# Simulation of Heat Transfer through Fins Using Computational Fluid Dynamics

**Kallappa Kudagi**[1]**, V K Pravin**[2]
[1, 2] PDA college of Engineering

***Abstract-*** *Performance of various devices are based on heat transfer and widely used in the many industries, especially in power distribution sector (transformers), Automobile sector (engine cooling), Power Plant Sector, electric components, space industry etc. One of the useful methods to take away heat transfer from surface area of thermal device was extended surface or fins. Pin fin is suitable for numerous applications including heat transfer removal from air cooled I C engines, Electrical Small Transfers etc.*

*This study presents the results of computational numerical analysis of air flow and heat transfer in a considering two different morphology pin fins. A numerical study using Ansys fluent was conducted to find the optimum pin shape based on minimum pressure drop and maximizing the heat transfer across the Fin bodies. The results indicate that the trapezoidal shaped pin fins show improved results on the basis of heat transfer and pressure drop by comparing other fins. The reason behind the improvement in heat transfer by trapezoidal shape pin fin was increased wetted surface area and delay in thermal flow separation from trapezoidal shape pin fin. Therefore from the trapezoidal fin the maximum heat transfer is obtained.*

***Keywords***- CFD, Continuum, Fluent, Optimization, Simulation, Turbulence

## I. INTRODUCTION

Heat transfer is the transition of thermal energy from a hotter mass to a cooler mass. When an object is at a different temperature than its surroundings or another object, transfer of thermal energy, also known as heat flow, or heat exchange, occurs in such a way that the body and the surroundings reach thermal equilibrium; this means that they are at the same temperature. Heat transfer always occurs from a higher-temperature object to a cooler-temperature one as described by the second law of thermodynamics or the Clausius statement. Where there is a temperature difference between objects in proximity, heat transfer between them can never be stopped; it can only be slowed.

### Extended Surfaces: Fins

Extended surfaces are often used to reduce the thermal resistance at a surface and thereby increase the heat transfer rate from the surface to the adjacent fluid. They are also referred to as fins. It is also not possible to increase the temperature difference (T1 – Tf) because temperatures in the system are fixed by other constraints. The only option is to increase the area. This is done by using extended surfaces or fins.

The use of fins is very common and they are fabricated in a variety of shapes. A familiar application is the use of circumferential fins around the cylinder of motorcycle engine. This geometry is shown in the figure for two shapes. In the figures shown below, one show the fins having a rectangular cross section, while in the other figure shows them having a triangular cross section.

Fins are most commonly used in heat exchanging devices such as radiators in cars and heat exchangers in power plants. They are also used in newer technology such as hydrogen fuel cells. Nature has also taken advantage of the phenomena of fins. The ears of jackrabbits and Fennec Foxes act as fins to release heat from the blood that flows through them. Aluminum Fins take 30% & Copper fins takes 35% less time for cooling from 350 degree centigrade to 100 degree

## II. LITERATURE REVIEW

This section presents a brief look at the research that has been conducted prior to the writing of this report. This literature review includes a discussion of current state-of-the-art issues and optimization techniques involved with thermal management in compact heat exchangers and e-cooling.

One of the most referenced works is that of C. L. Chapman and Seri Lee paper [5]. They carried out comparative thermal tests using aluminumheat exchangers made with extruded Fin , cross-cut rectangular s and elliptical shaped s in low air flow environments. They developed an elliptical Pin fin heat exchanger with specific design parameters, maintaining large exposed surface area for heat transfer and minimizing vortex flow by incorporating an airfoil design. The approach taken in the paper was to compare this elliptical shapedheat exchanger with a conventional extruded Pin fin heat exchanger of equal volume. They used thermal resistance and amount of flow bypass terms to measure the effects of different thermal conductivity, flow characteristics and pressure drop onheat exchanger performance. Figures 1.5 and 1.6 show the schematic diagram of differentheat exchangers that they used and their experimental test set-up.They found that there was 40% more air flowing through the rectangular design, yet the thermal resistances were virtually equal and the elliptical Fin enhanced the heat transfer.

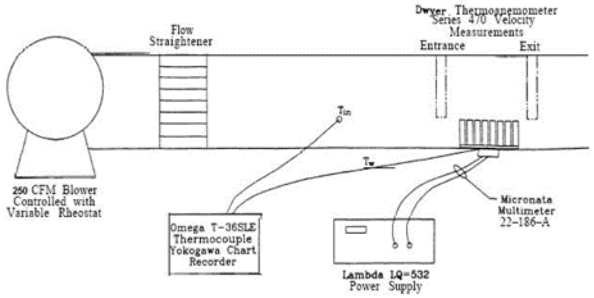

Figure 1. Schematic diagrams of differentheat exchangers for tests [5]

In internally Fin ned tubes the flow can also be turbulent. The turbulent flow was analysed by Patankar et al. [29], Webb and Scott [30], Kim and Webb [31]. The results found were compared with available experimental data.

Patankar et al. [29] used a mixing length model for analysing an internally Fin ned tube. For the analysis the Nusselt number and friction factor were derived. The results also showed that the Fin s were a better transfer surface than the tube wall.

An internally Fin ned tube was analysed for three design cases by Webb and Scott [30]. The design cases were:

- Reduced tube material volume for equal pum g power and heat transfer.
- Increased heat transfer for equal pum g power and length of heat exchanger tubing.
- Reduced pum g power for equal heat transfer and length of heat exchanger tubing.

The results showed that although material savings were found using axial internal Fin s, much better savings were obtained when the Fin s had a helix angle (twisted).

A comparison with the empirical Carnavos correlations was made in the analytical model used by Kim and Webb [31] in their article about axial internal Fin tubes. The friction factor and heat transfer equations were derived and predicted the Carnavos friction data within 10% and the Carnavos heat transfer data within 15% accuracy.

## SIMULATION

### What is Computer Simulation???

The simulation of an industrial system on computers involving mathematical representation of the physical process undergone by the various component of the system, by a set of governing equations (usually differential equations) transformed to difference equations which are in turn solved as a set of simultaneous algebraic equations.

### Advantages of Computer Simulation:-

Here are some of the important advantages of computer simulation or also known as numerical simulation:-

- It is possible to see simultaneously the effect of various parameters and variables on the behavior of the system since the speed of computing is very high. To study the same in an experimental setup is not only difficult and time consuming but in many cases, may be impossible.
- It is much cheaper than setting up big experiments or building prototypes of physical systems.
- Numerical modeling is versatile. A large variety of problems with different level of complexity can be simulated on a computer.
- A numerical simulation allows models and hence physical understanding of the problem to be improved. It is similar to conducting experiments.
- In some cases, it is the only feasible substitute foe experiments, for example, modeling loss of coolant accident (LOCA) in nuclear reactors, numerical simulation of spread of fire in a building and modeling of incineration of hazardous waste.

However, it is to be emphasized that not every problem can be solved by computer simulation. Experiments are still required to get an insight into the phenomena that are not well understood and also to check the validity of the results of computer simulation of complex problems.

### Application of Fluid Flow and Heat Transfer

Fluid flow and heat transfer plays a very important role in nature, living organism and in a variety of practical situations. More often than not, flow and heat transfer are coupled and rarely an engineer solves a problem of either pure fluid flow or pure heat transfer. The various applications of fluid flow and heat transfer are:-

- Power production e.g. thermal, nuclear, hydraulic, wind, and solar plants.
- Heating and Air-conditioning of buildings.
- Chemical and metallurgical industries, e.g. furnace, heat exchangers, condensers and reactors.
- Design of an I.C engine.
- Optimization of heat transfer from cooling fins
- Aircraft and spacecraft.
- Design of electrical machinery and electronic circuits.
- Cooling of computers.
- Weather prediction and environment pollution.
- Material processing such as solidification and melting, metal cutting, welding, rolling, extrusion, plastics and food processing in screw extruders and laser cutting of materials.
- Oil exploration.
- Production of chemicals such as cement and aluminium oxide.
- Drying.
- Processing of solid and liquid waste.
- Bio-heat transfer

## A COMPARATIVE STUDY OF EXPERIMENTAL, ANALYTICAL, AND NUMERICAL METHODS:-

1. Experimental Method: - Experimental methods are used to obtain reliable information about the physical process that are not well understood example combustion and turbulence. The major disadvantage this method are high cost, measurements difficulties and probe errors. Often, small scale models do not always simulate all the features if the full scale set up. The advantage of this method is that it is more realistic.
2. Analytical Method :- Analytical methods or methods of classic mathematics are used to obtain the solution of a mathematical model consisting of a set of differential equation which represents a physical process within the limit of assumption made. Only a handful of analytical solutions are available in heat transfer because of the complexity involved in the handling of the complex boundary and non-linearities in differential equation or boundary conditions. Furthermore, the analytical solution contains infinite series, special functions, transcendental equations foe Eigen values etc. thus the numerical evaluation becomes quite cumbersome.
3. Numerical Method: - The major advantages of numerical solution are its abilities to handle complex geometry and non linearity in the governing equation and boundary condition. Other advantages of numerical method are briefly described below :-

- Low cost
- High speed
- Complete information
- Ability to simulate realistic conditions
- Ability to simulate ideal conditions

The disadvantage of the numerical predictions is the associated truncation error and round off error and the difficulties in simulating the complicated boundary condition.

## METHODS OF DISCRETIZATION

### A. The Finite Difference Method

Because of the importance of the heat equation to a wide variety of fields, there are many analytical solutions of that equation for a wide variety of initial and boundary conditions. However, one very often runs into a problem whose particular conditions have no analytical solution, or where the analytical solutions even more difficult to implement than a suitably accurate numerical solution. Here we will discuss one particular method for analytical solution of partial differential equations called the finite difference method.

The usual procedure for deriving the finite difference equations consists of of approximating derivatives in the differential equation via the truncated Taylor series. The method includes the assumption that the variation of the unknown to be computed is somewhat like a polynomial in x, y or z so that higher derivatives are unimportant. The great popularity of the finite difference method is mainly due to their straight-forwardness and relative simplicity by which a newcomer in the field is able to obtain solutions of a simple problem.

### Disadvantages of Finite Difference Method

Several shortcomings and limitations of the FDM were discovered when researchers tried to solve problems with increasing degree of physical complexity such as :

1) Flow at higher Reynolds's numbers.
2) Flows around arbitrarily shaped bodies.
3) Strong time-dependant flows.

This led to the development of superior methods, particularly in the area where finite difference method have some disadvantage.

These Methods can be divided into two main categories
a) Finite Element Method.
b) Spectral Method.

### B. Finite Element Method:-

Finite Element Method (FEM) basically seeks solution at discrete spatial regions called elements by assuming that the governing differential equations apply to the continuum within each element. Their introduction and ready acceptance was due to the relative ease by which flow problems with complicated boundary shapes can be modelled, especially when compared with finite difference method. However disadvantage of FEM arises from the fact that more complicated matrix operation are required to solve the resulting system of equations. Furthermore, meaningful variational formulations are difficult to obtain for high Reynolds number flows. Hence, variation principle based FEM is limited to solutions of creeping flow and heat conduction problems.

Glarkin’s weighted residual method, which is also another FEM is a powerful method and circumvents the difficulties faced by variational calculus based FEM. Much research is in progress in the use of this method.

### C. Spectral Method:-

Spectral methods are generally much more accurate than simple first or second order finite difference methods. In this method, the approximation is based on expansions of independent variables into finite truncated series of smooth functions. A disadvantage of the spectral method is their relative complexity in comparisons with standard finite difference methods. Also the implementation of complex boundary conditions appears to be a frequent source of considerable difficulty.

### D. Control Volume Formulation:-

In this method, the calculation domain is divided into number of non overlapping control volumes such that there is one control volume surrounding each grid point. The differential equation is integrated over each control volume.

The major advantage of this method is its physical soundness. The disadvantage is that it is not as straightforward as finite difference method.

### FINITE DIFFERENCE METHOD

There are three numerical techniques:

1. Finite –difference (numerical solution)
2. Finite –element
3. Finite volume (in CFD)

This project uses finite-difference method to get temperature distribution, which is then verified using finite-Slice the fins at right angle to the length, defining ΔX=L/M
For interior elements the node is centre having width Δx , while for two end elements the node is at the edge having width Δx/2.

## III. PROBLEM FORMULATION

### ONE DIMENSIONAL STEADY STATE PROBLEM:

Considering the two-dimensional steady state heat conduction in an isolated rectangular horizontal fin as shown figure below. The base temperature is maintained at T=T0 and the tip of the fin is insulated. The fin is exposed to a convective environment (neglecting the heat transfer by radiation from the tip of the fin) which is at T∞ (T∞<T0).

The average heat transfer coefficient of the fin to the ambient is ‘h’. the length of the fin is ‘L’ and the coordinate axis begins at the base of the fin. The one dimentionality arises from the fact that the thickness of the fin is much small as compared to its length and width can be considered either too long or the sides to the fin to be insulated.

### GOVERNING DIFFERENTIAL EQUATION:

The energy equation for the fin at the steady state (assuming constant k) is

$$\frac{d^2T}{dx^2} - \frac{hP}{kA}(T - T_\infty) = 0$$

Where
P = Perimeter of the fin
A= Cross sectional area of the fin

**Boundary conditions**:

The governing equation is a linear, second order ordinary differential equation, two boundary conditions are needed to completely describe this problem( which is a boundary value problem).

Boundary conditions are:

B.C.1: At x=0, $T=T_0$
B.C.2: At x=L, $\frac{dT}{dx} = 0$

**Discretization:-**

The governing equation in non-dimensional form is discretized at any interior grid point ***i*** using central difference for $\frac{d^2\theta}{dx^2}$ as follows

$$\left(\frac{d^2\theta}{dx^2}\right)_i - \{(mL)^2\theta\}_i = 0$$

$$\frac{\theta_{i-1} - D\theta_i + \theta_{i+1}}{(\Delta x)^2} - m^2L^2\Theta_i = 0$$

$\Theta_{i-1} - D\Theta_i + \Theta_{i+1} = 0$ .........(A)
i=1,2,3.......

Where $D = 2 + (mL)^2(\Delta X)^2$
$\Theta_{i-1} - 2\Theta_i + \Theta_{i+1} - (mL)^2\Theta_i\,(\Delta X)^2 = 0$
$\Theta_{i-1} - [2 + (mL)^2(\Delta X)2]\Theta_i + \Theta_{i+1} = 0$

**Handling of the boundary condition:-**

At x = L i.e. i=M
The above equation reduces to
$\Theta_{M-1} - D\Theta_M + \Theta_{M+1} = 0$

Hence $\Theta_{M+1}$ represents a fictious temperature at point M+1 which lies outside the computational domain. Hence here the image point technique is used.

**How to obtain a solution in ANSYS?**

A solution can be obtained by following these nine steps:
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

Results were plotted using different materials by simulating them in the ANSYS package. The results show obtained were compared with experimental data.

## IV. RESULTS AND DISCUSSIONS

**Results obtained from ANSYS**

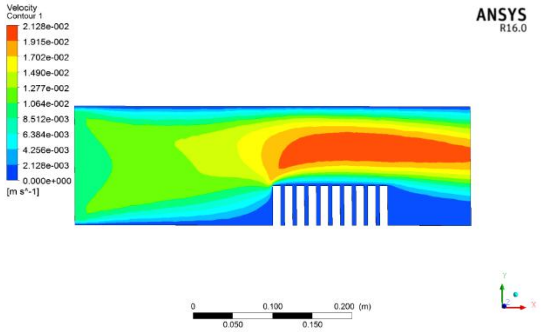

Figure 2. Velocity Contour With The Velocity of Re=100

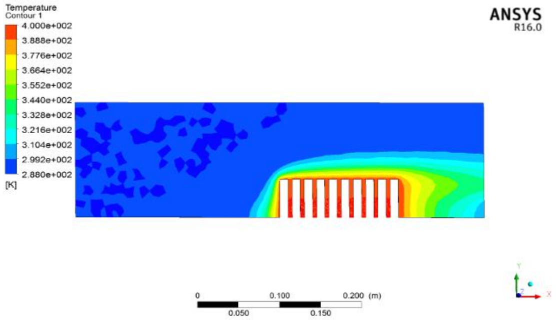

Figure 3. Temperature Contour Of Flat Fin With Re=100

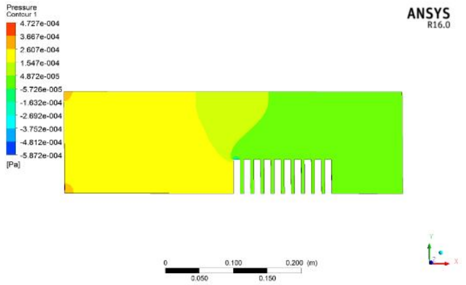

Fig 4. Pressure Contour Of Flat Fin with Re=100

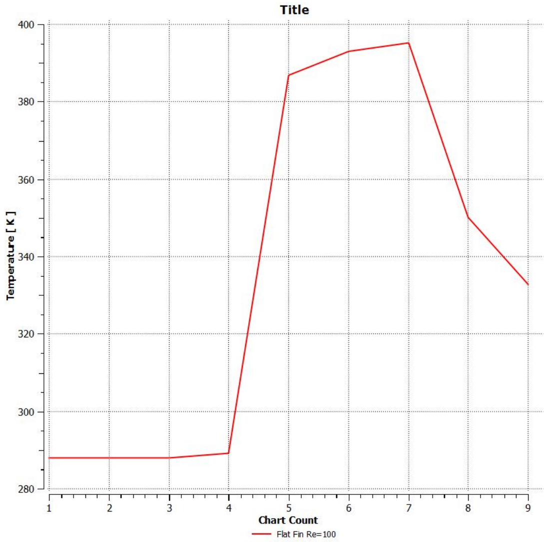

Figure 4. Chart 1 Temperature Vs Chart Count Re=100

## Flat Fin Results Re=200

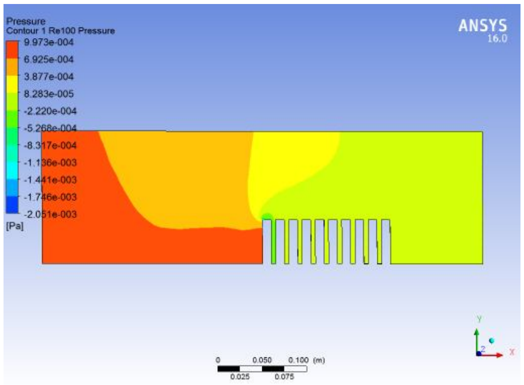

Figure 5. Pressure Contour Of Flat Fin with Re=200

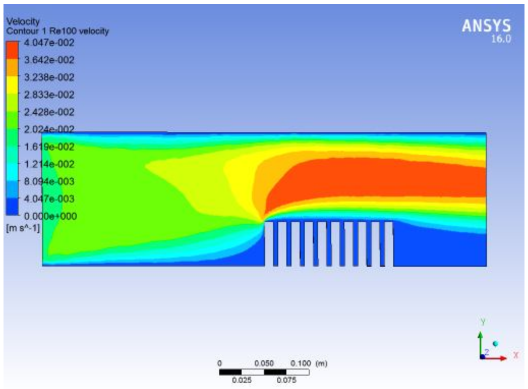

Figure 6. Velocity Contour With The Velocity of Re=100

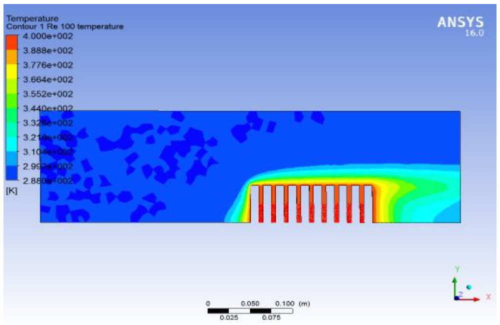

Figure 7. Temperature Contour Of Flat Fin With Re=100

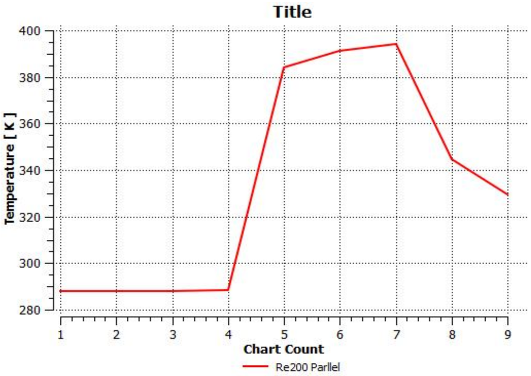

Figure 8. Chart 1. Temperature Vs Chart Count Re=200

## Results of The Trapezoidal Fin With Re=100 and Re=200

### Mesh Report

Table 1. Mesh Information for Trapi Re 100

| Domain | Nodes | Elements |
|---|---|---|
| trapezoidal_fin | 80040 | 73621 |

### Physics Report

Table 2. Domain Physics for Trapi Re 100

| Domain - trapezoidal_fin | |
|---|---|
| Type | cell |

Table 3. Boundary Physics for Trapi Re 100

| Domain | Boundaries | |
|---|---|---|
| trapezoidal_fin | **Boundary - fin_wall** | |
| | Type | WALL |
| | **Boundary - inlet** | |
| | Type | VELOCITY-INLET |
| | **Boundary - outlet** | |
| | Type | PRESSURE-OUTLET |
| | **Boundary - symmetry 1** | |
| | Type | SYMMETRY |
| | **Boundary - symmetry 2** | |
| | Type | SYMMETRY |
| | **Boundary - wall** | |
| | Type | WALL |

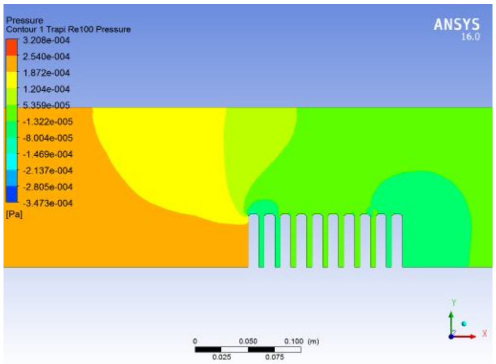

Figure 9. Pressure Contour Of Trapezoidal

### Fin With Re=100

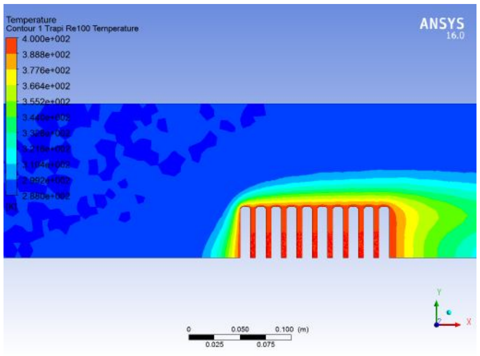

Figure 10. Temperature Contour Of Trapezoidal Fin With Re=100

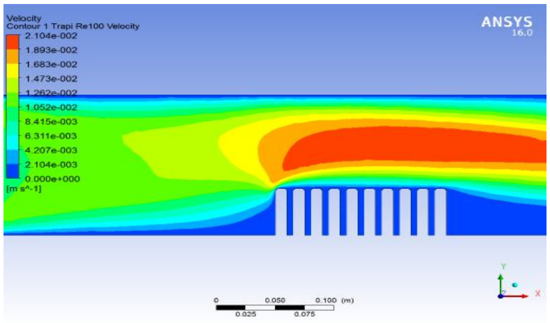

Figure 11. Velocity Contour Of trapezoidal fin With Re=100

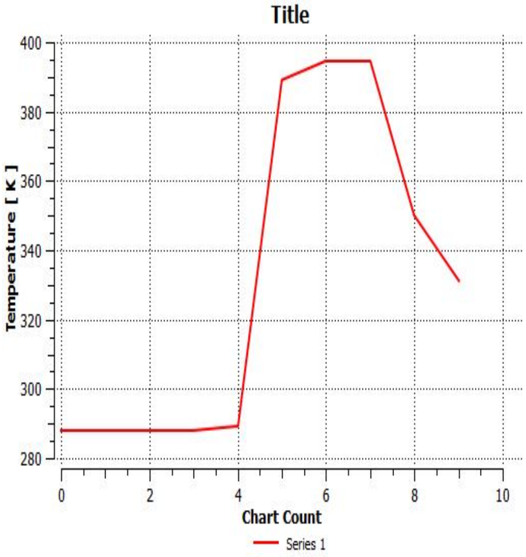

Figure 12. Chart 1.Chart Count Vs Temperature Trapezoidal Fin With Re =100

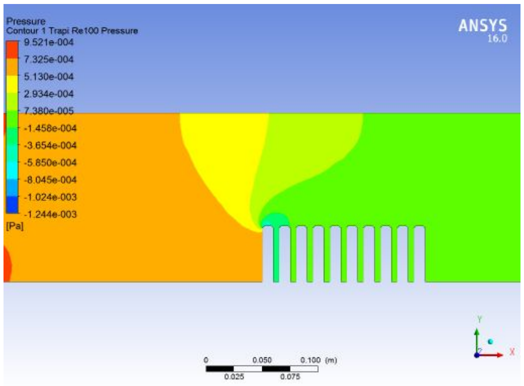

Figure 13. Results of Trapezoidal Fin With Re=200

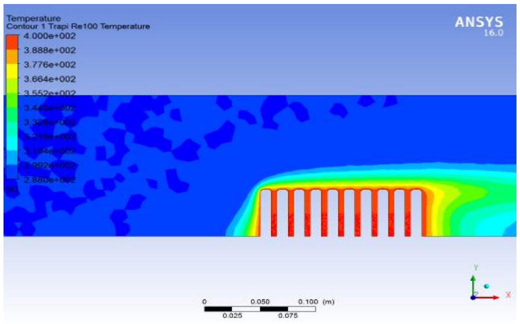

Figure 14. Temperature Contour Of Trapezoidal Fin With Re=200

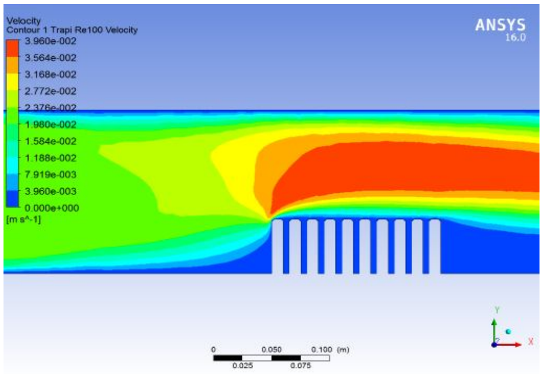

Figure 15. Velocity Contour Of trapezoidal fin With Re=100

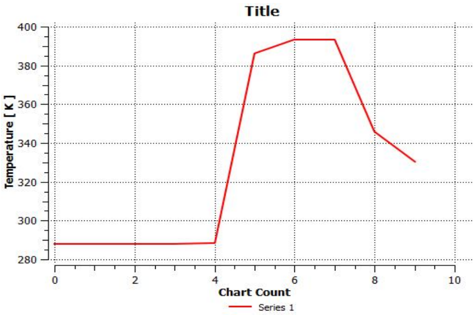

Figure 16. Chart 1. Chart Count Vs Temperature Trapezoidal Fin With Re =100

## V. CONCLUSIONS AND FURTHER SCOPE

A numerical was developed to solve the one-dimensional differential heat equation and plot the variation of temperature along the entire length of the pin fin of Different Reynolds Number.

The results obtain thought the Computational awere verified by using the CFD tool of ANSYS. Thus a comparative study was done for each of the Reynolds Number to see the temperature variation along the length of the pin fin in different cases.

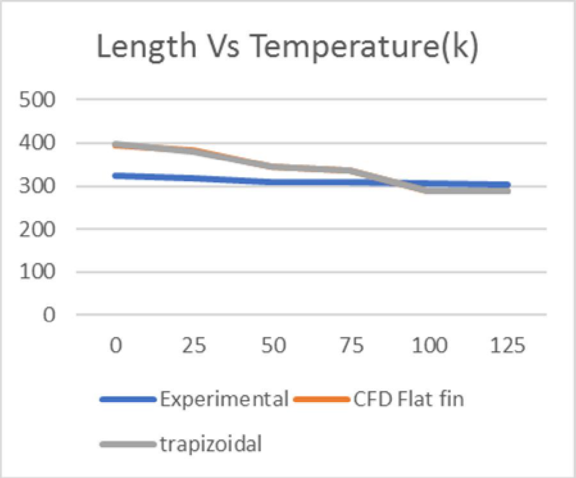

Figure 17.

With the above analysis we could see the difference in heat transfer of different Reynolds Number in Two dimension and could predict the most suitable Velocity for this purpose around the surface of the fin.

## REFERENCES

[1] HEAT TRANSFER BY SP SUKHATME

[2] GETTING STARTED WITH MATLAB BY RUDRA PRATAP

[3] HEISLER,M.P 1947.TEMPARATURE CHARTS FOR INDUCTION AND CONSTANT TEMPARATURE.

[4] SCHNEIDER,P.J,1955 CONDUCTION HEAT TRANSFER.READING,MASS:ADDISON-WESLEY PUBLISHING CO.

[5] GARDNER,KA,1945.EFFICIENCY OF EXTENDED SURFACES.TRANS.ASME,VOL 67:621

[6] LEON ,OCTAVIO.,MEY,GILBERT DE., JAN VIERENDEELS,ERIK DICK ..”COMPARISON BETWEEN THE STANDARD AND STAGGERED LAYOUT FOR COOLING FINS IN FREE CONVECTION COOLING ”,

[7] ASME SEPTEMBER 2003 BEBNIA ,MASUD,NAKAYAMA,WATARU.”CFD SIMULATIONS OF HEAT TRANSFER FROM A HEATED MODULE IN AN AIR STREAM”

[8] NAKAYAMA ,MATSOUKIS, HACHO,YAKIMA “A NEW ROLE OF CFD SIMULATION IN THERMAL DESIGN OF COMPACT EQUIPMENT :APPLICATON OF THE BUILD-UP APPROACH TO THERMAL ANALYSIS OF BENCH MARK MODEL”