ENHANCEMENT OF COOLING USING DESIGN MODIFICATION ON SINGLE WALL MOUNTED CUBE EXPOSED TO IMPINGING JET AND CROSS FLOW

[1]S ABDULLAH, [2]MEHUL VARSHNEY, [3]YAGNESH P PARJIYA

[1]University of Malaya, ZHCET
[2]Aligarh Muslim University
[3]Central Queensland University
E-mail: [1]abdul.miet@gmail.com, [2]mehulvarshney@zhcet.ac.in, [3]yagneshparjiya207@gmail.com

Abstract - The aim of the present work is to enhance the cooling effectiveness of a single wall mounted cube exposed to imping jet and cross flow. The arrangement is analogous to electronic cooling configuration. For a particular value of cross flow velocity corresponding to Reynolds number (Re_H) of 3410, three different jet velocities are being tested which corresponds to Reynolds number (Re_j) of 1705 and 3410. The governing equation of continuity, momentum and energy have been solved using commercial package Fluent®. The cooling effectiveness enhancement has been done using modifications in the design of the cube which involve normal grooving. The design changes bring about a change in the flow patterns which have been studied in the scope of this work. It can bring about the heat transfer enhancement by 1.3 times.

Keywords - Electronics Cooling, Heat Transfer, Jet Impingement, Thermal Management

I. INTRODUCTION

The rapid development of electronics industry is responsible for the development of future microprocessors. Main cause of failure of these devices is overheating which decreases the efficacy. Dates to 1965 the Moore's law states that the number of transistors per square inch of integrated circuits would keep on doubling in every two years [1]. Electronics industry needs to deal with advancement of transistor design as well as heat transfer enhancement. Thermal management technologies have boosted due to demand of heat dissipation requirement. Thermal management market is likely to expand in global market with significant level of 12.4 billion to 16.3 billion by 2023[2].
In this regards lot of basic research has been carried out to enhance heat transfer and flow behavior in micro as well as in macro level. Predominant electronic cooling has been required in microchannel reactor [3], gas turbine blade cooling [4], rocket engines precooling [5], hydrogen storage cooling [6] and laser diode cooling [7,8]. In the next case of heat transfer enhancement has been done through multi inlet channel in electronic devices [9]. In these regards use of jet from the top of the electronic components is used for the electronic cooling system by changing the shape of electronic component [10].

Numerical study of heat transfer and fluid flow had been done on heated block with the help of jet from the top recently presented by Arquis et al [11].

Various studies had been done to improve cooling efficacy of heated block where fluid hydrodynamics play important role in cooling effectiveness. In a similar vein, this study is focused on enhancement of heat transfer through the cubical block. The geometry of this study is like experimentally studied by Masip et al [12] here top surface is manipulated by creating rectangular groove.

II. CASE DESCRIPTION

As mentioned before, geometrical details and computational parameters are like those in the experimental work of Masip et al. [12]. The complete domain can be seen in Figure 1. The main rectangular channel is 2000mm*300mm*30mm in dimensions. The cube is placed exactly at the center of the bottom wall. The inlets to the domain happen mainly from two faces. One is the channel inlet and the other is the jet inlet. To ensure that the jet delivers the fully developed flow by the time it enters the channel domain, the inlet length of the jet has been kept sufficiently long (240mm) which is approximately 20 times the jet diameter.

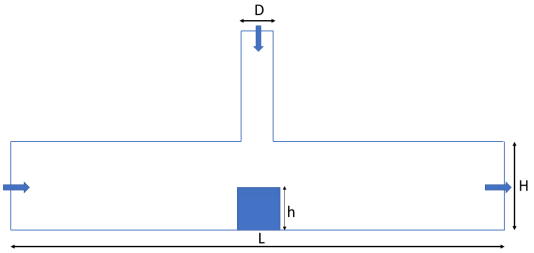


Figure 1: Computational Domain

The details of the dimensions of the computational domain have been shown in Table 1.
For heat transfer enhancement, the normal grooves have been done on the top of the cube. The details of the grooves also have been mentioned in Table 1. Figure 2 shows the cube with grooves.

