iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** X **Month of publication:** October 2017

DOI: http://doi.org/10.22214/ijraset.2017.10321

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue X, October 2017- Available at www.ijraset.com

Mathematical Modelling and Analysis of Flat Louvers on the Air Duct Intake Using CFX

Sudhir Singh[1], Yogesh Tembhurne[2]
[1]M. Tech Scholar Mech. Eng. Dept. Bhabha Engineering Research Institute Bhopal
2 Asst. Prof. Mech. Eng. Dept. Bhabha Engineering Research Institute Bhopal

Abstract: Louver configuration of pride in designing an air inlet frame work is important part. Gas turbine power station applications, the fins can remove the air present in the compressor and means for cooling the interior of the structure. That it can be shaped and formed louvers will affect the performance of a gas turbine. Discomposed air over the turbine inlet air louvers current work to produce drop-off prompts can cause pressure drop during this test, we are accessible to analysis CFD code by exploiting the influence of Airfoils width variation. In the present study we CFD simulations through 2 mm gap width of louvers from 10mm to 18mm with various cases separately. We have found that the simulation results obtained from the width greater than the pressure drop in the previous research recognized CFD Simulations. From the simulation results obtained we found that greater the width induce greater pressure drop which is validate to previous researches. Cause for pressure drop is the formation of vortex due to flow separation across the louvers and it is easily visualized from velocity contours. Larger width provides increased distance for reattachment point, for creating larger vortex below the louvers and hence higher magnitude of flow separation is achieved. This vortex narrowed down the flow path between louvers causing increased velocity due to turbulence within that region, reducing the pressure as stated by Bernoulli's Principle.
Keywords: Louver, Airfoil, Air Duct, CFD, CFX etc.

I. INTRODUCTION

Air intake system is important in order to a turbine operation. Air suctioned into the system is employed as oxidant for engine combustion and as a cooling medium for turbo machineries element of a turbine. Poor air intake system can lead to poor overall performance of

The Turbine. For that reason, it's necessary to make sure that the flow of air within the system to be "as smooth as" potential. an air intake system could accommodate weather hood with an array of louvers, multiple stages of filter and inlet duct with silencer. Close air with atmospheric pressure is initially coming into the weather hood, where it bypasses the filters before travel further downstream into the inlet duct into the compressor. The air could experience the reduction of atmospheric pressure, because of the disturbances and friction present throughout the intake system. Massive pressure drop across the system can increase the engine's fuel consumption and reduce the turbine work output as a result of low air inlet mass flow and engine's combustion temperature. Weather hood is put in upstream of the air inlet system, wherever it operates to deflect massive particles together with rain and snow droplets from the inlet air stream, and an equivalent time, directing close air into the system. It's designed by an array of 2-dimensional flat louvers pointed upward at an angle in relation to the flow direction. An economical weather hood should be able to take away large particles from the flow stream and at a similar deadline the enormous scale of pressure loss within the flow of air across its louvers. As the compression ratio is mounted for a turbine compressor, so low air pressure at the compressor inlet can lead to comparatively low air pressure into the combustion chamber which consequently can turn out low work output [1]

A. Louver Fin

A louver is a window visually impaired or screen with level supports that are calculated to concede light and air, however to keep out downpour, direct daylight, and commotion. The point of the supports might be flexible, generally in blinds and windows, or altered. Louvers began in the Middle Ages as light like developments in wood that were fitted on top of rooftop gaps in substantial kitchens to permit ventilation while keeping out downpour and snow. They were initially rather rough developments comprising just of a barrel. Later they advanced into more intricate outlines made of earthenware, taking the state of countenances where the smoke and steam from cooking would spill out through the eyes and mouth, or into developments that were more similar to present day louvers, with supports that could be opened or shut by pulling on a string. Present day louvers are frequently made of aluminium, metal, wood, or glass. They might be opened and shut with a metal lever, pulleys, or through mechanized administrators. Louvers are once in a while seen as essential configuration components in the dialect of present day engineering, but instead basically a specialized gadget.

