World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:7, No:3, 2013

Numerical and Experimental Investigations on Jet Impingement Cooling

Arun Jacob, Leena R., Krishnakumar T.S., and Jose Prakash M.

***Abstract*—**Effective cooling of electronic equipment has emerged as a challenging and constraining problem of the new century. In the present work the feasibility and effectiveness of jet impingement cooling on electronics were investigated numerically and experimentally. Studies have been conducted to see the effect of the geometrical parameters such as jet diameter (D), jet to target spacing (Z) and ratio of jet spacing to jet diameter (Z/D) on the heat transfer characteristics. The values of Reynolds numbers considered are in the range 7000 to 42000. The results obtained from the numerical studies are validated by conducting experiments. From the studies it is found that the optimum value of Z/D ratio is 5. For a given Reynolds number, the Nusselt number increases by about 28% if the diameter of the nozzle is increased from 1mm to 2mm. Correlations are proposed for Nusselt number in terms of Reynolds number and these are valid for air as the cooling medium.

***Keywords*—**CFD, heat transfer coefficient, Nusselt number, ratio of jet diameter to jet spacing (Z/D), Reynolds number, turbulence model.

I. Introduction

THE demand for high powered electronics has increased and we require faster, smaller, and reliable electronic components. These lead to high heat flux that must be removed to avoid the failure. The traditional cooling techniques such as heat sink, heat sink with fan, heat pipes reached their limit. Jet impingement cooling is one of the very efficient solutions of cooling hot objects in industrial processes as it produces a very high heat transfer rate through forced convection. Jet Impinging is widely used for cooling, heating and drying in several industrial applications due to their high heat removal rates with low pressure drop.

Industrial applications are drying of food products, textiles, films and paper processing of some metals and glass, cooling of turbine blades etc. Over the past 30 years, experimental and numerical investigations of flow and heat transfer characteristics under jet impingement remain a very dynamic research area. The effects of jet diameter, jet-to-surface spacing, Reynolds number etc. on flow and heat transfer have been studied by both experimentally and numerically.

Most industrial application jets are concerned with turbulence flow in the downstream of a nozzle. Modeling of turbulence flow is the greatest challenge for rapidly and accurately predicting impinging heat transfer under a single round jet over the past years ,no single mode has been universally accepted therefore research are going on to develop various turbulence model for the prediction of impingement flow and heat transfer.

Due to many industrial applications of impinging jet extensive prior research has been conducted to understand their flow and heat transfer characteristics. Liu et al. [1] investigated experimentally convective heat transfer by using impingement of circular liquid jet in laminar and turbulent flow conditions for different Prantl number. Baughn et al. [2] studied the heat transfer and fluid flow for impinging jets by investigating experimentally the entrainment effects of jets for circular jet. The result from this research have been summarized by Jambunathan and Viskanta et al. [3,4]. Lytle and Webb [5] carried out an experimental study to investigate the effect of Prantl number and jet to plate spacing for stagnation point Nusselt number. Garinella in a review paper presented a detailed discussion of heat transfer and flow fields in confined jet impingement [6]. The flow and heat transfer characteristics of laminar impinging rectangular slot jet were investigated by Sezai and Mohammed[7].The effect of jet velocity profiles on the flow and thermal fields of laminar confined and swirling jet were investigated by Shuja et al.[8].Juan et.al [9] conducted experimental research on heat transfer of confined air jet with tiny size. H.G. Lee [10] investigated numerically the unsteady two-dimensional fluid flow and heat transfer in the confined impinging slot jet for different Reynolds numbers of 50-500 and different height ratios of 2-5.It is found that the unsteadiness gives a big impact on the flow and the temperature fields and as a result the pressure coefficient, skin friction and Nusselt number in the unsteady region show different characteristics from coefficient those in the steady region. Yahya Erkan Akansu [11] studied experimentally the effects of inclination of an impinging of two dimensional slot jets on the heat transfer from a flat plate. It showed that as the inclination angle increases, the location of the maximum heat transfer shifts towards the uphill side of the plate and the value of the maximum Nusselt number gradually increases at lower jet to plate spacing.

