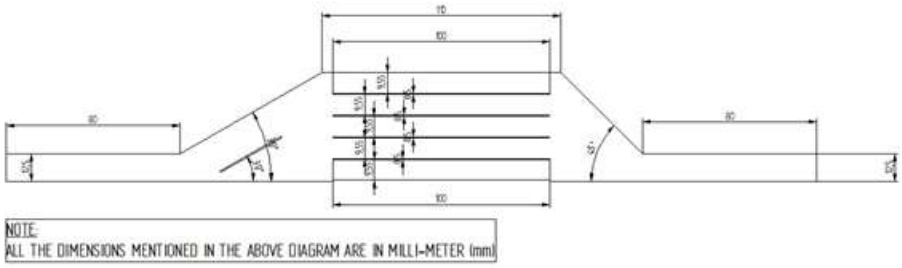


Fig 2.2: Heat sink with diverter

Channels are separated with walls of 0.5mm thickness. The semi-converging angle of outlet port has been kept constant at 45^0 as it has least effect on flow distribution through the channels. Heat sink geometry modeled for 2D analysis without diverter is shown in Fig 2.1 whereas with diverter is shown in Fig 2.2. As the geometry is symmetrical about the centre line only the portion above the center line is considered for solution to reduce the computations.

Parameters	Linear Measurements in mm Angular measurements in deg
Width of inlet duct	12.5
Length of inlet duct	80
Heat sink width	110
Heat sink height	10
Length of the channels	100
Width of the channel	9.55
Wall Thickness of channels	0.5
Inlet port angle (semi-converging)	30^0
Outlet port angle (semi-diverging)	45^0
Length of outlet duct	80
Width of outlet duct	12.5

Table 2.1 - Geometrical parameters of the considered heat sink

It is to be observed that diverters are provided only in the inlet port of the heat sink not at the exit of the channels. This has been followed because it is observed that geometry at the exit port does not affect much of the flow distribution in the channels and it does not have significant effect on the flow around the diverters. Various cases have been generated with different inlet port geometries. Flow through each of the geometrical configuration is simulated with four different inlet velocities of air. Each of the geometrical configurations of inlet port is framed by varying length and inclination of diverter.

Mesh convergence tests are to be conducted to know least dense mesh leading to consistent and accurate results. Parameters of meshing such as refinement, sizing and inflations are maintained the same for further simulations with various other parameters. Refinement of mesh has been done at the inlet and at edges of walls, walls of channels, diverters. Mainly inflations are given to the edges of the walls of channels and diverters, where steep velocity gradients are expected, so that relatively accurate results can be expected.

Fig 2.3 shows the discretized flow domain for the flow through the heat sink channels without diverters, whereas Fig 2.4 shows the mesh for the same with diverter. The 2D quadrilateral elements are used for meshing. The number of elements used depends on each case analyzed and is in the range 2.5×10^4 to 3.2×10^4, whereas the inflation given is 12. Maximum layers with a growth rate of 1.2 at the walls, plates and diverter.

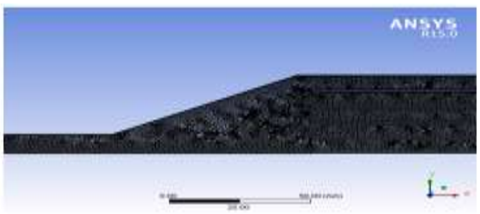


Fig 2.3: Unstructured mesh for 2D analysis of fluid flow without diverters

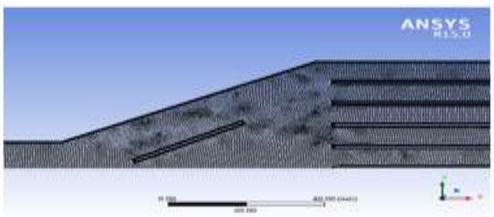


Fig 2.4: Unstructured mesh for 2D analysis of fluid flow with diverter

2.1 Boundary conditions and models

Modelled flow domain is meshed and imported into Fluent as per the validated refinement. Simulations are made with following boundary conditions with fluid as air.