II. LITERATURE REVIEW

An air intake system is a section in gas turbine that draws ambient air and route it into the compressor. Air is an essential consumables for a gas turbine operation, thus air ingested into the device must be of high quality, clean and have sufficient pressure. To meet these criteria, the air intake system is often equipped with components including weather hood, a series of filters, ductwork, and silencer. All of the components must be designed with as little flow disruption as possible to the air entering the system because large disruption on the airflow will result in large pressure drop, consequently reducing the combustion temperature hence increasing the engine's fuel consumption and producing less work from the turbine [2]. Air first enter the weather hood at local pressure and velocity, and continue downstream to multiple stages of filters which normally includes insect screen, anti-icing, inertial separator, pre-filter, and high efficiency filter. The airflow continues into the ductwork which convoys the air into the gas turbine compressor. In each of these stages pressure drop occurs due to formation of vortices, friction and turbulence airflow. Numerous studies have been conducted to investigate the pressure drop across these components.

Khorshand et al. [3] investigates the flow pattern and pressure loss of Khoramshar Power Plant v94.2.5 Gas Turbine air intake units. Its aim is to improve the pressure loss and flow characteristics of the gas turbine compressor inlet. The components involved in the study are weather hood, filter house, bending duct and silencers, and transition piece and compressor inlet cone. The model used in the analysis for this research is as shown in figure 1. A 3-dimensional simulation of flow past components in air intake system was conducted by

Using commercial numerical CFD software, FLUENT. Simulation on weather hood shows vortex formation in louvered part of the weather hood, where caused the flow velocity to increase, reducing the pressure of air pass through the weather hood. Although pressure loss in intake system is more significant across filter and ductwork, the inlet louvers of weather hood also record a considerable amount pressure differential.

Hughes et al. [4] investigated the effect of altering the external angle of wind vent louvers against the internal Pressure and velocity within the device and microclimate velocity. The aim of the research is to obtain the angle with lowest pressure loss in order to optimize fresh air delivery rate to the occupied space. The method used in the research is CFD modelling, and the flow of air past the wind vent louvers were simulated and analyzed at louvers angle 10o to 45o with 5o increment. The wind velocity is set at 4.5 m/s which represent the local wind velocity. Results of the research indicate that by altering louver angle the velocity and pressure performance is improved to desirable levels to achieve optimum microclimate velocity levels. It was shown that louver angle of 35o provides optimum performance at given parameters. Turnbull et al. [5] conducted a study on the impact of air filtration onto compressor efficiency, and Muller [6] commented on how air flow simulation can help in reducing pressure drop in gas turbine inlet duct. Flat louver is the common geometry of louvers in weather hood design.

III. METHODOLOGY

CAD Modelling: Creation of CAD Model by using CAD modelling tools NX8.5 for creating the geometry of the part/assembly.

Meshing: Meshing is a critical operation in CFD. In this operation, the CAD geometry is discretized into large numbers of small Element and nodes. The arrangement of nodes and element in space in a proper manner is called mesh. The analysis accuracy and duration depends on the mesh size and orientations. With the increase in mesh size (increasing no. of element) the CFD analysis speed decrease but the accuracy increase.Governing equation- The governing equations used were 2-dimensional Navier-Stokes equation and continuity equation since the flow is less than the compressibility Mach number of 0.3, the effect of flow compressibility was not considered hence the airflow in this research was treated as incompressible flow. The whole structure of weather hood model was taken as the computational domain for CFD simulation. Boundary conditions were specified at the front and rear surface of the weather hood model with respect to flow direction. Flow velocity of 5 m/s was set as the inlet boundary condition while static pressure was set as the outlet boundary condition.

Type of Solver- Pressure Based.

Physical model- k-ε model Turbulence model is used.

Material (Fluid)- AIR

Boundary Condition- Velocity of air at inlet is 5m/s.

Solution Method- Second order

Solution Initialization- Initialized the solution to get the initial solution for the problem.

Run Solution: Run the solution by giving 1000 no of iterations.