Arun Jacob is with the MES College of Engineering, Kollam, Kerala. India (phone:-8893780880; fax: 0474-2590700; e-mail: arunjacobcool@gmail.com).

Leena R is with the T.K.M College of Engineering, Kollam, India. She is now with the Department of Mechanical Engineering, University of Kerala, India (e-mail: leenakalidas@gmail.com).

Krishnakumar T.S. is with the Mechanical Engineering Department, T.K.M College of Engineering, Kollam, University of Kerala, India (e-mail: krishnakumarts@gmail.com).

M. Joseprakash is with the Mechanical Engineering Department, T.K.M College of Engineering, Kollam, University of Kerala, India (e-mail: jpmmech@yahoo.co.in).

II. Experimental Setup

A. Experimental System

Experiments have been conducted to see the effect of the various parameters on the heat transfer coefficient. The key parameters determining the heat transfer characteristics of a single jet impingement jet are the diameter of the nozzle, spacing between the jet and the target, Reynolds number and the ratio of jet spacing to diameter of the jet (Z/D).

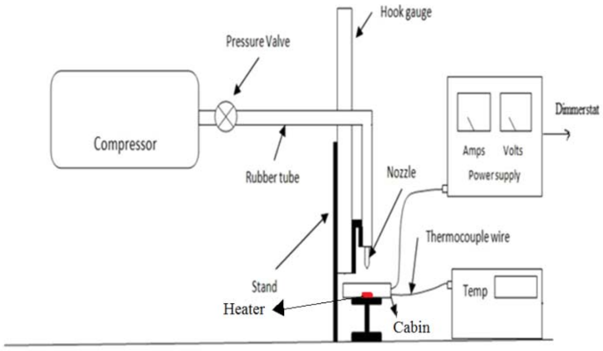

Fig. 1 Schematic of Experimental Setup

The schematic of the experimental setup is shown in Fig. 1. The cooling medium is air and the same is supplied from a reciprocating compressor. The thermal environment of the microprocessor is simulated using a copper plate of size 5cm X 5cm X 1cm with a heater placed underneath. The cabinet is made of aluminum and its dimensions are 45cm X 40cm X 15cm and one of the smaller sides is open. The power input to the heater is varied using a dimmerstat and the voltage and current are measured using voltmeter and ammeter respectively. T-type thermocouples (36 SWG) were used to measure the temperature at various locations.). The temperature data were acquired using a temperature scanner. The jet to target spacing is adjusted using a hook gauge. The flow rate of air is measured using an orifice meter.

B. Experimental Procedure

Prior to experimental measurements the thermocouples were calibrated using a constant temperature bath (JulaboF25). Experiments have been conducted by varying the nozzle diameter, nozzle to plate spacing, jet spacing to diameter ratio and the Reynolds number.

III. Numerical Analysis

A. Geometry and Boundary Conditions

The computational domain considered for the present study is shown in Fig. 2. It consists of an enclosure which has the same dimensions as that of the cabinet used in the experimental studies. The copper plate which has the same dimensions as that of a typical computer processor (5cm X 5cm X 1cm)is placed at the centre of the bottom surface of the enclosure. A tube having diameter same as that of jet is protruding in to the enclosure and positioned in such a way that the jet impinges on the centre of the top surface of the copper plate. The thickness of the enclosure is not considered and adiabatic boundary condition is applied to all walls except the top one.

Heat flux boundary condition is applied to the bottom surface of the copper plate. Mass flow boundary condition is applied to the inlet of the tube and temperature of air entering is 300. The flow is assumed to be steady, incompressible and three dimensional over the entire computational domain. K.

