R V COLLEGE OF ENGINEERING®, BENGALURU-59

(Autonomous Institution Affiliated to VTU, BELAGAVI)

Project Report 2018-19

Project Title:

MATRIX FIN BASED TRANSPLY EFFUSION COOLING SYSTEM

Submitted in partial fulfillment for the award of degree of

Bachelor of Engineering

in

Mechanical Engineering

Submitted by

SANGAMESH GUDDA	**1RV15ME094**
SHREYAS ASHWIN SUNDER	**1RV15ME104**
THOUSIF ULLA KHAN	**1RV15ME114**

Under the guidance of

Prof. Keshav M

Department of Mechanical Engineering

R V College of Engineering®

Bengaluru – 560059

R V COLLEGE OF ENGINEERING®, BENGALURU – 560059

(Autonomous Institution Affiliated to VTU, Belagavi)

DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE

This is to certify that the project work titled **Matrix Fin Based Transply Effusion Cooling System** carried out by **Sangamesh Gudda (1RV15ME094), Shreyas Ashwin Sunder (1RV15ME104), Thousif Ulla Khan (1RV15ME114),** in partial fulfillment for the award of degree of **Bachelor of Engineering** in **Mechanical Engineering** of the Visvesvaraya Technological University, Belagavi during the year **2018-2019**. It is certified that all corrections/suggestions indicated for the internal Assessment have been incorporated in the report. The project report has been approved as it satisfies the academic requirements in respect of Project Work (12ME81) prescribed by the institution for the said degree.

Prof. Keshav M	**Dr. M. Krishna**	**Dr. K. N. Subramanya**
Assistant Professor,	Professor and HOD	Principal
Dept. of Mechanical Engineering	Dept. of Mechanical Engineering	R V College of Engineering®,
		Bengaluru – 560059.

External Viva:

Name of the Examiner **Signature with date**

1.

2.

R V COLLEGE OF ENGINEERING®, BENGALURU – 560059

(Autonomous Institution Affiliated to VTU, Belagavi)

DEPARTMENT OF MECHANICAL ENGINEERING

DECLARATION

We, **Sangamesh Gudda (1RV15ME094), Shreyas Ashwin Sunder (1RV15ME104), Thousif Ulla Khan (1RV15ME114)** students of Eight Semester B.E, Mechanical Engineering, R. V. College of Engineering®, Bengaluru hereby declare that the project titled **Matrix Fin Based Transply Effusion Cooling System** is carried out by us and submitted in partial fulfilment for the award of the Degree of Bachelor of Engineering in Mechanical Engineering for the academic year 2018-2019.

Further we declare that the content of the project report has not been submitted previously by anybody for the award of any degree or diploma to any other University.

We also declare that any Intellectual property rights generated out of this project carried out at RVCE will be property of R. V College of Engineering®, Bengaluru and we will be only one of the co-authors of the same.

Place: Bengaluru

Date:

1. Sangamesh Gudda

2. Shreyas Ashwin S

3. Thousif Ulla Khan

ACKNOWLEDGEMENT

The successful completion of this project work was made possible through valuable contribution of number of people. To say thank you to all of them is not even enough to express my gratitude.

Our first debt of gratitude must go to our guide **Prof. S Srinivasan**, Professor of Dept. of Aerospace Engineering for his valuable and inspiring guidance throughout the course of this work.

We would also like to thank **Prof. Keshav M**, Professor of Dept. of Mechanical Engineering for his advice and encouragement throughout the project work.

We also thank department of Mechanical Engineering and IEM for allowing us to manufacture matrix fins in the workshop. They have driven us to an everlasting debt of gratitude through their valuable guidance and support in bringing out this work.

We take this platform to thank Head of the Department**, Dr. M Krishna,** for his valuable suggestions and support to carry out the entire project work in the facilities of the Department for their advice and encouragement throughout the project work.

We thank **Dr. K N Subramanya**, Principal, RVCE, for being supportive throughout our course. Our special thanks go to our parents for their blessings have been great source of inspiration for us.

ABSTRACT

The aviation market is expected to achieve a compound annual growth rate of 3% over the forecast period 2019 to 2024 and the manufacturers are looking for efficient jet engines to reduce the fuel consumption which in turn reduces the cost of the travel. The transpiration cooling method has many inherent disadvantages due to its micro sized holes and complexity of manufacturing. In this context, many research works have focused on developing a cooling system which can produce the same effect as transpiration cooling but lesser number of inherent problems. The main objective of this project is to increase the cooling effectiveness of the transply effusion cooling system using matrix fins.

The geometric parameters of the matrix fin are optimized using MATLAB software. The optimized parameters are height of the fin is 0.7mm, thickness is 0.5mm, distance between fin array is 0.865mm, angle between top and bottom fin array is 45^0. These parameters are used to design the matrix fin within the effusion cooling system. The design of the effusion cooling system was taken from the standard designs which are currently in use. The geometric parameters of the holes present in the effusion cooling system are diameter 0.5mm, streamwise pitch 1.03, spanwise pitch 0.91mm. These optimized designs are used to conduct the thermal analysis in ANSYS fluent.

The cooling effectiveness of the matrix fin based transply effusion cooling system was calculated using the temperature values that are obtained from the analysis for different conditions of inlet and outlet fluid temperature. For hot gas inlet temperature 500K the values obtained are, maximum temperature 383.315K, minimum temperature 331.22K and the average temperature 356.011K. Similar analysis is conducted on the effusion cooling system with inlet fluid temperature 500K the values obtained are, maximum temperature 425.135K, minimum temperature 334.105K and average temperature equal to 384.966K. To see the effect of increase in the temperature of the combustion chamber the hot fluid inlet temperature is increased to 850K the values obtained are, maximum temperature 417.14K, minimum temperature 319.829K and average temperature 357.612K. The outcome of the analysis was that the cooling system with the matrix fins had better cooling effectiveness than the cooling system without the matrix fins.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 Background

In the current situation turbofan engines are used to achieve lower fuel consumption, increase in the fuel price forcing the manufacturers to design highly efficient jet engines, and many companies are focused on developing the more economical cooling devices for combustor liner. The efficiency of the jet engine can be improved by the use of higher compression ratios and turbine entry temperature. Without changing the design of the combustion chamber these conditions require increased liner cooling flows and hence a decrease in quantity of air available for pattern factor control. Since there is little hope for dramatic improvement in material properties, apart for development of ceramic composites and thermal barrier coatings which are able to withstand high temperature but not enough to withstand the temperature of the combustion chamber; thus, a lot of interest is given in the field of efficient cooling of combustion liner, and many companies are focused on developing the more economical cooling devices for combustor liner. Various methods are used to maintain the temperature of combustion liner within the safe range, most of these methods involve film cooling technology where the boundary layer formation characteristic of the fluid is used to prevent hot fluid inside the combustion chamber coming in direct contact with the inner wall of the liner wall. Based on the design and fluid interaction with the liner wall these are classified between forced convection and transpiration cooling system.

In recent years, continuous increase in combustor operating temperature has emphasised the need to achieve efficient cooling technologies that utilises very less coolant air and yet achieve acceptable wall temperature of the combustion liner. The latest cooling technology being film cooling technology where annulus air is used as coolant, the annulus air is made to pass through the porous or multi holed combustion liner where a boundary layer is formed inner side of the liner which protects the wall from coming in direct contact with the combustion zone hot environment. The success of the film cooling technology relies on providing a protective film of cooling air between the wall and the hot combustion gases by injecting the air along the inner surface of the liner. Combustor is designed in such a way that many numbers of coolant holes are provided along the length of the combustor to overcome the destruction of the boundary layer due to the turbulent mixing of the combustion gases.

1.2 Global scenario-global market/growth rate

The aviation industry is expected to achieve a gross annual growth rate of more than 3% over next five years as shown in the **Fig 1.1**

Fig 1.1 Comparison of CAGR of Aviation Market

- With developing countries advancing to strengthen their defense structure and military bases, it increases the demand for military aircrafts and technologies.
- Increasing population in countries like India and China demands for increased air travel, lower air fares, growing living standards of middle-class in large.
- Replacing older commercial aircrafts and their technology with newer ones are some of the reasons for the growth of aviation industry.

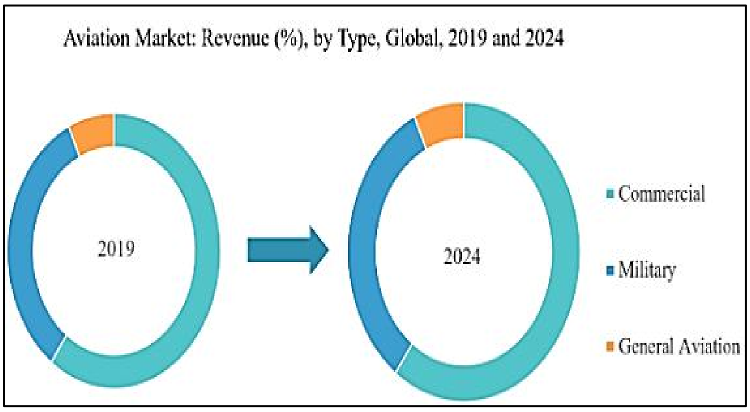


Fig 1.2 Graphical representation of Aviation Market Revenu

In **Fig 1.2** it can be depicted that the commercial aviation market dominates all the other sectors, resulting for more than half the market revenues of 2018. This segment is predicted to dominate the aviation field in coming years, because of increasing demand for new aircrafts to serve the increased air travel. Apart from that, several airlines are set to replace their old fleets with new airlines for fuel efficiency.

With so many factors influencing the aviation growth, the demand for technology also rises rapidly. Commercial aviation sector is all set to procure and implement new technologies to enhance better and comfort travel for the passengers. Different aircraft manufacturing companies are hereby set to increase their market shares by grabbing the most of new technologies and innovations.

1.3 Literature review

An intensive experimental survey was performed to study the heat transfer and friction performance of different matrix cooling geometries in view of a possible application as internal cooling systems of both nozzles and vanes of industrial gas turbines [1]. This activity has been carried out within collaboration between the Department of Industrial Engineering of the University of Florence and GE Oil & Gas [2]. Among the Gas Turbine manufacturers form the western world, the matrix fins system is relatively unknown. This is because the application of the matrix structures as cooling systems in gas turbine air foils originates from former Soviet design engineering system and were largely unknown until the recent dissolution of the Soviet Union in the 1980’s, at which time Russian research works began to appear in the international literature [3].

Currently, Flame Tube Cooling method is being used in the combustion chamber of a gas turbine jet engine for heat regeneration [4]. The process [4,5] involves different types of techniques and principles when it comes to meet the objective of heat regeneration in the combustion chamber of jet engine [5] viz. machined cooling ring, corrugated strip cooling, splash cooling strip and transpiration cooling [6].

The use of Matrix Fin cooling system [6,7]; which is also called as latticework cooling system for heat regeneration in Combustion chamber of Jet Engines is still considered a better choice because of the use of thermal efficient and high performing Aluminium Silicon Carbide (AlSiC) as a fin material when compared to other fin materials such as aluminium and copper (Aluminium (Al) and Copper (Cu) reinforced by SiC is used in various industries due to its outstanding thermo-physical properties such as high thermal conductivity, low Coefficient of thermal expansion (CTE) and improved mechanical properties such as better wear resistance, specific modulus and higher specific strength,) [7].

The chief alloying elements added to aluminium are magnesium, copper, manganese, nickel, silicon and zinc. These chief alloying elements are added in pure aluminium to increase its strength. Two different classes of alloys may be considered for increasing the strength. The first are the 'wrought alloys', which are casted in ingots or billets and hot and cold worked mechanically into extrusions, forgings, sheet, foil, tube and wire, while the second class, the 'cast alloys' in which the alloys are directly casted into their desired form by either of these three methods (i.e., sand-casting, gravity die casting or pressure die casting).

The main classes of aluminium alloys are the 2000 series (Al-Cu alloys), which are high-strength materials and light in weight mainly used in the aircraft industry, the 3000 series (Al-Mn alloys) are mainly used in the canning industry , the 5000 series (Al-Mg alloys) which have good corrosion resistance is used in structural applications of navy, , and the 7000 series aluminium alloy (Al-Zn-Mg alloys) which have light weight and very good strength and very good at corrosion resistance because of these properties 7000 series is used for military vehicle and aircraft applications.

The alloy used in any desired particular application will depend on factors such as the physical and mechanical properties required, the cost of the material and the service environment involved. If a finishing treatment has to be applied, then the suitability of the alloy for producing the particular finish desired will be an extra factor to be taken into account. The great advantage of Aluminium is that it can be casted into different alloys having desired mechanical and physical properties suitable for any application [8, 9].

The alloys of Aluminium are highly malleable which makes them easy and best suitable for manufacturing of heat sink profiles and manufacturing of the fins. Aluminium is an excellent choice for heat exchange applications because Aluminium is a good thermal conductor. Alloys of Aluminium come in wide range of tempers at minimal cost when compared to any other metal alloys. An alloy of Aluminium can be shaped into fins, foil or sheets making it perfect for heat sink applications of all pattern, due to its excellent malleability.

Cooling techniques often materials with low weight and high strength. Only a perfect alloy combination of Aluminium offers the right choice at affordable cost. Aluminium has rightly proven to possess a higher strength-to-weight ratio than copper, brass or steel. Heat sinks of Aluminium can be electrically grounded which is an important consideration in many industries. On top of that, Aluminium also has low melting point and low density, which makes it easier to stamp or cast into different shapes with different complexities, making it even more suitable for industrial applications.

The Aluminium alloys used in fin design are listed below;

- Aluminium Silicon Carbide (AlSiC)
- Aluminium Boron Nitride (AlBN)
- Aluminium alloy 1050A, 6060 (low stress), 6061 and 6063[10,11]

The major properties on which the application of aluminium is based are its high corrosion resistance even though it is in the pure form of the metal, and it is very easy to achieve an excellent mechanical strength by alloying with suitable element and heat treatments and low mass density (approx. 2.7g/cm^3) as shown in **Fig 1.3**. Other important properties include high thermal and electrical conductance, its reflectivity which plays very important role in radiation heat transfer, high ductility and low cost of working and maintenance.