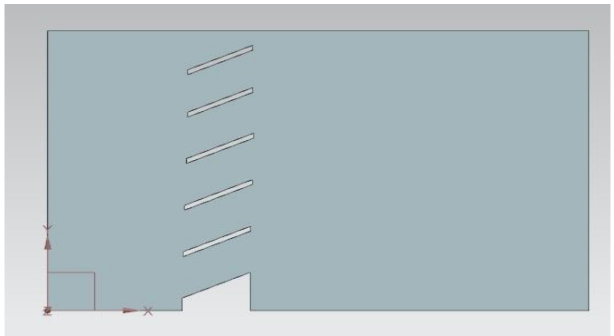


Figure: 1 CAD Model of 10mm case

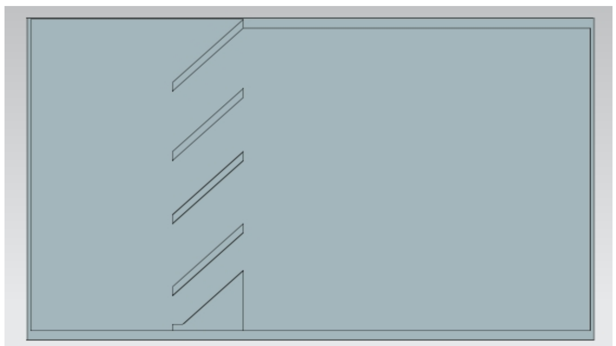


Figure: 2 CAD Model of 12mm case

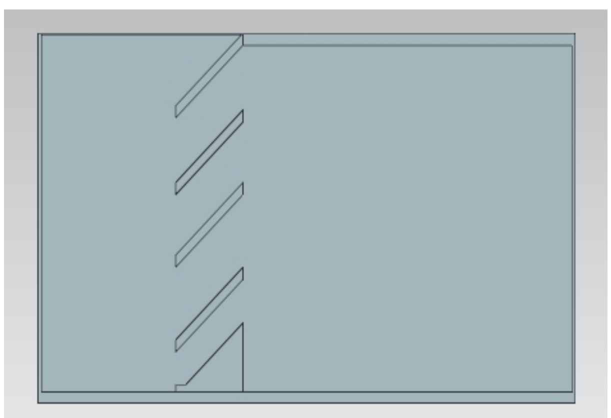


Figure: 3 CAD Model of 14mm case

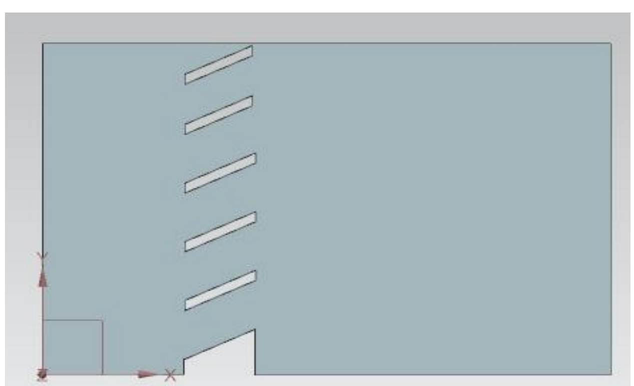


Figure: 4 CAD Model of 16mm case

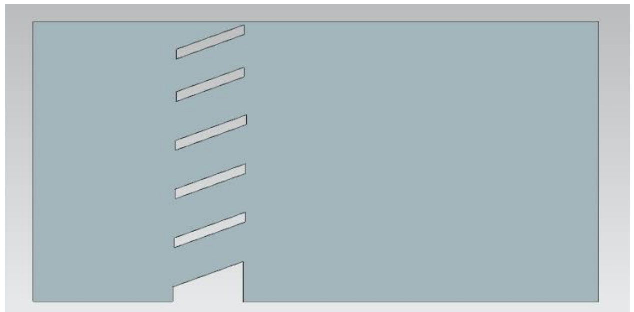


Figure: 5 CAD Model of 18mm case

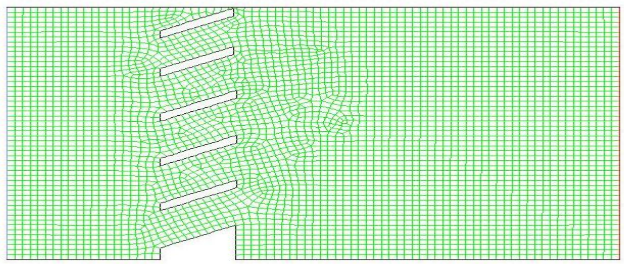


Figure: 6 MESH Model

IV. RESULTS

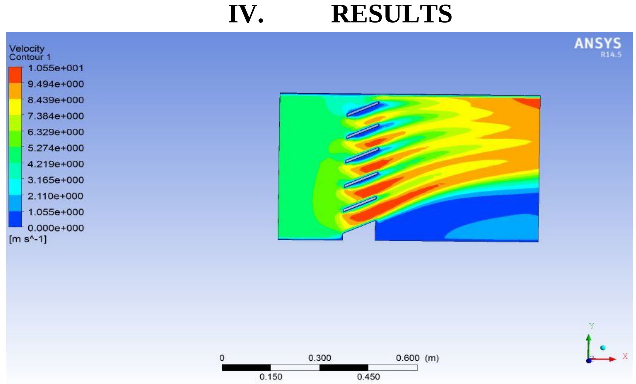


Figure: 7 Velocity profile in 10mm case

