CHAPTER - 1

INTRODUCTION AND LITERATURE REVIEW

This thesis intends to understand the phenomena of multi air-jet impingement and its effects on fluid flow and heat transfer characteristics on various configurations of Pin Fin Heat Sinks. Heat transfer from impinging jets is a relatively new development that has attracted the interest of engineers involved in the design of electronic systems. In this chapter, the role of heat sinks and jet impingement techniques to tackle today’s electronic cooling problem are described briefly.

1.1 ELECTRONIC COOLING

In the present scenario of many advanced technologies, use of electronic equipment has become almost inevitable. These electronic equipments play a vital role in many critical areas of technology and resulted in high density of components in small volume. Therefore, there has been a steady increase in heat dissipation rate from electronic components for the last few decades. As these devices consume electric power, this power needs to be dissipated or otherwise heat will be accumulated and temperature of device may exceed to dangerous levels. Figure 1.1 shows that major cause of electronic failures is temperature (Ndao et al., 2009).

The continuing increase of power densities in microelectronics and the simultaneous drive to reduce the size and weight of electronic products have led to the increased importance of thermal management issues in these applications. Over the time, the size and cost of typical electronic device has drastically decreased while the required functionality, reliability and operating temperatures have significantly increased. Also day by day the rate of automation is increasing significantly in all the fields, for example today’s average new automobile content is about 40% of electronics (Klier and Rubenstein, 2011). The temperature at the junction of an electronics package (chip temperature) has become the limiting factor determining the lifetime of the package. The most common method for cooling packages is the use of passive heat sinks (Fig.1.2a) or active heat sinks (Fig.1.2b). Passive heat sinks used in natural convection applications where normal heat dissipation load is about 5 – 30 W, which are relatively simple, and their usage does not require external power.
Fig. 1.1: Major Causes of Electronic Failures (Ndao et al., 2009).

Fig. 1.2: Types of Heat Sinks.

(a) Passive
http://www.dansdata.com/images/c3ezra/via
sink220.jpg

(b) Active
http://cache-www.intel.com/cd/00/00/14/97/149748_149748.jpg

Fig. 1.2: Types of Heat Sinks.
Active heat sinks are used in forced convection applications with normal load limit 20-150 W which require external power. These heat sinks provide a large surface area for the dissipation of heat and effectively reduce the thermal resistance of the package. They often take less space and contribute less to the weight and cost of the product. For these reasons, they are widely used in applications where heat loads are substantial and/or space is limited. They are also found to be useful in situations where the direction of the approaching flow is unknown or may change. They offer a low cost, convenient method for lowering the thermal resistance and in turn maintaining junction temperature at a safe level for long term reliable operation.

The overall performance of a heat sink depends on a number of parameters including the dimensions of the base plate and pin-fins, thermal joint resistance, location and concentration of heat sources. These parameters make the optimal design of a heat sink very difficult. Traditionally, the performance of heat sinks is measured experimentally or numerically and the results are made available in the form of design graphs in heat sink catalogues. Analytical and empirical models for the fluid friction and heat transfer coefficients are used to determine optimal heat sink design. Obviously the traditional natural/forced convection is insufficient to solve the overload conditions of today's system requirements and thus an efficient cooling of electronic system remains a challenge. In this situation the air jet impingement has turned out to be an attractive option which can be attained using heat sinks in conjunction with jet impingement cooling technique. The jet impingement heat transfer has become well established as a high performance technique for heating, cooling and drying a surface (Sarkar and Singh, 2003).

1.2 MOTIVATIONS

Compact heat sinks have been subject of extensive research, because of their importance in a wide variety of engineering and industrial applications. Fins are playing a vital role in such equipments to enhance their performance. Performance reliability and life expectancy is inversely proportional to the component temperature. Therefore, long life and reliable performance of a component is achieved by effectively controlling the device temperature. Earlier, electronic cooling was not addressed at the design stage. It was addressed only when there was a thermal failure of an electronic device. Later on, as the power densities increased and reliability of
electronic devices became an issue, it became imperative to address thermal management at the design stage.

Presently, the rate of heat fluxes in any typical electronic component is about $70 \text{ W/cm}^2$, which is a common high-end commercial application. In the coming years, this value is expected to be above $120 \text{ W/cm}^2$ (Kraus and Bar-Cohen, 1983). Developments and improvements of present cooling system for these future applications will be almost impossible as they require very intricate cooling circuits. The impinging jet technique is very simple, moderate and expected to provide a workable solution. Although several researchers have attempted to address this problem at the design stage, unfortunately the speed of invention of cooling mechanism has not kept pace with ever increasing requirement of heat removal from electronic system. As a result, analysis and design of heat sinks have been a major research topic for the thermal management community. The following studies could be carried out in order to improve heat transfer characteristic in electronic cooling using multi air jet impingement on pin fin array.

Over the last decade, electronics thermal design practices have evolved to a high reliance on virtual prototyping using numerical predictive techniques. Their applications, now widespread within the electronics industry, has been enabled by increase in computational power, and contributed significantly to reduce both prototyping costs and development of cycle times. The use of CFD-based methods is the most realistic approach for the prediction of conjugate heat transfer in electronic equipment. The onus is on the CFD user to employ both the correct modeling strategy and flow modeling approach for the application under analysis.

Our motivations are to find those variances that have not been investigated yet. Published studies, so far, include the regular jets arrays impinging on flat target surface, with variations on jet-to-jet spacing and air velocity. As far it is known, situations involving multi-jet array arrangement impinging on pin fin array heat sink, haven’t been investigated by both experimental and numerical approach.

### 1.3 HEAT TRANSFER ENHANCEMENT TECHNIQUES

The way to improve heat transfer performance is referred to as heat transfer enhancement (or augmentation or intensification). The research in this field was strongly stressed by the need of developing high performance thermal systems. The
development of heat transfer enhancement during the past 2-3 decades is such that enhanced surfaces are used routinely in refrigeration, automotive, electronic industries, food industries, textile industries and even more and more often in process industries. The improvement of the heat transfer coefficient requires quite different approach according to the phase of the fluid (gas or liquid) and to the process type: techniques are quite different for only sensible heat transfer or for phase change such as evaporation or condensation. This justifies a classification as represented in Fig. 1.3 (Webb, 1994).

Heat transfer enhancement techniques can be classified as active methods, which require external power, or passive methods, which require no direct application of external power. The major passive cooling solutions are obtained through conduction (heat spreaders, thermal interface materials), natural convection (heat sinks, liquid immersion), radiation (coating, surface treatments) or phase change (heat pipes, phase change materials). However, passive cooling techniques have low cooling performance requiring a large device size. Consequently, high-power systems require active techniques, which require input power but have larger heat removal capacity. The major active techniques are forced convection (fans, active heat sinks), pumped loops (heat exchanger, liquid cold plates, micro-channels, jet spray), thermoelectric cooling (TEC) and refrigeration. Fig.1.4 shows comparison between various electronics cooling mechanisms. From these jet impingement is an attractive and promising cooling mechanism due to the capability of achieving high heat transfer rate. The various passive and active techniques are described below:

1.3.1 Passive Techniques

Treated surfaces: This method involves the fine-scale alteration of the surface finish which affects single-phase heat transfer. This method is used for condensing and boiling.

Rough surfaces: This application is generally chosen to promote turbulence rather than heat transfer enhancement and its application is directed to single-phase flow. These surfaces are produced in many configurations ranging from random sand-grain-type roughness to discrete protuberances.

Extended surfaces: This technique is one that is currently the focus of many studies (including this study). The method involves the extension of the surface and examples of this method that are being used in practice are micro-fin tubes and most recently,
Heat Transfer Enhancement

- Single phase convection (gas/vapour or liquid)
- Boiling or evaporation
- Condensation

Fig.1.3: Heat Transfer Coefficient Enhancement Technique (Webb, 1994).

Fig.1.4: Comparison between Electronics Cooling Mechanisms. (Mahmoud, 2007)
herringbone type tubes.

**Displacement enhancement devices:** These devices are inserted into the flow channel so as to improve the energy transport indirectly at the heated surface and these devices are used with forced flow.

**Swirl-flow devices:** Examples of such devices are coiled tubes, inlet vortex generators, twisted tape inserts and axial-core inserts. These devices create a rotating flow and/or a secondary flow.

**Surface tension devices:** These devices consist of wicking or grooved surfaces to direct the flow of liquid during condensing or boiling.

**Additives for liquids and gasses:** Additives for liquids include solid particles and gas particles in single-phase flow, while for gas additives, liquid droplets or solid particles are used.