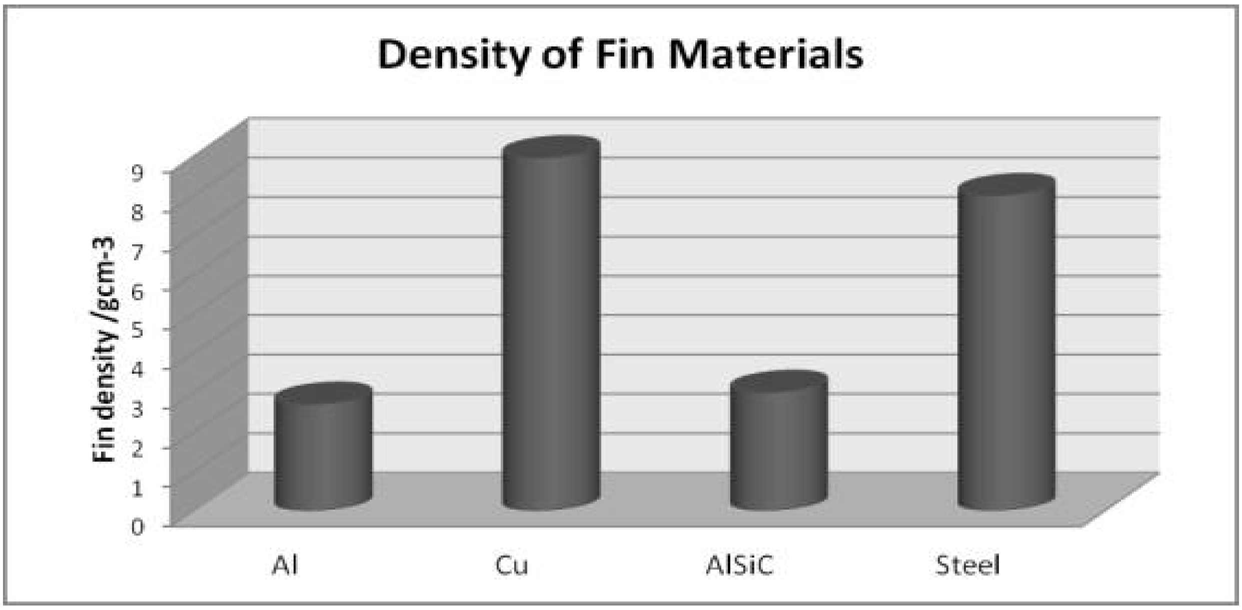


Fig. 1.3 Density of fin materials

Aluminium in its pure form is a relatively soft metal with yield strength of 34.5 N/mm^2 and tensile strength of $90N/mm^2$. However, with the development of wide range of alloys, very varied strengths and ductility can be achieved. And this has led to the many applications of today. For example, these range from the use of very thin foil material in the packaging industry, ductile materials for drink containers, and highly conductive alloys for electrical purposes, to relatively low-strength alloys for the building industry and high strength materials for aircraft and armour vehicles. The metal in its pure state has a relatively high corrosion resistance and needs less protection than most metals. On the other hand, the commercial metal and its alloys, though resistant, are distinctly more sensitive to corrosion, and the development of high strength light alloys, containing quantities of heavy metals such as copper, zinc or nickel, has heightened the need for protective surface treatments. The addition of the heavy metal influences the nature appreciably influences the alloy's susceptibility to corrosion, and it is observed that alloying material which increases the mechanical strength happens to decrease the corrosion resistance after the alloying. The development of satisfactory protective finishes for these metals has been, therefore, of very great importance [12,13].

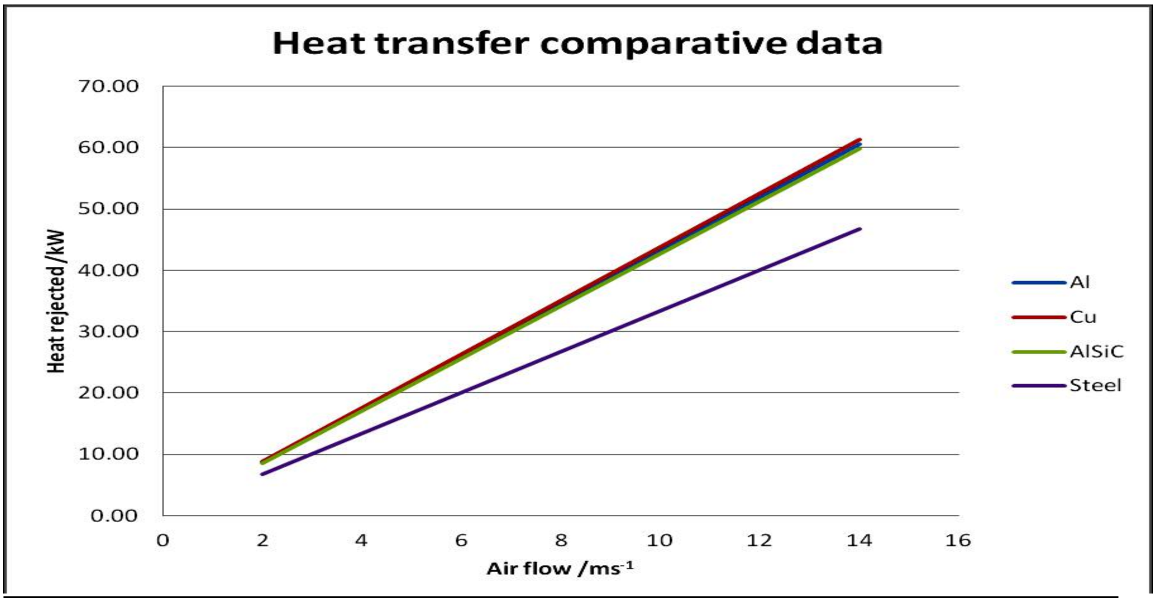


Fig. 1.4 Heat transferred from fin materials

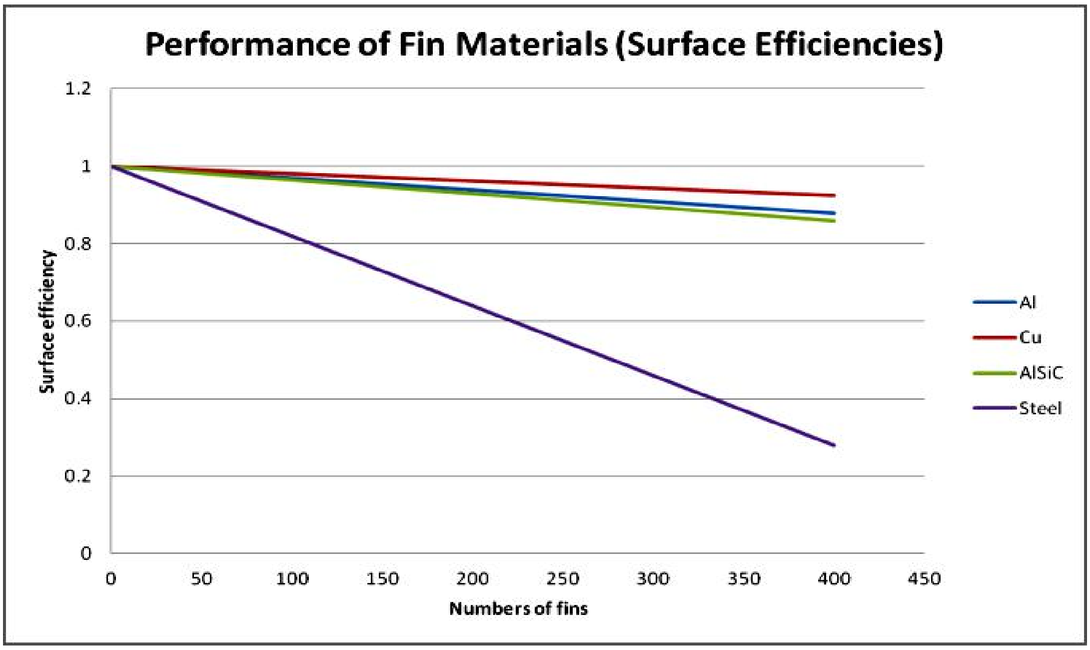


Fig. 1.5 Comparing surface efficiencies

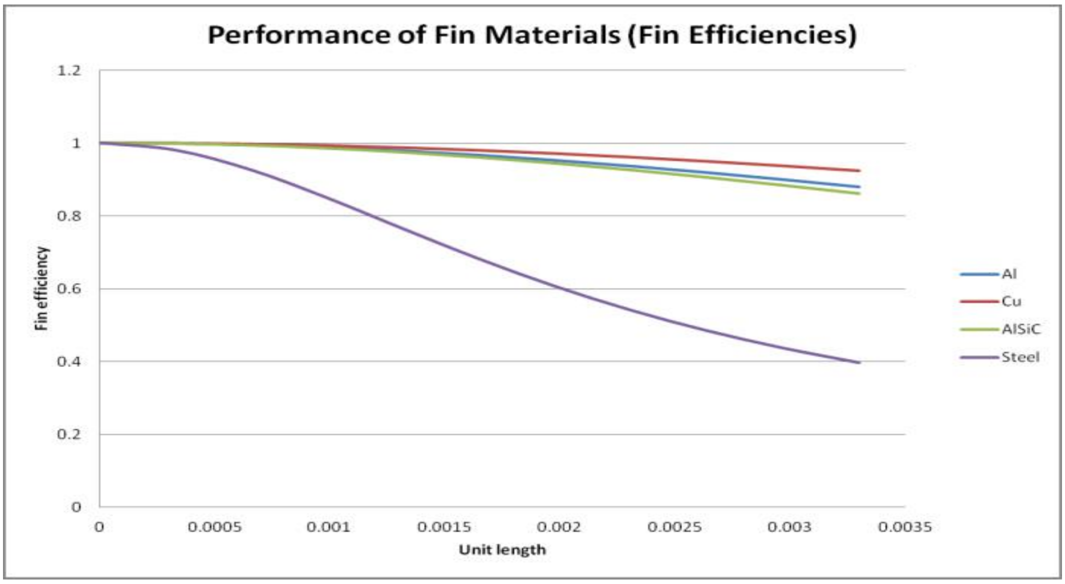


Fig. 1.6 Comparing fin efficiencies

Results obtained for the heat transferred, surface efficiency and fin efficiency are presented in **Fig 1.4, Fig 1.5 and Fig 1.6** [14].

To increase the efficiency of gas turbine there is gradual increase in the inlet temperature. The material used for the components cannot withstand high temperature so it must be well cooled [15]. Effusion cooling is recognized as one of the most effective cooling methods there has been many developments in the film cooling of the combustion liner using guiding ring to the hot side [16] where guiding ring is used to prevent the mixing of the cooling fluid and hot fluid, using micro-channels where transply is produced by laminating two or more laminates containing pattern of holes and channels [17-20]. Many methods are investigated to increase the cooling effectiveness by increasing the internal convection within the combustion liner [21]. There has been extensive work on the optimization of the effusion hole diameter, hole angle inclination, inter-row streamwise distance, and interrow longitudinal distance

In 2004 the tests were conducted on matrix fin to study the pressure losses and heat transfer characteristics. Acrylic models were tested in the first test where the effect of increased surface area for heat transfer was not considered as the acrylic material was an insulating rib. Heat transfer and pressure variations on the matrix fin were investigated. Liquid crystal technique was used to measure the variations in temperature.

In the second test metal models were tested on the matrix fin where the effect of increased surface area for heat transfer was considered as the metal was conducting. Temperatures were measured in this method using infra-red camera. Heat transfer and pressure variations on the matrix fin channels were investigated. Heat transfer from the matrix fin in first method was compared with the second method using Dittus-Boelter Equation and following conclusions were drawn from the test results,

1. Average heat transfer enhancement factor K on duct shell was equal to 3.5. After turns the local K value reached up to 3.
2. Narrow sub channels in the matrix fin provide higher overall heat transfer enhancement, with $K \approx 3$, than wider sub channels of the matrix fin which have less turn effects.
3. The effect of increased surface area for heat transfer was of great significance.

To describe heat transfer and friction enhancement factors, Nagoga compared the matrixes with different channel orientation, he investigated a matrix with straight channels to describe heat transfer and friction enhancement factors, i.e. for which $\beta = 0$. And he also investigated a matrix with angular orientation. After comparing the results of both experiment Nagoga found that a matrix with angular orientation increases the heat transfer when compared to matrix fin with straight channels, i.e. with $\beta = 0°$.

Heat transfer in the basic section was higher than that in a straight duct. The local enhancement factor of the matrix fin varied from 1.28 to 3. For increased Reynolds number and ratio of length to width ratio of the channel, the enhancement factor decreased. The local Nusselt number and average Nusselt number over the length of the channel depend on β. There was increase in the average Nusselt number with increased in the β angle up to $\beta = 45°$. The average Nusselt number decreased with the increase in the channel length. The local Nusselt number reaches a maximum right after the turn and then reduces along the channel. Nagoga concluded that the rib pitch of matrix fin, relative channel depth of the matrix fin, form of channel cross-section, did not affect local and average heat transfer from the matrix fin in the scope of interest for turbine blades and vanes [22,23].

CHAPTER 2

OBJECTIVES AND RESEARCH METHODOLOGY

2.1 Main Objective of the Project

The main objective of our project is to increase the cooling effectiveness of combustion liner of steam and gas engines using matrix fins.

2.2 Report organization

The project report is represented in seven chapters:

Chapter 1: Introduction presents an overview of the transply effusion cooling system and study of matrix fins. It includes the present market scenario of scope of Aviation field. This is followed by the literature review pertaining to the current project. The chapter concludes by describing the main objective of this work.

Chapter 2: This chapter deals with the basic concepts of the combustion chambers and cooling systems used. Basics concept related to matrix fin and equations that describe the heat transfer

Chapter 3: Here using mathematical expressions of heat transfer is used to code the algorithms and MATLAB is used to obtain the optimised parameters

Chapter 4: Using the optimised parameters model is designed in the solidworks.

Chapter 5: Concepts related to numerical analysis like meshing, mesh quality, convergence and boundary conditions are discussed

Chapter 6: In this chapter discussion of the Ansys results are done using figures and graphs.

Chapter 7: Conclusions are listed against project objectives. Scope for future work is discussed.

2.3 Project Objectives

1) To design and optimize the matrix fin cooling system by studying the heat transfer characteristics.

2) To implement the matrix fin pattern in the transply effusion cooling system and study the cooling effectiveness.

