International Journal of Mechanical Engineering and Technology (IJMET)
Volume 9, Issue 6, June 2018, pp. 1129–1140, Article ID: IJMET_09_06_125
Available online at http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=9&IType=6
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

EXPERIMENTAL ANALYSIS OF JET IMPINGEMENT ON ALUMINIUM HEAT SINK

Sk. Esub Basha
PG Scholar, Department of Mechanical Engineering,
Lakireddy Balireddy College of Engineering, Mylavaram,
Krishna Dist, Andhra Pradesh, India

K. Lakshmi Prasad
Assistant professor, Department of Mechanical Engineering,
Lakireddy Balireddy College of Engineering, Mylavaram,
Krishna Dist, Andhra Pradesh, India

G. Kartheek
Assistant professor, Department of Mechanical Engineering,
Lakireddy Balireddy College of Engineering, Mylavaram,
Krishna Dist, Andhra Pradesh, India

K.V.Viswanath
Assistant Professor, Department of Mechanical Engineering,
Lakireddy Balireddy College of Engineering, Mylavaram, Krishna Dist,
Andhra Pradesh, India

ABSTRACT

Heat sinks in electronic systems are devices that transfer heat from the hotter body by dissipating heat to fluid medium, generally air. These are basically heat exchangers that are used to transfer heat from the components to the surroundings so as to keep away from the problem of overheating. So number possible ways are implemented in all fields.

This study presents, heat transfer rate at different heat input conditions for a aluminium flat fin as a heat sink. The maximum heat dissipation rate for the different Z/d ratio's and velocity was optimized by using taguchi's method by taking orthogonal array design as L_{16} *at different heat input conditions. The analysis part was done by using ansys fluent CFD programme. The studies of experimental and numerical were carried out for a aluminium heat sink with same nozzle diameter, two different velocity rates at four different input conditions. The variations of heat transfer rate to z/d ratio's and for Re-Nu analysed and compared. Finally the maximum heat transfer rate was observed at optimum parameter's. Hence plots are drawn using experimental as well* *as numerical results. It was observed that both numerical and experimental results were taken as a good agreement.*

Key words: Taguchi Method, Heat Transfer, Impingement Jet, Experimental Analysis, Computational Fluid Dynamics.

Cite this Article: Sk. Esub basha, K. Lakshmi Prasad, G. Kartheek and K.V.Viswanath, Experimental Analysis of Jet Impingement on Aluminium Heat Sink, International Journal of Mechanical Engineering and Technology, 9(6), 2018, pp. 1129–1140.
http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=9&IType=6

Nomenclature

A area (m^2)

C_p pressure coefficient

D Nozzle diameter (m)

Q heat dissipation rate (w)

h convection coefficient(w/m^2k)

I Current (A)

K thermal conductivity (w/mk)

L length of the base plate (m)

N total number of fins

Nu Nusselt number

P Pressure (pa)

Pr prandtl number

Re Reynolds number

T Temperature (k)

U velocity (m/sec)

V Voltage (V)

W width of base plate (m)

υ Kinematic viscosity (m^2/sec)

z distance between nozzle and heat sink

S/N signal to noise ratio

MSD mean standard deviation

n number of parameters

Subscripts

a air

ave average

cond conduction

conv convection

in inlet

out outlet

rad radiation

s surface

tot total

1. INTRODUCTION

The growth of energy consumption all over the world is leads to use efficient manner by adopting new techniques. So in order to looking forward some of energy conditions like heat transfer's used in daily life and industries till now. The studies to improve the heat transfer, maximum heat dissipation to surroundings have not enough increasing mass production and rising costs have resulted in addition problem. As a result of the studies, heat transfer by impinging jet is allocated to be an efficient also economically good method, dissipation is very fast and can be applied to even small surface areas.

More importantly, the heat dissipation from fins are considerable because of different parameters like area, thermal conductivity of the material and the difference of temperature. The perfect fins for the different heated parts can also be seen by past research field. Impinging on a aluminium heated concept having anonymous applications because of low cost high result manner. For the higher heat transfer rate is depends on flow instabilities and distances from finned area to nozzle. Generally finned surface region are used as heat sinks.