Jet diameter	*D*	12 mm
Component Height	*h*	15 mm
Channel Height	*H*	30 mm
Groove width/Height	*Gw/Gh*	1.25 mm
Number of Groove	*Ng*	6
Computational domain length	*Sx*	2000 mm
Computational domain width	*Sy*	300 mm
Reynolds number based on channel cross-section	*ReH*	3410
Reynolds number based on jet cross-section	*Rej*	1705,3410

Table 1: Computational domain dimension



Figure 2: Slotted Cube

The slotting of the cube has only been done normal to the channel flow direction in the scope of this study. The depth and width of the slots have been maintained uniform. Table 2 shows the description of the cases which have been solved.

	Rej/ReH (Baseline)	*Rej/ReH* (With grooves)
Case 1	0.5	0.5
Case 2	1	1

Table 2: Description of the cases studied

This case has been solved in 3-D to account for the flow disturbances induced in third direction by putting grooves. The first point near the wall has been kept as per the requirement of the turbulence model employed. The total cell count for the case is 3 million.

III. GOVERNING EQUATIONS & BOUNDARY CONDITIONS

The present simulations were conducted using the ANSYS-Fluent package, where the solution of the Reynolds Averaged Navier-Stokes (RANS) and energy equations is obtained using the finite volume method on body-fitted hexahedral structured grid. A co-located layout is employed in which the pressure, turbulence, and velocity variables share the same location and continuity equation is coupled through a pressure correction scheme, and several implicit first- and second-order accurate schemes are implemented for the space and time discretization.

In the present work, convection terms are discretized with a second order except near discontinuities, where it reduces to first order to preserve boundedness. The turbulence closure is achieved by use of the well-known Shear Stress Transport (SST) k omega-based model proposed by Menter [13].

For the simulation, velocity inlet with the value of velocity based on Reynolds number has been prescribed on the channel inlet and jet inlet. The outlet has been specified as pressure outlet with pressure of 101325 Pa. The channel length in the upstream and downstream direction is sufficient such that the results are not influenced by the placement of the boundary. All the walls of the domain other than cube walls has been specified the temperature of 293 K. The temperature on the walls of the cube is fixed at 333 K. The solid conduction has not been considered and hence the mesh has not been created inside the cube. Air has been used as working media. Since the temperature of the wall is 333 K therefore temperature near the wall can influence the air density and viscosity. Hence, the density of the air has been modeled using ideal gas law and the viscosity has been modeled using Sutherland Law. It was seen that the simulation did not exhibit any transient effects and hence the steady state method was used to run the simulation.

To ensure that the results obtained by the simulation are not a function of grid size, a grid independence test was performed. The global mesh size along with the mesh size near the wall was refined such that the overall cell count reached upto 5 Million. It was seen the velocity profile did not improve any further compared to what has been shown in the section of validation. So, the 3D grid with 3 Million cells was used to run the simulation.

IV. VALIDATION

The validation of the present method has been done with the experimental results of Masip et al. [12]. The experimental work is done using Particle Image Velocimetry (PIV). The comparison of non-dimensional streamwise velocity with respect to non-dimensional height of the channel has been done in the streamwise center plane of the channel. The non-dimensonalization of the streamwise velocity and height has been done with respect to jet velocity (Uj) and the block height (h). The comparison has been done at three locations mentioned as A, B & C in Figure 3.

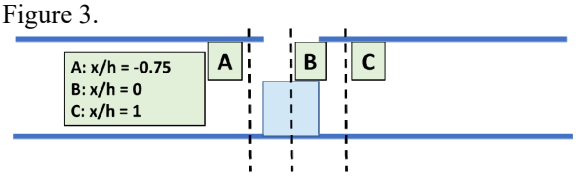


Figure 3: Location of velocity comparison

Figure 5 & Figure 6 shows the comparison of the data at Rej/Re_H of 0.5 & 1 respectively. The quantitative plots show a very good comparison of the streamwise velocity at downstream, upstream and wall location. However, there are certain differences in the comparison which can be attributed to the experimental errors and the turbulence model limitations. Nevertheless, the comparison was good

enough to proceed the problem using limited computational power. The profiles show flow reversal happening near the wall which the simulations were able to capture. The flow reversal becomes dominant in the wake region of the cube.