### 1.3.2 Active Techniques

**Mechanical aids:** These aids stir the fluid by mechanical means or by rotating the surface. Equipment with rotating heat exchanger ducts is found in commercial practice (Bergles and Chyu, 1988; Lasance, 1997).

**Surface vibration:** To improve single-phase heat exchange, the surface is vibrated at either low or high frequencies.

**Fluid vibration:** The fluid is vibrated at pulsations of 1 Hz with ultrasound. This is the most practical type of vibration enhancement and is used in single-phase flow.

**Electrostatic fields:** These fields are applied in many different ways to dielectric fluids. The electrostatic fields can be directed to cause greater bulk fluid mixing in the vicinity of heat transfer surfaces.

**Injection:** This method involves the supplementation of gases to a flowing liquid through a porous heat transfer surface, or by injecting another liquid into the liquid upstream.

**Suction:** Here liquids or gases are removed by suction through a porous heat transfer surface. Table 1.1 shows the range of $h$ for different modes of cooling.

### 1.4 JET IMPINGEMENT TECHNIQUE

Jet impingement cooling is a mechanism of heat transfer by means of collision of fluid molecules on to a surface. The impinging jet is defined as a high-velocity jet of cooling fluid forced through a hole or slot which impinges on the surface to be
Table 1.1: Convection Heat Transfer Regimes. (Incropera and Dewitt, 2006)

<table>
<thead>
<tr>
<th>Sr. No.</th>
<th>Modes of Cooling</th>
<th>Heat Transfer Coefficient (W/m²-K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>01.</td>
<td>Boiling, Water</td>
<td>$10^4$ to $3 \times 10^5$</td>
</tr>
<tr>
<td>02.</td>
<td>Condensation, Water</td>
<td>$8 \times 10^3$ to $10^5$</td>
</tr>
<tr>
<td>03.</td>
<td>Condensation, Organic vapor</td>
<td>$9 \times 10^2$ to $4 \times 10^4$</td>
</tr>
<tr>
<td>04.</td>
<td>Liquid metals, Forced convection</td>
<td>$9 \times 10^3$ to $8 \times 10^5$</td>
</tr>
<tr>
<td>05.</td>
<td>Water, Forced convection</td>
<td>$9 \times 10^2$ to $10^4$</td>
</tr>
<tr>
<td>06.</td>
<td>Organic liquids, forced convection</td>
<td>$2 \times 10^2$ to $2 \times 10^3$</td>
</tr>
<tr>
<td>07.</td>
<td>Gases 200 atm, Forced convection</td>
<td>$4 \times 10^2$ to $2 \times 10^3$</td>
</tr>
<tr>
<td>08.</td>
<td>Gases 1 atm, Forced convection</td>
<td>10 to $4 \times 10^2$</td>
</tr>
<tr>
<td>09.</td>
<td>Gases, natural convection</td>
<td>5 to 50</td>
</tr>
</tbody>
</table>
cooled, which results in high heat transfer rate between the wall and the fluid. ‘Impingement’ means ‘collision’ that the coolant flow collides into the target surface and guarantees a thin stagnant boundary layer at the stagnant core for cold coolant contacting the hot surface without damping. Impinging jets are used in many engineering applications to enhance heat transfer for cooling or heating purposes or mass transfer for vapor deposition. Typical heat transfer applications include food drying, paper drying, metal annealing, sheet metal treatment, cooling of gas turbine blades, cooling in grinding processes and cooling of photovoltaic cells. Jet impingement has also become a viable candidate for high-powered electronic and photonic thermal management solutions and numerous jet impingement studies have been aimed directly at electronics cooling.

In most of these applications, arrays of jets are used in a range of configurations and shapes with the objective on optimizing heat transfer rates. Convective heat transfer to impinging jets is known to yield high local and area averaged heat transfer coefficients. Impingement jets are of particular interest in the cooling of electronic components where advancement relies on the ability to dissipate extremely large heat fluxes. A jet impingement device can produce a flow field that can achieve relatively high local heat transfer rates over a surface to be cooled or heated. Jets have wide applications in industry because of some attractive features like simplicity of equipments, flexibility of control of mass flow rate and attainment of high heat transfer rates. In many applications a turbulent jet of liquid or gas is directed onto target area of interest.

Thus, impinging jets are often used where high rates of heat transfer are recommended. There have been number of ideas of complimenting jet impingement with other technique such as cross-flow fin cooling, ribs and turbulators. Attempts have been made to optimize each method in order to obtain effective heat transfer with low pressure losses. In order to augment the heat transfer, the boundary layer has to be thinned or partially broken and restarted.

1.4.1 Hydrodynamics of Jet Impingement

The physical model of impinging jet is shown in the Fig. 1.5a (Shi, 2004). When the coolant air is ejected from the nozzle, the jet forms a shear layer with the surrounding, relatively stagnant air. This shear layer is unstable and therefore generates turbulence in the form of a recirculation region directly adjacent to the jet’s
core. The eddies that result from the shear interaction are then propagated downstream along with the spent coolant, helping to enhance heat transfer in the regions between jets. The core of the jet impinges upon the target surface, and the region directly beneath the jet is known as the stagnation region. Here, essentially only the effects of that single jet are realized, and it is therefore the highest area of heat transfer. Upon impingement, the jet must flow radially outwards, and thus, forms a wall jet along the target surface.

The wall jet creates a thin boundary layer, which is due to the column of jet air “pressing down” on the boundary layer as it moves outward. This causes some local acceleration with higher heat transfer coefficients. The wall jet region, outside the impinging jet region, is characterized by a deceleration and disbursement of coolant as it interacts with the recirculation region, which may cause turbulent transition of the laminar boundary layer; however, locally, the influence of the wall jet enlarges and progressively decreases the heat transfer effectiveness.

The flow field of an impinging jet has been classified into 3 regions (Fig.1.5b), the free jet region, the stagnation region and wall jet region (radial flow region). The free jet region is further classified into three sub regions: the potential core region, developing flow region, and developed flow region. The potential core region is the part of the flow where there is no vorticity in the jet. Vorticity is introduced in the flow because of the free shear between the impinging jet and the stagnant air. As a result, the jet tends to become turbulent, and the potential core is eroded. The rate of energy dissipation and the length of potential core largely depend upon the shape of the nozzle, exit velocity, length of nozzle, and the sharpness at nozzle exit (Martin, 1977; Polat et al., 1993). This region of mixing is called the developing flow region, which eventually manifests as the developed flow region (Fig.1.5b). The turbulence in the free jet region and associated dissipation of energy is an important factor contributing to heat transfer in the jet further downstream (stagnation and wall jet).

The jet zone is situated directly beneath the nozzle. The fluid issuing from the nozzle mixes with the quiescent surrounding fluid and creates a flow field, which is up to a certain distance from the wall identical with the flow field of a submerged non-impinging jet. The jet flow is non-developed up to six or seven nozzle diameters from the nozzle lip. Consequently, in most applications, the nozzle-to-plate distance is too small to enable the developed jet flow conditions. A shear layer forms around the jet. Its properties depend strongly on the nozzle type. In most situations, except a
Fig. 1.5: (a) Physical Model of Impinging Jet (Shi Yuling, 2004)

Fig. 1.5: (b) Flow Zones of Free Jet
laminar flow from a tube nozzle, the shear layer is initially relatively thin compared to the nozzle diameter $d$ and therefore its dynamical behavior is similar to that of a plane shear layer. The shear layer thickness becomes comparable with the jet diameter downstream and the behavior of the layer changes considerably.

The flow issuing from the nozzle is either laminar or turbulent, depending on the nozzle type and the Reynolds number. The initially laminar flow undergoes a turbulent transition. The transition begins in the shear layer, which is unstable. The roll-up of vortices is the first stage of this transition, if the jet Reynolds number is moderate. The vortices are convected downstream by the flow and they grow, pair, lose the symmetry and finally break-up in eddies. Finally a turbulent flow is developed. In many practical situations, the nozzle-to-plate spacing is small and the jet is still in a transitional state when it impinges on the wall. If the velocity profile in the nozzle exit is flat enough, there is a potential core in the centre of the jet. The potential core is the flow region, in which the mean velocity is still the same as in the nozzle exit. The fluid inside the core did not yet transfer its momentum to the surroundings. However, the instantaneous velocity is not constant in the core. The flow is pulsating due to the velocity induction from the vortices passing in the shear layer. The potential core flow has inviscid character.