2.4 Methodology

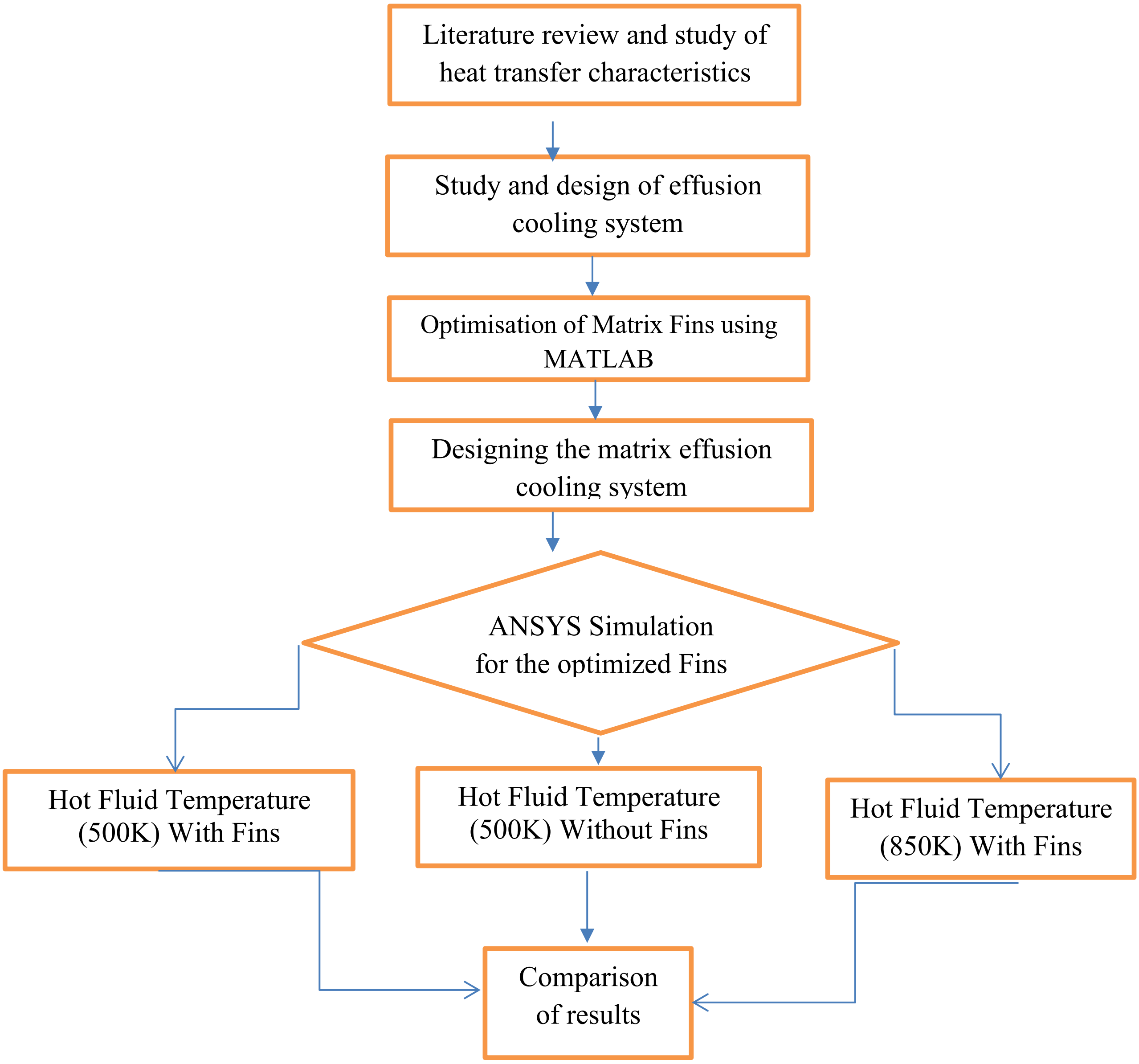


Fig 2.1 Detailed methodology of the project

CHAPTER 3

THEORY AND CONCEPTS

3.1 Combustion Chamber

The combustion chamber of the jet engine is as shown in the **Fig 3.1** it burns the fuel that has been supplied through the fuel spray nozzles. Very large quantity of the sir is supplied along with the fuel by the compressor, this air and fuel mixture will undergo chemical reaction releasing very large amount of heat, this expands the excess air in such a way that it gives smooth acceleration which then is used to run the gas turbine. During these processes of combustion and acceleration of the gas the processes must be accomplished such that there should be minimum pressure loss and the heat released by the combustion reaction per unit volume should be very large for high work output of the turbine.

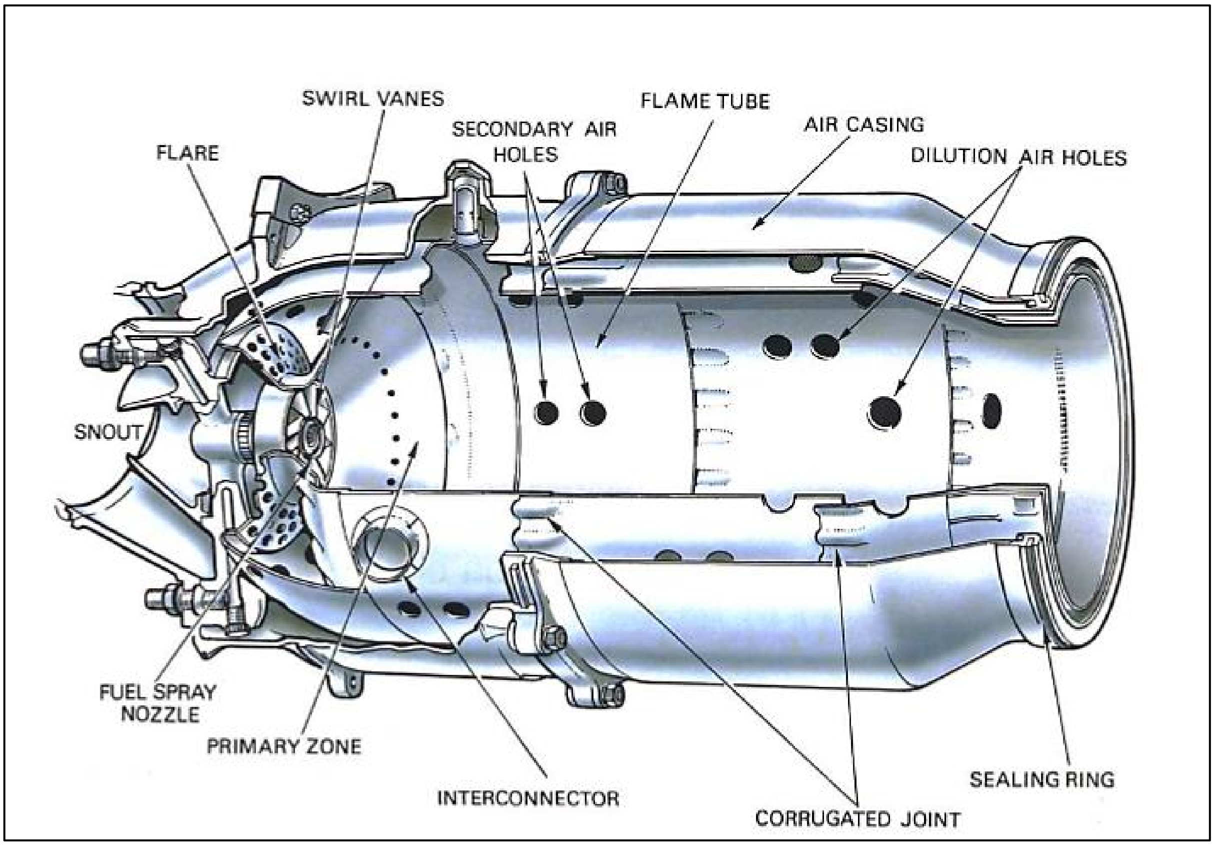


Fig 3.1. Combustion chamber of jet engines

The temperature rise of the combustion chamber depends on the metallurgical properties of the material used. So, the amount of the fuel is calculated according to the temperature requirement and the melting point of the material used. The maximum temperature of the combustion zone and the by-products is from 850 to 1700 deg. C. before combustion zone itself the compressor increases the temperature of the atmospheric air from 200 to 550 deg. C. due to the work done by compression prosses, which means the remaining temperature rise of 650 to 1150 deg. C. should be done by the chemical reaction of the fuel and compressed air. The temperature of the gas at the exit depends on the thrust that has to be produced by the engine so the combustion chamber should be designed in such a manner that it should take wide range of the working thermal loads. Increase in the fuel prices forcing the manufacturers to design the jet enginee which are efficient. The cooling methods that are used for the combustion chamber have the positive effect on the overall efficiency of the jet engine, the more the efficient cooling method the efficient the engine will be.

3.2 Combustion Process and Cooling of The Liner

Atmospheric air is drawn into the compressor where it is compressed to a very large pressure ratio, the compressed air then enters the combustion zone, the are that enters the combustion zone has velocity 500 feet per second, which is higher than the flame propagation velocity to maintain the flame the air velocity inside the combustion chamber is reduced using the diffuser, the process of diffusing the air will increase the static pressure of the air. The flame propagation speed of burning fuel at normal air fuel mixture is only 80 feet per second, so the air that enters the combustion chamber should have the velocity of 80 feet per second for the continuous operation of the jet engine.

During the operation condition of the combustion chamber the air/fuel ratio can very between 45:1 and 130:1, but the fuel that is used will burn efficiently at the ratio 15:1 or at least close to the ratio. Because of the constrain on the air fuel ratio only part of the air that enters from the compressor is used for the combustion process. This combustion zone is called the primary zone.

Only 20% of the compressed air flow is taken in by the snout or entry section as shown in the **Fig 3.3.** after the entry section air passes through the swirl vanes, after the swirl vanes air passed in to the primary zone of the combustion chamber. The swirling vanes help in the air recirculation and smooth upstream flow towards the secondary zone of the combustion chamber. The air mass flow that is not picked up by the swirler is made to flow into the annular space, the annular space is the space that is provided between the flame tube and the air casing. The air the is directed in to the annular region is used for the cooling of the flame tube.

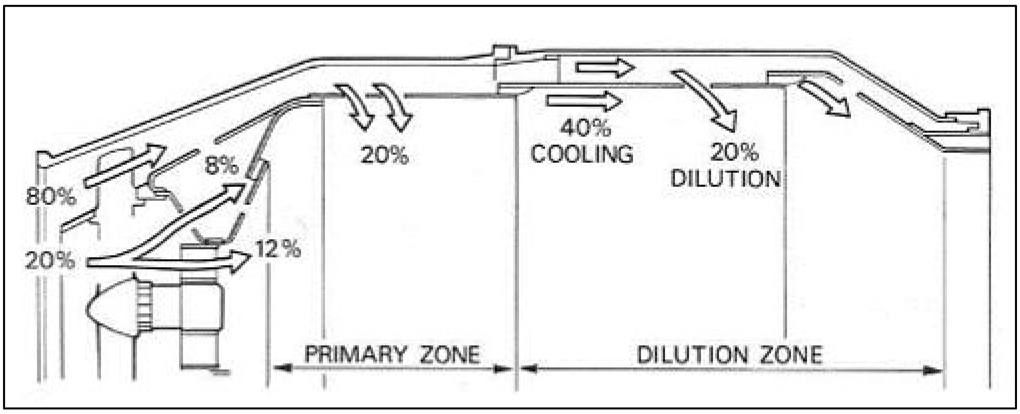


Fig 3.2 Apportioning the airflow

There are holes present in the wall of the flame tube adjacent to the primary combustion zone, these holes are called secondary holes. Through these holes further 20% of the annular air is passed in to the primary combustion zone to facilitate the complete combustion the fuel that is present in the primary combustion zone. The air that enters from the secondary holes will interact with the air that is picked up by the swirl vanes and creates the region of low velocity recirculation zone. This recirculation zone will take the form of toroidal vortex which looks like donut of smoke, which gives the flame stability and flame anchoring as shown in **Fig 3.3**. The fuel is beaked in to fine droplets and mixed with the recirculating air to facilitate the efficient burning of the fuel air mixture, breaking the fuel particle into fine droplets will increase the temperature of the fuel rapidly to the ignition temperature. The breaking of the fuel particle into fine droplet is carried out by using the conical shaped spray nozzle, together with the turbulence that is present in the primary zone will create fine droplets of the fuel and mixed it with the incoming air.

The combustion process will produce large amount of heat which raises the temperature of the gases from 1,800 to 2,000 deg. C. this temperature is far too hot to enter the guide vanes of the turbine. The air present in the annular region which is not used for combustion amounts to about 60% of the total airflow is used for the cooling of the flame tube and dilution of the chot fluid. One third of the annular air is used to lower the temperature of the gas in the dilution zone, which then enters into the turbine and nozzle. The remaining of the annular air is used for cooling of the flame tube or combustion liner. This is achieved by a film of cooling air flowing along the inside surface of the flame tube wall, insulating it from the hot combustion gases.

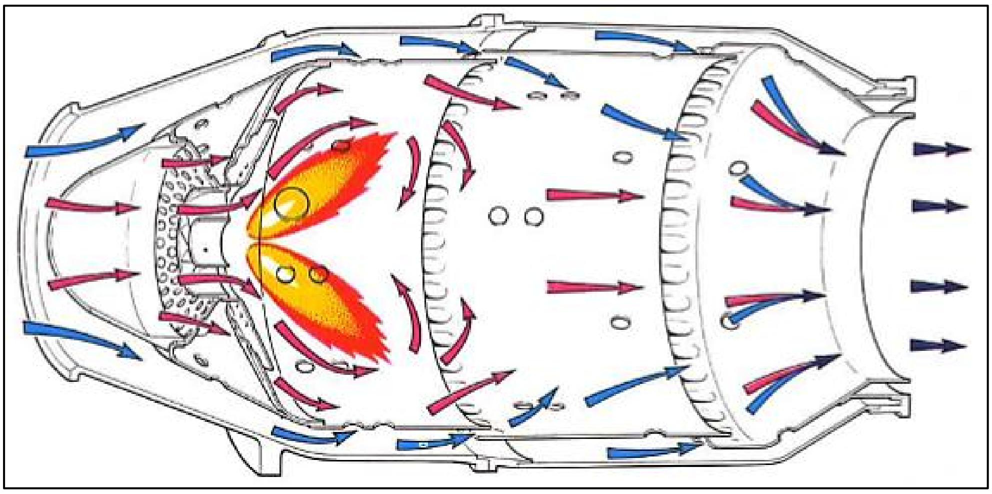


Fig 3.3 Flame stabilizing and general airflow pattern inside the combustion chamber