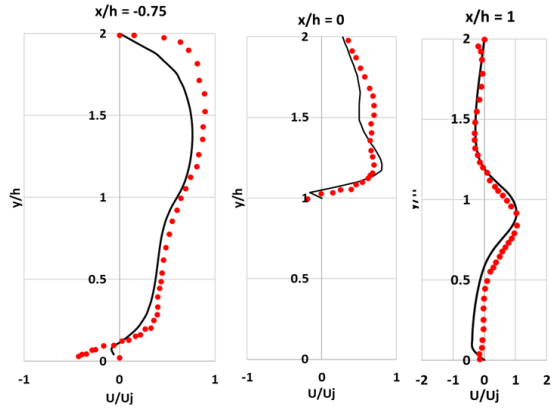


Figure 5: Comparison of streamwise velocity profile at *Rej/ReH*=0.5

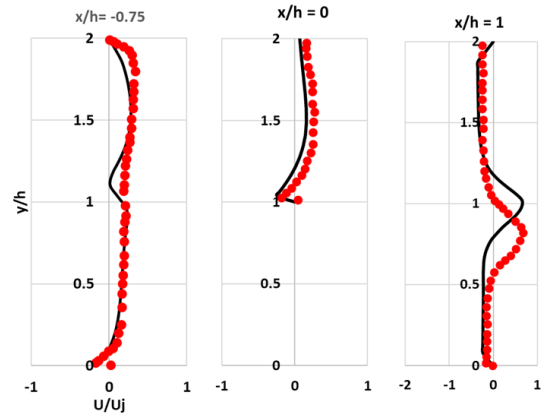


Figure 6: Comparison of streamwise velocity profile at *Rej/ReH*=1

The validation ensured the size of the grid, solver settings, boundary conditions, properties that can be used to carry out the further studies. For validation, energy equation was not solved since the experimental results also do not have consideration of effects of temperature in the domain.

V. RESULTS & DISCUSSION

The introduction of the grooves in the present work leads to heat transfer enhancement as can be seen in Table 3. It can be observed that the heat transfer rate increases by almost 26% which is a considerable increase.

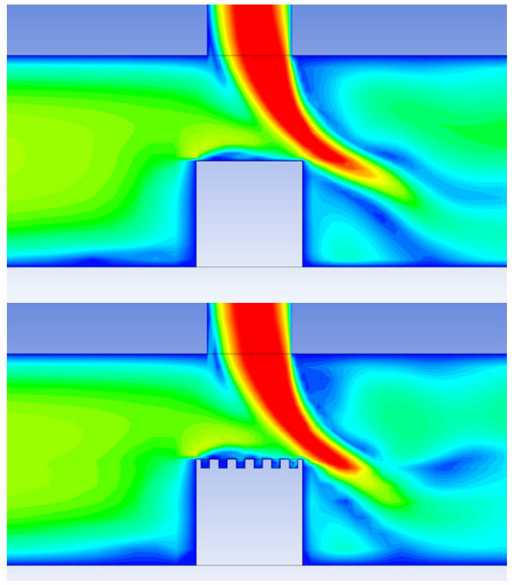
Rej/Re_H	Baseline	Grooves	% Change
0.5	1.33	1.44	8.271
1	1.85	2.33	25.946