Near the stagnation point, there is a stagnation zone. It is characterized by pressure gradients, which stop the flow in axial direction and turn it radially outward. The pressure gradients also re-laminarize the flow on its arrival to the stagnation region. The boundary layer around the stagnation point is laminar because of the favorable pressure gradient. The increase of the velocity along the wall keeps the boundary layer thin and consequently, the heat transfer rates are high. The wall jet zone is free of gradients of the mean pressure. The flow decelerates and spreads here. The initially laminar boundary layer undergoes the turbulent transition that is induced by the impingement of large eddies created in the jet shear layer. This turbulent transition is believed to increase locally the heat transfer rate, which sometimes exhibits peaks in this zone. Downstream, the heat transfer rate diminishes progressively.

1.4.2 Classification of Jet Impingement

The detailed classification of jet impingement is depicted in Fig.1.6. Jets can be broadly classified as submerged or non-submerged. When medium of jet is same
Fig. 1.6: Classification of Jet Impingement
as the surrounding medium (e.g., both are air), it is called as a submerged jet (Fig.1.7a), whereas in non-submerged jets (Fig.1.7b), both the fluids are of different nature (for example, water jet issuing in air). Dynamics of the both cases is different.

In submerged jets, a shear layer forms at the interface between the jet and the surrounding fluid. This shear layer is unstable and it generates turbulence. In free jets (Fig.1.7c), this kind of instability is usually not important, and the turbulent motions in the shear layer do not have a substantial effect on the flow. The impinging jet (Fig.1.7d) is a simple flow configuration, in which the fluid issuing from a nozzle hits (usually normally) a wall (Fig. 1.5a). A characteristic feature of this flow arrangement is an intensive heat transfer rate between the wall and the fluid. It predetermines the fluid jets to be widely used in industrial applications where intensive transfer rates are needed, for example for cooling of turbine blades, laser mirrors and electronic components, for paper drying etc. The flow issuing from the nozzle is either laminar or turbulent, depending on the nozzle type and the Reynolds number. For the confined geometry a confinement plate installed parallel to the impingement target plate with a separation distance (Fig.1.7e). After being released from a nozzle tube perpendicular to an impinging surface, the jet fluid is confined to flow radially outward in a channel between two parallel plates. Whereas, in unconfined case there is no plate installed parallel to the impingement target plate (Fig.1.7f).

Confinement, which is common in industrial applications, causes the flow recirculation around the jet. The steady impinging jet stabilizes the boundary layer on the surface to be cooled (Fig.1.7g). Enhancement in the convective heat transfer coefficient is possible by disruptions of the stabilized boundary layer. Due to pulsating the impinging jet; i.e. by making the jet unsteady, the boundary layer can be disrupted periodically. The pulsated impinging jet is characterized by the pulse amplitude and pulse frequency (Fig.1.7h). After impinging the jets on a target plate, impinged fluid must be removed from the target surface without mixing to each other.

Fig.1.8 shows various ways of discharging air after impingement i.e. exit flow condition. The presence of a cross flow tends to disturb the impinging jet pattern, thicken wall boundary layers, and degrade transfer rates. Therefore after impinging air on the target surface, air should be discharged out efficiently. In minimum cross flow, air is allowed to discharge from all the four sides, whereas, in Semi cross flow condition, air is allowed to exit along longitudinal direction by installing sidewalls. In maximum cross flow condition cross-flow scheme is formed by restricting air to pass
(a) Submerged jets

(b) Non-submerged jets.

(c) A Free Jet

(d) An Impinging Jet

(e) Unconfined impinging jet

(f) Confined impinging jet

(g) Steady state jet

(h) Pulsating jet

Fig. 1.7: Types of Jets (Zuckerman and Lior, 2006)
along only one direction and placing sidewalls along remaining three directions. Note that the arrows represent the cross-flow directions.

The impingement heat transfer schemes involve either round jets or slot jets. Jets can be single or array of jets. The 3D views of single round and slot jets, as well as regular arrays of round and slot jets are shown in Fig. 1.9. Single jet impingement provides an effective means where highly localized cooling is required and multiple impinging jets or slot jets can be used for applications involving large surface areas and where a single jet is usually not sufficient for cooling it.

1.4.3 Multi-Jet Impingement

Compared to single jet impingement, multi-jet impingement involves complex flow pattern as shown in Fig. 1.10. The heating or cooling of large areas with impinging jets requires arrays; however, the flow and geometrical parameters have to be carefully selected to provide both a sufficiently high average heat transfer coefficient and uniformity of the heat transfer over the impingement surface. The need for uniformity is important in applications such as drying of textile and paper, annealing and tempering of glass, cooling of turbojet engine structure to avoid local hot spots, and spot cooling of electrical apparatus. These and other applications motivated the research. The flow from arrays of impinging nozzle has the same three flow region free jet, stagnation, and wall jet- as the single impinging jet. However, there are some basic differences in the fluid mechanics of single and multiple jets that complicate the use of single jet heat transfer results for the design of multiple jet systems.

The multiple jets systems can be subdivided into three different kinds of arrays viz. round jets from free tube (e.g. In-line and staggered), (round or slot jets from perforated plate with or without spent air holes, and rows from hole channels which can be considered as mixture of perforated plate and free jet. Many additional factors influence the heat transfer in multiple jets impingement systems. These factors include separation distance, jet-to-jet spacing distance, kind of array, geometry of jet, and diameter of jet and impingement surface form. For arrays perforated plate impingement jets, a cross flow is formed by the spent air from the impinging jets in a confined space, and the amount of cross flow increases as the flow moves downstream. Turbulent intensity of impinging jets is increased because the cross flow disturbs impinging jets at downstream region. Therefore, the local heat transfer rate
Fig. 1.8: Exit Flow Condition (Obot and Trabold, 1987)

Fig. 1.9: Single / Multi Array Round and Slot Impinging Jets

Fig. 1.10: Complex Flow Pattern within an Array of Impinging Jets

(Glaser, 1962)

a- Jet entry,  b- Confining nozzle plate,  c- Free jet,  d- Stagnation point,
  e- Stagnation zone,  f- Decelerated flow,  g- Recirculating flow,  h- Vortices
around the stagnation region is enhanced. However, at the mid-way region, the heat/mass transfer is decreased because the spent fluid upstream jets in an array can sweep away the downstream jets and delay impingement. Also the thermal boundary layer is developed in the cross flow at this region. Therefore, the heat/mass transfer coefficient is non-uniform over the overall impingement surface.

Although there exists a considerable amount of reviews on jet impingement heat transfer (Viskanta, 1993; Martin, 1977) the design of a multi-jet configuration with respect to heat transfer still remains a complex task due to following two reasons-

a. There is possibility of interference between adjacent jets prior to their impingement on the surface. This happens when jets are closely spaced ($X_p/d$, $Y_p/d$) and when $Z/d$ ratio is relatively larger. Thus the pitch (center-to-center positioning of jets) in an array, determines the degree of jet interaction.

b. There is collision of cross flows associated with adjacent impinging jets. These collisions are increased when jets are closely spaced, $Z/d$ ratio is small and jet velocity is larger.

To overcome above difficulties, proper design of multiple jets and proper system to remove spent air after impingement is very important.

1.5 EXTENDED SURFACE - HEAT SINK

The use of impinging jets on finned heat sinks leads to substantial enhancement in heat transfer. This happens because of the increase in the surface area of the heat sink. However, there are certain limitations. The fins should not be very closely spaced; otherwise they block the flow of fluid to the heat sink. In this condition to get the desired cooling Reynolds number has to be increased. The increase in fin height beyond a certain limit does not necessarily increase the heat transfer rate. Proper selection of height of impingement, in combination with jet Reynolds number and fin height can substantially improve the heat transfer from the heat sink.

Whenever the available surface is found inadequate to transfer the required quantity of heat with the available temperature drop and convective heat transfer coefficient, Extended Surfaces or Fins are used. This practice, invariably, is found necessary in heat transfer between a surface and gas as the convective heat transfer
coefficient is rather low in these situations. The finned surfaces are widely used in various industrial applications. In practice all kinds of shapes and sizes of fins are employed; some common types of fin configurations are shown in Fig. 1.11.

1.5.1 Material for Heat Sinks

The most common heat sink material is Aluminium. Chemically pure Aluminium is not used in the manufacture of heat sink, but rather Aluminium alloys. Aluminium alloy 1050A has one of the higher thermal conductivity values at 229 W/mK. However, it is not recommended for machining, since it is relatively soft material. Aluminium alloys 6061 and 6063 are the more commonly used Aluminium alloys, with thermal conductivity values of 166 and 201 W/mK, respectively. The above mentioned values are dependent on the temperature of the alloy. Copper is also used since it has around twice the conductivity of Aluminium, but is three times as heavy as Aluminium. Copper is also around four to six times more expensive as Aluminium. Aluminium has the added advantage that it is able to be extruded, while copper cannot. Copper heat sinks are machined and skived.