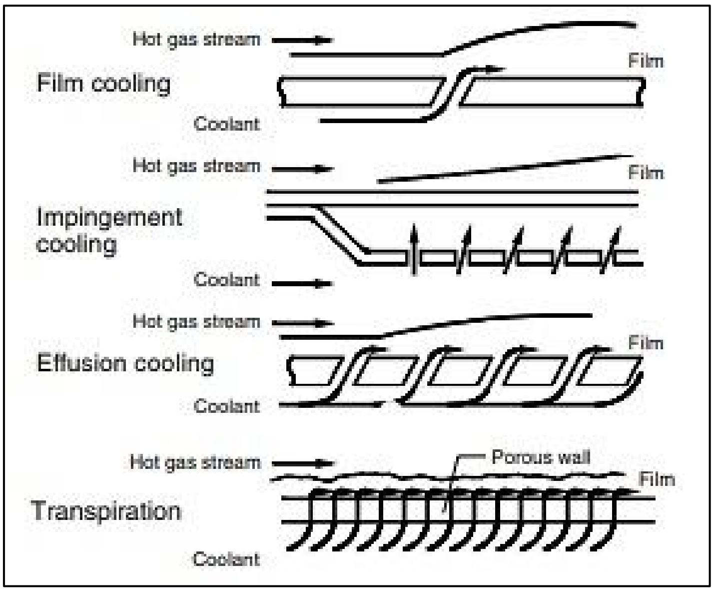


Fig 3.4: Main cooling technologies using intermediate fluids

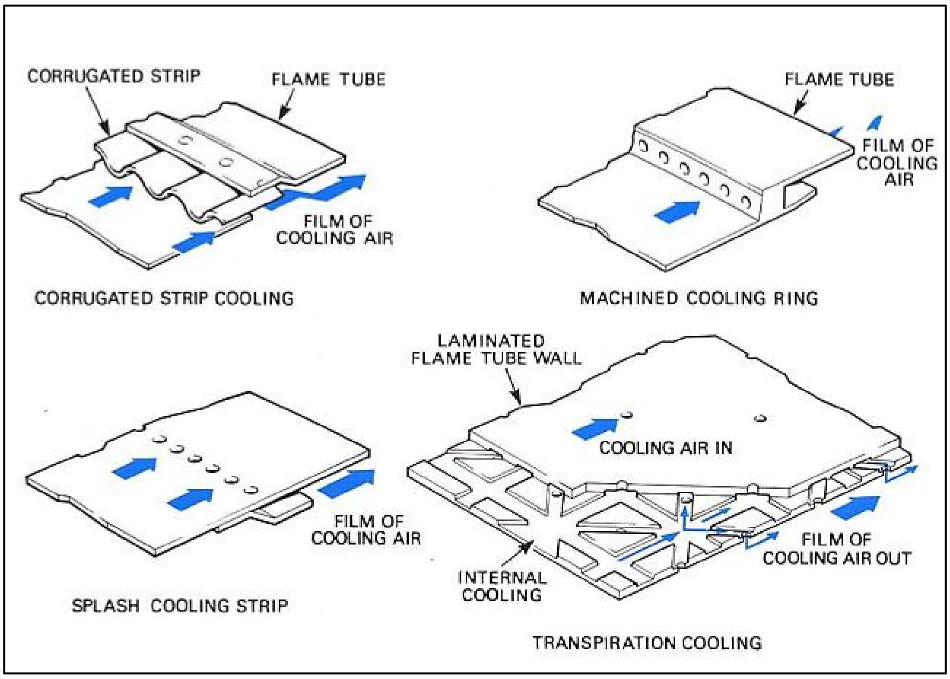
s

Fig 3.5 Flame tube cooling methods

There are many methods of cooling the combustion liner as shown in **Fig 3.4**. To increase the internal convection of the combustion liner many developments are done to design network of microchannels and passages, through which the cooling air will flow and produce the effective cooling of the combustion liner. By doing so the amount of air required to cool the flame tube is reduced up to 50%. There are many film cooling methods as shown in **Fig 3.5**. The combustion process should be completed in the primary zone and secondary zone before the dilution air enters the dilution zone otherwise the dilution air will cool the fuel and incomplete combustion will occur.

The flame tube walls and parts of the combustion chamber should withstand high temperature of the combusted gas in the primary zone. Along with the cooling methods that are mentioned above other methods also used to achieve this. Best heat resisting materials are used to design the flame tube and combustion chamber, and thermal barrier coating are used to resist the high temperature of the combustion zone, these coatings are coated on inner side of the combustion chamber wall. The combustion chamber should also resist the corrosion due to the products that are release by the combustion process, and creep failure that arises due to large temperature gradients. It should also withstand the fatigue failure due to the vibrational stresses.

3.3 Matrix Fins

A matrix fin system has two layers of longitudinal ribs which are opposite angled. Each layer of ribs acts like arrays of fins by creating a system of channels as shown in the **Fig 3.6**. These channels are diagonal in direction in which the cooling air flows, the direction of the flow is continuously changed due to the oppositely angled layer of the fin arrays.

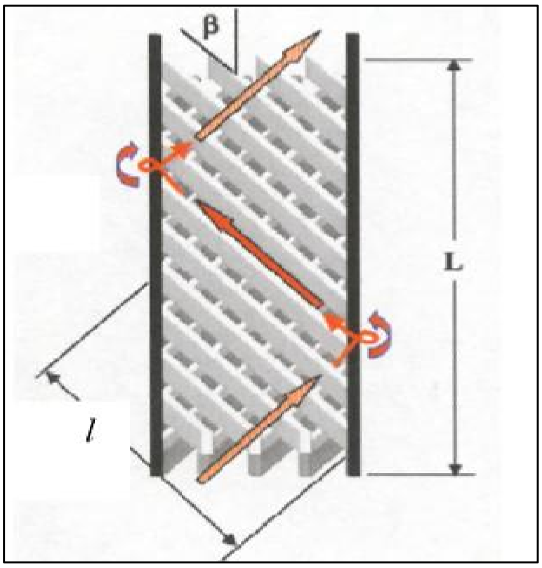


Fig 3.6 Example of a closed Matrix Geometry

The convective heat transfer coefficient is increased due to the turbulent effect that is produced by the site direction of the longitudinal ribs. The air that passes through the channels due to the matrix fin configuration of the ribs the flow will pass from one channel to another channel, this creates swirl of air, which then increases the turbulence of the flowing air. This increase in the turbulence has the positive effect on the Nusselt number. These longitudinal ribs also increase the overall surface area due to this heat transfer is also increased. When it comes to the strength, the longitudinal ribs provide extra stability and strength to the component, they behave like honey comb structure due to the same design resemblance. The angle β of the channels has a large influence on the matrix heat transfer enhancement.

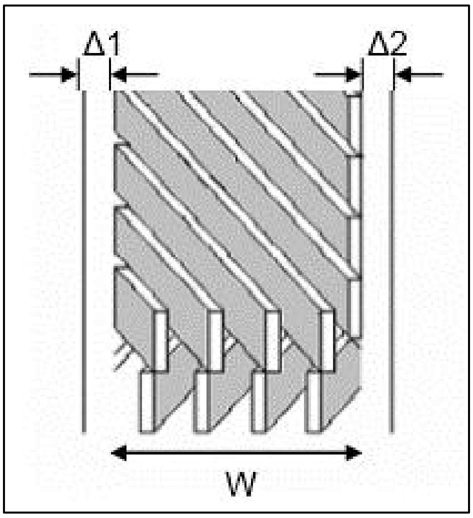


Fig 3.7 Open Matrix Geometry

There are two types of matrix configurations, the matrix shown in the **Fig 3.6** is called closed matrix. Here the channels created by the longitudinal ribs reach the side of the wall. In closed matrix configuration, the air flows till the end of the channel and through the direction change the flow will move to the channel that is present on the top layer, the air flow from the two channels will not mix. Another type of matrix design open matrix, here there exist a clearance between the channels and the side the wall which are called Δ1 and Δ2 in the **Fig 3.7.** In the open matrix configuration, the air mass flow reaches the end of a channel and mixes with the other air mass flow in the clearances Δ1 or Δ3. The air mass that is mixed in the clearance will flow in to new channels that starts at the clearances. Matrix fin cooling system is used in the trailing edge of turbine vanes and blades. Usually open matrix fin configuration is used in the turbines that are produced by the Siemens. The flow in the open matrix is assumed to be axil in direction and radial in closed matrix.

3.4 Basic concepts of convection heat transfer

The convection heat transfer from any surface involves very complex processes, usually these processes is comprised of two mechanisms, one is the transfer of the energy due to random molecular motion also called diffusion process, second transfer of the energy by the bulk motion or macroscopic motion of the fluid.

The convective heat transfer coefficient of the matrix fin is the function of geometric parameters like angle between the longitudinal ribs and fluid properties like Nusselt number, Prandtl number and Reynolds number. The equation of the Nusselt number is given below

$$Nu_l = 0.0361 \cdot \left[1 + 12.77 \cdot sin^2(2\beta)\right] \cdot Re_l^n \cdot Pr^{0.4} \cdot \left(\frac{T_w}{T_f}\right)^{-0.55}$$

Where

$$n = 0.8 \cdot \left\{1 + 0.2 \cdot \left[\left(\frac{4\beta}{\pi} - 1\right)^2 - 1\right]\right\}$$

Where

β is the angle between the matrix fin

Re is Reynolds number

Pr is Prandtl number

T_w wall temperature

T_f fluid temperature

CHAPTER 4

OPTIMIZATION OF MATRIX FIN

3.1. Optimisation of Fin Arrays

The idea of applying the matrix fin in the effusion cooling system might increase the cooling effectiveness but we need the optimised geometric parameters like height, thickness and the spacing between fin array so that we have maximum heat transfer from the matrix fin. Convection heat transfer is the product of the convection coefficient and total surface area for constant temperature difference between fin surface and surrounding. Heat transfer can be increased by increasing either one of them or both of them but increasing the surface area further has negative effect on the Nusselt number. Increasing the number of fins increases the surface area but it does not necessarily increase the convection heat transfer it is because of the reduction in the convection coefficient due to the negative effect on the Nusselt number.

4.2 MATLAB Coding

Using the mathematical models that describe the heat transfer from the matrix fin we developed the algorithm which we used in MATLAB coding to conduct the iterations that gives the optimum point. To obtain the optimum point for maximum heat transfer we used the product of convection coefficient and surface area of the fin and we called it factor, to which multiplying with temperature will give the convection heat transfer. MATLAB software bundle is used to conduct the iterations for different geometric parameter to see their effect on the convection heat transfer. These geometric parameters are used as variables in equations that describe the heat transfer from the fin to the surrounding air. Using the mathematical expressions that describe the heat transfer algorithms are coded to simulate them in the MATLAB, **Table 4.1** shows the algorithm that we used in the optimisation of the matrix fin while coding the equations are adjusted such a way that the spacing between the fin is maintained not to overlap with the effusion holes.

Table 4.1 MATLAB code

```
hf=0.0007;
p=1.185;
v=20;
u=18.385*10^-6;
len=0.0707;
k=0.02634;
a=[];
j=1:0.1:4;
for t=0.0003:0.00001:0.00087
a=[];
for i=1:0.1:4
g=i*0.001375-t;
d=(4*hf*g)/((2*hf)+g);
Re=p*v*d/u;
Nu=0.0367*(Re^0.88)*(0.702^0.4)*(1.678^-0.55);
No=0.050/(t+g);
A=2*hf*len+2*(t*hf)+(t*len);
H=Nu*k/len;
x=H*A*No;
disp(['At thickness= ', num2str(t), ', Factor = ', num2str(x)]);
a = [a; x];
end

plot(j,a);
hold on
end
```

4.3 Results and Discussion

After the iterations, we obtained multiple curves for different thickness as shown in the **Fig. 4.1**, where spacing between is taken along the x-direction and factor along the y-direction, each different coloured curve is for different thickness. From the graphs we obtained the optimum thickness 0.5mm and spacing between fin 0.865mm. These analyses are conducted by keeping the velocity of the fluid constant, to see the effect of the velocity on the convection rate we changed the velocity only on the obtained optimum parameters. The iterations show that when the fluid velocity increases the factor increases but not the optimum point this is shown in the Figures which means for all velocity of the fluid the optimum parameters remain same but only the factor changes due to its positive effect on the Nusselt number.

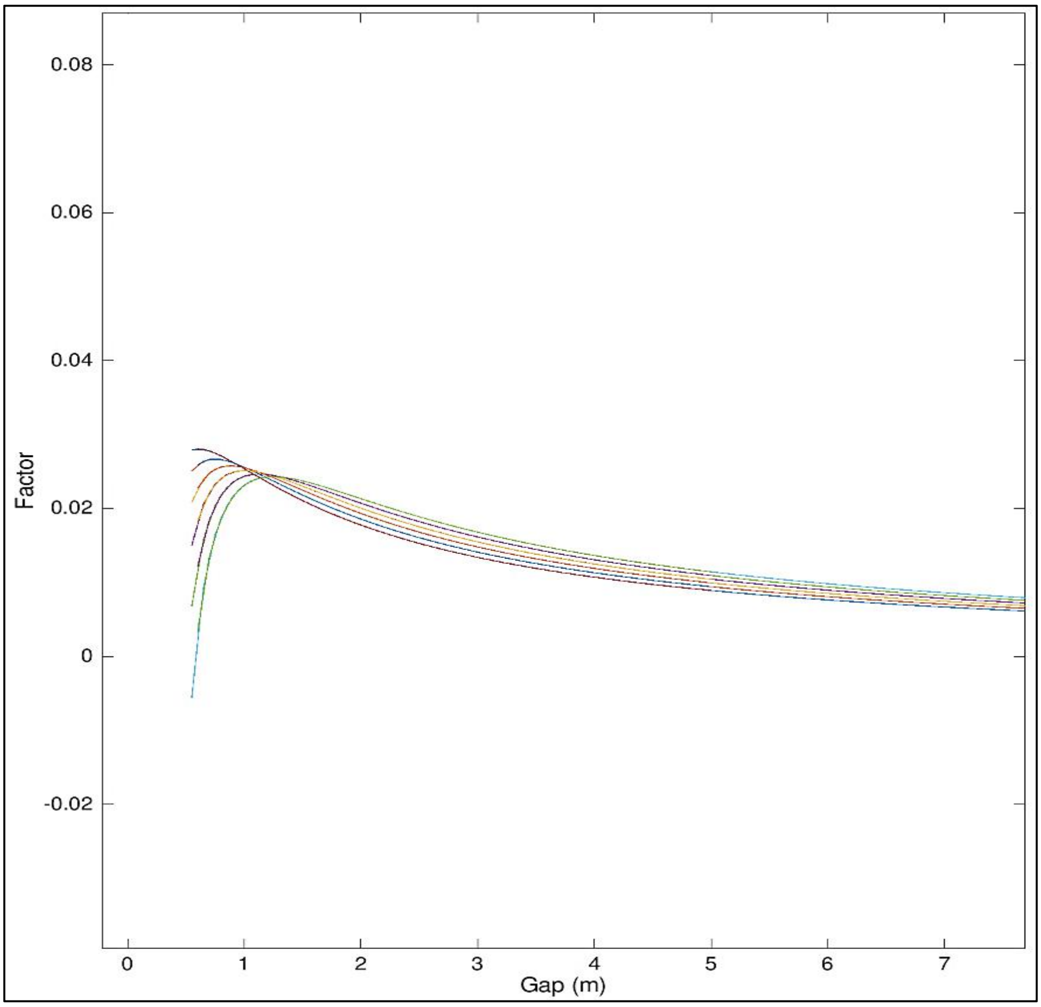


Fig. 4.1 Convection heat transfer variation with fin thickness and spacing

In the **Fig. 4.2** we can see the variation of the heat transfer for the velocity of the fluid 20m/s and the same variation can be seen in the **Fig. 4.3** where the analysis is done for the velocity of the fluid 60m/s. The graphs show that there is no change in the positions of the points due to the change in the velocity, instead the whole graph is lifted upwards due to increase in the velocity of the fluid which in turns increases the convective heat transfer.