Boundary	Condition
Inlet	Inlet Velocity m/s
Outlet	Pressure at outlet = 0Pa (Gauge pressure)
Wall	No slip wall condition
Separating Plates	No slip wall condition
Diverters	No slip wall condition
Flaps	No slip wall condition
Axis / Centre line	Planar – Symmetry

Table 2.2: Boundary conditions used for 2D fluid flow analysis of heat sink with diverters and flaps

The software package in ANSYS FLUENT 15 has several turbulence models that could be used for the analysis. Majority of them are two equation models involving turbulent kinetic energy, dissipation, Reynolds stresses etc. The most widely used model is k-ω model when the flow is fairly well defined. During the present analysis, two models namely k-ε and k-ω model were used. It was observed that k-ω model resulted with better agreement with exact solutions for flow through pipe and hence this model has been chosen for further analysis. In the present analysis, convergence criteria for the residuals are chosen as 10E-6 for all the equations. The computed velocity and pressure fields are used to calculate different parameters like Average velocity, Pressure drop, Standard Deviation, Coefficient of Variance.

2.2 Measure of uniformity of flow rate

Coefficient of variance considered as the measure of uniformity of flow in this work. Average velocities of flow in each of the channels are base parameter in computing the coefficient of variance. It indicates how velocities in the channels are differing from average velocity. Higher values of coefficient of variance indicate non-uniform flow and the value of coefficient of variance tending towards 0 (zero) indicates uniform distribution of flow among the channels. As lower values of coefficient of variance are desired, inlet port design resulting in the same is considered to better, even, pressure drop across the heat sink has to be taken into account.

Standard Deviation
$$\sigma^2 = \frac{\sum_{i=1}^{n}\left(V_i - V_{avg}\right)^2}{n}$$

Coefficient of variance
$$COV = \frac{\sigma}{V_{avg}}$$

Where,
n = number of channels in the heat sink
V_i = Velocity at the respective channels

III. RESULTS AND DISCUSSION

Study involves the numerical analysis of forced flow of cooling air through channels with and without diverters and flaps. Flow of air through the channel without diverters and flaps is simulated initially and subsequently further studies are carried out with insertion of diverters and flaps in order to study the effect of the following parameters on flow velocity and pressure drop across the channels. As the main objective of this work is to attain near uniform flow in all the channels of the heat sink, various geometrical configurations are framed for inlet port. Length and inclination are the geometrical parameters varied in the analysis in both of the cases involving diverter and flaps. Efforts are made to keep the analysis dimensionless to make the results applicable for all dimensions and semi-divergent angle of the inlet port.

3.1 Geometrical parameters of the considered heat sink with diverter

The length L is base linear dimension in terms of which length of the diverter is varied. It is the length of the inlet port along the inlet axis or the centreline, which holds a linear dimension of 64.95mm in the present geometry. In all of the cases involving diverter, the diverter is placed as such that it is pivoted through its centre and at the centre of length L. In this work, diverter length as a parameter is varied with the values of 0.3, 0.4 and 0.5 times of the length of divergent inlet port.

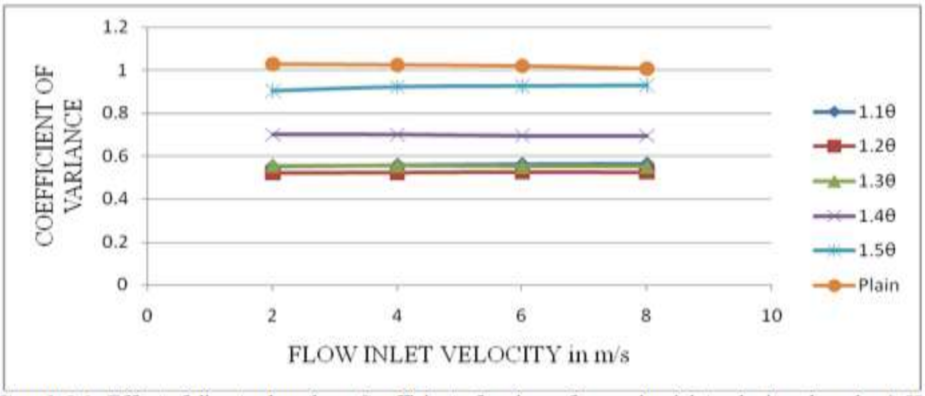
The angle θ is the base angular dimension in terms of which inclination of the diverter is varied. It is the semi-divergent angle of the inlet port, which holds an angular dimension of 30^0 in the present geometry considered in the work. The semi-divergent angle is considered to be 30^0 as it is shown by many researchers that flow tends to separate from diverging surface above an angle of 10^0 and lesser diverging angle leads to lengthier inlet port. As length of the inlet port is a practical constraint, efforts have been made in this work to attain uniform flow distribution with a semi-divergent angle of 30^0. Flow through the heat sink of all the geometrical configurations has been numerically simulated for four inlet velocities. In this work, diverter angle as a parameter is varied with the values of 1.1, 1.2, 1.3, 1.4 and 1.5 times of the semi-divergent angle of the inlet port.