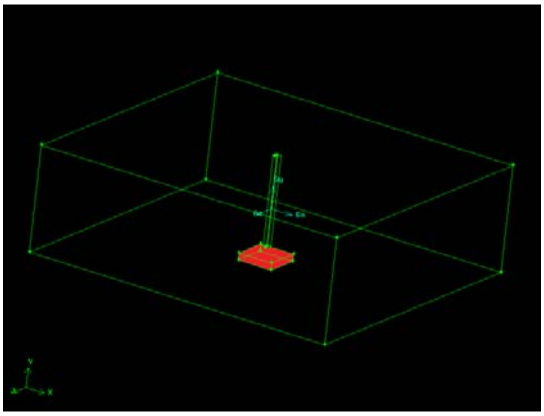

Fig. 2 The computational domain/model

B. Numerical Procedure

The computational domain is modeled using GAMBIT 2.3.16 software and the same is meshed with Tet/hybrid elements. The number of elements is about 19 lakhs. No slip condition is applied to the wall surfaces. Fig. 3 shows the computational domain with mesh.

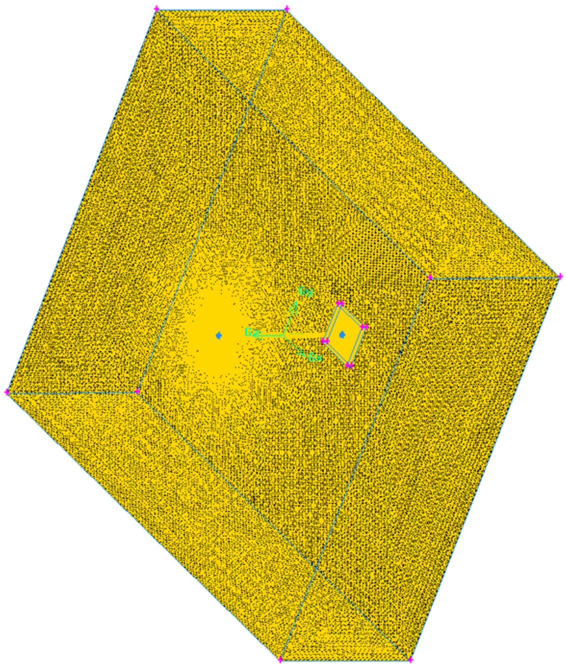

Fig. 3 The computational domain with mesh

C. Analysis

The software used for analysis is fluent 6.3.26. The turbulence model used is SST (k- ε) model which is found to be the best among available turbulence models for this type of flow configurations. The interfaces are coupled together by using the grid interface tool. The solver used is pressure base segregates solver. The materials selected for the enclosure, chip and tube are aluminum, copper and brass respectively and the coolant medium is air.

Analysis have been conducted to see the effect of Reynolds number (Re), jet spacing to jet diameter (Z/D) ratio on cooling effectiveness.

IV. Results and Discussion

Numerical and experimental investigations have been carried out on jet impingement cooling for its effectiveness. The results are validated by conducting experiments.

A. Effect of Z/D Ratio on Heat Transfer

Fig. 4 shows the variation of temperature at the top surface of the copper plate. The heat flux at the bottom of the copper plate is 12000 W/m^2 The results obtained from numerical and experimental studies are shown for comparison.Here the diameter of the jet is 2 mm, the velocity of flow of air is as 300 m/s and the corresponding Reynolds number is 36000.There is only marginal difference in the values of temperatures obtained numerical and experimental studies. It is observed that the temperature is minimum when the Z/D ratio is 5.

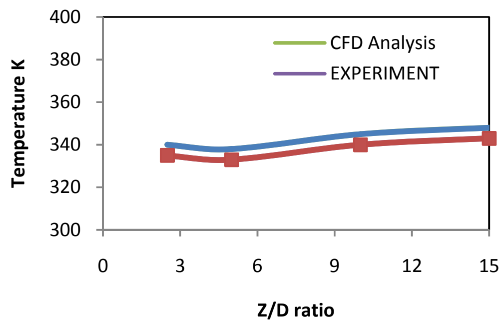

Fig. 4 Variation of temperature at the top surface of the heat sink. Diameter of the jet is 2 mm

In case of higher Z/D ratio there is a higher momentum exchange between impinging fluid and surrounding fluid due to this the jet diameter becomes broader and spreads over more surface area. In case of low Z/D ratio the same amount of fluid spreads over lesser surface area causing a higher heat transfer rate. If the Z/D ratio is very low it will prevent the free flow of jet.