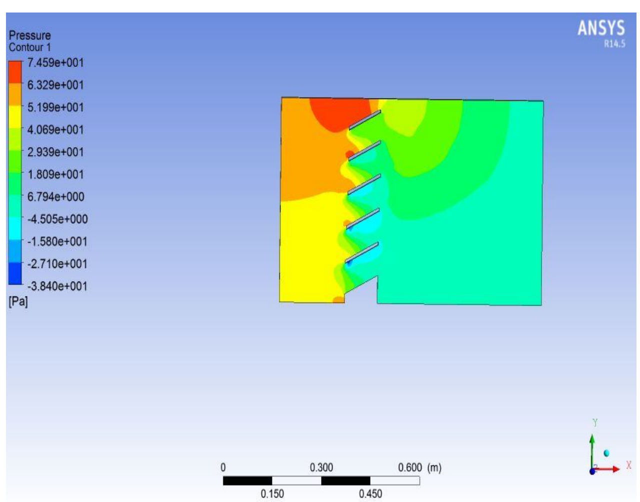


Figure: 8 Pressure drop in 10mm case

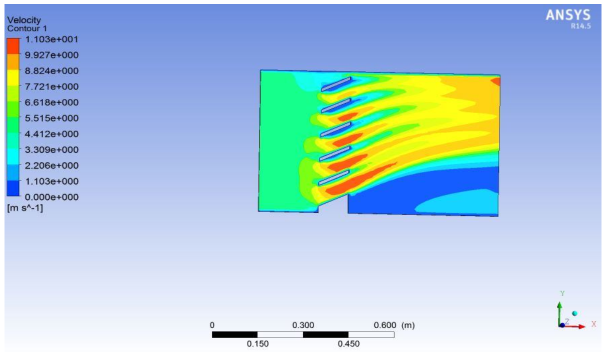


Figure: 9 Velocity profile in 12mm case

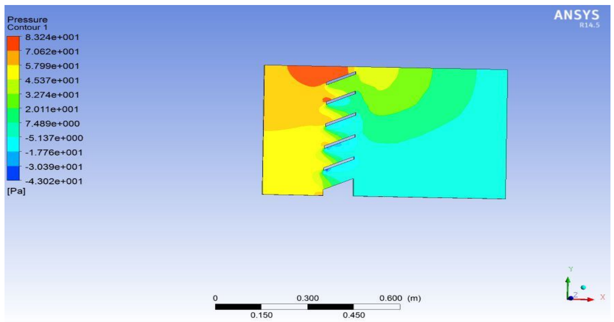


Figure: 10 Pressure drop in 12mm case

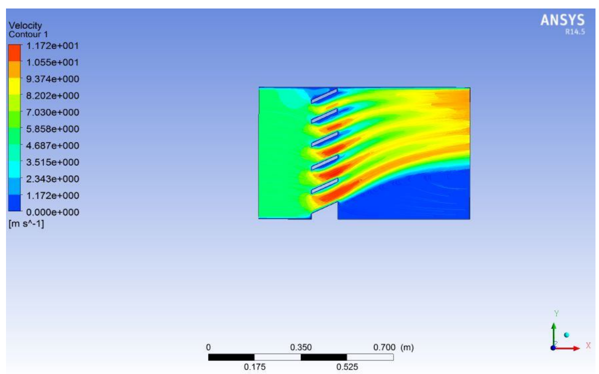


Figure: 11 Velocity profile in 14mm case

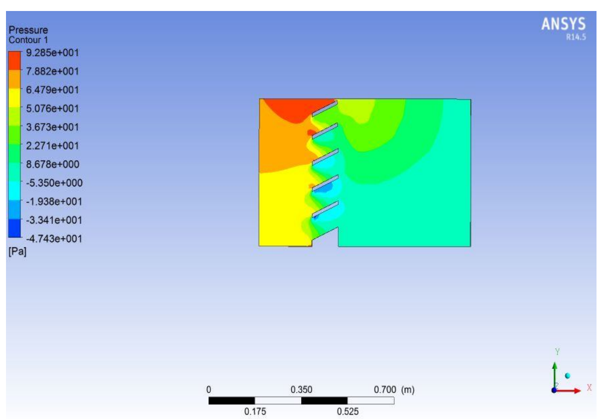


Figure: 12 Pressure drop in 14mm case

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue X, October 2017- Available at www.ijraset.com