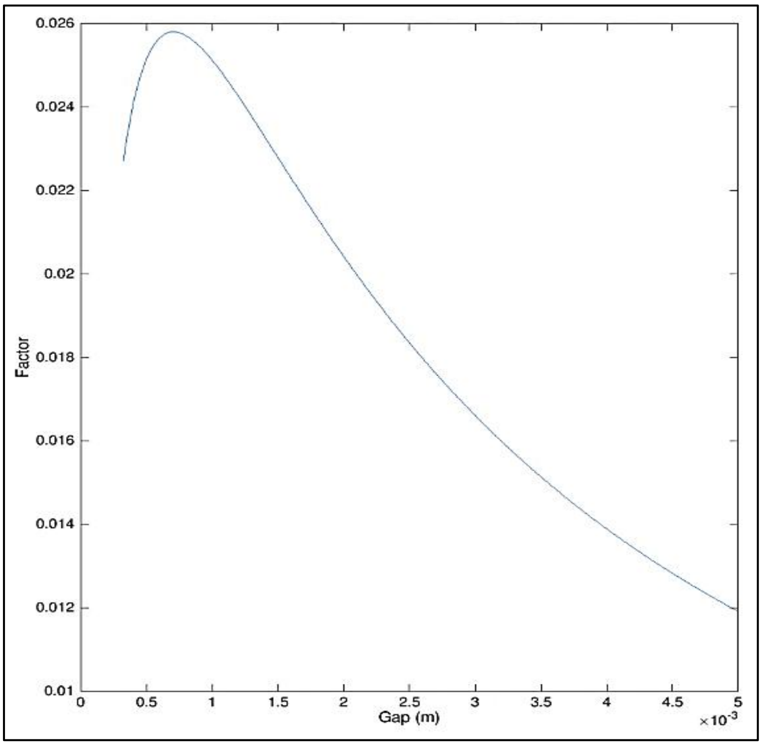


Fig. 4.2 Convection heat transfer variation for fin thickness 5mm, spacing 0.865mm, and velocity 20 m/s

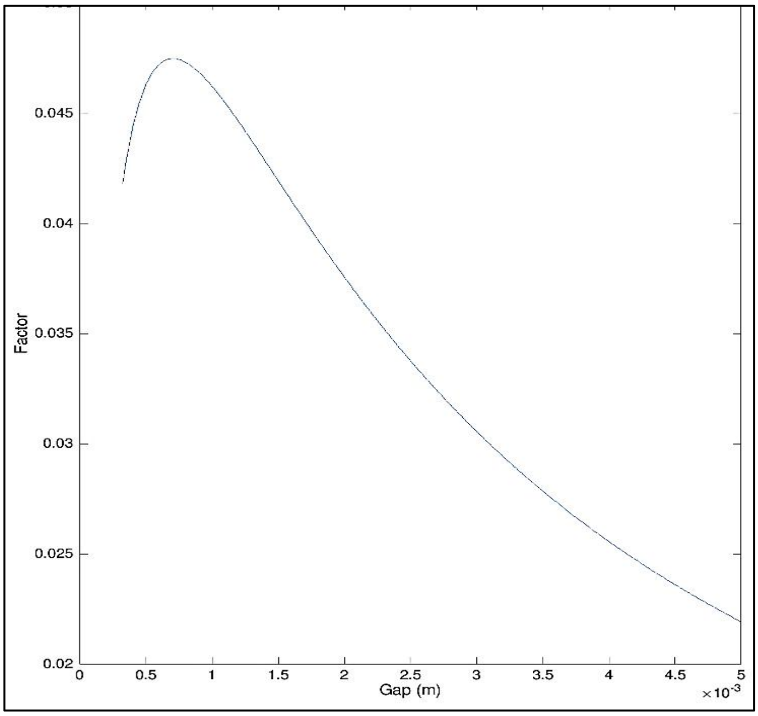


Fig. 4.3 Convection heat transfer variation for fin thickness 5mm, spacing 0.865mm, and velocity 60 m/s

CHAPTER 5

OPTIMIZED DESIGN

The final optimized parameters of the matrix fin have been obtained from the MATLAB simulations and the dimensions of the hole diameter and their arrangement was taken from the research papers using these geometric parameters solid model is designed in the Solidworks 2016. **Table 5.1** shows the dimensions used for the designing of the model.

Table 5.1: Optimized fin parameters

Serial number	Parameter	Value
1	Fin Height	0.7mm
2	Fin Thickness	0.5mm
3	Distance Between Fins	0.865mm
4	Hole Diameter	0.5mm
5	Streamwise Pitch	1.03mm
6	Spanwise Pitch	0.91mm
7	Thickness of Effusion Plate	0.5mm

A test area of 20mm X 20mm was taken for the analysis. It consisted of 100 holes and 20 fins. SolidWorks was used to design the optimized model. The top view of the optimized design is shown in **Fig. 5.1** from the figure we can see that the matrix fin is designed such a way that the effusion holes run in between two fin arrays.

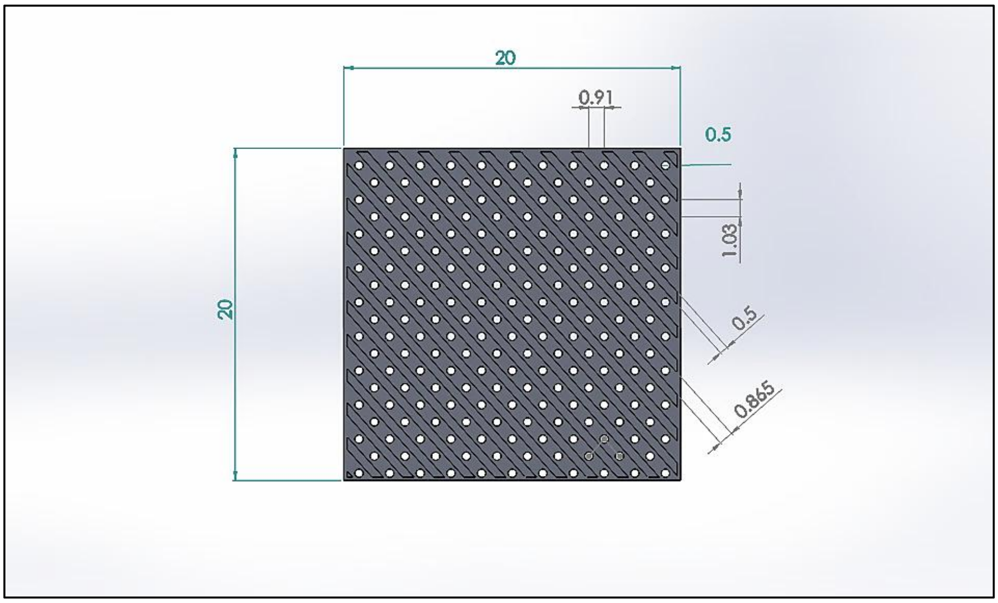


Fig. 5.1 Top view of optimized part

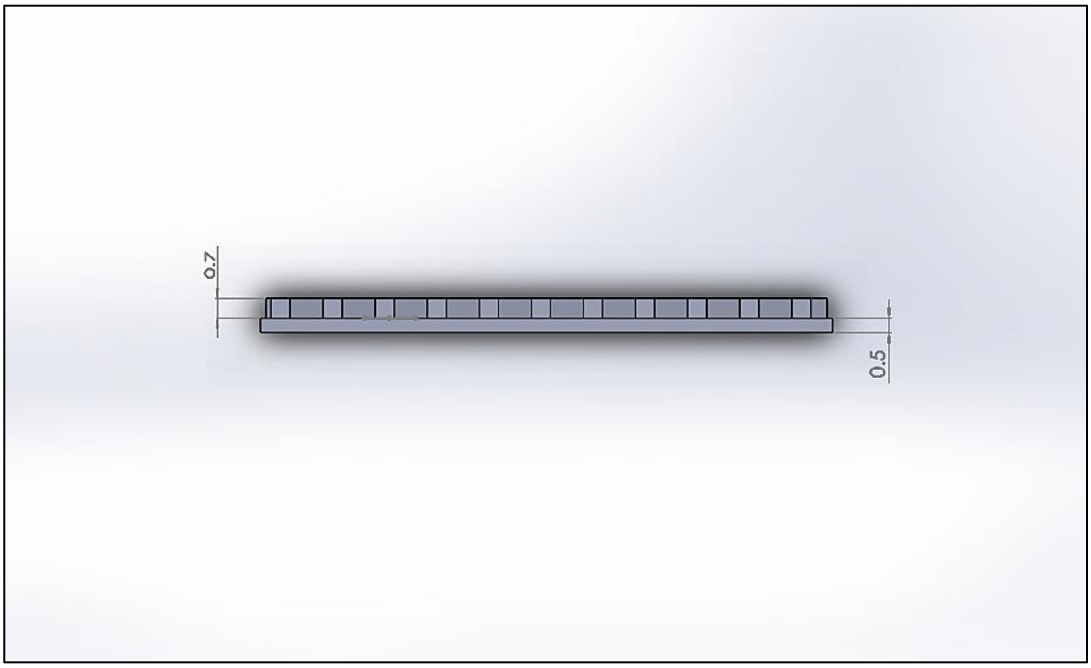


Fig. 5.2 Front view of optimized part

The front view of the optimized matrix fins design is shown in **Fig. 5.2** the height of the fin is 0.7mm and the thickness of the effusion plate was 0.5mm two such parts are created to laminate them to obtain the final model this method of laminating the effusion sheet with the microchannels is call the transply method, that’s why the name given is matrix fin transply effusion cooling system.

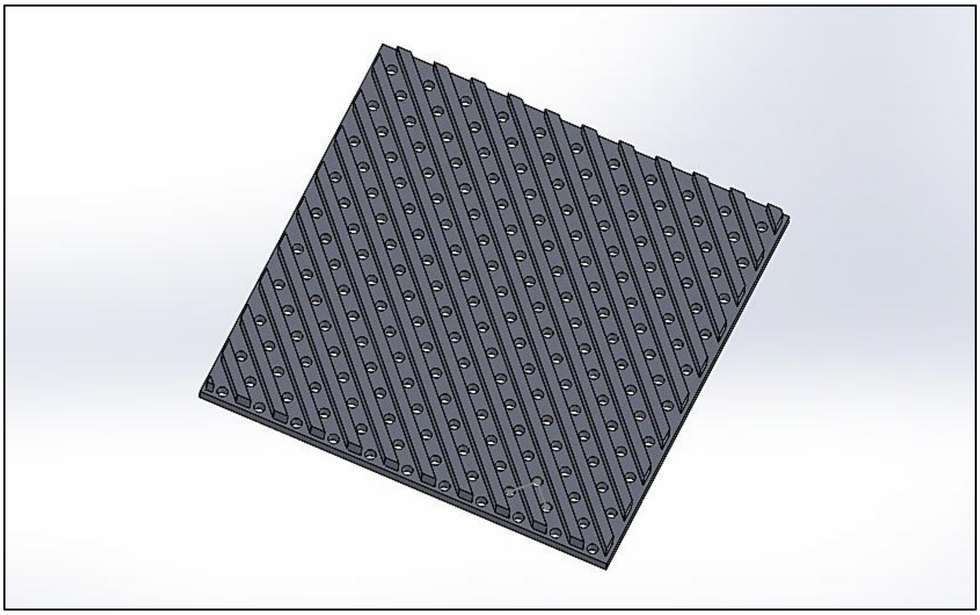


Fig. 5.3 Isometric view of optimized part

The isometric view of the matrix fin design is shown in **Fig. 5.3** here only one sheet of the cooling system is shown, two such sheets are created and laminated to obtain the final part.

The matrix fin plates are aligned together with the fins from each plate intersecting at an angle equal to the forty-five degree. The corner view of the combination of the matrix fin plates is shown in **Fig. 5.4.**

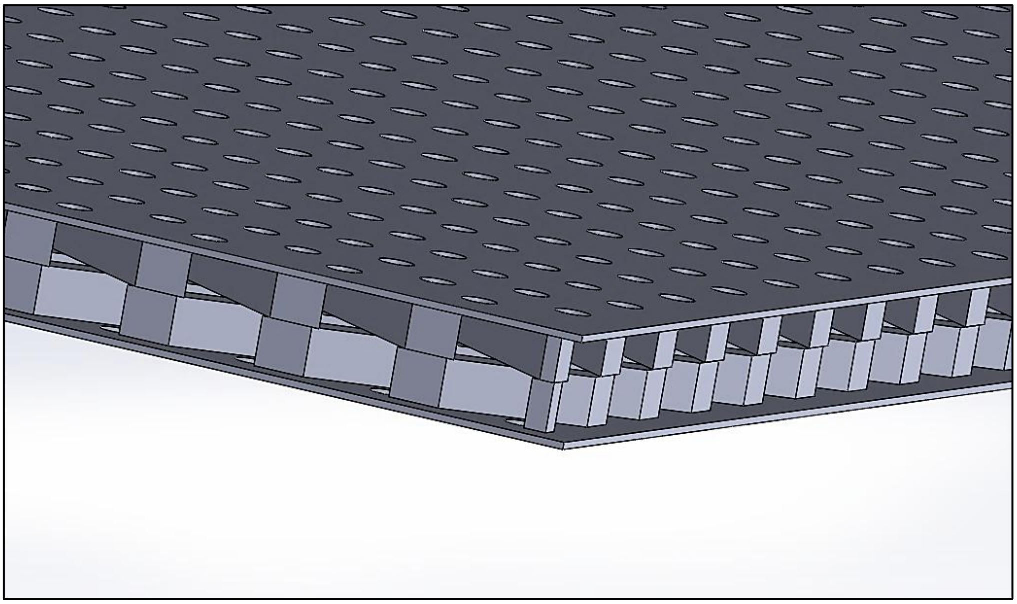


Fig. 5.4: Corner view of the final optimized part

CHAPTER 6

NUMERICAL ANALYSIS

6.1 CFD analysis

ANSYS fluent software is used to conduct the CFD analysis, optimised design parameters are used to design the model cooling system in the solid works 2016, which then is imported into the fluent geometry where Boolean command is used to create the fluid domains around the model to simulate the fluid domain and the interaction between the solid surface and fluid that is surrounding for different boundary conditions. Different temperature of inlet hot fluid is used to compare the cooling effectiveness.

6.2 Meshing and Boundary Conditions

The accuracy of the numerical analysis depends on the meshing and its quality so it is very important to have good quality of meshing. The results that are obtained from the numerical analysis are not exact but accurate because of the presence of truncation error (error of higher order terms), and it is equal to the difference between partial derivatives of the governing equations and their discrete approximations. The rapid change in the volume of the cells which are adjacent will result in larger truncation errors, so it is very important to obtain the convergence with very minimum residues. There is interaction between the solid surface and the fluid, to study the convection heat transfer from the solid surface to the surrounding fluid mesh interface is created between solid surface mesh and the fluid domain mesh. We used velocity boundary condition as inlet and pressure boundary condition as outlet. To produce the annulus cold air and combustion hot gas condition we had two inlet one for each cold and hot environment. From the literature survey we got the velocities of both inlets for hot and cold fluid, atmospheric pressure is used as outlet. For the setup of the analysis we used laminar model with energy equation, steady state analysis and pressure-based solution type is used. The convection heat transfer between the solid and fluid is achieved by the mesh interfaces. The meshing element size is decided by the study of zones where the variations of the properties are high or low, if the zone is simple and less important, we don’t have to refine mesh for that particular zone however if variation of properties are very large and the study of the zone is very important, we have to do the fine meshing to increase the accuracy. Since the whole model is the zone of interest so used auto meshing and increased the number of elements and nodes by making the meshing very fine, the meshing quality of the mesh is checked by evaluating the orthogonality, skewness, aspect ratio, and mesh quality. the detail study of the mentioned parameters is discussed below.