Table 3: Comparison of heat transfer rate with and without grooves

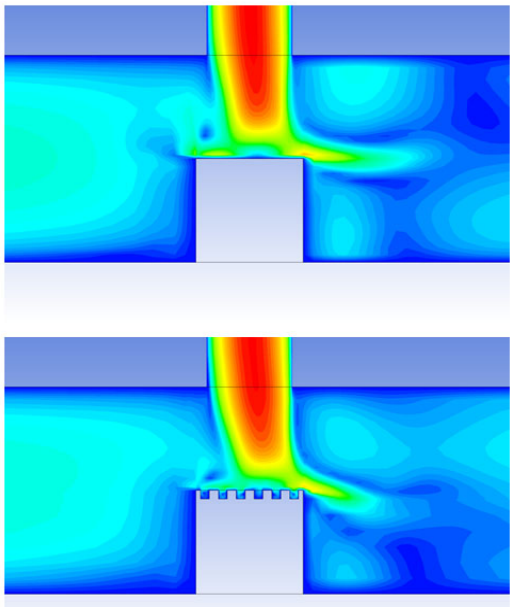
Since this is a case of multiple inlet hence there will be two faces of the cube that are exposed to cooled stream. The stream of the jet hits the grooved surface. The heat transfer rate increases at higher velocity of the jet. This can be attributed to enhanced mixing that happens due to increased jet velocity. The introduction of the grooves increases the vorticity and enhances the mixing. Residence time can be defined as the time for which the air flow remains in contact with the solid surface. The grooves increase the residence time and hence the air gets more time to extract heat from the surface of the cube. The grooves also reduced the flow separation happening over the surface of the cube. Also, the reattachment length behind the cube has reduced significantly.

The thermal boundary layer around the cube is also significantly modified. Slots have reduced the overall area of influence of high temperature.

Figure 7(a) shows the comparison of non-dimensional velocity magnitude (made dimensionless using jet velocity) on the streamwise center plane. For the ratio of 1, the comparison can be seen in Figure 7(b).



(a)



(b)

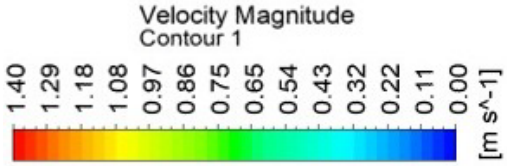
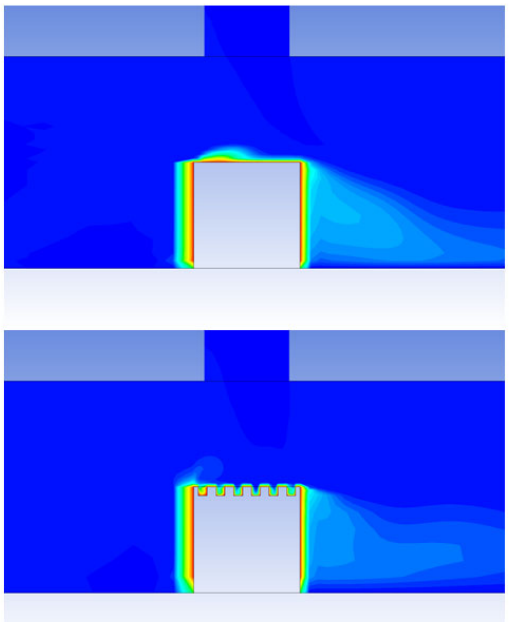
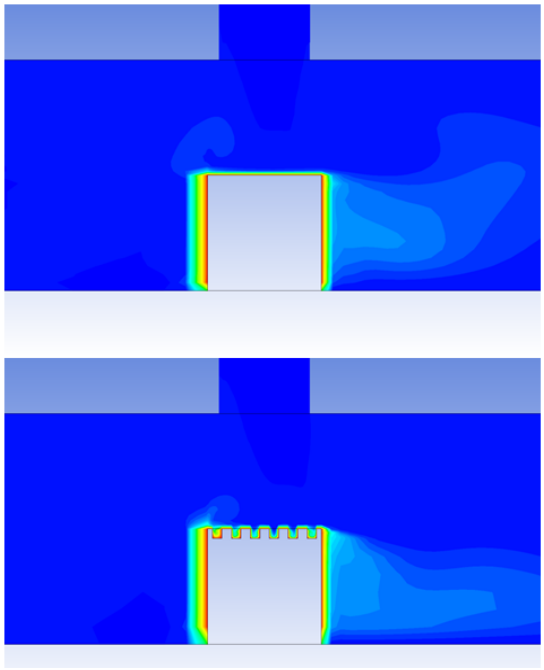