3.2 Results and Discussion

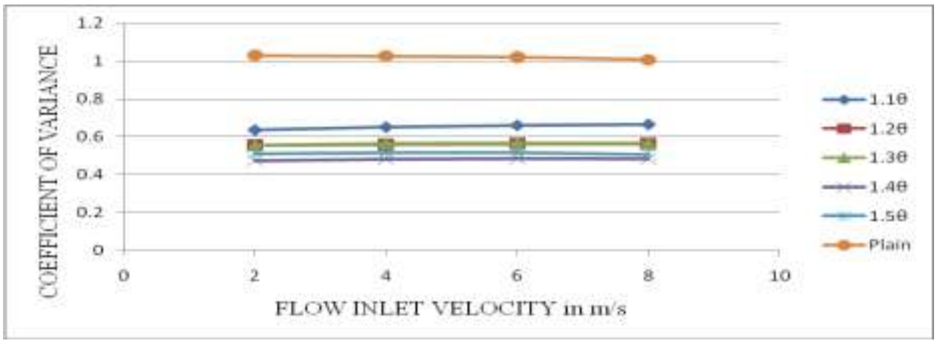
As it is discussed, Coefficient of variance is the measure uniformity in the flow distribution among the channels of the heat sink. Therefore, in the discussions, Coefficient of variance is considered as the dependent variable which depends on flow parameter velocity and geometrical parameters length and angle of diverter and flaps.

3.2.1 Effect of inlet velocity and diverter angle on coefficient of variance for a given diverter length

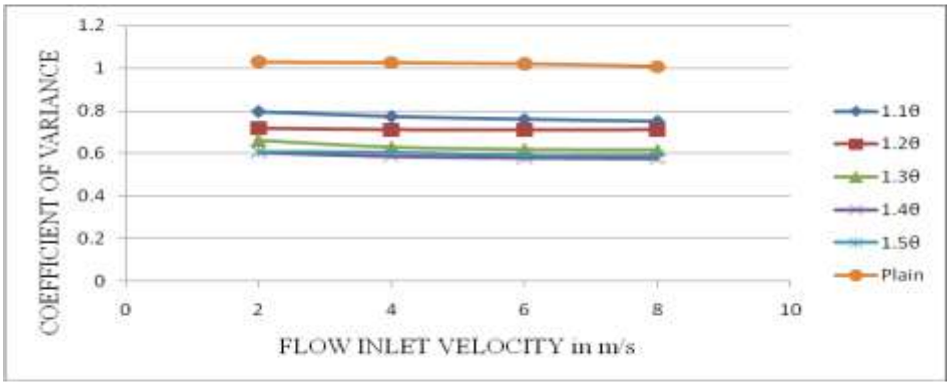
Graphs 3.1, 3.2 and 3.3 depict the effect of inlet velocity and diverter angle and on coefficient of variance for a given diverter length in heat sink along with a heat sink without diverter on coefficient of variance. It is clear from the graphs that uniformity of distribution in the channels has very little dependency on the inlet velocity.



Graph 3.1: Effect of diverter length on Coefficient of variance for varying inlet velocity where l = 0.5L



Graph 3.2: Effect of diverter length on Coefficient of variance for varying inlet velocity where l = 0.4L



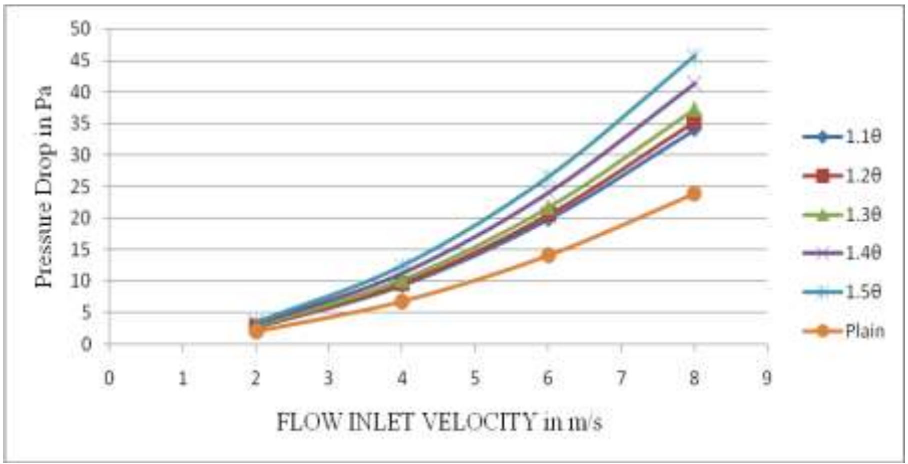
Graph 3.3: Effect of diverter length on Coefficient of variance for varying inlet velocity where l = 0.3L