6.3.1 Skewness

Skewness is the angular measure of element quality with respect to the angles of ideal element types. Skewness is considered as one of the quality measures to check the mesh quality. Skewness determines how close a face or cell is to ideal **Fig 6.1** (i.e., equilateral or equiangular). The **Table 6.1** lists the range of values of skewness and the corresponding cell quality.

There are two different methods for calculating Skewness for elements. First method is Calculation of Skewness for Triangular/Quadrilateral Elements (Angular Measure) and second method is Calculation of Skewness for Triangular/Quadrilateral Elements (Normalized angle deviation). In the normalized angle deviation method, the skewness is defined as the maximum ratio of Angular deviation from Ideal element.

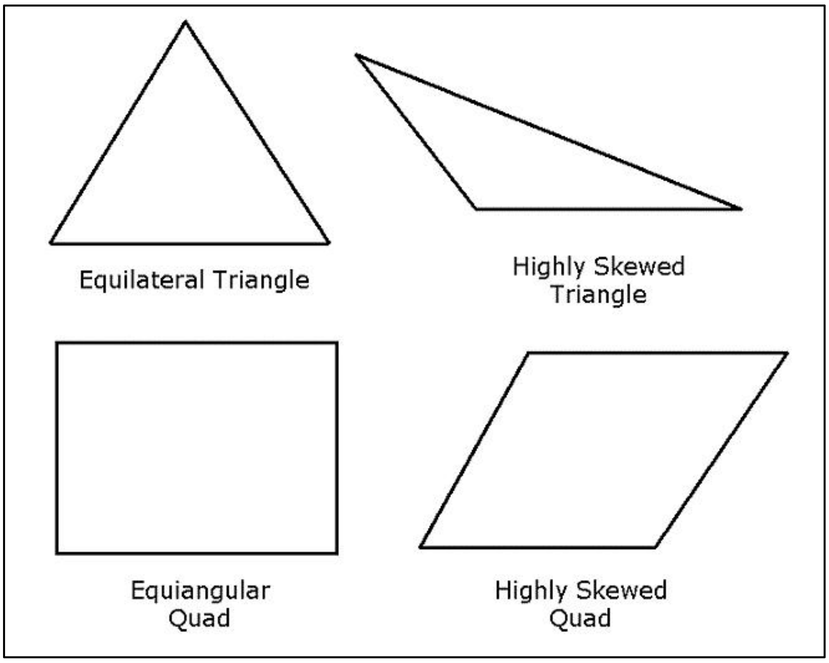


Fig 6.1 Ideal and Skewed Triangles and Quadrilaterals

Table 6.1 Range of skewness values and the corresponding cell quality

Value of Skewness	Cell Quality
1	**degenerate**
0.9 — <1	**bad (sliver)**
0.75 — 0.9	**poor**
0.5 — 0.75	**fair**
0.25 — 0.5	**good**
>0 — 0.25	**excellent**
0	**equilateral**

There is a general rule that the maximum skewness for a triangular/tetrahedral mesh in most cases should be kept below 0.95, and an average value should be less than 0.34. skewness value above 0.95 may lead to convergence difficulties.

6.3.2 Aspect Ratio

Aspect ratio is defined as the ratio between largest and smallest characteristic dimension of an element. As the aspect ratio increases the inaccuracy of finite element analysis also increases, and this will have significant effect on convergence of Finite Element Solutions.

The aspect ratio of an ideal element is 1.0. in tetrahedral element the aspect ratio is the ratio of the longest edge to the shortest normal dropped from a vertex to the opposite face. For better convergence there should not be more than 10% of the elements with an aspect ratio higher than 10. For the larger value of aspect >> 40 the elements should be closely examined.

6.3.3 Orthogonal Quality

The orthogonal quality of a cell is another measure to check the meshing quality. orthogonal quality for cells is computed using the face normal vector, the vector from the cell centroid to the centroid of each of the adjacent cells, and the vector from the cell centroid to each of the faces. The range for orthogonal quality is from zero to one. where orthogonal quality zero is considered as worst and a value of one is considered as best. For better convergence most of the cells should have the orthogonal quality nearer to value one and the worst cells will have an orthogonal quality closer to zero.

CHAPTER 7

RESULTS AND DISCUSSION

CFD analysis is conducted on the optimised model with fin and without fin within the thickness of the effusion cooling system to compare the effect of the matrix fin on the effectiveness of the cooling system. And the temperature of the inlet hot fluid is changed by keeping all other boundary conditions same CFD analysis is conducted on the model with matrix fin within the thickness to see the effect of the temperature of the hot combustion environment on the cooling effectiveness of the model. The main aim of these analysis is to study the effects of the matrix fin on the cooling effectiveness of the combustion liner.

7.1 For hot gas Inlet Temperature 500K

The optimised model of matrix fin effusion cooling system which was designed in the Solidworks 2016 **Fig 7.1** is imported in to the ANSYS Fluent geometry. Inlet velocity of the hot fluid 11.3m/s with temperature 500K is used as hot fluid inlet boundary condition, inlet velocity of cold fluid 20m/s with temperature 300K is used as cold fluid inlet boundary condition **Fig 7.2.** Meshing is done in the fluent setup where Tetrahedrane meshing type is used, number of iterations are conducted till we good convergence with residue of the order 10^{-5}. CFD post is used to analyse the results with the help of the coloured contours and graphs.

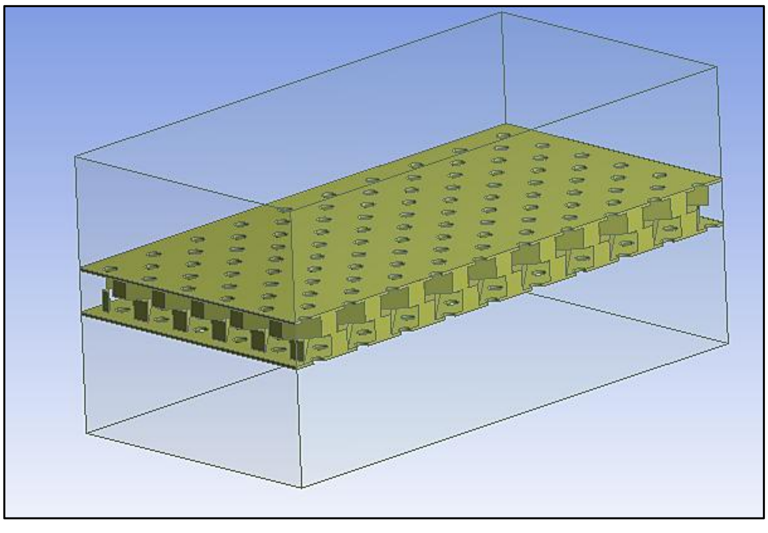


Fig 7.1 Analysis setup of the matrix fin transply effusion cooling part

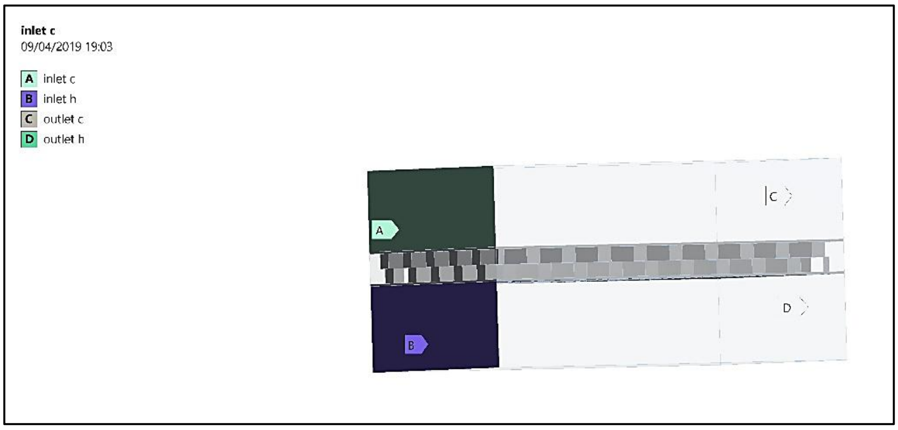


Fig 7.2 Fluid domains with inlet and outlet boundary conditions

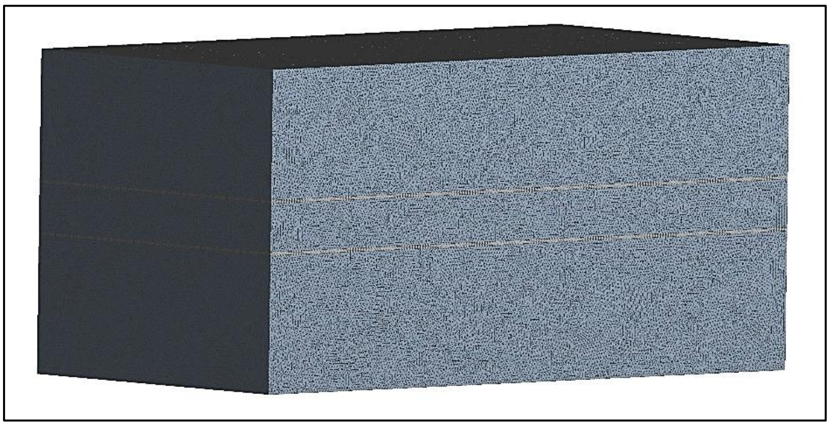


Fig 7.3 Meshing of the model

In the **Fig 7.3** we can see the model with the meshing, the quality and the statistics of the mesh is shown in the graphs. The quality of the mesh is decided by the aspect ratio which was shown in the **Fig 7.4**. For accurate results aspect ratio should be closer to one. The element quality is shown in the **Fig 7.5** for accurate results elements matrix should be closer to the one. The skewness is the main parameter to judge the quality of the meshing **Fig 7.6** for good quality, element matrix should lie nearer to one. In the **Fig 7.7** orthogonal quality of the meshing it is another parameter that decides the mesh quality closer to the unity shows the good quality of the meshing