Over the few decades many experiments is done on the jet impingement heat transfer. Ridvan yakut [1] (2016) Experimental and numerically investigated on impingement air jet for a heat sink. He was observed that nussult number increases with increase of reynlds number and nussult number also increases with decreasing of fin height. Angiolletti et. al.[2] (2005) numerically analyzed the heat transfer by laminar and submerged transition on the target plane with free gas jet compression. It is observed that if Reynolds number is equal to 1000 the k-ω SST turbulence model is better, and if Reynolds number is equal to 4000 then the k-ε RNG and RSM turbulence models are better. Bayraktar and Smith [3] (2005) discussed the hot turbulent jet flow in cross-flow using computational fluid dynamics (CFD). The standard k-ε turbulence model is used, and the flow field is considered as three-dimensional. Deoganda et. al. [4] (2014) studied the wall static pressure of sealed air by impinging it on a flat plate. They measured the average velocities and the wall static pressures for nozzle-plate openings in the range of 0.25-4 for the Reynolds numbers in the range of 18000-40000. Yucel and Ozmen [5] (2013) numerically studied the heat transfer and flow characteristics in the two-dimensional flow field created by the bounded impinging air jet stream for the nozzle openings in the range of 1-10 in the distance between the jets from 2 to 6 and for the case where the Reynolds number is 30000, for the realizable k-ε and standard k-ω turbulence models. With experimental data, they observed that the realizable k-ε model is more compatible. Katti and Prabhu [6] (1983) these are done by an experiments for effects of jet exit to plate distance and Reynolds numbers, based on the local heat transfer co- efficient(h).They reports the when increase of jet to plate distance as a flat plate. When increasing of temperature in all directions and decrease of Nusselt numbers. Hrylak et al [7] (1983) reports the experimental results of heat transfer of circle turbulent jets impinges on flat plate .The target plate distance of different nozzles they observed that increase the nozzle to plate spacing they remains independent on potential core. Huber and Viskanth [8] (1994) they both are reported the effects of jet to jet distance on heat transfer performance form the impingement jets and for large plate spacing decreasing of the heat transfer of the target plate.

Most of the studies was done on impinging jet on a heat sink in order to maximize the number of possibilities that carry out in a practical mode as well as numerical models. In this study heating of aluminium flat fin taken as a heat sink given different heat input conditions at four different Z/d distances at two different velocity conditions. Furthurly taguchi method was taken for optimization. Hence CFD model performed after optimizing technique by considering results of taguchi method which was done on minitab software, and simulation was done on Ansys-Fluent 18.0 version.

2. EXPERIMENTAL SETUP

Experiments were performed for a fixed nozzle diameter(d=20mm), at a different z/d distances(z/d=4,6,8,10), and at two different velocity values(U=8,9), for the four different heat input voltage(V=40,60,80,100).As a result of those experiments the main aim is to determine most effective heat transfer parameters by using taguchi's optimization method where the orthogonal array $L_{16}(4^2\ 2^9)$ for the cooling applications.

For the Experiments, the entire setup whose z/d ratio can be adjustable were used optimized fin heat sink. In this study air was used as fluid in the system. The reason of using aluminium material is having higher thermal conductivity and which is used all cooling applications. The values which are taken from optimizing technique and plotted for maximum heat transfer rate to Reynolds number. Graphs are drawn for heat sinks with blower and without blower.

1. Experimental setup table. 4. Aluminium fin (heat sink).7.Thermo couples.

2. Air blower. 5. Voltage regulator.8.Flow direction of air.

3. Electrical heater. 6. Digital thermometer. 9. Supporter. 10. Connecting wire.

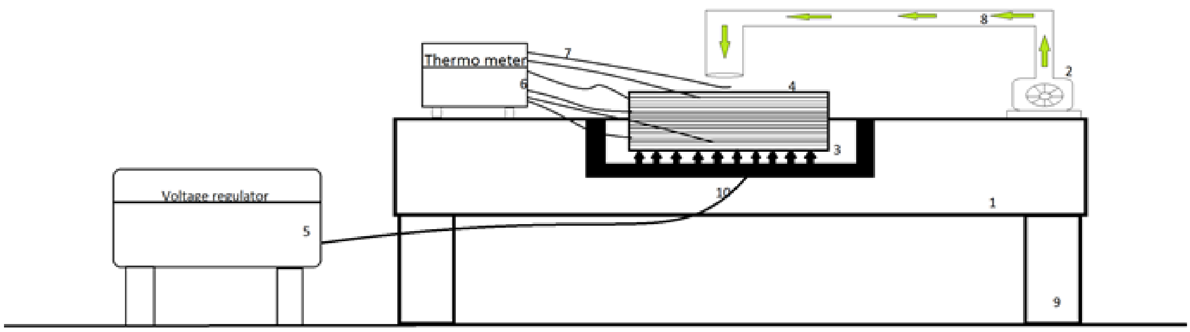


Figure 1 Schematic diagram of the experiment

The electrical heating surface 300x300 mm^2 which is flat and uniform surface was placed on slotted portion of setup table, heat input can be alter by using dimmer start. An air blower NWB type of capacity $1.5m^3$/min was used and the flow of discharge can be changed by using valve. It is very important to be considerable minimizing gap between heater and heat sink which was perfectly filled by applying heat sink compound like thermal paste, in this study too. Thermal paste compounds were used of 12gm each. Eight pieces of thermocouples used for this experiment. Three pieces were arranged on the plate to determine the average surface temperature, two are used to determine to fin average wall temperature, two are used to determine fin average base temperature. One thermocouple is used for both ambient and air temperature of nozzle. The total eight pieces of thermocouples attached to a digital measuring thermocouple for good accuracy. Whereas the air flow measuring by using anemometer having fan type flow calculator in m/sec.