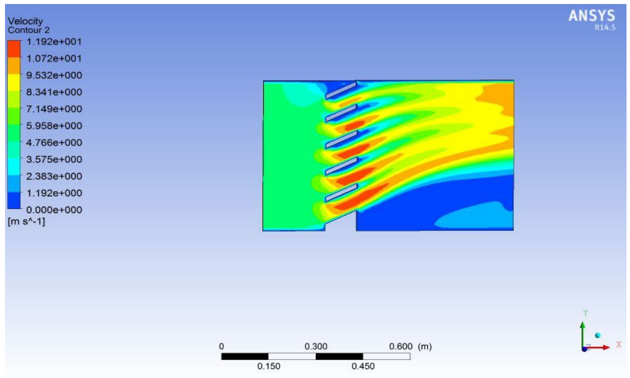


Figure: 13 Velocity profile in 16mm case

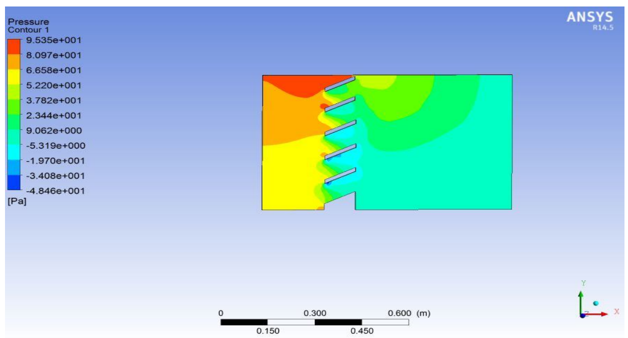


Figure: 14 Pressure drop in 16mm case

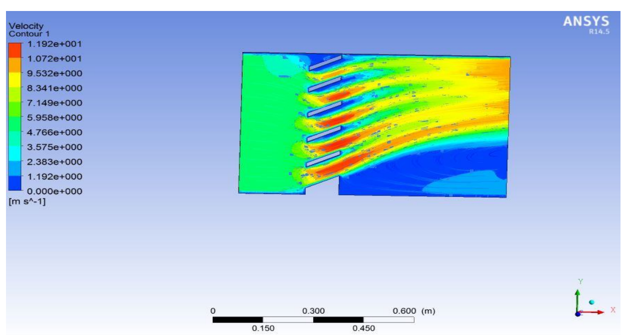


Figure: 15 Velocity profile in 18mm case

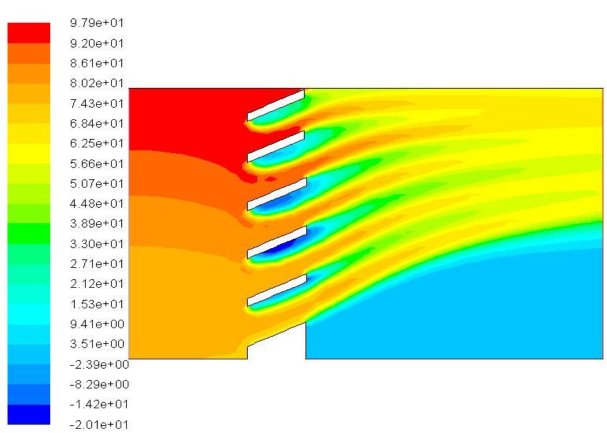


Figure: 16 Pressure drop in 18mm case

V. CONCLUSION

In present study we investigate the different cases by varying the width of louvers from 10mm to 18mm with 2mm intervals through CFD Simulations. From the simulation results we found that greater the width induce greater pressure drop which is validate to previous researches. Cause for pressure drop is the formation of vortex due to flow separation across the louvers and it is easily visualized from velocity contours. Larger width provides increased distance for reattachment point, for creating larger vortex below the louvers and hence higher magnitude of flow separation is achieved. This vortex narrowed down the flow path between louvers causing increased velocity due to turbulence within that region, reducing the pressure as stated by Bernoulli's Principle.