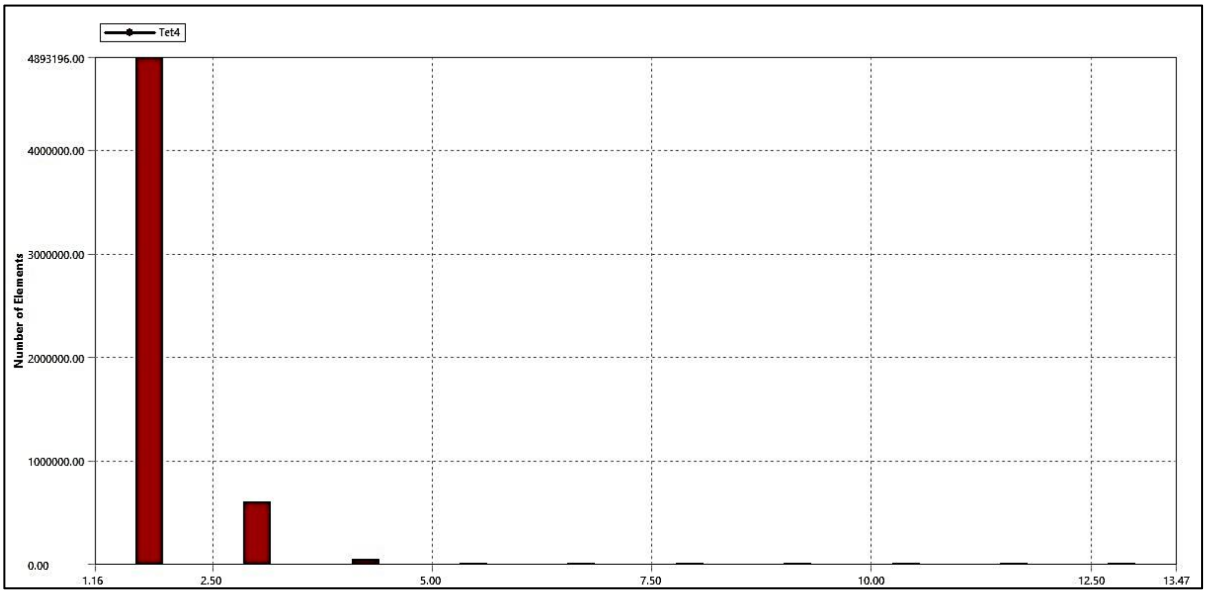


Fig 7.4 Aspect ratio of the meshed elements

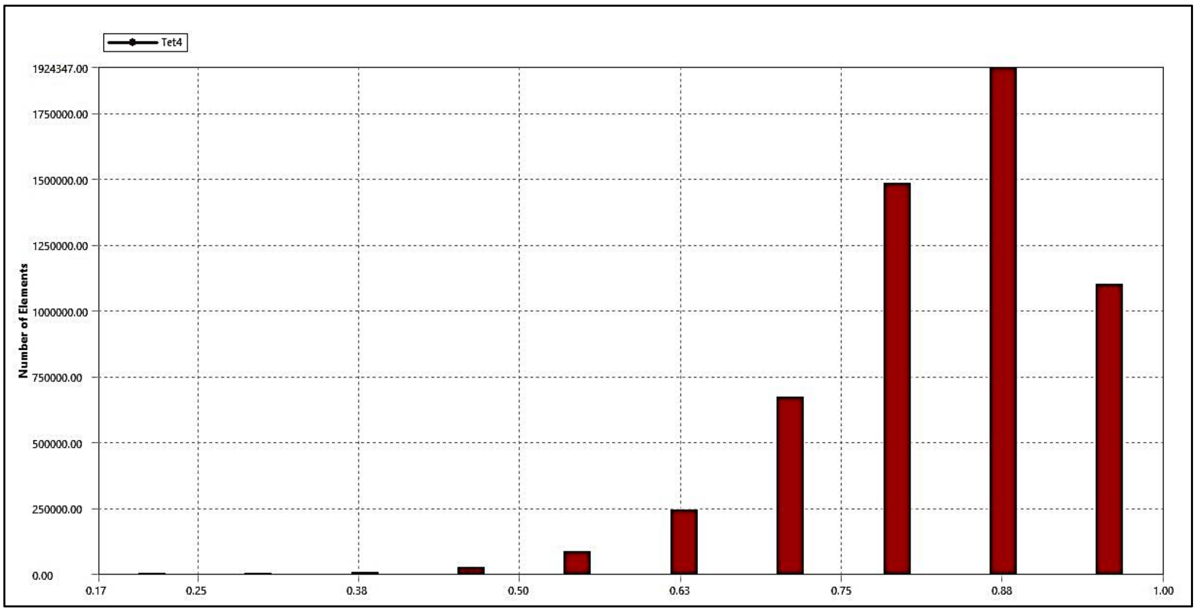


Fig 7.5 Element quality of the meshed element

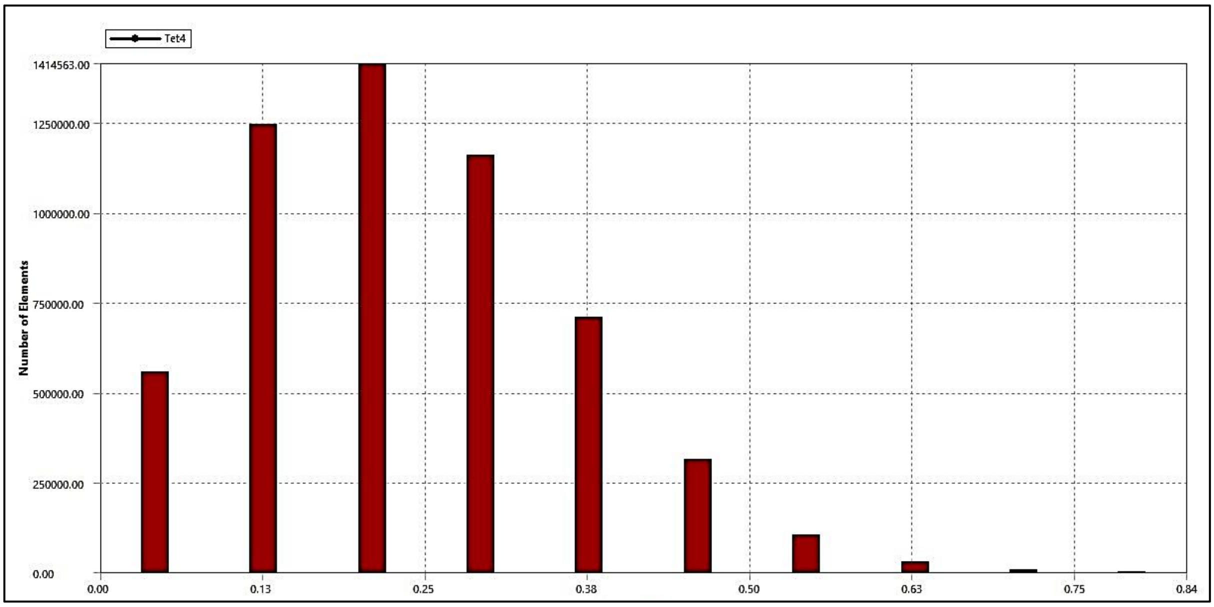


Fig 7.6 Skewness of the meshed elements

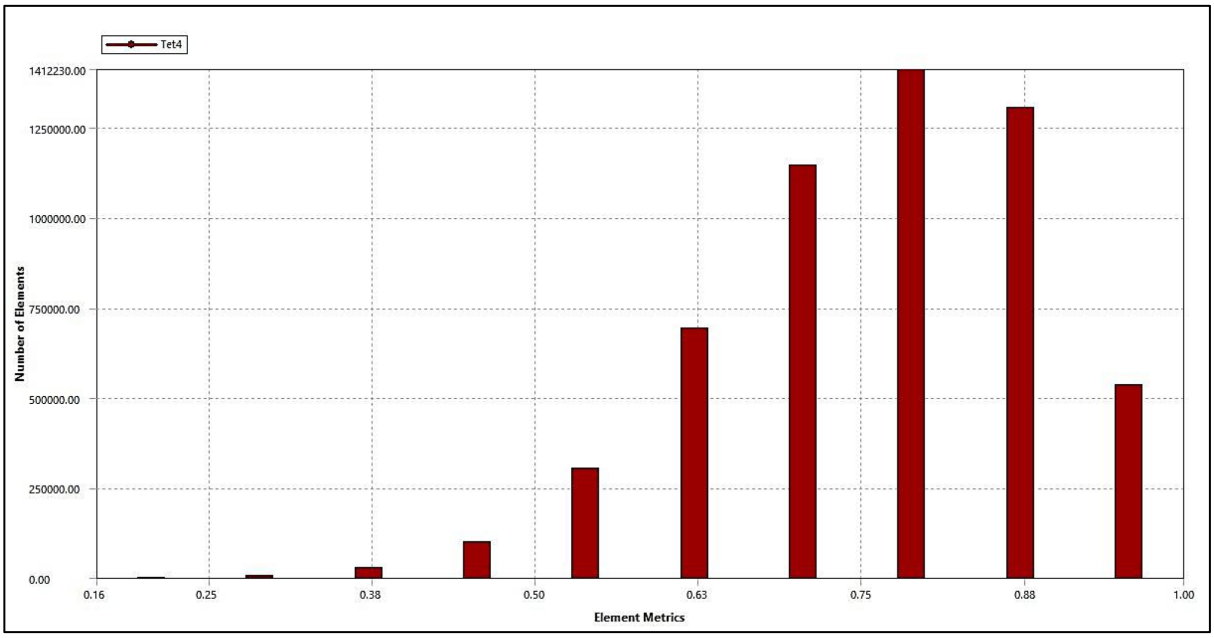


Fig 7.7 Orthogonal quality of the meshing

In CFD post we obtained temperature profile of the matrix fin effusion cooling model, the **Fig 7.8** shows the temperature contour of the model. It can be seen in the figure that the leading edge of the model is hot compare to the trailing edge it is because of the absence of the boundary layer on the hot side of the model

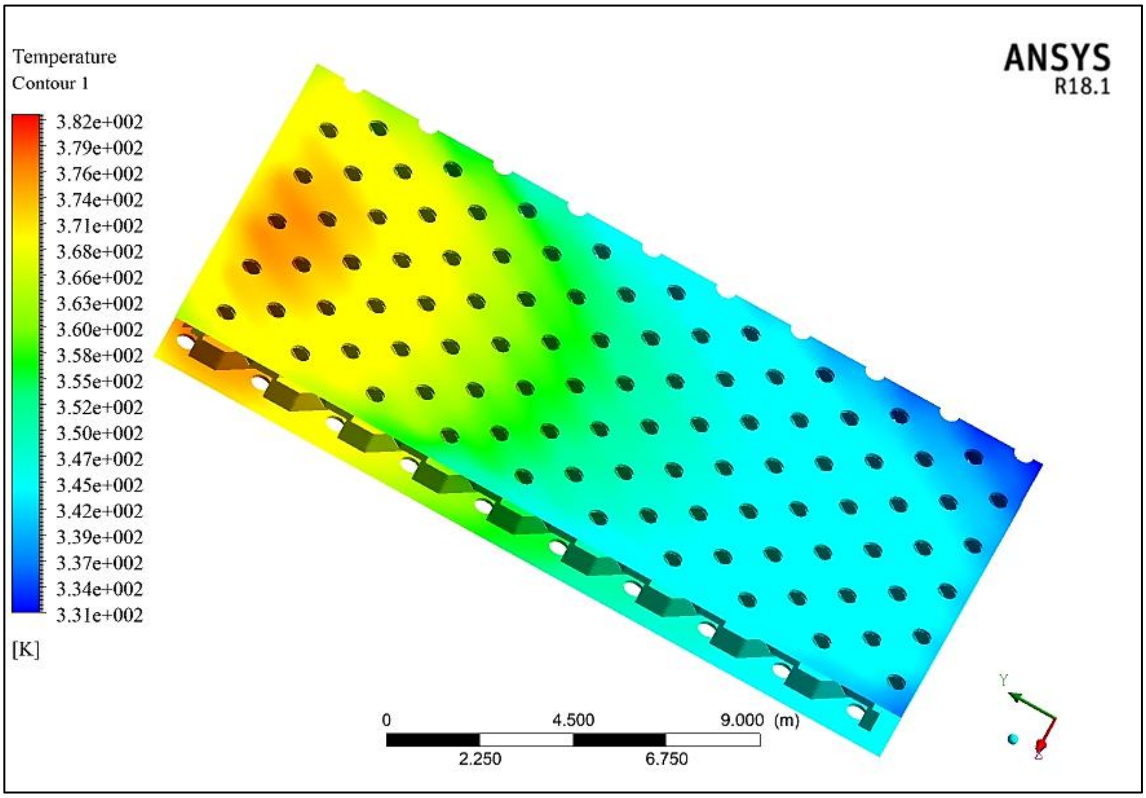


Fig 7.8 Temperature profile of the model

Since the temperature is varying along the length of the model average temperature of the model is used to calculate the cooling effectiveness. Using the functional calculator, we are able to get the maximum temperature equal to 383.315K, minimum temperature equal to 331.22K, and average temperature equal to 356.011K. We can see form the analysis that the maximum temperature is 383.315K that is way below the temperature of the hot combustion environment this is due to the formation of the boundary layer of the cold fluid on the solid surface which is facing the combustion zone. These boundary formations can be seen in the **Fig 7.9** where the cold fluid enters through the effusion holes from the top and pass through the matrix fin and forms the cold fluid boundary layer on the hot side of the plate such that it prevents the hot fluid of the combustion coming in direct contact with the combustion liner.

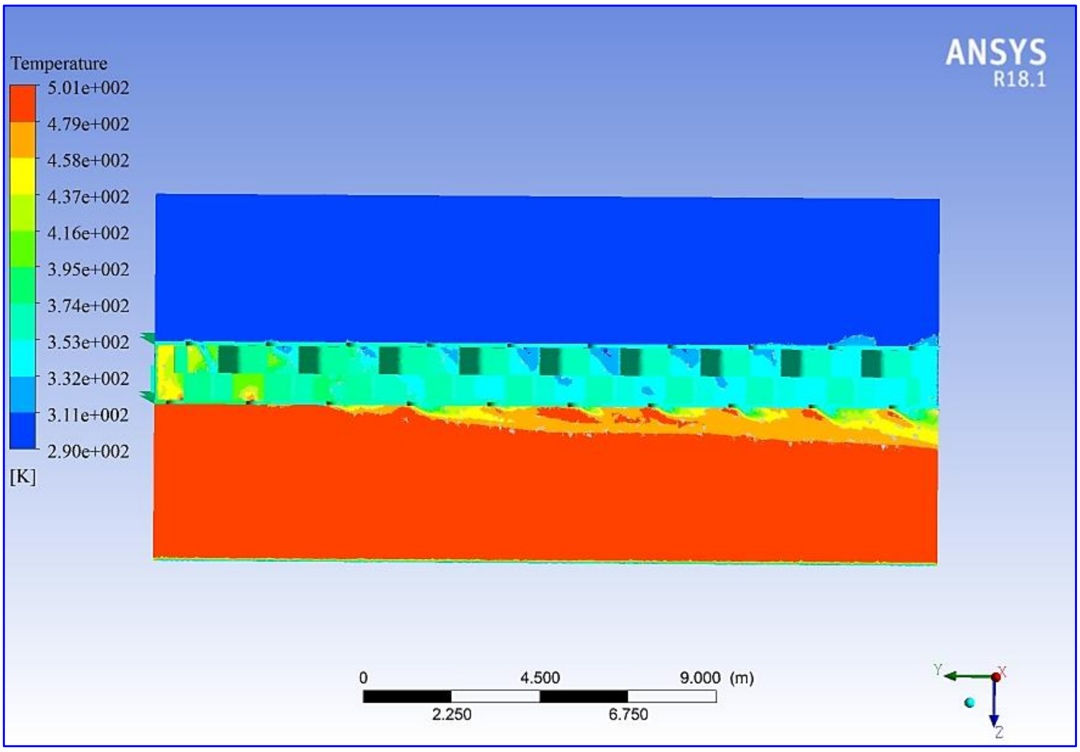


Fig 7.9 Boundary layer formation.

The continuity of the boundary layer can be seen in the **Fig 7.10** where two perpendicular planes are created to see how the boundary layer is attached to the surface along longitudinal and lateral direction of the model. In the we can see that there is a uniform growth in the boundary layer of the cold fluid which then mixes with the combusted air in the secondary zone and enters the turbine with hot air. In lateral direction the depth of the cold fluid is maximum in the region where the holes are present and have grown to cover the whole surface with the cold air boundary layer

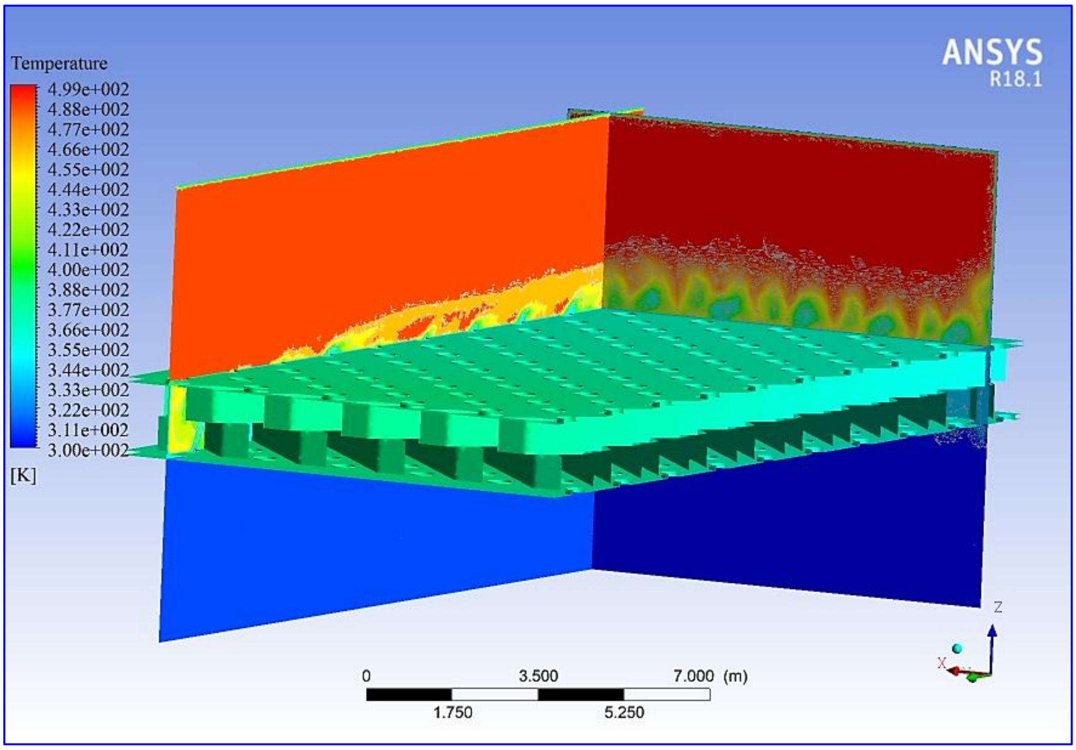


Fig 7.10 Growth of the boundary layer in the lateral direction

In the **Fig 7.11** we can see the temperature variation of the model along the z-direction. The 300K constant temperature line represents the cold fluid temperature that is created to simulate the cold air that is present in the annulus region, the constant temperature line ends at the top surface of the model. From the surface of the model temperature starts to increase as we move towards the hot fluid side, the rise in the temperature is slow for the top fin arrangement due to the direct contact of the cold fluid which is just above the surface. But the temperature rise in bottom fin arrangement is fast when compared to the top fin arrangement it is due to the fact that the bottom surface is facing the hot combustion zone. The temperature of the zone immediate next to the bottom surface is not 500K as it should be since there is hot fluid zone with 500K temperature but instead the temperature is increasing from the bottom surface it is because of formation of boundary layer of cold fluid on the bottom surface, from the graph it is very much clear that the boundary layer formed is preventing the hot fluid coming in direct contact with the surface of the liner. **Fig 7.12** shows the turbulence which is increasing the value of the convection coefficient the internal convection within the thickness of the liner due to the interaction of the fluid with the matrix fin, in the figure we can also observe the turbulence created due to the criss-cross pattern of the matrix fin system