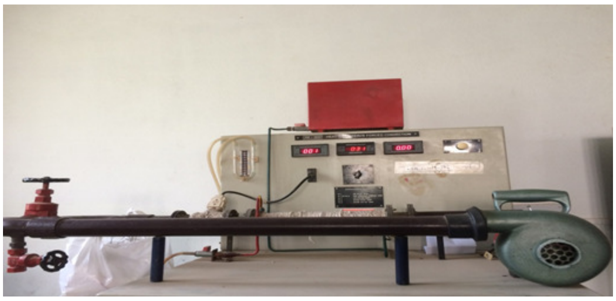
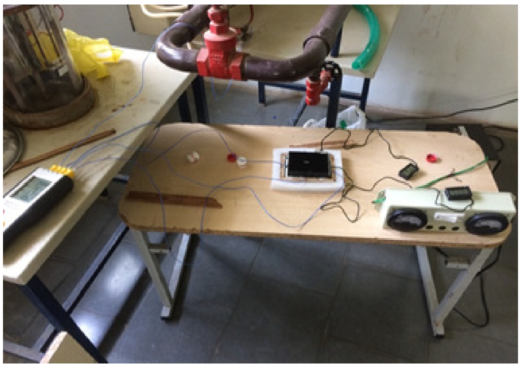


Figure 2 Air Blower.



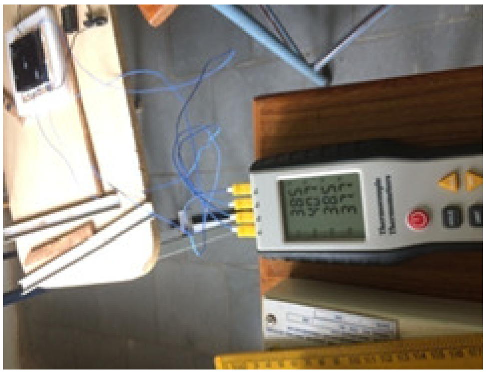


Figure 3 Experimental setup **Figure 4** Digital thermometer

The total number of experimental variables inspected in this study are given in below table.1

Table 1 Parameters used in experiment.

Parameters	Optimum elements
1.Area of heater(mm^2)	8000
2.Fixed diameter of nozzle(mm)	20
3.Height to diameter ratio (z/d)	4,6,8,10
4.Fin height from base(mm)	9
5.Span wise distance between fins(mm)	5
6.Fin width(mm)	82

2.1. Calculations

Generally heat transfer in study state through the air expressed as

$Q_{tot} = Q_{cond}+Q_{conv}+Q_{rad}$ ..... (I)

Hear Q_{conv} =heat transfer in convection mode.

Q_{cond}=heat transfer in conductive mode.

Q_{rad}=heat transfer in radiation mode.

According to convection the heat transfer written by

$Q_{conv}=h_{avg}\ A_s\ (\Delta T)$

$Q_{conv}=h_{avg}\ A_s\ (T_{avg}-T_{jet})$ .... (II)

The heat transfer by radiation made was less than 5% due to the heater and heat sink were made of aluminium, so transfer of heat by radiation for similar metals are neglected also here in this setup majorly concentrated about forced convection heat transfer as the blower having high capacity air dischargeable so conduction heat transfer can be neglected.

The equation I follow as

$Q_{tot}=Q_{conv}$

The average base temperature can be calculated as the experiment was conducted for different heat inputs and at the different velocities, calculating film temperature

Now the properties of air can be taken at the value of film temperature (T_f)ºC.like

Table.2 Properties of air at film temperature.

1.Kinematic viscosity(ʋ) m^2/sec
2.Density (ρ) kg/m^3
3.prandtl number (pr)
4.Specific heat(c_p) J/kgºk
5.Thernal conductivity (k) w/mk

Now Reynolds number can be calculated as follows and it is defined as the ratio of inertia force to the viscous force.

$$\boxed{Re_L=\frac{UL}{\upsilon}} \quad \text{(III)}$$

Where inertia force U×L i.e U-velocity (m/sec).

L-length of fin (m).

Viscous force means kinematic viscosity (ʋ) m^2/sec.

In this setup Reynolds number value obtained as $Re<5x10^5$. i.e laminar flow.

Now nusselt number can be calculated as follow

$Nu_L=0.332\ (Re_L)^{0.5}(pr)^{0.333}$ (IV)

The above expression is taken for the Pr lies in between 0.6 to 50.

$$\boxed{Nu_L=\frac{hL}{K}} \quad \text{(V)}$$

Where h=heat transfer coefficient (w/m^2k).

L=length of the fin (m).

K=thermal conductivity of air (w/mk).

By equating IV and V we get the value of heat transfer coefficient (h).