CASE NO	PRESSURE DROP (ΔP)	VELOCITY m/s
Width10 mm	74.59	10.55
Width 12 mm	83.24	11.03
Width 14 mm	92.85	11.72
Width 16mm	95.35	11.92
Width 18 mm	97.90	11.92

REFERENCES

[1] Mohd Hafiz MohdNoha, HelmiRashidb, Ahmad Hussein Abdul Hamidb, Muhammad Faisal Iskandarb “Comparison of Numerical Investigation on Airfoil and Flat Louvers on the Air Duct Intake” International Symposium on Robotics and Intelligent Sensors 2012 (IRIS 2012).

[2] Wilcox, M.; Baldwin, R.; Garcia-Hernandez, A.; and Brun, K. (2010). “Guideline for Gas Turbine Inlet Air Filtration Systems”, Release 1.0. New York, US. Gas Machinery Research Council.

[3] Khorsand, K.; Karimian, S. M.; Varmaziar, M.; and Sarjami, S. (2008). “Investigation of Flow Pattern and Pressure Loss of Av94.2.5 Gas Turbine Air Intake System Using 3D Numerical Modelling”. Department of Engineering. Department of Aerospace. Amir Kabir University. Tehran, Iran.

[4] Hughes, B. R.; Ghani, S. A. A. (2010). “A Numerical Investigation into the Effect of Wind vent Louver External Angle on Passive Stack Ventilation Performance”, The Journal of Building and Environment 45: 1025-1036.

[5] Turnbull, G.; Clement, E.; and Ekberg, Tord.; “Gas Turbine Combustion Air Filtration: Its Impact on Compressor Efficiency and Hot End Component Life”, Position Paper. European Turbine Network A.I.S.B.L.

[6] Muller, F. (2006). Simulation Helps Reduce Pressure Drop on Gas turbine Inlet Duct by 50%. Texas, US. Mueller Environmental Design Inc.

[7] Toshikazu Nakanishi Tamotsu Nakamura Youichirou Watanabe KatsumasaHandou Takahiro Kiwata “Investigation of Air Flow passing through Louvers” Komastu Technical Report 2007 ② VOL. 53 NO.160

[8] S.A. Wilfred Rajkumar, V.Vijayaraghunathan, S.Karthikeyan, I.SharonMarishka “Comparative Study of Air Flow Analysis of A Car Louvre” International Journal of Innovative Research in

[9] Science, Engineering and Technology An ISO 3297: 2007 Certified Organization Volume 4, Special Issue 3, March 2015Primož Pogorevc, Breda Kegl “DESIGN OF INTAKE SYSTEM WITH SPECIAL DEMANDS” University of Maribor, Faculty of mechanical engineering, Smetanova 17, SI-2000 Maribor,Slovenia Journal of KONES Internal Combustion Engines 2002/No/12/ISSN/123/400

[10] D.Ramasamy, Zamri.M, S. Mahendran, S.Vijayan “Design Optimization of Air Intake System (AIS) of 1.6L Engine by Adding Guide Vane” IMECS Vol- II March 2010