Fig. 5 shows variation of Nusselt number at the top surface of the heat sink with Z/D ratio. Here also the heat flux is 12000 W/m^2 and the Reynolds number is 36000. It is clear from the graph that the Nusselt number increases with increase in Z/D ratio and then decreases. The optimum value of Z/D ratio is 5.

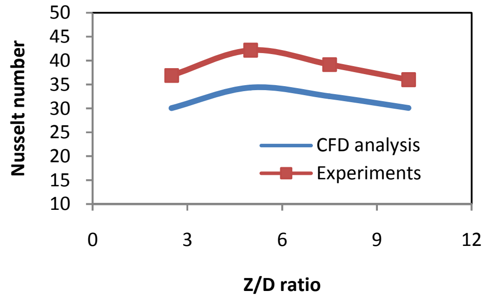

Fig. 5 Variation of Nusselt number on top surface of the heat sink with Z/D ratio. Diameter of the jet is 2mm.

For further analysis the value of Z/D ratio is taken as 5 i.e. the optimum value.

B. Effect of Reynolds Number on Temperature

Fig. 6 shows variation of maximum temperature at the top surface of heat sink. The heat flux at the bottom surface of the heat sink is 12000 W/m^2. It is observed that the surface temperature decreases with increase in Reynolds number and the chip can be maintained within the safe temperature limit if the Reynolds number is 20000.

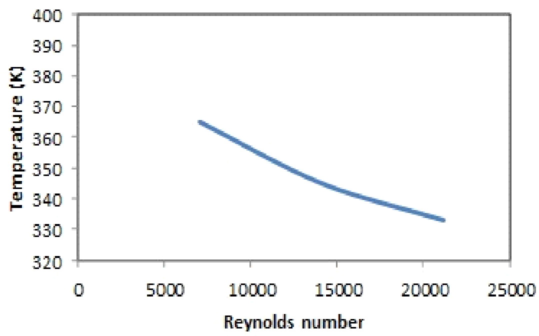

Fig. 6 Variation of the temperature on the top surface of heat sink with Reynolds number. Diameter of the jet = 1mm

Fig. 7 shows the variation of maximum temperature at the top surface of heat sink. Here the diameter of the jet is 2mm. and the heat flux 12000W/m^2. It is observed that if the Reynolds number is more than 19000 the chip can be maintained at the safe temperature limit. For the same Reynolds number, there is only marginal difference in the value of temperatures if the jet diameter is increased from 1mm to 2mm, but the volume flow rate of air increases by 100 percentages.

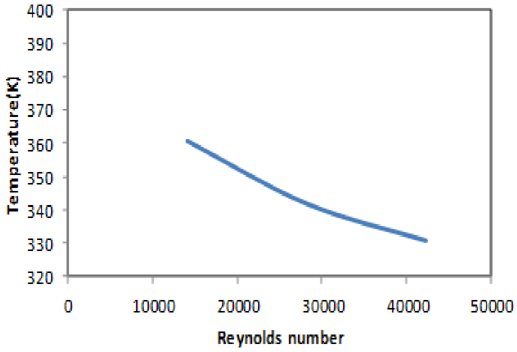

Fig. 7 Variation of the temperature at the top surface of heat sink with Reynolds number. Diameter of the jet = 2mm

C. Effect of Reynolds Number on Heat Transfer Coefficient

Fig. 8 shows the variations of the average heat transfer coefficient at the top surface of heat sink with various Reynolds number. The heat flux is 12000 W/m^2 and the diameter of the jet is 1 mm. The average heat transfer coefficient increases by 114 percentages if the Reynolds number is increased from 7000 to 21000.