$h=(0.332\ (Re_L)0.5(Pr)^{0.333}\ k)/L\ w/m^2k$ (VI)

Once we get convection heat transfer coefficient simply substitute in equation II

i.e $\boxed{Q=hA\Delta T}$ ((or)

heat flux $\boxed{\frac{Q}{A}=h\Delta T}$ w/m^2

$$\boxed{\frac{Q}{A}=h(T_{avg}-T_{jet})} \ w/m^2 \quad \text{(VII)}$$

The total heat transfer rate (Q) and heat flux (Q/A) can be calculated for different values which were obtained experimentally for different z/d ratios at different input conditions. The maximum heat transfer rate obtained and results were plotted.

3. TAGUCHI'S METHOD

One of the best technique for optimizing different number of parameters. Perfect output for the optimal values when we have considerable parameters usage of taguchi's method in all fields by consider loss function. Orthogonal arrays (OA) are simplified method putting to gather an experiment.

In this study consult about to maximum heat transfer for z/d ratio and different heat input conditions. Hence the way of calculating in taguchis technique is larger and the better it was the thirds S/N ratio.

S/N_L ratio for larger the better

$S/N_L = -10\log_{10}(MSD)$ (or)

$S/N_L = -10\log_{10}(\Sigma(1/y^2)/n)$ (VII)

The entire taguchi design was performed by minitab software.

Details of taguchi's design.

Taguchi orthogonal array design.

Number of factors =3

Runs =16

Columns of $L_{16}(4^2\ 2^9)$ array.

Input values given as per the experimental calculations as well as for the order specified by the software. The results of heat transfer (Q) versus voltage (V), z/d ratio, velocity (U) are obtained as shown in figure.5. and figure.6.

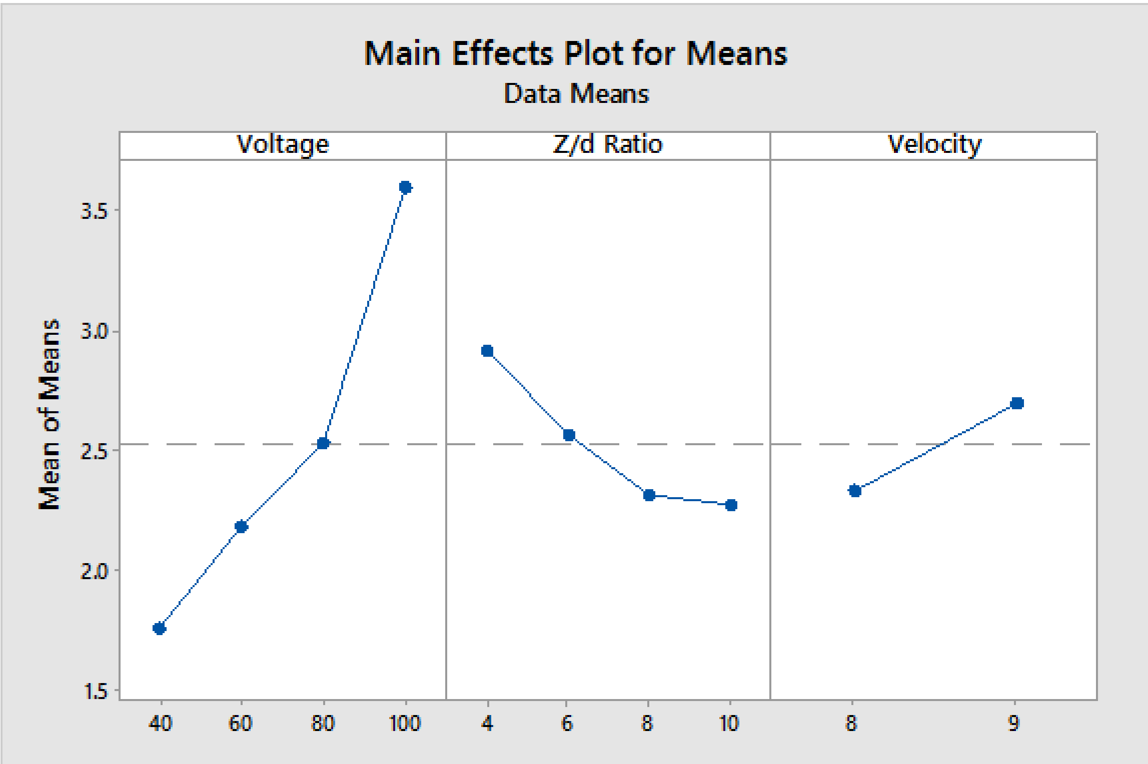


Fig-5: Results of Taguchi's method 1

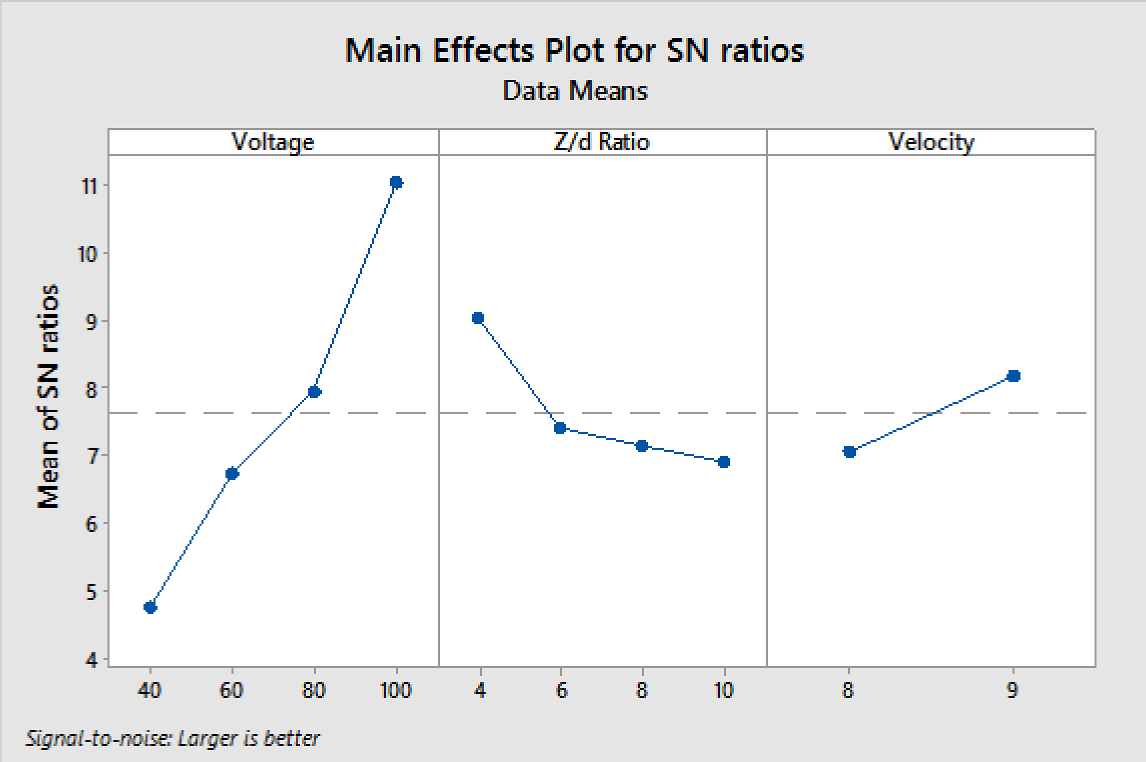


Fig-6: Results of Taguchi's method 2.

4. COMPUTAIONAL MODEL

The geometrical part of heat sink was created as per the dimensions of aluminium fin for numerical analysis, with fixed nozzle diameter (20mm), four different z/d distances (z/d=4,6,8,10).The numerical solutions were considered for constant surface heat flux boundary conditions.

In numerical analysis of heat and mass transfer with impinging air commonly used turbulent models were examined k-ε reliable turbulence model was chosen. The aluminium heat sink has a good geometrical symmetry with perfect dimensions, so the models were generated a tighter mesh and completed with fine mesh conservation of mass and momentum equations were solved with ansys-fluent 18.0 software package by giving exact boundary conditions like heat flux, wall temperature, fin base temperature and after iterations also very important in simulations. The optimum number of iterations to be around 100 and solution is converged. The experimental and numerical results were observed in a good agreement.

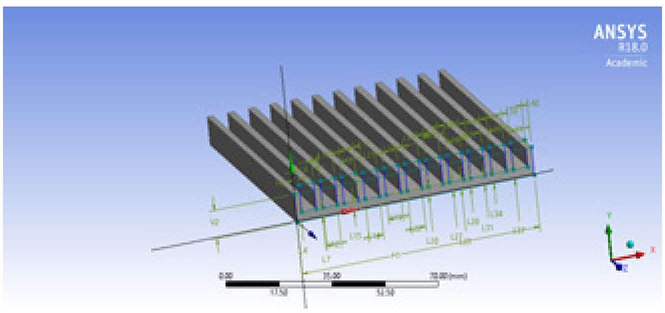


Fig-7: Fin geometry.

4.1. Boundary Conditions

The following boundary conditions are taken for this analysis those are follows,

Velocity inlet: when inlet velocity applied as two different cases (U =8,9 m/sec) and constant temperature (T) = 300K .But single nozzle the velocity is varying based on the Reynolds numbers (Re at 5000, 10000, 15000, 20000, 25000).from finding the velocity based on that Reynolds number (Re = ρvd/μ).

At wall: There are no slip conditions at wall side and adiabatic conditions.

Heat flux: In that model uniform and constant heat flux (q) =2000 W/m^2.

Insulated: In that insulation is given by four sides of target plate for there are no heat losses for that.

Interface: In these flow domain to solid interface at heat transfer boundary conditions is applied.

4.2. NUMERICAL VALUES OF GOVERNING EQUATIONS:

The following governing equations are taken from these models that are momentum, energy, continuity equations at steady state three dimensional flow equations in the ansys fluent software 18.0. For turbulence modals the two equations are taken that are shear stress transformation (SST) and k-ῲ modals. In this second order up wind scheme is used for momentum and energy equations for turbulence. In that residuals values is taken from this modals 1 x10^{-04} for continuity equation and 1 x 10^{-08} for energy equation.

5. RESULTS AND DISCUSSION

The results and pictures of ansys work like construction of geometry, fine meshing, fin wall and fin base temperature variation are shown in below figures:(8,9,10,11,12,13,14,15).and also observed that maximum heat dissipation occurring at Z/d=6 among all heat input conditions like voltage (60V) at velocity (U=8m/sec).

5.1. Simulation Figures

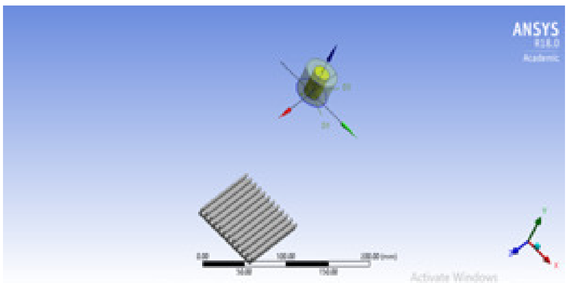


Fig-8: Nozzle geometry.

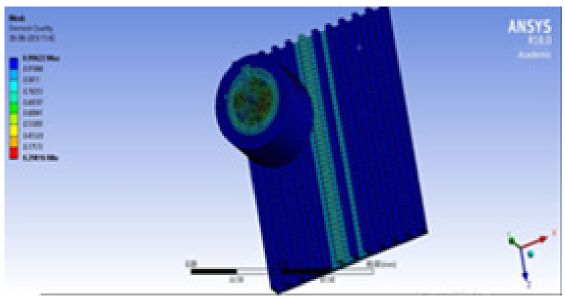


Fig-9: Creating fine mesh.

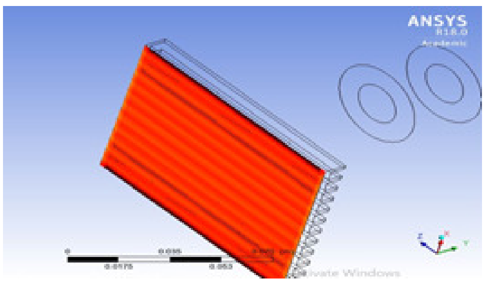


Fig-10: Variation of temperature on fin base.

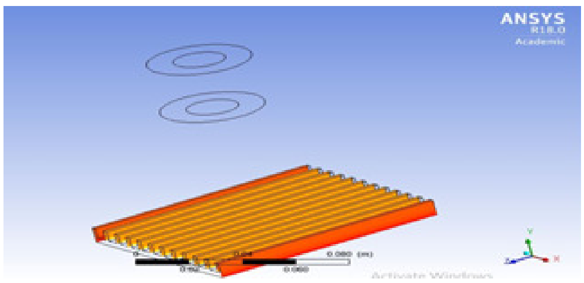


Fig-11: Variation of temperature on fin walls.

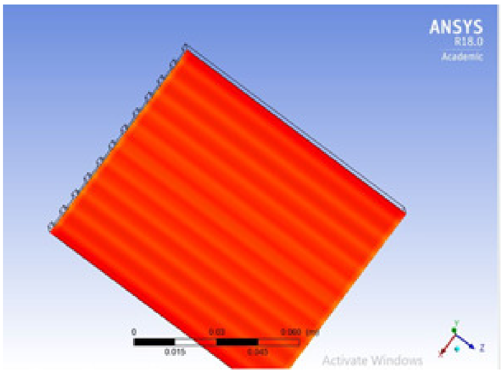


Fig-12: On Heat flux base fin variation.

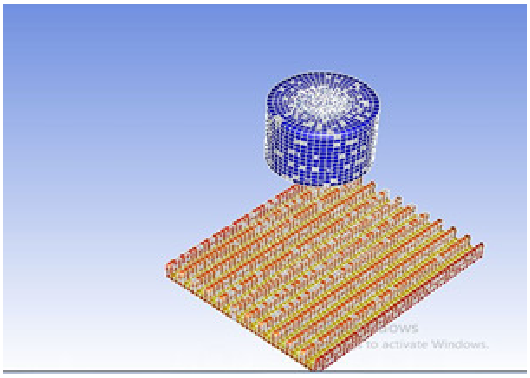


Fig-13: Static temperature on meshing.

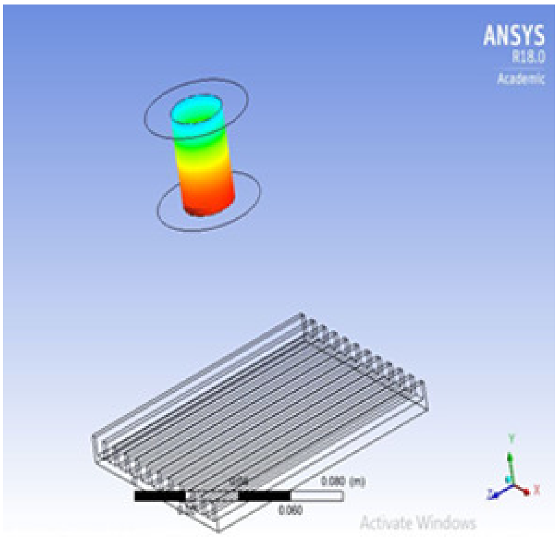


Fig-14: Air distribution in nozzle.

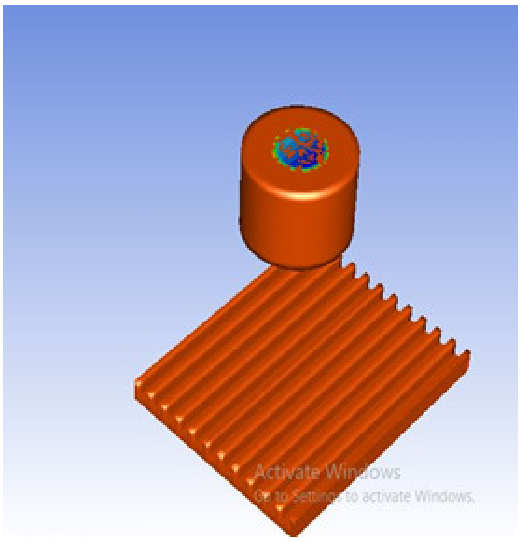
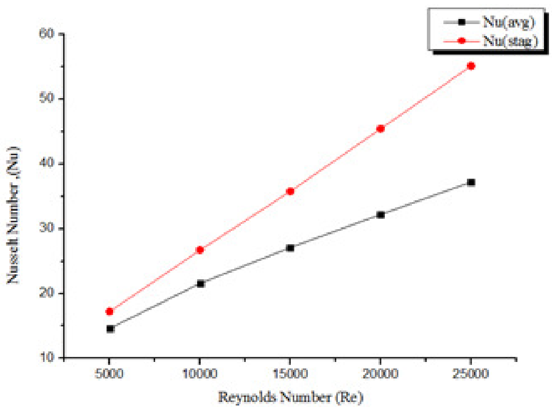


Fig-15: Tangential velocity.

5.2. Graphs:



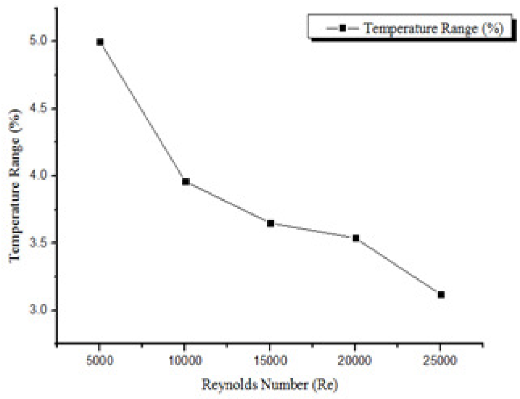
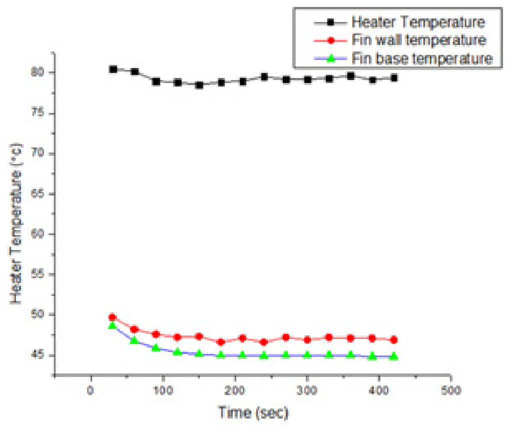


Fig-16: Effect of Reynolds number on Nusselt number. **Fig-17:** Effect of Reynolds number on temperature.



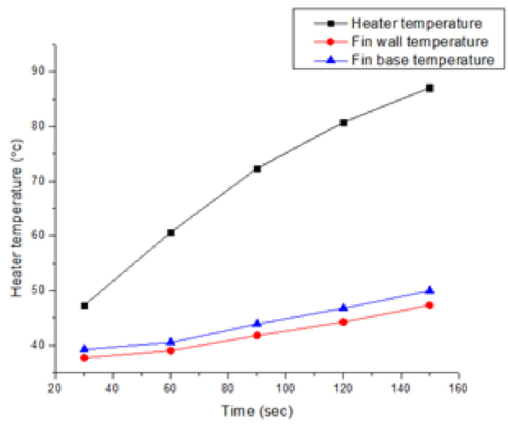
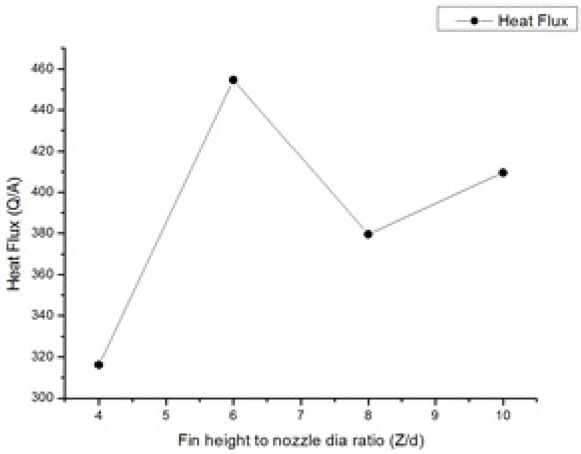


Fig-18: Heater and fin temperatures without blower At Z/d=6 and U=0m/sec. **Fig-19:** Heater and fin temperatures with blower At Z/d=6 and U=8m/sec.



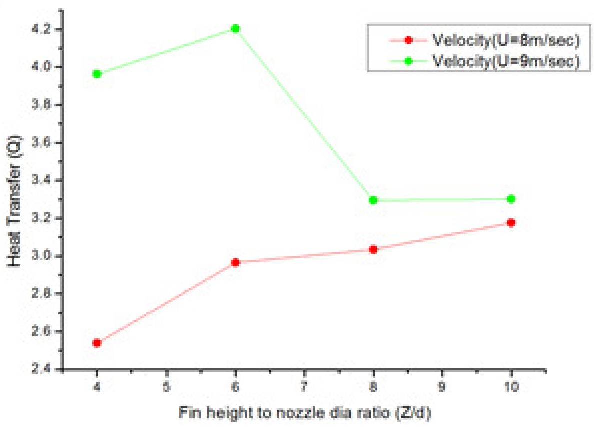


Fig-20: Z/d to heat flux (Q/A=W/m^2). **Fig-21**: Z/d to Heat transfer (Q=W).

The results of both numerical and experimental are shown that maximum heat dissipation from aluminium fin material or heat sink with thermal conductivity 256w/mk observed at Z/d=6 for the velocity of air at 8m/sec. The numerical results also observed as a good manner by observing fig.16 and fig.17.as the Reynolds number increases Nusselt number also increases.The stagnation nussult number can be calculated at minimum temperature of the fin at z/d=6 and velocity U=8m/sec. The values obtained by simply substitution of values in equation numbers V, VI and VII. Well it was clear that by looking fig.20 and fig.21 the maximum heat dissipation rate at Z/d =6 with air flow velocity from the fixed nozzle diameter d=20mm is U=8m/sec.

6. CONCLUSIONS

This paper summarises the computational and experimental results of jet impingement on aluminium heat sink for the maximum heat dissipation rate at variable parameters like fin height to nozzle diameter ration (Z/d), heat input conditions, different velocity cases.

- The highest Nusselt number were obtained for Z/d=6 and U=8m/sec.
- The highest heat flux and heat transfer rate is obtained at Z/d=6 for both numerical and experimental results.
- The Produced nusselt number correlations from experimental and numerical results were very close to each other and have high accuracy rate.

REFERENCES

[1] Ridvan Yakut el,, al.(2016) '' Experimental and Numerical Investigations of Impingement Air Jet for a Heat Sink'' Procedia Engineering International Conference on Computational Heat and Mass Transfer, ICCHMT2016.

[2] Angioletti, M., Nino, E. and Ruocco, G., 2005, 'CFD Turbulent Modelling of Jet Impingement and Its Validation by Particle Image Velocimetry and Mass Transfer Measurements, '' International Journal Of Thermal Sciences, 44(4), pp. 349-356.

[3] Bayraktar S. and Yılmaz, T., 2005, 'The Investigation of Vortices by Using Second Order Turbulence Closure,'' 15. National Heat Science and Technique Congress, Trabzon, Turkey.

[4] Deogonda, P., Chalwa, V. N., Murugesh, M.C. and Dixit, V., 2014, ''Wall Static Pressure Distribution Due to Confined Impinging Circular Air Jet,'' International Journal of Research in Engineering and Technology, 3(3), pp. 591-597.

[5] Yücel, O. and Baydar, E., 2013, '' A Numerical Investigation on Confined Impinging Array of Air Jets,'' Journal of Thermal Science and Technology, 33(2), pp. 65-74.

[6] Katti and Prabhu (1983) "Distribution between smooth flat surface and impinging air jet from a circular straight pipe nozzle," Int. J. Heat Mass Transf. 51 (17) (2008) 4480– 4495.

[7] P. Hrycak, (1983) "Heat transfer from round impinging jets to a flat plate," Int. J. Heat Mass Transf. 26 (12) (1983) 1857–1865

[8] A.M. Huber, R. Viskanta, (1994) "Effect of jet-jet spacing on convective heat transfer to confined, impinging arrays of axisymmetric air jets," Int. J. Heat Mass Trans. 37 (18) (1994) 2859–2869.