Another method of manufacture is to solder the fins into the heat sink base. Copper fin gives the best performance, but if the weight and cost of the heat sink is a constraint, then Aluminum fin would be preferable (Zeinab and Abdel, 2007). Another heat sink material that can be used is diamonds. With a value of 2000 W/mK it exceeds that of copper by a factor of five. In contrast to metals, where heat is conducted by electrons, lattice vibrations are responsible for diamond's high thermal conductivity. For thermal management applications, the outstanding thermal conductivity and diffusivity of diamond is an essential. Nowadays CVD diamond is used as surmounts for high-power integrated circuits and laser diodes. Composite materials can also be used. Examples are a copper-tungsten pseudo alloy, AlSiC (silicon carbide in Aluminium matrix), Dymalloy (diamond in copper-silver alloy matrix), and E-Material (beryllium oxide in beryllium matrix). Such materials are often used as substrates for chips, as their thermal expansion coefficient can be matched to ceramics and semiconductors. Aluminium is the most common, followed by copper (which is 4-6x more expensive, 3x as heavy, but has 1.5x the conductivity).

1.5.2 Fin Efficiency and Effectiveness

Fin efficiency is one of the parameters which make a higher thermal conducti-
Fig. 1.11: Types of Fins.
vity material important. A fin of a heat sink may be considered to be a flat plate with heat flowing in one end and being dissipated into the surrounding fluid as it travels to the other. As heat flows through the fin, the combination of the thermal resistance of the heat sink impeding the flow and the heat lost due to convection, the temperature of the fin and, therefore, the heat transfer to the fluid, will decrease from the base to the end of the fin. This factor is called the fin efficiency and is defined as the actual heat transferred by the fin, divided by the maximum possible heat transfer obtained from same fin.

Now according to definition, Fin efficiency \( \eta_f = \frac{Q}{Q_{\text{max}}} \)

\[
\eta_f = \frac{\tanh(mH)}{\frac{kH}{\sqrt{mH} H}} = \frac{\tanh(mH)}{mH}
\]

… (1.1)

Fins are used to enhance heat transfer and the use of fins on a surface cannot be recommended unless the enhancement in heat transfer justifies the added cost and complexity associated with the fins. In fact, there is no assurance that adding fins on a surface will enhance heat transfer. The performance of the fins is judged on the basis of enhancement of heat transfer relative to the no fin case called as the fin effectiveness (Holman, 1976). It is the ratio of heat transfer rate with fin to the heat transfer rate that would be obtained without fin which is given by- \( \varepsilon_t = \frac{Q_{\text{fin}}}{Q_{\text{unfin}}} \). The use of fins cannot be justified unless \( \varepsilon_t \) is sufficiently larger than 1. Finned surfaces are designed on the basis of maximizing effectiveness of a specified cost or minimizing cost for a desired effectiveness.

1.6 LITERATURE REVIEW

It can be stated that the impinging jets appear in many industrial applications, they provide high heat transfer rates, they have many features, which are common for the flows in the stagnation zone over blunt bodies, the impinging jets have simple geometric configuration and they are easy to study. These features explain why the impinging jet flow became an evergreen of scientific and engineering literature. The research in this field is predominantly aimed at improving heat transfer rate by various modifications in pin fin array and nozzle jets. As these modifications form a significant part of research work, a brief review of recent literature is desirable and is included below. Depending upon the objectives of this study, the literature review is
divided into three main sections, i.e. fluid flow (hydrodynamic), heat transfer (thermal) and optimization (Fig. 1.12). Each section reviews analytical, experimental and numerical studies about single/multi jet impingement on Flat plate/ Pin fin heat sink in detail. This review provides insights into convection heat transfer effects due to boundary layer separation, Reynolds number, spacing between nozzle plate and target surface and exit flow condition.

### 1.6.1 Single Jet Impingement on Flat Plate

For laminar fluid flow, Sparrow and Lee (1975) used a solution for the inviscid flow field as a boundary condition to determine the viscous flow along the impingement surface. They showed that with this method the Nusselt number \( Nu \) is proportional to the Reynolds number to the 0.5 power. Saad et al., (1977) solved the full Navier-Stokes equations using a finite difference approximation. They concluded that the Nusselt number was proportional to \( Re^{0.36} \) for a parabolic velocity profile and to \( Re^{0.5} \) for a flat velocity profile in the range of the \( Re \) from 900 to 1950. These numerical computations show the importance of the velocity profile in the stagnation region and also in the wall jet region. The heat transfer from a parabolic impinging jet is higher than that from a uniform impinging jet in both stagnation and wall jet regions.

The fact that a turbulent impinging jet yields a higher heat transfer than a laminar jet has been recognized for some time (Huang, 1963; Donaldson et al., 1971; Gardon and Arfirat, 1965). Donaldson et al., (1971) compared the predicted laminar heat transfer and the measured heat transfer revealed that the measured turbulent heat transfer rate is 1.4 - 2.2 times as high as the laminar rate. The Nusselt and Reynolds numbers for air at the stagnation point are usually expressed as \( Nu_{st} = c. Re^n \). The Reynolds number exponent \( n \) from laminar boundary layer theory for a uniform exit velocity profile is 0.5. Polat et al. (1993) compared the values of the exponent \( n \) that were determined by various numerical and experimental studies and found that there is considerable scatter. The value of \( n \) ranges from as low as 0.23 to as high as 0.67 and depends on whether the inlet velocity profile is flat or parabolic. The differences in the exit velocity profiles make a direct comparison of experimental and numerical results. The stagnation point heat transfer within the plate to jet (nozzle-to-plate ratio, \( Z/d \leq 5 \)) is in good agreement with laminar boundary layer for Reynolds number of 1050 and 1860 (Popiel et al., 1980).
Fig. 1.12: Flow Chart of Literature Review.
In the developed region $Z/d \geq 8$, strong free jet turbulence effects were observed, that augmented the convective heat transfer (Failla et al., 1999; Baydar, 1999). Zu et al., (2007) developed a co-relation for stagnation Nusselt number as-

$$Nu_s = 0.423 \cdot Re^{0.642} \cdot (Z/d)^{-0.3} \cdot e^{-[0.055(L/d)\cdot]}$$

... (1.2)

for $5 \leq \frac{L}{d} \ll 50; 1 \ll \frac{Z}{d} \ll 6; 10000 \ll Re \ll 30000$.

Donaldson et al. (1971) obtained the relation for the Nusselt number

$$Nu_s = (0.5c \cdot Pr)^{0.5} \cdot Re^{0.5}$$

... (1.3)

where, the value of $c$ is 1.13 for fully developed free jets.

The experimental data of Hoogendoorn (1977) clearly demonstrated that not only the stagnation point but also the local Nusselt number depend on the nozzle design and on the nozzle-to-plate ratio $Z/d$. The effects of ratio $Z/d$ are now being understood by (Angioletti et al., 2003) and found that for $Z/d > 4$ the maximum heat transfer coefficient occurs at the stagnation point of the jet. This optimum separation distance, $Z/d$, apparently coincides with the length of the potential core. Beyond the potential core, the jet velocity decays and the heat transfer coefficient falls. Beyond the potential core, the jet velocity decays and the heat transfer coefficient falls. Jung-Yang, (2006) concluded that a stagnation Nusselt number is proportional to the 0.638 power of the Re and inversely proportional to the 0.3 power of the $Z/d$. Stagnation Nusselt number decreases with an increase of $Z/d$, because a larger spacing between the jet plate and impingement plate allows a longer distance for the jet to mix with the recirculation flow before the jet reaches the impingement plate. This flow mixing would deteriorate the heat transfer. Jambunathan et al. (1992) concluded that the Nusselt number is independent of $Z/d$ up to a value of 12 diameters at $X_p/d > 6$ from the stagnation point.