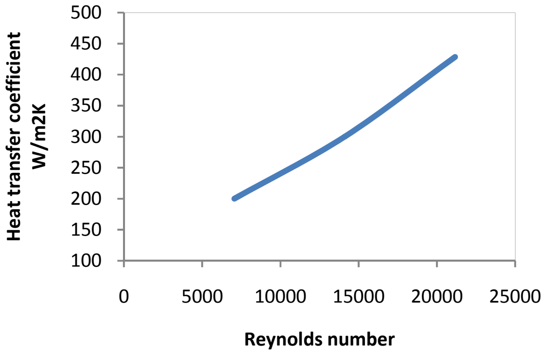

Fig. 8 Variations of the heat transfer coefficient at the top surface of heat sink with Reynolds number. Diameter of the jet = 1mm

Fig. 9 shows the variation of average value of heat transfer coefficient at the top surface of heat sink with Reynolds number. Heat flux is 12000 W/m^{2}and the diameter of the jet is 2 mm. The average heat transfer coefficient corresponding to a Reynolds number of 14000 is 215 W/m^{2}K and increases by 114 percentage if the Reynolds number is increased from 14000 to 42000.

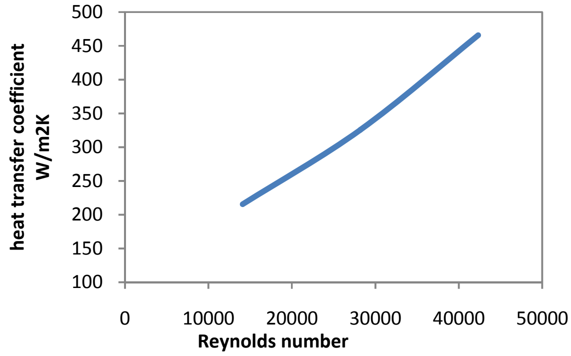

Fig. 9 Variation of the heat transfer coefficient at the top surface of heat sink with Reynolds number. Diameter of the jet = 2mm

D. Effect of Nusselt Number with Reynolds Number

Fig. 10 shows the variation of the average Nusselt number at the top surface of heat sink with Reynolds number. The diameter of the jet is 1 mm. The Nusselt number corresponding to a Reynolds number of 7000 is 7.38 and it increases by 50 percent if the Reynolds number is doubled.

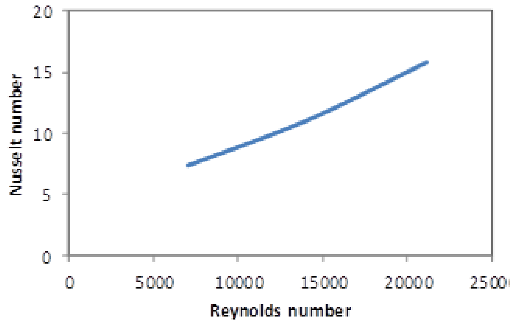

Fig. 10 Variation of the Nusselt number at the top surface of heat sink with Reynolds number. Diameter of the jet = 1mm

Fig. 11 shows the variation of the average Nusselt Number at the top surface of heat sink with different Reynolds number for a jet diameter 2 mm.

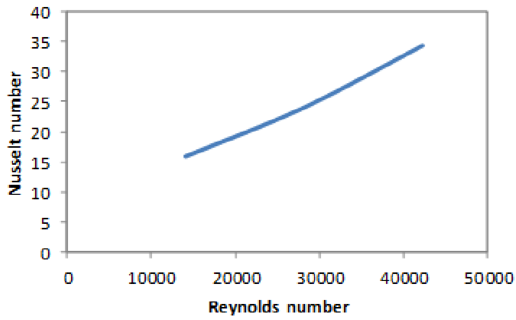

Fig. 11 Variations of the Nusselt number at the top surface of heat sink with Reynolds number. Diameter of the jet =2mm

The Nusselt number increases from 7.38 to 34.if Reynolds number is increased from 7000 to 42000 at Z/D = 5. The average heat transfer coefficient increases with increase in Reynolds number at a given Z/D ratio. At higher Reynolds number, turbulence level increases and along with this spread of jet also increases. The net amount of fluid that comes out of the jet is also higher for higher Reynolds number which causes better heat transfer performance.