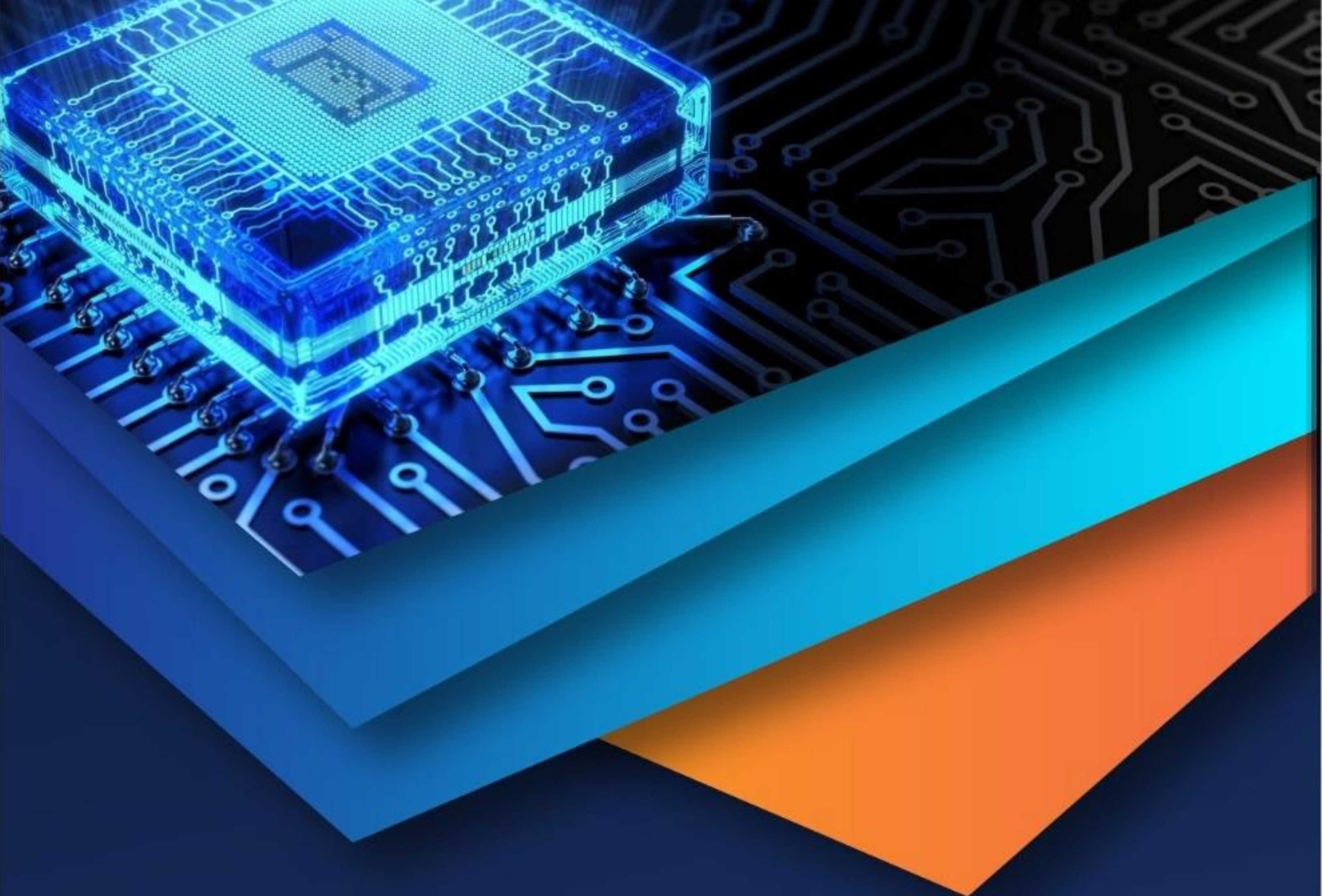
[11] M. Kim and C. W. Bullard, “Air-side thermal hydraulic performance of multi-louvered fin aluminum heat exchangers ´ air des e ´ changeurs de chaleur a Performance hydraulique co ˆ te ` persiennesailettes en aluminium a,” vol. 25, pp. 390–400, 200

[12] X. Zhu, “Air Flow and Heat Transfer in Louver-Fin Round-Tube Heat Exchangers,” vol. 129, no. February, pp. 200–210, 2007.

[13] L. Tian, Y. He, Y. Tao, and W. Tao, “International Journal of Thermal Sciences A comparative study on the air-side performance of wavy fin-and-tube heat exchanger with punched delta winglets in staggered and in-line arrangements,” International Journal of Thermal Sciences, vol. 48, no. 9, pp. 1765–1776, 2009

[14] [14] K. D. Bellows and A. Conditioning, “Flow Visualization of Louvered-Fin Heat Exchangers Amana Refrigeration , Inc . Carrier Corporation Caterpillar , Inc . Copeland Corporation Dayton Thermal Products Delphi Harrison Thermal Systems Eaton Corporation Ford Motor Company Frigidaire Company Gen,” vol. 61801, no. July, 1997.

[15] N. Flow, “An Experimental Study of Low-Reynolds- Number Flow and Heat Transfer in an Array of Louvers at a Non-Zero Angle of Attack,” vol. 61801, no. 217, 2000.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY