

# CFD studies on heat transfer analysis of straight and backward fin with cylindrical channel

<sup>1</sup>Kunal Dadhania, <sup>2</sup>Chirag Maradiya

<sup>1</sup>Student MEFGI,Rajkot, <sup>2</sup>Assistant Proffesor, MEFGI,Rajkot

<sup>1</sup>Mechanical Department,

<sup>1</sup>Marwadi Education Foundation group of Institute, Rajkot, India

**Abstract - CFD Analysis** are conducted to assess turbulent forced convection heat transfer and behaviors for air flow through a constant heat flux channel fitted with different shaped fins. The fin cross-sections used in the present study are straight and backward shapes. Two rib arrangements, namely, in-line and staggered arrays, are introduced. Measurements are carried out for a cylindrical with single fin height 6 mm and 8 mm, fin pitch is 40 mm. The analysis results show a significant effect of the presence of the fins on the heat transfer rate and friction loss over the smooth wall channel. The in-line fin arrangement provides higher heat transfer enhancement. Our analysis carried out with CFD fluent technique. Here analysis set-up carried out with the cylinder, straight and backward fin and heating coil. The use of rib turbulators completely results in the change of the flow field and hence the variation of the local convective heat transfers. The presence of a transverse fin assists to induce the main stream separation first, and to generate a recirculation zone ahead of it and then reattachment over the fin itself. If several fins exist and their pitch is sufficiently larger than the fin height, these flow patterns will reoccur along the channel wall.

**Index Terms – Finite Volume Method (FVM),CFD,ANSYS**

## I. INTRODUCTION

The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define many single-phase (liquid or gas, but not both) fluid flows. These equations can be simplified by removing mathematical terms describing viscous actions to yield the Euler equations. In the further simplification, by removing other terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations[1]. Historically, methods were first developed to solve the linearized potential equations. Two-dimensional (2D) methods, using conformal transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s[2].

An intermediate step between Panel Codes and Full Potential codes were codes that used the Transonic Small Disturbance equations. In particular, the three-dimensional WIBCO code which is very useful code and it is seen that its use is very higher[3].

Developers turned to Full Potential codes, as panel methods could not calculate the non-linear flow present at transonic speeds. The first description of a means of using the Full Potential equations was published by Earl Murman and Julian Cole of Boeing. Frances Bauer, Paul Garabedian and David Korn of the Courant Institute at New York University (NYU) wrote a series of two-dimensional Full Potential airfoil codes that were widely used.. Many Full Potential codes emerged after this, culminating in Boeing's Tranair (A633) code[4]. which still sees heavy use.so, many person have contributed their skills in this field of CFD[5].

The next step was the Euler equations, which promised to provide more accurate solutions of transonic flows[6]. The methodology used by Jameson in his three-dimensional FLO57 code was used by others to produce such programs as Lockheed's TEAM program and IAI/Analytical Methods' MGAERO program. MGAERO is unique in being a structured cartesian mesh code, while most other such codes use structured body-fitted grids (with the exception of NASA's highly successful CART3D code, Lockheed's SPLITFLOW code and Georgia Tech's NASCART-GT). Antony Jameson also developed the three-dimensional AIRPLANE code which made use of unstructured tetrahedral grids[7].

In the two-dimensional thing, Mark Drela and Michael Giles, then graduate students at MIT, developed the ISES Euler program (actually a suite of programs) for airfoil design and analysis[8]. This code first became available in the year of 1987 and has been further developed to analyze, design and optimize single or multi-element airfoils, as the MSES program[9]. MSES sees wide use throughout the world[10]. A derivative of MSES, for the design and analysis of airfoils in a cascade, is MISES, developed by Harold "Guppy" Youngren while he was a graduate student at MIT[11].

## II. CFD ANALYSIS

In in memory usage and solution speed, especially for large problems, High Reynolds number turbulent flows, and source the CFD analysis we have used a finite volume method. The finite volume method (FVM) is a common approach used in CFD codes, as it has advantage term dominated flows.

There are several steps to solve any problem in ANSYS fluent module.