Martin (1977) worked for range of parameters $Z/d$ from 2 to 12 and $Re = 2000$ to 400000. He observed that for $\frac{Z}{d} \geq 5$ the distribution is characterized by a bell shaped curve for which Nu monotonically decays from a maximum value at stagnation point, whereas for $\frac{Z}{d} < 5$ the distribution is characterized by a second maximum, whose value increases with increasing jet $Re$ and may exceed that of the
first maximum. He recommended the average Nu correlation for single round nozzle as

\[ \frac{Nu_a}{Pr^{0.42}} = F_1\left(\frac{X}{d}, \frac{Z}{d}\right) \cdot F_2(Re) \]  

\[ \text{... (1.4)} \]

Where, \( F_1 = 2 \cdot Re^{0.5}(1 + 0.005 \cdot Re^{-0.55})^{1/2} \) and \( F_2 = \frac{d}{X} \cdot \frac{(1-1.1d/X)}{1+0.1 \left(\frac{Z}{d}-6\right)d/X} \).

Obot \textit{et al.} (1979) found that the effect of entrainment on the heat transfer to a turbulent jet is strongly dependent on nozzle configuration. Most of the published literature pertains to jets generated with well characterized nozzles. However, in many applications, the square nozzle configuration is preferred primarily because of ease of fabrication and installation, especially for multiple jet systems. The experimental data of Hoogendoorn (1977) clearly demonstrated that not only the stagnation point but also the local Nusselt number depend on the nozzle design and on the nozzle-to-plate ratio \( Z/d \). The effects of nozzle-to-impingement surface spacing are now being understood and the local heat transfer coefficient depends on several factors whose variation is complex (Lytle and Webb, 1991). For \( Z/d > 4 \) the maximum heat transfer coefficient occurs at the stagnation point of the jet (Livigood and Hrycak, 1973). This optimum separation distance, \( Z/d \), apparently coincides with the length of the potential core. Beyond the potential core, the jet velocity decays and the heat transfer coefficient falls. Depending on the jet Reynolds number and separation distance two radial peaks have been observed for circular air jets (Lytle and Webb, 1991). The inner peak is located at approximately \( X_p/d = 0.5 \). For \( Z/d < 0.25 \), global mass continuity requires that the fluid accelerate between the nozzle and the impingement surface. The resulting acceleration produces local thinning of the boundary layer, explaining the peak seen at \( X_p/d = 0.5 \) (Angioletti \textit{et al.}, 2003). The discussion of prior work by these authors suggests that the local maximum in the Nusselt number at \( Z/d \) is strongly dependent on separation distance and Reynolds number.

Jambunathan \textit{et al.} (1992) have critically reviewed experimental data for the rate of heat transfer from impinging turbulent jets with nozzle exit Reynolds numbers in the range of 5000–124000 from the considerable body of literature available on the subject. The geometry considered is that of a single circular jet impinging orthogonally onto a plane surface for nozzle-to-plate distances from 2–16 nozzle diameters and over a flow region up to six nozzle diameters from the stagnation point.
Existing correlations for local heat transfer coefficient express Nusselt number as a function of nozzle exit Reynolds number raised to a constant exponent. However, the available empirical data suggest that this exponent should be a function of nozzle-to-plate spacing and of the radial displacement from the stagnation point. A correlation for Nusselt number of the form suggested by this evidence has been derived using a selection of the data. The review also suggests that the Nusselt number is independent of nozzle-to-plate spacing up to a value of 12 nozzle diameters at radii greater than six nozzle diameters from the stagnation point. The results from a simple extrapolation for obtaining heat transfer coefficients in the wall jet region compare favorably with published data.

Garimella and Rice (1995) investigated experimentally the local heat transfer from a small heat source to a normally impinging jet in confined and unconfined configuration for $Re = 4000 – 23000$, $Z/d = 1$ to 14 and nozzle diameter $(d)$ 0.79 to 6.35 mm. Secondary peak in local heat transfer is observed at $X_p/d = 2$ were more pronounced at smaller spacing and larger nozzle diameters for a given $Re$, and shifted radially outward from the stagnation point as the spacing increased. The results indicate that for a given $Z/d$ and $Re$, the smaller nozzles generally produce higher heat transfer coefficients and for larger nozzle diameters secondary peaks are pronounced.

Juan et al. (2009) investigated experimentally the heat transfer of confined air jet impingement using tiny size round nozzle (diameter no more than 1mm). By comparing the heat transfer effects of 1.5 mm nozzle and 1mm nozzle, a rule was found that at the same Reynolds number, the bigger diameter has higher Nusselt number. For 1 mm nozzle, when the Reynolds number increased above 14000, the Nusselt number does not increase obviously. It may be showed that for strongly confined air jet with tiny size round nozzle, when Reynolds number increased to a definite value (such as about 14000), the high pressure caused by the high velocity air flow makes the quantity of heat stay at the area between impinging plate and target plate. So the increase of Reynolds number has little effect on the Nusselt number.

Khummongkol et al. (2004) employed $2^3$ factorial designs to plan the experiment, and an analysis of variance to analyze the effects of nozzle diameter, nozzle-to-surface distance and the air velocity on heat transfer coefficient. The analysis of variance indicates that among the three factors investigated, air velocity has strongest effects on the heat transfer between the impinging air jet and the impinged surface. The air velocity increases the heat transfer coefficient. The next
influential factor is nozzle-to-surface distance. It decreases the heat transfer rate. The nozzle diameter does not statistically affect the heat transfer of the jet impingement. There are also no two-factor interactions.

Most industrial applications of impinging jets are concerned with turbulent flow in whole domain downstream of a nozzle. Modeling of turbulent flow presents the greatest challenge for rapidly and accurately predicting impingement heat transfer even under a single round jet. Over the past decades, although no single model has been universally accepted to be superior to all classes of problems, various turbulent models have been developed successfully to roughly predict impingement flow and heat transfer. However, there are only a limited number of studies concerned with comparisons of the reliability, availability and capability of different turbulent models for impingement flows.

Chattopadhyay (2004) predicted heat transfer characteristics of laminar jets impinging on a surface using annular and circular jet. An axi-symmetric formulation was used for solving mass, momentum and energy equations with SIMPLE algorithm. It was found that heat transfer from annular jet was about 20% less compared to the circular jet and the distribution pattern of the $Nu$ over the impinging surface scales with $Re^{0.55}$. Thakare and Joshi (2000) evaluated twelve versions of low Reynolds number $k-\varepsilon$ models and two low Reynolds stress model (RSM) for heat transfer in turbulent pipe flows. Their comparative analysis between the $k-\varepsilon$ models and RSM models for the Nusselt number prediction is in favour of the applicability of the $k-\varepsilon$ models. Shi et al. (2002) presented simulation results for a single semi-confined turbulent slot jet impinging normally on a flat plate. The effects of turbulence models, near wall treatments, turbulent intensity, jet Reynolds number, as well as the type of thermal boundary condition on the heat transfer were studied using the standard $k-\varepsilon$ and RSM models. Their results indicate that both standard $k-\varepsilon$ and RSM models predict the heat transfer rates inadequately, especially for low nozzle-to-target spacing. For wall-bounded flows, large gradients of velocity, temperature and turbulent scalar quantities exist in the near wall region and thus to incorporate the viscous effects it is necessary to integrate equations through the viscous sublayer using finer grids with the aid of turbulence models. In the study of Wang and Mujumdar (2005), five versions of low Reynolds number $k-\varepsilon$ models for the prediction of the heat transfer under a two-dimensional turbulent slot jet were
analyzed by comparison with the available experimental data. Effects of the magnitudes of the turbulence model constants were also carried out.

Quan et al. (2008) used k–ε turbulence model. The optimal separation distance (Z/d) for stagnation region Nu is found to be about 5. Badra et al. (2007) used the commercial Fluent package for numerical simulation. They used many turbulent models and found that Shear-Stress Transport (SST) k –ω model can perform very well in a range of Z/d (5 < Z/d < 30) and Re (5000 < Re < 30000), in order to give confidence in its use as a predictive tool. The first peak of local Nu appears at Xp/d = 0.5 and the second peak is at Xp/d = 2 for Z/d < 5. At Z/d = 6 secondary peak disappear. The stagnation Nusselt number is maximum at Z/d = 6. Zu et al. (2007) analyzed numerically using CFD commercial code FLUENT 6.1. The relative performance of seven versions of turbulent models is investigated by comparing the numerical results with available benchmark experimental data. It is found that SST k –ω model and Large Eddy Simulation time-variant model can give the better predictions of fluid flow and heat transfer properties; especially, SST k –ω model is recommended as the best compromise between the computational cost and accuracy.