In graph 3.1, for the diverter length of 0.5L, longest of the diverters considered in this work, it is seen that with the increase in the inclination of diverter COV increases for all the values of inlet velocities considered. The least value of COV 0.52 is obtained for a diverter inclined at an angle of 1.2θ and further decrease in the inclination also leads increase in COV.

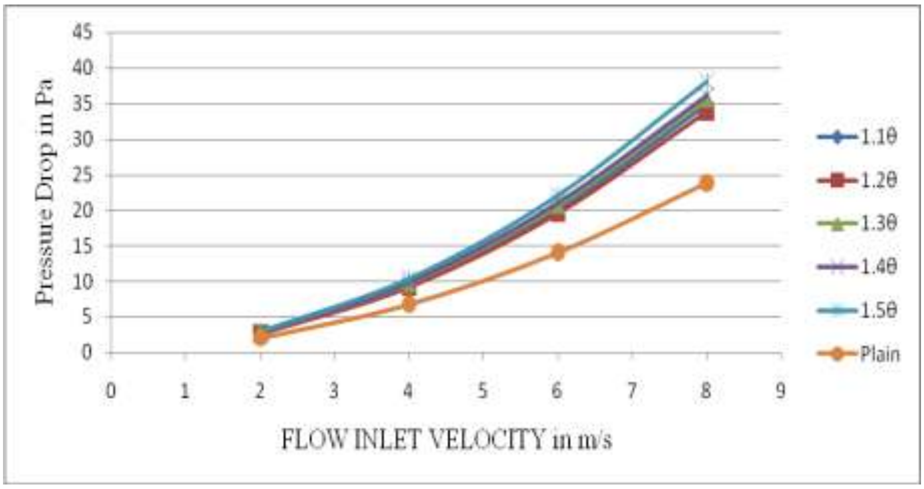
Studying comprehensively, from the graphs 3.2 and 3.3, for medium and shortest of the diverters, it is seen that with the increase in the diverter inclination the COV decreases up to a diverter angle of 1.4θ and increases on further increase in the angle. From all the above three graphs it can be concluded that the lowest value of COV 0.47 is obtained for a heat sink with a diverter of length 0.4L and inclination 1.4θ for a inlet flow velocity of 2m/s. Also, diverter of length 0.4L resulted better in terms of uniformity in flow distribution for most of the inclinations and inlet velocities considered. In general, it is clear that insertion of diverter in the inlet port has improved the flow distribution by huge margin.

5.2.2 Effect of inlet velocity and diverter angle on pressure drop for a given diverter length.

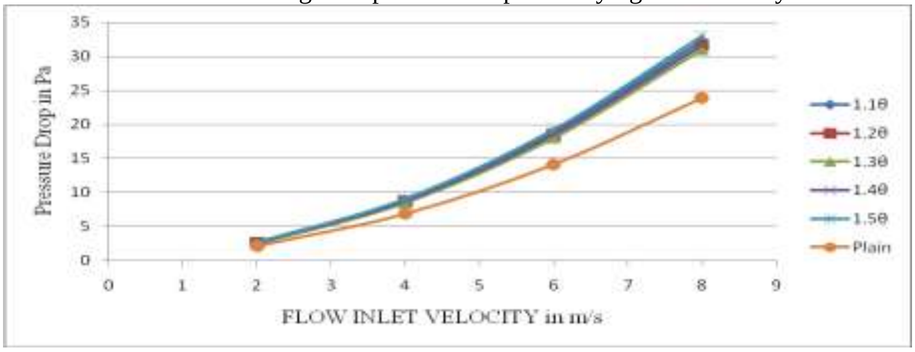
Graphs 3.4, 3.5 and 3.6 depict the effect of inlet velocity and diverter angle and on pressure drop for a given diverter length. It is obvious from the all the plots that irrespective of inlet port geometry the pressure drop increases considerably with inlet velocity. Insertion of diverter has little effect on pressure drop at low inlet velocities but aggravates as the inlet velocity increases. In graph 3.4, for diverter length 0.5L, longest of the diverters, it is seen that the pressure drop increases with the increase in the inclination of the diverter for all the inlet velocities considered. In graph 3.5, for diverter length 0.4L, diverter of medium length, it is studied that the pressure drop decreases with increase in the inclination of the diverter up to an angle of 1.2θ (36^0) and further increases with increase in the inclination. In graph 3.6, for diverter length 0.3L, shortest of the diverters, it is seen the pressure drop value decreases with increase in the inclination of the diverter up to an angle of 1.3θ (39^0) and further increases with increase in the inclination.