Firstly we have to know about the analysis model.

Specification of the model:

- Cylinder length: 480 mm
- Cylinder diameter: 63.55 mm
- Heating coil length: 480 mm
- Heating coil diameter: 21.05 mm
- Triangular fin length: 20 mm
- Triangular fin height: 6 mm, 8mm

1) Prepare a geometry.

Firstly, we have a define a line diagram of the geometry. This is a line diagram of the straight fin. Here we change the flow of this cylinder , that was called a backward fin flow.

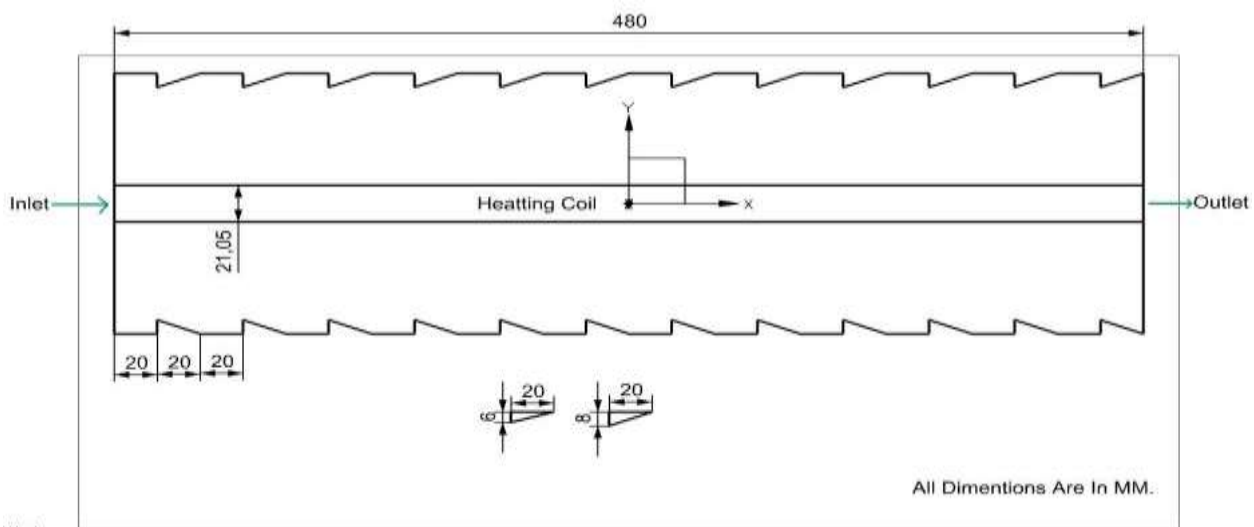


Fig 1 A line diagram of the straight fin

Here this is the ANSYS fluent geometry

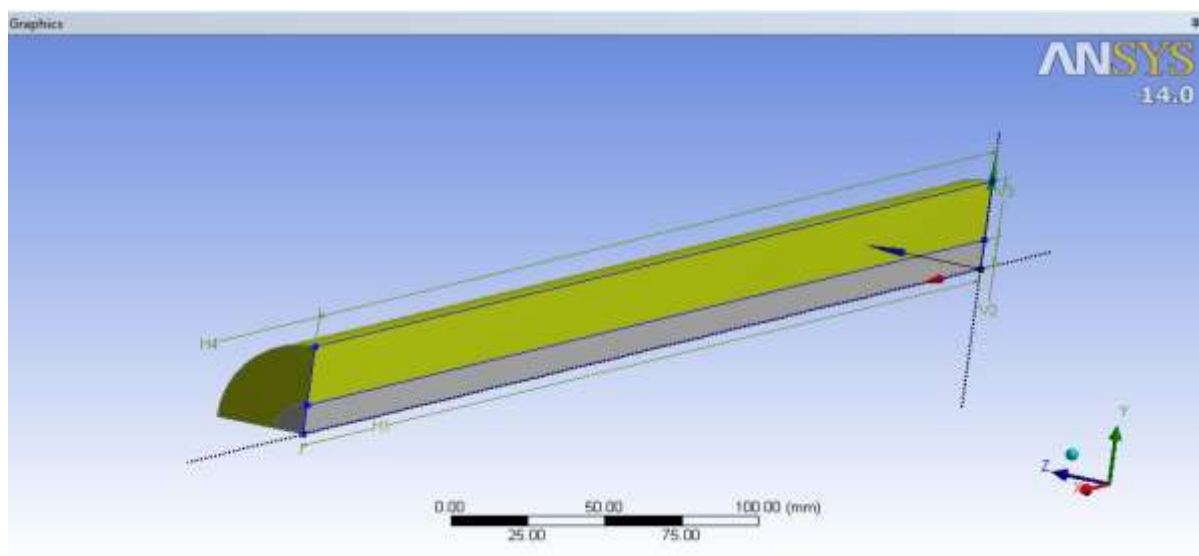


Fig 2 ANSYS fluent geometry

Here this is a fin geometry with the heating coil.

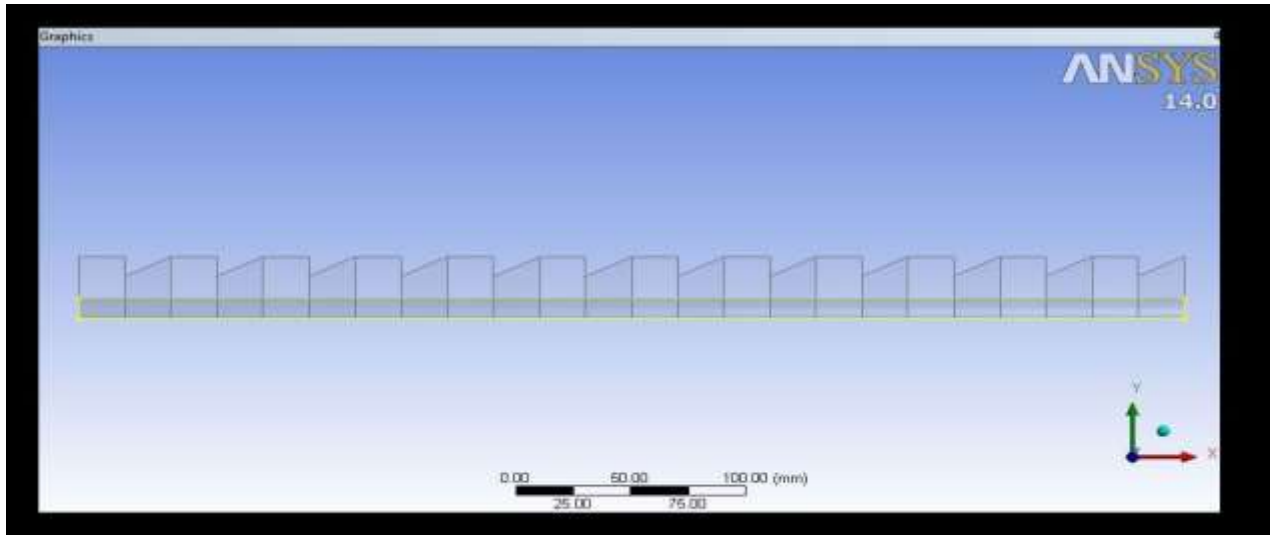


Fig 3 A fin geometry with the heating coil

2) Meshing of the geometry:

Here, we define that the second step of the module, that was the Meshing. In this meshing we define the all the specification of geometry. This step is the most important step of the analysis. By use of this step we have particularly define that the flow separation of Air and how to heat have been transfer in the geometry.

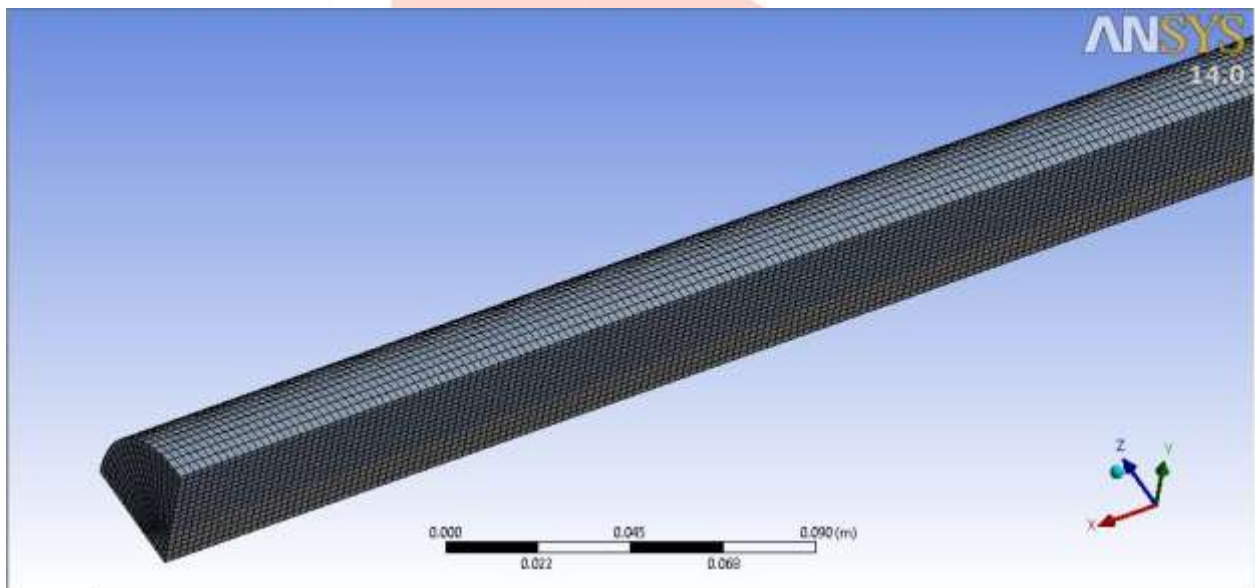


Fig 4 The flow separation of Air and how to heat have been transfer

Data	6 mm	8 mm
Nodes	34705	78103
Element	29532	70472
Growth rate	1.2	1.2

**III 6 MM STRAIGHT FIN**

Boundary Condition:-

Inlet: velocity Inlet

Velocity Magnitude: 0.01 m/s

Turbulence kinetic energy (k and epsilon): 0.01m/s

Turbulence Dissipation rate: 0.1 m/s

Outlet: pressure outlet

Backward turbulence K.E: 0.01 m/s

Backward turbulence K.E: 0.1 m/s

Source term: 1 (constant)

Heater:  $200000 \text{ W/m}^3$

Here this is a boundary condition of this geometry.