1.6.2 Multi-Jet Impingement on Flat Plate

In case of multi jet impingement geometry can be characterized by non-dimensional parameters, Xp/d and Z/d which are normalized by the jet nozzle diameter (d). In addition to these factors, the cross-flow is another important parameter that affects jet impingement heat transfer performance. Cross-flow is the spent jet flow upstream of the local jet and reinforced by the local jet flow after its impingement. Strong cross-flow is a non-desirable factor in impingement heat transfer, because cross flow tends to push the impinging air flow downstream and dilutes the impinging jet intensity. Several experiments and numerical simulations have been performed to analyze the effectiveness of steady multi jet impingement cooling on flat plate. Some of them are described as below:

The flow from arrays of impinging nozzle has the same three flow region free jet, stagnation, and wall jet- as the single impinging jet (Goldstein and Behbahn 1982; and Hollworth and Berry, 1983). However, there are some basic differences in the fluid mechanics of single and multiple jets that complicate the use of single jet heat transfer results for the design of multiple jet systems. The individual jets that make up a multiple jet system may be influenced by two types of interactions that do not occur
for single jets. First, there is possible interference between adjacent jets prior to their impingement on the surface. The likelihood of such interference effects is enhanced when the jets are closely spaced and when the $Z/d$ is relatively large. Second, there is an interaction due to collision of surface flows associated with the adjacent impinged jets. These collisions are expected to be of increased importance when the jets are closely spaced, the jet orifice impingement plate separation is small, and the jet velocity is large (Gao et al., 2003; Koopman, 1975).

For any impingement jet array situations, cross flow is inevitable. Some studies have shown how cross flow affects heat transfer. These studies presented correlations that account for regular arrays of impingement jet holes with low to moderate cross flow effect. Florschuetz et al. (1984) also studied on the effect of initial cross flow on impingement heat transfer. Their results show that the initial cross flow lowers the impinging heat transfer performance. As the initial cross flow rate increases, convective heat transfer will be more dominated by the cross flow. Huang et al. (1998) studied the effect of spent air flow direction on impingement heat transfer, when the feeding flow is parallel to the spent flow. They found when the spent flow has an opposite direction to incoming flow, Nusselt number peak occurs at leading section of the heat transfer target wall. When spent flow has the same direction as the incoming flow, the Nusselt number at the trailing part will be slightly higher than the leading part, but with overall performance 40% lower. According to their study, when both directions are allowed for spent airflow, Nusselt number is uniformly high, which is the best case. Some researchers proposed to remove spent airflow right after impinging in order to reduce the cross flow. Hollwarth and Dagan (1980), conducted experiments on this geometry. They drilled holes on the target plate at the same position with impingement jet holes and the positions in between. They reported 20-30% higher heat transfer rate compared with side venting case. Ekkad et al. (1999) also studied a jet impingement plate with holes to reduce cross-flow effect. They saw that the presence of holes on target surface increases overall heat transfer. A combined experimental and numerical investigation of the heat transfer characteristics within an array of impinging jets has been conducted by Yunfei et al., (2010). The effects of the variation in different cross-flow schemes (minimum, semi and maximum cross flow) were examined and found that the heat transfer rates for maximum cross-flow case were clearly much lower than the others. Compared to single jet impingement, multiple jet impingements provide both high average heat
transfer coefficient and uniformity of heat transfer over impingement surface (Badra et al., 2007; Hollworth and Berry, 1983).

Goldstein et al. (2003) performed experimental as well as numerical studies on steady impinging jets. They presented a general review of heat transfer in both axisymmetric and planar jets. The numerical studies investigated different turbulent models for axisymmetric submerged air jet. This review summarized the results with the effects of nozzle-to-plate spacing and Reynolds number in all three zones. The results are presented in terms of Nusselt number. Cooper et al. (1993) reported an extensive set of measurements of an orthogonal impinging turbulent jet onto a large plane surface. Two Reynolds numbers have been considered $2.3 \times 10^4$ and $7 \times 10^4$, while the height of the jet discharge above the plate ranged from 2 to 10 jet diameters, with emphasis on 2 and 6 jet diameters. All the turbulent parameters, such as turbulent kinetic energy profile and radial RMS turbulent velocity profiles, were measured. Turbulent kinetic energy profile and radial mean velocity profiles were compared with numerical results.

Morris et al. (1996) used FLUENT package to investigate local heat transfer coefficient distribution due to a normally impinging, axisymmetric confined and submerged jet. The test fluid was liquid, FC-77, which is widely used as a coolant in liquid cooled electronic systems. Numerical predictions were made for nozzle diameters of 3.18 and 6.35 mm with several nozzle-to-source height spacing and Reynolds number in the range of 8500 to 13000 in the turbulent zone. Various near-wall models were used to improve the local heat transfer coefficients.

Metzger et al. (1979) investigated experimentally another important difference between the heat from a single jet and an array of jets. They found that for a jet-to-jet spacing distance of 1.67-6.67 nozzle diameters, a maximum in the average heat transfer coefficient was observed for a separation distance ($Z/d$) of about one. A value of $P/d = 4$ was recommended by Freidman and Mueller (1991) to reduce adjacent jet interference and maximum heat transfer over the surface for large separation distance $Z/d \geq 8$, while Martin (1977) recommended an optimum value of roughly 7 diameter for $Z/d = 5.4$.

The heat transfer in the downstream of the target plate was also lower because the strong cross-flow decreased the heat transfer of the jets. The local minima value for the maximum cross-flow scheme in the downstream region was lower than the others because the cross-flow reduced the jet-to-jet interferences. The jet-to-plate
spacing $Z/d = 3$ achieved the highest overall heat transfer coefficients with both inline and staggered patterns. The inline pattern always performs better than to the staggered pattern. These trends were also found by Florschuetz et al. (1984) and Metzger et al. (1979). For the inline pattern, the jets are protected from the oncoming cross-flow by the upstream jets. For the staggered pattern, the crossflow influences the jets more directly, which causes stronger diffusion and leads to a reduced overall heat transfer performance.

Young et al. (2005) investigated experimentally the effects of the pore density of aluminum foam heat sinks, the jet velocity, the jet-to-jet spacing and the nozzle plate-to-heated surface separation distance in a $3 \times 3$ square multi-jet impinging array on the averaged Nusselt number. Thermal performances of 10, 20, and 40 PPI (pores per inch) aluminum foam heat sinks and a conventional plate-fin heat sink are evaluated in terms of the averaged Nusselt number. The jet Reynolds number is varied in the range of $Re = 1000–13650$. The highly permeable 10 PPI aluminum foam heat sink shows higher Nusselt numbers than the 20 and 40 PPI aluminum foam heat sinks both in the multi-jet and the single jet impingements. For the single jet impingement, the aluminum foam heat sinks display 8–33% higher thermal performance compared to a conventional plate-fin heat sink while the enhancement is 2–29% for the multi-jet impingement. The multi-jet impingement shows higher heat transfer enhancement than the single jet impingement at Reynolds number equal to at 13650.

Badra et al. (2007) conducted numerical simulation using the commercial package Fluent and Gambit has been validated against experimental data. Multi-jet case has been evaluated against experimental data. It showed very good qualitative agreement with the available data set. Ashok Kumar and Prasad (2009) reported a computational study on flow and heat transfer from a single row of circular air jets impinging on a concave surface with either one or two rows of effusion holes or without effusion holes. They found that the pressure distribution is higher for the configuration with the air exit only through effusion holes. When a single row of impingement holes is aligned with a single row of effusion holes (all in-line cases), heat transfer is the lowest. However, when the impingement holes and effusion holes are staggered, heat transfer increases significantly by up to about 3.45 times that of inline arrangement.

Zu et al. (2007) have used seven versions of turbulent models, including the standard $k – \varepsilon$ model, the renormalization group $k – \varepsilon$ model, the realizable $k – \varepsilon$
model, the standard $k - \omega$ model, the SST $k - \omega$ model, the Reynolds stress model and the Large Eddy Simulation, for the prediction of this type of flow and heat transfer. For $10000 \leq Re < 30,000$, $5 \leq W/d \leq 40$, $10 \leq L/d \leq 150$ and $1 < Z/d < 6$ they have presented correlation as:

$$Nu_s = 0.423 \cdot Re^{0.64} \left( \frac{Z}{d} \right)^{-0.3} \cdot e^{-\left[0.044(W/d) + 0.011\left(\frac{Z}{d}\right)\right]} \ldots (1.5)$$

A combined experimental and numerical investigation of the heat transfer characteristics within an array of impinging jets has been conducted by Yunfei et al. (2010) for both inline and staggered patterns. Local jet temperatures were measured at several positions on the impingement plate to account for an exact evaluation of the heat transfer coefficient. The effects of the variation in different cross-flow schemes (minimum, semi and maximum cross-flow), $Z/d$ ratio, and Reynolds number on the distribution of the local Nusselt number and the related pressure loss were investigated experimentally. In addition to the measurements, a numerical investigation was conducted which showed that state-of-the-art CFD codes that can be used as suitable means in the thermal design process of such configurations. The heat transfer rates for maximum cross-flow case were clearly much lower than the others.

1.6.3 Single Jet Impingement on Pin Fin Heat Sink

Hung et al., (2007) has investigated effects of width, height, $Z/d$ ratio and Re number on the thermal performance of heat sink with confined impingement cooling, is measured by infrared thermography. He concluded that Re number plays an important role in thermal resistance. Increasing width of fins increases total exposure surface of heat sinks which enhances heat convection but on the other side, inter-fin flow passages decreases, consequently increasing the flow resistance. Therefore increasing width of fins combined with an appropriate Re number can improve thermal performance. Enhancement of thermal performance can also be done by increasing height but height of fins beyond critical value might also impede the penetrating ability of impinging jets. Thermal resistance is decreased more effectively by increasing width of fin than by increasing height. The range of parameters considered are $Z/d = 12$, $d = 12 \ mm$, $W = 80 \ mm$, $D = 6.5$, 8 and 9.5 $mm$, $H = 35$, 40 and 45 $mm$, $N \times N = 6 \times 6$ array, fin arrangement- Inline, fin shape- rectangular and $Re = 500 – 30000$. Experimental results were obtained by Hani and Garimella,
for confined air jet impingement on square heat sink of 5 × 5, 7 × 7 and 9 × 9 arrays with inline and staggered arrangement for Reynolds number varying from 8000 to 450000. Heat sink base temperature is measured by T type thermocouples at four different positions which are equally spaced along diagonal. The heat transfer coefficient \( h \) obtained with pin fin heat sinks are higher compared to unpinned cases. At fixed flow rate, decrease in nozzle diameter resulted in an improvement in heat transfer, however effect of nozzle diameter was much more pronounced for unpinned heat sinks. Total fin effectiveness serves as measure of heat transfer enhancement effected by fins compared with unpinned surface. Total effectiveness is a ratio of \( h_p \) to \( h_{up} \). It is in the range of 3 to 5. Jim and Henry (2002) has given emphasis to the enhancement of heat transfer by optimizing fin geometry and nozzle to sink spacing using fin diameters 3, 5 and 7 mm with 9000 < Re < 26000. From geometry obtained relation to find number of array.

\[
N d + (N-1) P = W \quad \text{(1.6)}
\]

where \( N \) is number of fins along \( W \), \( d \) is diameter of fin and \( P \) is spacing between fins, \( W \) is fin distribution area. The optimal air impingement thermal performance occurs when \( P/W = 0.05 \), \( D/W = 0.1 \) and \( H/W = 0.53 \) with 7 × 7 design. For constant Re number, Nusselt number increases with increasing \( Z/d \) for 2 to 8 and remains relatively constant for range 8 to 12.

Patil and Devade (2011) have performed experimental work on various square pin fin heat sinks using single jet impingement method. By varying heat flux from 50 W/cm\(^2\) to 100 W/cm\(^2\), average temperature of heat sink is from 37 to 40 °C which is the requirement of electronic circuits to serve efficiently for long time. It is observed that with increase in Re, heat transfer coefficient increases as flow pattern covers the component surface completely to cool it off and return flow also escape with the opening available. At higher Re (5000) and \( Z/d = 5 \) maximum value of \( h \) is noted. With low \( Z/d \) the air after impingement does not get space to spread because of close impingements. Sanyal et al., (2009) investigated numerically for both steady and pulsating condition. Due to combined effect of fins and pulsation, thermal enhancement is up to 15 to 20% but considering noise and cost of pulsation steady jet is better.

Jung and Maveety (2000) performed experimental and numerical study to investigate and optimize three pin fin heat sink array geometries (5 × 5, 7 × 7 and 9 ×
9) over the range $7800 < Re < 19700$ by using the standard $k-\varepsilon$ turbulence model. This optimization study considers the effect of heat sink geometries and flow rate on the cooling performance. The $7 \times 7$ heat sink provide optimal performance due to maximum convective surface area, largest heat transfer coefficient and producing best turbulent flow. Jung and Maveety (2002) performed numerical experiments to obtain optimal fin height when $Z/L = 0.53$, beyond this gain could be offset by material and manufacturing costs. For dense finned array, little air penetration to cool heat sink. By contrast small finned array also reduces cooling effectiveness.

Yue-Tzu and Huan-Sen, (2008) presented the numerical simulation of heat sink of 12 type un-uniform fin height design for Reynolds number from 15000 to 25000. It is found that cooling performance can be enhanced by increasing fin height near center of heat sink. The result also shows that there is a potential for optimizing the un-uniform fin height design. Heng-Chien et al., (2001) studied a variety of pin fin parameters to predict optimum thermal performance. They argued that under fix fin numbers and same flow rate, thick fin will increase cooling area but cause more pressure drop. Higher fin has better cooling ability but the worse efficiency.

Ho-Chul et al., (2002) used an experimental method to study 20 types Aluminum square pin fin heat sinks with inlet velocity ranging from 1 to 5 m/s. As the fin height increases, the pressure drop across a heat sink decreases to an asymptotic value and the thermal resistance of a heat sink decreases. As surface porosity increases, the pressure drop across a heat sink decreases and the thermal resistance of a heat sink increases. Larger porosity and smaller inlet velocity are recommended for the case that the pumping power and the thermal resistance are equally important in the design of shrouded square pin fin heat sinks. Dong-Kwon et al., (2009) compared thermal performances of plate-fin and pin-fin heat sinks commonly used in the electronic equipment industry. Optimized pin-fin heat sinks possess lower thermal resistances than optimized plate-fin heat sinks when dimensionless pumping power is small and the dimensionless length of heat sinks is large. On the contrary, the optimized plate-fin heat sinks have smaller thermal resistances when dimensionless pumping power is large and the dimensionless length of heat sinks is small.

1.6.4 Multi-Jet Impingement with Effusion Holes/Slots

From above discussion it is clear that cross-flow of spent air decreases the average heat transfer coefficient. For the conventional array impinging jets, a cross-
flow is formed by the spent air from the impinging jets in a confined space, and the amount of the cross-flow increases as the flow moves downstream. Turbulence intensity of impinging jets is increased because the cross-flow disturbs impinging jets at downstream region. Therefore, the local heat/mass transfer rate around the stagnation region is enhanced. However, at the mid-way region, the heat/mass transfer coefficients are decreased because the thermal boundary layer develops in the cross-flow at this region and flow pattern is similar to the duct flow (Young et al., 2005). Therefore, the heat/mass transfer coefficients are non-uniform over the overall impingement surface, and this non-uniformity can induce thermal stress problem on the target plate.

Cho and Goldstein (1996) and Cho and Rhee (2001) investigated the effect of hole arrangements on local heat/mass transfer characteristics for the array jet impingement with spent air removal through the effusion holes on the target plate. They found that the high heat transfer rate is induced by strong secondary vortices and flow acceleration, and the overall heat transfer rate is approximately 45–55% higher than that for impingement cooling alone. Huber and Viskanta (1994) studied the effect of spent air exit in the orifice plate on the local and average heat transfer for 3 × 3 square array jet with 2 × 2 square spent air exit using the liquid crystal technique. They found that the interaction of adjacent impinged jets is reduced by spent air and the heat transfer on target plate is more enhanced. They also investigated the effect of jet-to-jet spacing on the heat transfer; it was found that, for large plate spacing, jet interference causes a significant degradation of the heat transfer.

Dong-Ho et al. (2003) investigated the effects of spent air flows with and without effusion holes on heat/mass transfer on a target plate for array impinging jets and reported that for small gap distances, heat/mass transfer coefficients without effusion holes are very non-uniform due to the strong effects of cross-flow and Re-entrainments of spent air. However, uniform distributions and enhancements of heat/mass transfer coefficients are obtained by installing the effusion holes.

1.6.5 Multi-Jet Impingement on Pin Fin Heat Sink

As per Literature survey it is observed except Hani and Garimella, (2000) nobody has worked on multi jet impingement on pin fin heat sink. They also studied multi jet case using 2 × 2 jet array by comparing pin-finned heat sinks with unpinned heat sinks. The heat transfer coefficient is increased by 10% by increasing $X_p/d = 2$ to
3. They concluded that the effectiveness of the pin-finned heat sinks is in the range of 2 to 3 times unpinned and multi jets yield higher heat transfer coefficients than the single jet.

1.6.6 Optimization of the Pin Fin Heat Sink

Wei-Chung et al. (2006) presented a novel electromagnetic optimization technique based on Taguchi method. He observed that compared to other optimization techniques, such as the genetic algorithm (GA) and particle swarm optimization (PSO), Taguchi method is easy to implement, and can quickly converge to the optimum solution. Senthilkumar et al. (2010) analyzed the heat pipe working parameters using Taguchi methodology. The Taguchi method is used to formulate the experimental work, analyze the effect of working parameters of the heat pipe and predict the optimal parameters of heat pipe such as heat input, inclination angle and flow rate. It is found that these parameters have a significant influence on heat pipe performance. The analysis of the Taguchi method reveals that, all the parameters mentioned above have equal contributions in the performance of heat pipe efficiency, thermal resistance and overall heat transfer coefficient. The L9 orthogonal arrays are used for the design of experiments and Analysis of Variance (ANOVA) is used to analyze the experimental data.

Ko-Ta et al. (2006) studied the effects of design parameters and the optimum design parameters for a Pin-Fin heat sink (PFHS) under single jet impingement with the multiple thermal performance characteristics have been investigated by using the grey-fuzzy logic based on the orthogonal arrays. Various design parameters, such as height and diameter of pin-fin and width of pitch between fins are explored by experiment. The average convective heat transfer coefficient, thermal resistance and pressure drop are considered as the multiple thermal performance characteristics. Through the grey-fuzzy logic analysis, the optimization of complicated multiple performance characteristics can be converted into the optimization of a single grey-fuzzy reasoning grade. In addition, the analysis of variance is applied to find the effect of each design parameter on the each or all thermal performance characteristics. Then the results of confirmation test with the optimal level constitution of design parameters have obviously shown that the fin height and the pin diameter are the significant influential factors.

Kenan et al. (2006) were investigated the effects of the heights, widths of the
hexagonal fins, streamwise and spanwise distances between fins and flow velocity parallel to base of heat sink on thermal resistance and pressure drop characteristics using Taguchi experimental design method. L18 orthogonal array was selected as an experimental plan for the five parameters mentioned above. While the optimum parameters were determined, due to the goals (above aims) more than one being, the trade-off among goals was considered. First of all, each goal was optimized, separately. Then, all the goals were optimized together, considering the priority of the goals, and the optimum results were found to be fin width of 14 mm, fin height of 150 mm, spanwise distance between fins of 20 mm, streamwise distance between fins of 10 mm and flow velocity of 4 m/s. The most important parameters affecting the thermal resistance are fin height, fin width and fluid velocity whereas the most effective parameter on the dimensionless pressure drop is found to be fin width.

Jim and Henry (2002) have investigated on square pin fin heat sink under single jet impingement cooling. Experimental and numerical results are presented for heat transfer from a C4 mounted organic land grid array (OLGA) thermal test chip cooled by air impingement. Five heat sink geometries were investigated for Reynolds numbers ranging from 9000 to 26000. The dimensionless nozzle-to-heat sink vertical spacing was varied between 2 and 12. In this study, they investigated the interactions between heat sink geometry, flow conditions and nozzle setting and how they affect the convective heat transfer and overall cooling of the test chip as measured by total thermal resistance. Optimizing fin arrays by minimizing the overall heat sink thermal resistance instead of focusing solely on maximizing the heat transfer from the fins is shown to be a better design criterion. Sahiti et al. (2007) optimized geometry of the various cross-section pin fin array with approach velocity parallel to heat sink base using commercial optimization software mode-FRONTIER (www.esteco.it.). It is shown that by subsequent use of the virtual solutions from the response surface modeling (RSM) of that software and their validation with Star-CD; a complete ‘Pareto-frontier solution’ can be obtained.

Some significant conclusions derived from the literature review of the jet impingement cooling are given below:

- The multiple jet systems enhance the heat transfer more than a single nozzle by approximately 75% for optimum spacing distance $X_p/d = 3$ to 4. Also, the interaction between adjacent jets both before and after
impingement is an important consideration in the design and characterization of multiple jet systems.

- Most of study is focused on minimum, semi and maximum cross flow arrangement of spent air, it is observed cross flow degrades the heat transfer coefficient.
- Spent air exit through the jet plate is effective. There is very less study on spent air exit through effusion slots in jet plate.
- There are very less investigation (rather no) on circular pin fin heat sink under multi jet air impingement in experimental as well as numerical studying area.
- There are potentials to improve heat transfer with air impinging jets provided an optimal combination of the governing parameters is to be identified.
- Exploring this problem is a challenge and its analysis may provide guidelines to design a cooling system for electronic equipments.

1.7 **SCOPE OF PRESENT WORK**

As study on the performance of pin-finned heat sink is scarcely found in literature, the present research involves a numerical and experimental investigation of heat transfer characteristic of different finned array using multi jet impingement. The goal of this research is to develop strategies for finding the optimal pin-fin geometry and operating parameter. To reach this goal, following action plan is done-

1. **Formulation of CFD Model** - To develop models that can be used to predict fluid flow and heat transfer from pin-fin heat sinks for a range of various operating parameters and pin fin geometries.
2. **Develop experimental setup** - for jet impingement which will be capable of predicting fluid flow and heat transfer for a pin-fin heat sink
3. **Parametric study** - Once CFD model is validated with experimental data, examine the influence of various parameters on overall performance of heat sink. Based on these studies, optimal selection of heat sink configurations will be investigated for various operating conditions. Finally, this research is targeted to expect the optimised design that will create the best available heat sink solution by improving heat transfer and fluid flow characteristics.
Most of study is conducted on simple geometries like flat plate or smooth cylinders. The various governing parameters of this problem are Reynolds number, jet-to-plate distances $Z/d$ ratio, heat sink geometrical parameters and nozzle geometrical parameters. By combining the parameters mentioned above, a significant augmentation in heat transfer can be expected with air as the cooling medium which has wide range of applications in thermal management of electronic systems.

1.8 ORGANIZATION OF THE THESIS

The thesis is divided into six CHAPTERS.

In CHAPTER 1 theory of different heat transfer enhancement techniques and particularly Jet impingement technique is described. Also a review of the related literature is presented. Some important conclusions are drawn from literature review leading to identifications of some gaps and defining the scope of the present work.

CHAPTER 2 explains the numerical modeling formulation for the problems of convection heat transfer using single and multi jet impingement on both flat plate and pin fin array heat sink of different configurations. Literature review reveals that there are a number of variables that control the heat transfer rate in impingement cooling such as Reynolds number, nozzle to target heater plate ratio ($Z/d$), pin fin heat sink configurations, exit flow condition. After ensuring grid and domain independence, computations were carried out for a range of geometric and other parameters.

In CHAPTER 3 as a part of the present investigation an experimental test facility is designed and developed and commissioned, and the description of the experimental setup and test procedure are presented. DAQ system is also designed to measure temperature, pressure, velocity. Observations are recorded for a range of geometric parameters and air velocities. Sample calculations are provided to deduce the results under multi-jet impingement on flat plate and pin fin array. Thus the experiment is intended to study the effect of Reynolds number, $Z/d$, exit flow conditions and pin fin heat sink geometric parameter on heat transfer rate.

CHAPTER 4 explains results and discussions of the forced convection heat transfer coefficient and the influence of other parameters on jet arrays. The effect of local and average Nusselt number is investigated by choosing different nozzles arrays and different pin fin heat sinks with minimum, semi and maximum exit flow condition. Steady state temperatures are measured and heat transfer coefficients are estimated for each case. The experimental results are compared with the numerical
results. The experimental results show good agreement with the numerical results. Some useful correlations are developed making use of the experimental and numerical results for Nusselt number as a function of various system parameters. Heat transfer characteristics are presented in terms of temperature contours and Nusselt contours. Fluid flow characteristics are presented in the form of velocity vectors and stream lines. Comparison between single and multi-jet impingement on flat plate and pin fin heat sinks are also made. A correlation has been developed for multi-jet impingement on pin fin heat sink with effusion slots on the nozzle plate. Finally comparison between traditional exit flow condition and effusion slot are discussed which remains as baseline for optimization work.

CHAPTER 5 describes the Taguchi method used for optimization of pin fin heat sink geometry under various operating conditions.

In CHAPTER 6 overall conclusions based on numerical and experimental results are summarized and presented in this chapter. Optimum selection of Pin Fin Heat Sink is reported and finally the possible future scope for the present work is defined.