Graph 3.4: Effect of diverter length on pressure drop for varying inlet velocity where l = 0.5L



Graph 3.5: Effect of diverter length on pressure drop for varying inlet velocity where l = 0.4L



Graph 3.6: Effect of diverter length on pressure drop for varying inlet velocity where l = 0.3L

Comprehensively, it can be stated that diverter inclination has its considerable effect on pressure drop only with longer diverters, but has very little effect on pressure drop with shorter diverters.

On combining all the three graphs 3.4, 3.5 and 3.6 it can be concluded that the pressure drop value increases with increase in inlet flow velocity and the pressure drop values are the least for heat sink without diverters. As the length of the diverter decreases and with increase in the inclination angle of the diverter the pressure drop values decreases up to a certain angle and increases further on that is in graph 3.4 for diverter length 0.5L least pressure drop value is2.67N/m is obtained for an diverter angle of 1.1θ (33^0), in graph 3.5 for diverter length 0.4L least pressure drop value is2.62N/m is obtained for an diverter angle of 1.2θ (36^0) and in graph 3.6 for diverter length 0.3L least pressure drop value is 2.44N/m is obtained for an diverter angle of 1.3θ (39^0) and increases further on.

IV. CONCLUSIONS

Outcome of the numerical simulation of flow through heat sinks with various inlet port geometry and inlet velocities can be analyzed and summarized as follows;

- Insertion of diverter in the inlet port improves significantly the uniformity in flow distribution in the channels.
- Inlet velocity has very little effect on uniformity of flow distribution in all the cases of inlet port geometry.
- Longer diverters at lower inclinations and shorter at higher inclinations resulted in better distribution of flow.
- In general, it can be concluded as, diverter of medium length 0.4L, resulted with better distribution of flow at most of the diverter inclinations and inlet velocities with least COV of 0.47.
- Negligible increase in the pressure drop is observed with the introduction of diverter at lower inlet velocities which increased with increase in inlet velocity.
- Longer diverters resulted in higher pressure drops with higher inclinations and negligible change in pressure drop is observed with variation in inclinations of shorter diverters.
- Least pressure drop was observed with diverter inclination of 1.2θ with most of the cases considered in this work.

- Comprehensively, it is clear that diverters of medium length placed at moderate angles provide better uniformity in flow distribution with little compromise in pressure drop.

REFERENCES

Journal Papers:

[1]. M.M. Mohamed "Air Cooling Characteristic of a Uni-Form Square Modules Array for Electronic Device Heat Sink", Journal of Electronic Cooling and Thermal Control, pp 486-493,2006

[2]. Yue-Tzu Yang, Huan-SenPeng "Investigation of Planted Pin Fin for Heat Transfer Enhancement in Plate Fin Heat Sink" Microelectronic Reliability 49, pp163-169,2009

[3]. Manasa N , Vijaya Raghu B , Dr. V Seshadri , Yogesh Kumar K J , Puneeth Kumar S B, "Numerical Analysis of Heat Dissipation through Rectangular Channel with Elliptical Pin Fins"

Books:

[4]. Patankar S V. "Numerical Heat Transfer and Fluid Flow" .New York: McGraw-Hill; 1980.

[5]. Yunus A. Cengel, and John M. Cimbala,"Fluid Mechanics: Fundamentals and Applications", 2nd edition, McGraw-Hill, 2010.

[6]. Frank .M.White, "Fluid mechanics" seventh edition, published by Tata McGraw-hill Publication Company, Special Indian edition 2011,2008.

[7]. H K Versteeg and W Malalasekera, "An introduction to Computational Fluid Dynamics, Finite Volume Method", 1995.