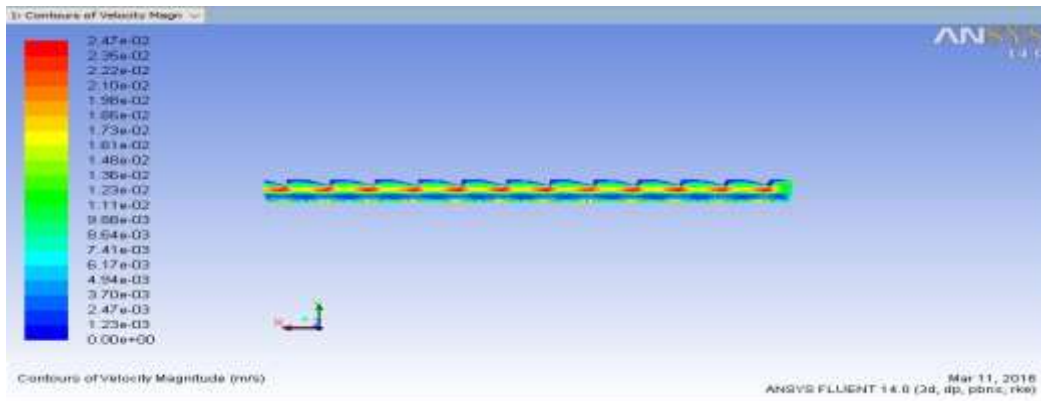


Fig 5 6mm Straight Fin – Velocity

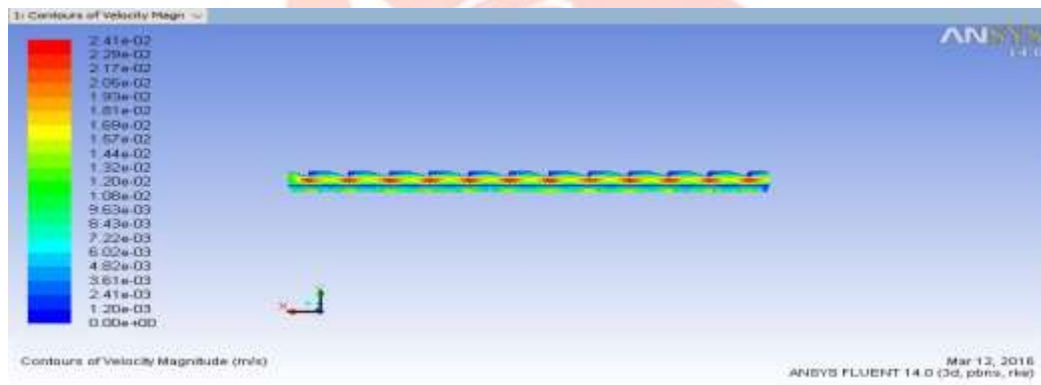


Fig 6 6mm Backward Fin – Velocity

	6mm Straight Fin		6mm Backward Fin	
Velocity(m/s)	0	0.02469	0	0.02407

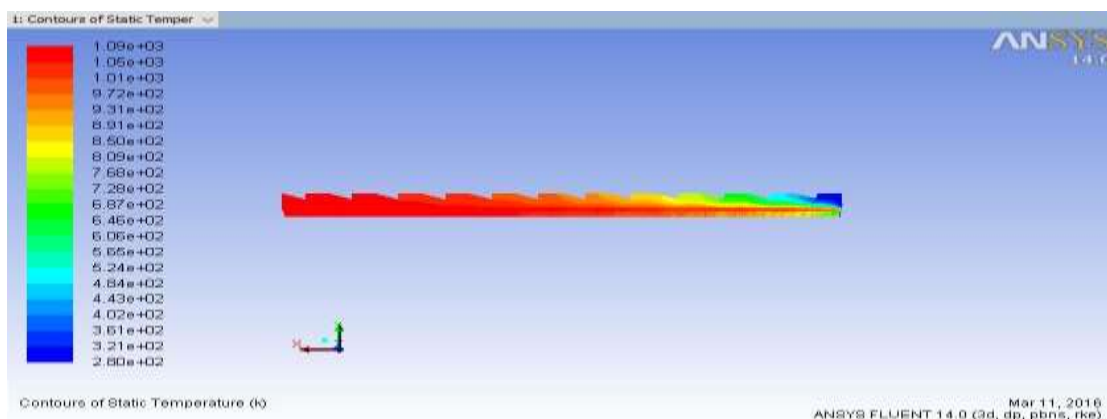


Fig 7 6mm Straight Fin – Temperature

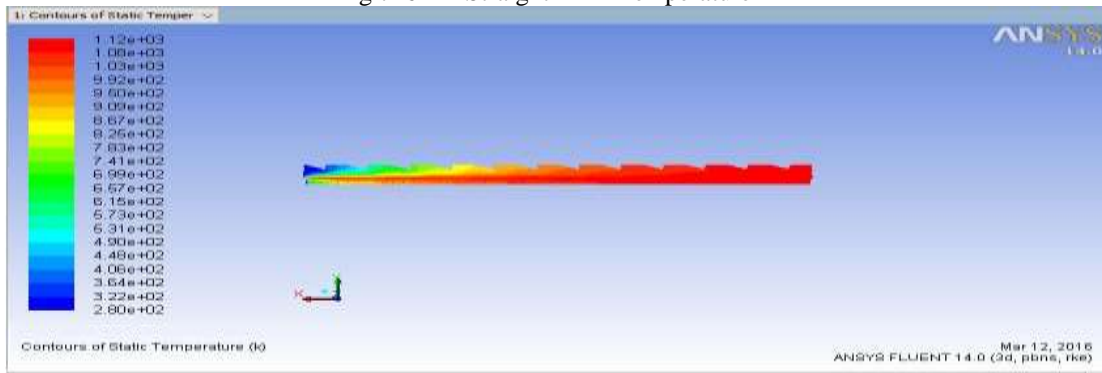


Fig 8 6mm Backward Fin – Temperature

	6mm Straight Fin		6mm Backward Fin	
Temperature (K)	280	1094.5	280	1118.04

**IV 8 MM STRAIGHT FIN**

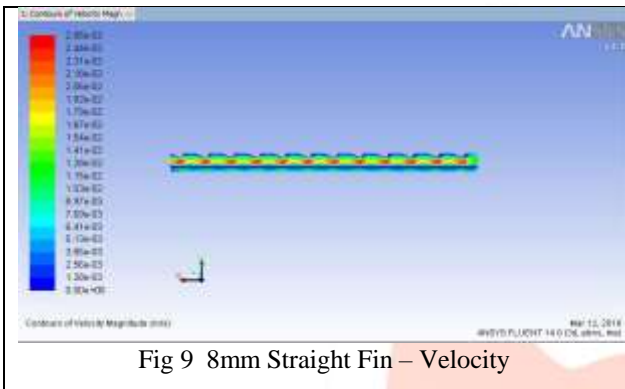


Fig 9 8mm Straight Fin – Velocity

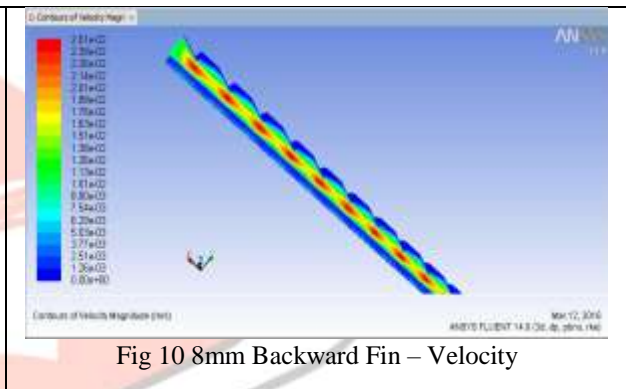


Fig 10 8mm Backward Fin – Velocity

	8mm Straight Fin		8mm Backward Fin	
Velocity (m/s)	0	0.0256	0	0.02514

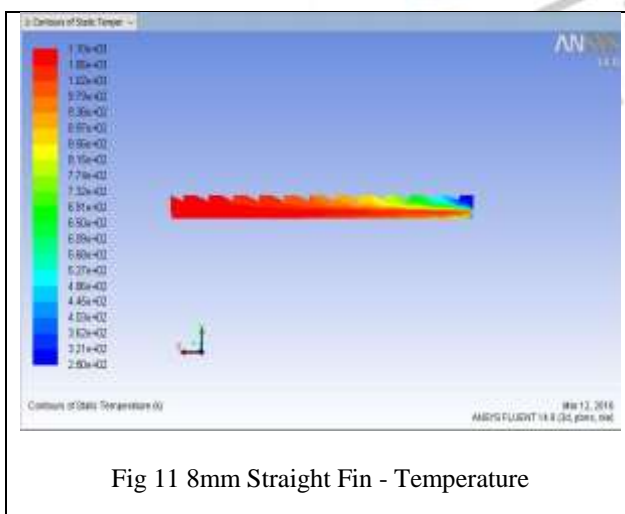


Fig 11 8mm Straight Fin - Temperature

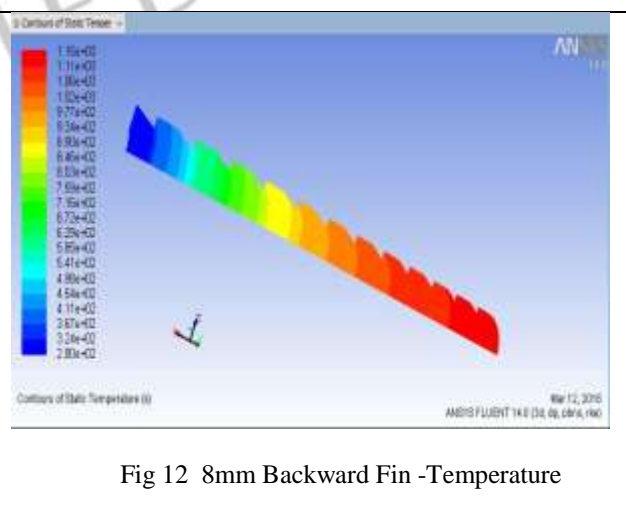


Fig 12 8mm Backward Fin -Temperature

	8mm Straight Fin		8mm Backward Fin	
Temperature (K)	280	1102.52	280	1151.35

## CONCLUSION

We have been compared the CFD analysis of the cylinder with Straight and Backward fin. Here we use the mainly two types of fin height. 