Figure 7: Velocity magnitude contours for (a) *Rej/ReH* = 0.5 (b) *Rej/ReH* = 1

Figure 8 shows the comparison of temperature on the streamwise center plane. It can be seen presence of grooves radically transform the thermal boundary layer. The majority of transformation has occurred over the top surface of the cube owing to the presence of the groove on that surface. The influence of the high temperature region has come down in the controlled case.



(a)



(b)

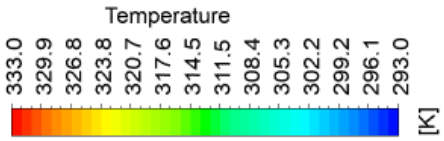
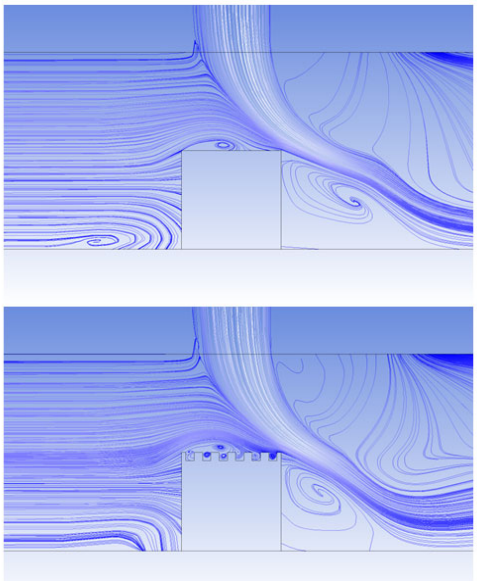
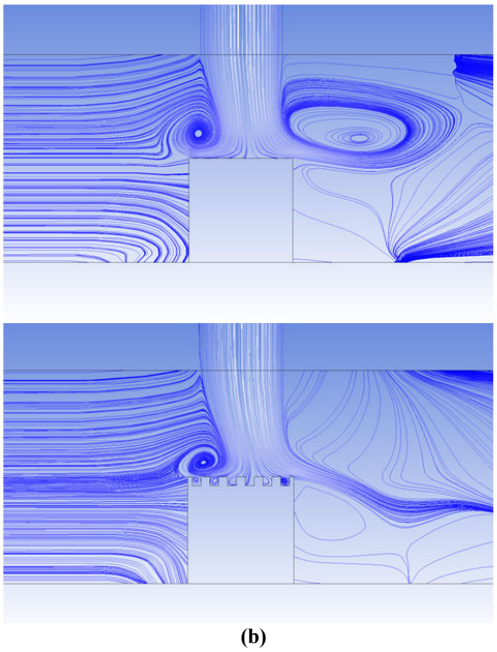


Figure 8: Temperature contours for (a) *Rej/ReH* = 0.5 (b) *Rej/ReH* = 1

Figure 9 shows the comparison of stream traces at the streamwise center plane. Figure 9(a) shows the flow separation and reattachment happening in the wake region of the cube. Also, the flow separation at the top of the cube can be seen in the baseline case. In the controlled case the flow separation at the top of the cube is not seen. However, small vortices entrapped in the grooves can be seen. For the uncontrolled case in figure 9(b) there are two major vortex. However, for the case with grooves only one major vortex appears.