V. Correlation

Correlations are proposed for Nusselt number in terms of Reynolds number and this is valid for air as the cooling medium. This correlation is based on the numerical experiments. The Nusselt number is calculated based on the average value of heat transfer coefficient and the heat transfer coefficient (h) is estimated using the correlation

$$h = q/(Tw - Ti))$$

here q is the heat flux, T_w is the average wall temperature and T_j is the temperature of jet.

The correlation proposed is Nu $=0.021Re^{0.69}$ and this is applicable for a jet diameter of 2mm. The correlation can be used in the range of Reynolds number 7000 to 42000. For a jet diameter of 1mm the correlation is $0.018Re^{0.68}$.

VI. Conclusion

Numerical and experimental investigations were carried out to study the effect of geometrical and flow parameters on the heat transfer characteristics. The conclusions derived out of the present study are (i) the optimum value of jet spacing to jet diameter is 5(ii) Numerical predictions based on k- ε turbulence model show good agreement with experimental results. (iii) for the same value of Reynolds number the heat transfer coefficient increases by about 28 percent if the diameter of the jet is increased from 1mm to 2mm but the volume flow rate increases by 100%. (iv) Correlations are proposed for Nusselt number in terms of the Reynolds number.

References

[1] X. Liu, V.J.H. Lienvard, and J.S. Lombara, “Convective heat transfer by impingement of circular liquid jets”, J. Heat Transfer vol.113, 1991, pp. 571 – 582.
[2] J.W. Baughn, A.E. Hechanova and X. Yan, “An experimental study of entrainment effects on the heat transfer from a flat surface to a heated circular impinging jet”, J. Heat Transfer vol. 113,1991, pp. 1023 - 1025.
[3] K Jambunathan, E. Lai, M.A. Moss and B.L. Button, “A review of heat transfer data for single circular jet impingement”, Int.J.Heat Fluid Flow vol.13, 1992, pp. 106 - 115.
[4] R. Viskanta, “Heat transfer to impinging isothermal gas and flame jets”. Exp. Therm. Fluid Sci.vol.6, 1993, pp. 111-134.
[5] D. Lytle and B.W. Webb, “Air jet impingement heat transfer at low nozzle-plate spacing’s”, Int. J. Heat Mass Transfer, vol.37, 1994, pp. 1687 - 1697.
[6] S.V. Garimella, “Heat transfer & flow fields in confined jet impingement”, Ann. Rev. Heat Transfer, vol.11, 1999, pp.413 - 494 ch.7.
[7] I. Sezai and A.A. Mohamed, “Three dimensional simulations of laminar rectangular impinging jets, flow structure and heat transfer”, ASME J. Heat Transfer, vol.121, 1999, pp.50 - 56.
[8] S.Z. Shuja, B.S. Yilbas, and M. Rashid, “Confined swirling jet impingement onto anadiabatic wall”, Int. J. Heat Mass Transfer, vol. 46, 2003, pp. 2947 - 2955.
[9] Juan.T, Jie-min.Z, Li.J and Ying. Y “ Experimental research on heat transfer of confined air jet impingement with tiny size round nozzle in high density electronics packaging model", ieeeexplore.com,2005.
[10] H.G. Lee, H.S. Yoon and M.Y. H, “A numerical investigation on the fluid flow and heat transfer in the confined impinging slot jet in the low Reynolds number region for different channel heights”, International Journal of Heat and Mass Transfer, vol. 51, 2008, pp. 4055 - 4068.
[11] Yahya Erkan Akansu, Mustafa Sarioglu , Kemal Kuvvet B and Tahir Yavuz, “Flow field and heat transfer characteristics in an oblique slot jet impinging on a flat plate”, International Communications in Heat and Mass Transfer, vol. 35, 2008, pp. 873 - 880.