6mm, 8mm. In this analysis we have found that the better heat transfer compared the backward fin to the straight fin. More velocity turbulence creates in the backward fin. When we increase the fin height then heat transfer also have been increased, but in the particular height they have been decreased in the backward.

## REFERENCES

- [1] Thermal performance assessment of turbulent channel flows over different shaped ribs, Pongjet Promvonge ,Chinaruk Thianpong, International Communications in Heat and Mass Transfer 35 (2008)
- [2] Turbulent flow and heat transfer enhancement in rectangular channels with novel cylindrical grooves, Jian Liu ,GongnanXie, Terrence W. Simon, International Journal of Heat and Mass Transfer 81 (2015).
- [3] Convective Heat Transfer Enhancement of a Rectangular Flat Plate by an Impinging Jet in Cross Flow, LI Guoneng, ZHENG Youqu, HU Guilin, ZHANG Zhiguo, Chinese Journal of Chemical Engineering, 22(5), 2014.
- [4] CFD and experimental studies on heat transfer enhancement in an air cooler equipped with different tube inserts, S.R. Shabaniyan , M. Rahimi , M. Shahhosseini , A.A. Alsairafi, International Communications in Heat and Mass Transfer 38 (2011).
- [5] Milne-Thomson, L.M. (1973). Theoretical Aerodynamics. Dover Publications. ISBN 0-486-61980-X.