(a)



(b)

Figure 8: Stream Traces in center plane for (a) *Rej/ReH* = 0.5 (b) *Rej/ReH* = 1

VI. CONCLUSION

In the present study, a slight modification in the design has been introduced to enhance the heat transfer. It was seen that the design change was effective in enhancing the heat transfer rate. Enhanced heat transfer hints at effective working of the electronic component which in turn ensures the smooth operation and increased lifetime of the equipment. The increase in the heat transfer rate was seen to be 26% of its baseline value. This method find its application in the cooling of electronic component like PCB. It can also help to make the component compact. Though the design change can incur additional manufacturing costs, but it can certainly be compromised by increased life and better performance of the equipment. The suggested design change in this work is practically feasible for manufacturing also. The future scope of the study involves determining the threshold ratio of Reynolds number upto which a design change can be effective in enhancing heat transfer rate.

REFERENCES

[1] Moore, G. E., 1965, "Cramming More Components onto Integra Circuits," Electronics, 38(8), pp. 114–117.

[2] McWilliams, A., 2019, "The Market for Thermal Management Technologie,", bcc Research Report Code: SMC024L.

[3] Tonkovich, A. Y., Zilka, J. L., LaMont, M. J., Wang, Y., and Wegeng, R. S., 1999, "Microchannel Reactors for Fuel Processing Applications. I. Water Gas Shift Reactor," Chem. Eng. Sci., 54(13–14), pp. 2947–2951.

[4] Nacke, R., Northcutt, B., and Mudawar, I., 2011, "Theory and Experimental Validation of Cross-flow Micro-channel Heat Exchanger Module with Reference to High Mach Aircraft Gas Turbine Engines," Int. J. Heat Mass Transf., 51(5-6), pp. 1224–1235.

[5] Wang, Z., Wang, Y., Zhang, J., and Zhang, B., 2014, "Overview of the Key Technologies of Combined Cycle Engine Precooling Systems and the Advanced Applications of Micro-Channel Heat Transfer," Aerosp. Sci. Technol., 39, pp. 31–39.

[6] Visaria, M., Mudawar, I., Pourpoint, T., and Kumar, S., 2010, "Study of Heat Transfer and Kinetics Parameters Influencing the Design of Heat Exchangers for Hydrogen Storage in High-Pressure Metal Hydrides," Int. J. Heat Mass Transf., 53(9–10), pp. 2229–2239.

[7] Beach, R., Benett, W. J., Freitas, B. L., Mundinger, D., Comaskey, B. J., Solarz, R. W., and Emanuel, M. A., 1992, "Modular Microchannel Cooled Heatsinks for High Average Power Laser Diode Arrays," IEEE J. Quantum Electron., 28(4), pp. 966–976.

[8] Missaggia, L. J., Walpole, J. N., Liau, Z. L., and Phillips, R. J., 1989, "Microchannel Heat Sinks for Two-Dimensional High-Power-Density Diode Laser Arrays," IEEE J. Quantum Electron., 25(9), pp. 1988–1992.

[9] Kumar, R., Abiev, R., Ribatski, G., Abdullah, S., and Vasilev, M., 2020, "New Approach of Triumphing Temperature Nonuniformity and Heat Transfer Performance Augmentation in Micro Pin Fin Heat Sinks," J. Heat Transfer, 142(6), pp. 1–12.

[10] Saleha, N., Fadèla, N., and Abbès, A., 2015, "Improving Cooling Effectiveness by Use of Chamfers on the Top of Electronic Components," Microelectron. Reliab., 55(7), pp. 1067–1076.

[11] Arquis E, Rady MA, Nada SA. A numerical investigation and parametric study ofcooling an array of multiple protruding heat sources by a laminar slot air jet. Int J Heat Fluid Flow 2007;28:787–805

[12] Masip Yunesky, Rivas Alejandro, Larraona Gorka S, Anton Raúl, Ramos Juan Carlos, Moshfegh Bahram. Experimental study of the turbulent flow around a single wall-mounted cube exposed to a cross-flow and an impinging jet. Int J Heat Fluid Flow 2012;38:50–71.

[13] Menter FR. Zonal two-equation k–eps turbulence model for aerodynamic flows. AIAA Paper 1993:93–2906.

★★★