- [6] Richardson, L. F.; Chapman, S. (1965). Weather prediction by numerical process. Dover Publications
- [7] Hunt (1998). "Lewis Fry Richardson and his contributions to mathematics, meteorology, and models of conflict". Annual Review of Fluid Mechanics 30. Bibcode:1998AnRFM..30D..13H. doi:10.1146/annurev.fluid.30.1.0
- [8] Harlow, F. H. (2004). "Fluid dynamics in Group T-3 Los Alamos National Laboratory:(LA-UR-03-3852)". Journal of Computational Physics (Elsevier) 195 (2): 414–433. Bibcode:2004JCoPh.195..414H. doi:10.1016/j.jcp.2003.09.031
- [9] F.H. Harlow (1955). "A Machine Calculation Method for Hydrodynamic Problems". Los Alamos Scientific Laboratory report LAMS-1956.
- [10] Carmichael, R. and Erickson, L.L., "PAN AIR - A Higher Order Panel Method for Predicting Subsonic or Supersonic Linear Potential Flows About Arbitrary Configurations," AIAA paper 81-1255, presented at the AIAA 14th Fluid and Plasma Dynamics Conference, Palo Alto California, June 1981.
- [11] Youngren, H.H., Bouchard, E.E., Coopersmith, R.M. and Miranda, L.R., "Comparison of Panel Method Formulations and its Influence on the Development of QUADPAN, an Advanced Low Order Method," AIAA paper 83-1827, presented at the AIAA Applied Aerodynamics Conference, Danvers, Massachusetts, July 1983.