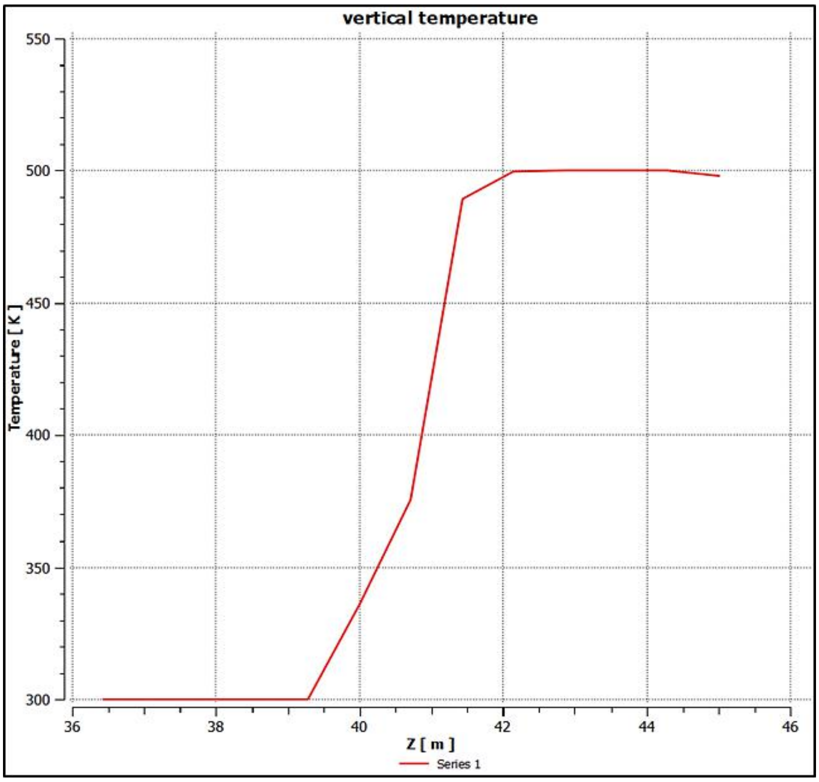


Fig 7.11 Temperature variation along the z direction

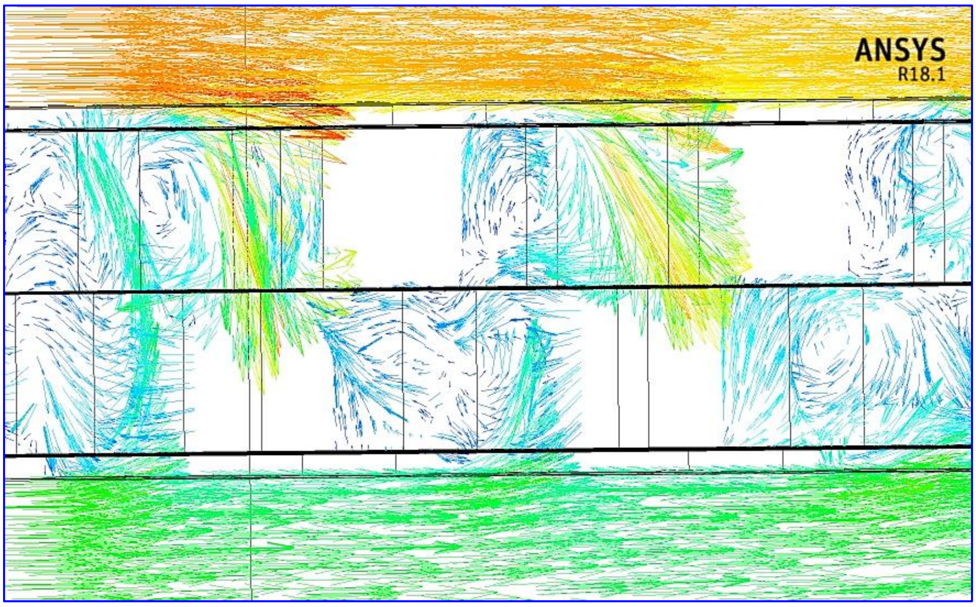


Fig 7.12 Velocity vectors of the flow in the fin channels

From the analysis we are able to obtain the temperature profile and get the average value of the temperature of the model, the obtained temperature is then used to calculate the cooling effectiveness using the formula

Cooling Effectiveness= $(T_G-T_W)/(T_G-T_C)$

Where T_G is hot fluid temperature

T_W is wall temperature

T_C is cold fluid temperature

Average temperature = 356.011 K

Maximum temperature = 383.315 K

Minimum temperature = 331.22 K

Cooling effectiveness $= (500 - 356.011)/(500 - 300)$

$= 0.7199$

7.2 For Hot Gas Inlet Temperature 500K, without the Matrix Fin

The geometries of the effusion cooling system are obtained by the literature review, using these parameters model was designed in the solid works 2016 and imported into the fluent geometry **Fig 7.13**. Fluid domain was created with two inlets and outlets. Inlet velocity of the hot fluid 11.3m/s with temperature 500K(A) is used as hot fluid inlet boundary condition, inlet velocity of cold fluid 20m/s with temperature 300K(B) is used as inlet boundary of cold fluid **Fig 7.14**.

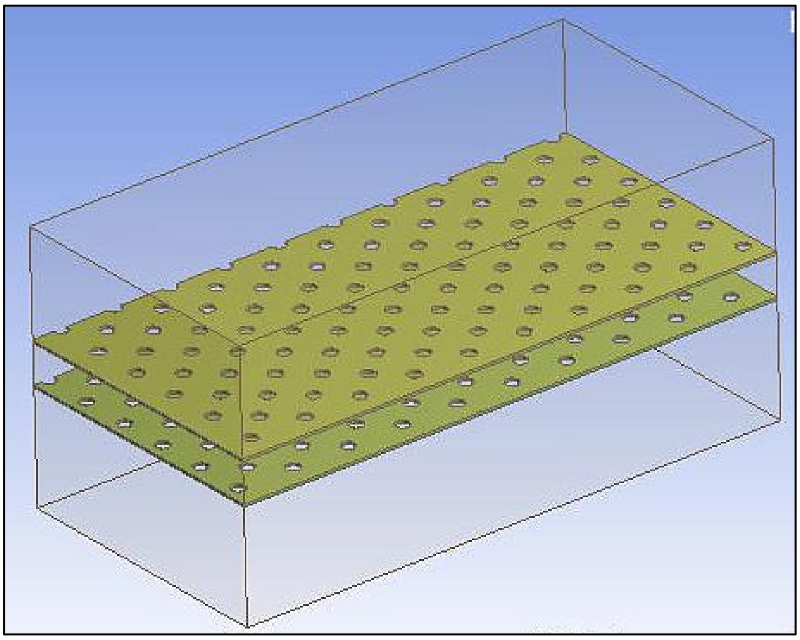


Fig 7.13 Analysis setup of the effusion cooling system

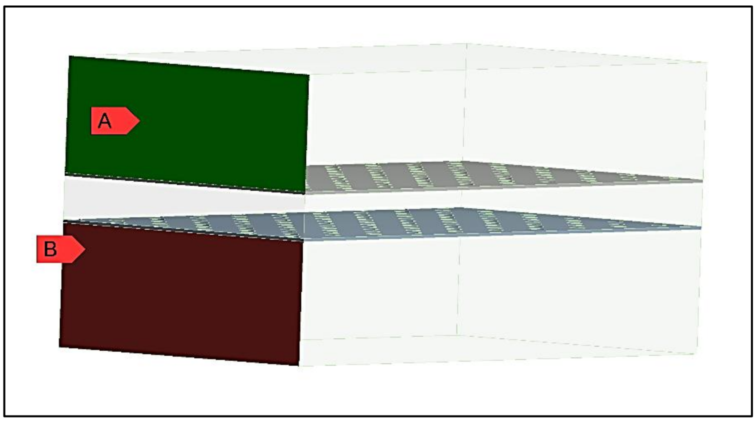


Fig 7.14 Fluid domains with inlet and outlet boundary conditions

In the **Fig 7.15** we can see the meshing of the model, the quality and the statistics of the mesh is shown in the graphs. The quality of the mesh is decided by the aspect ratio which was shown in the **Fig 7.16**. For accurate results aspect ratio should be closer to one. The element quality is shown in the **Fig 7.17** for accurate results elements matrix should be closer to the one. The skewness is the main parameter to judge the quality of the meshing **Fig 7.18** for good quality element matrix should lie nearer to one. In the **Fig 7.19** orthogonal quality of the meshing it is another parameter that decides the mesh quality closer to the unity shows the good quality of the meshing.

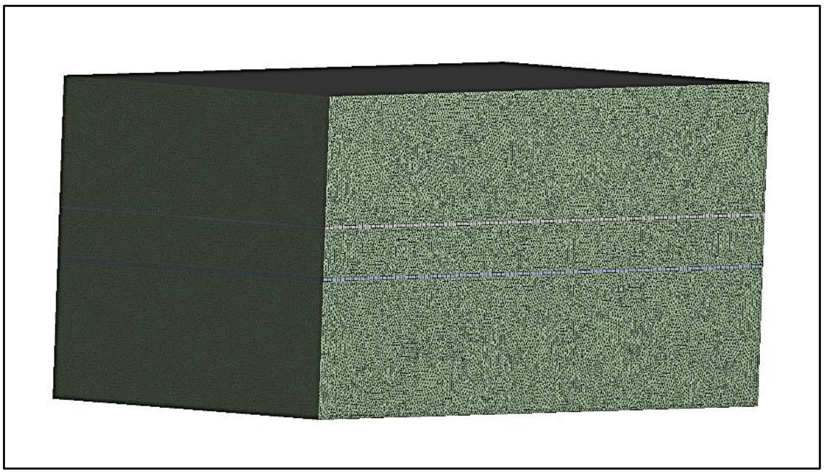


Fig 7.15 Meshing of the model

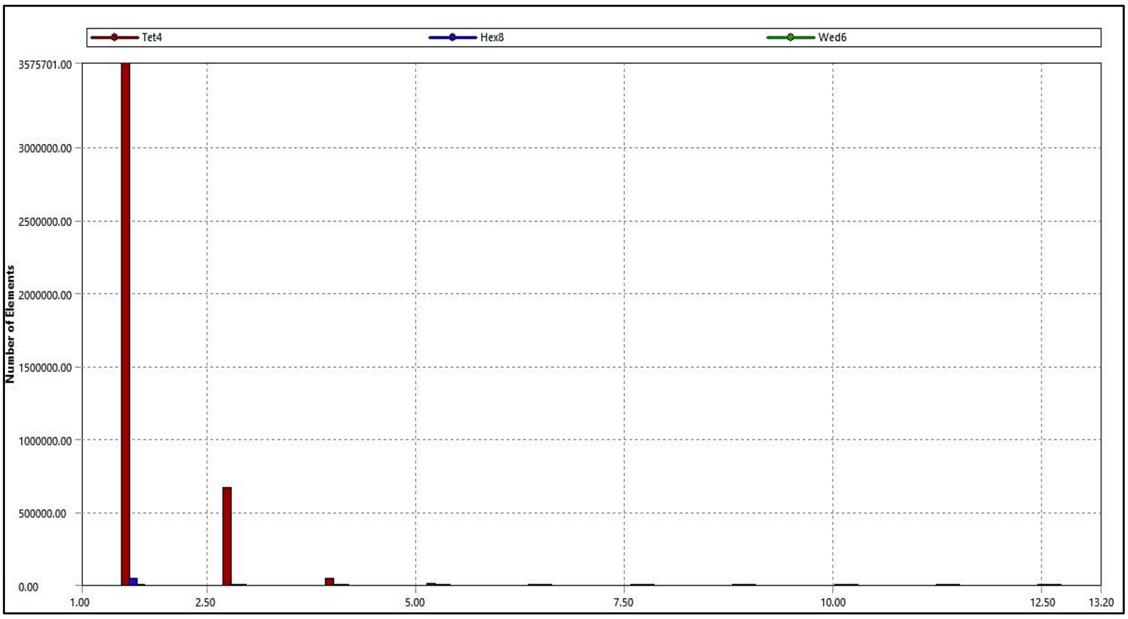


Fig 7.16 Aspect ratio of the meshed elements

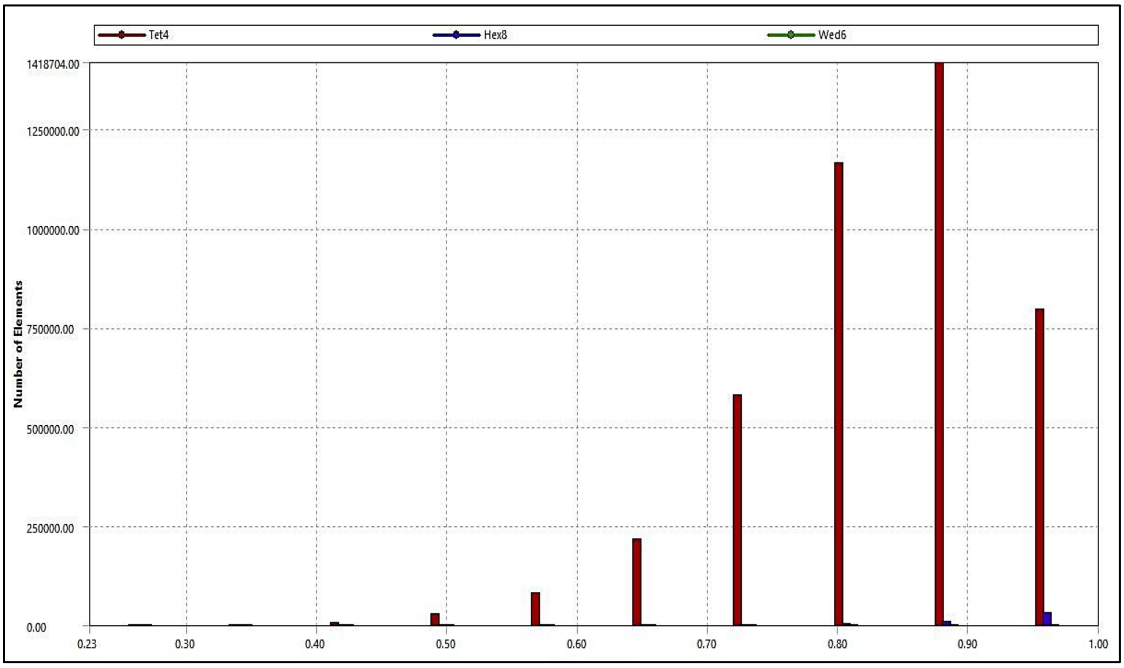


Fig 7.17 Element quality of the meshed element

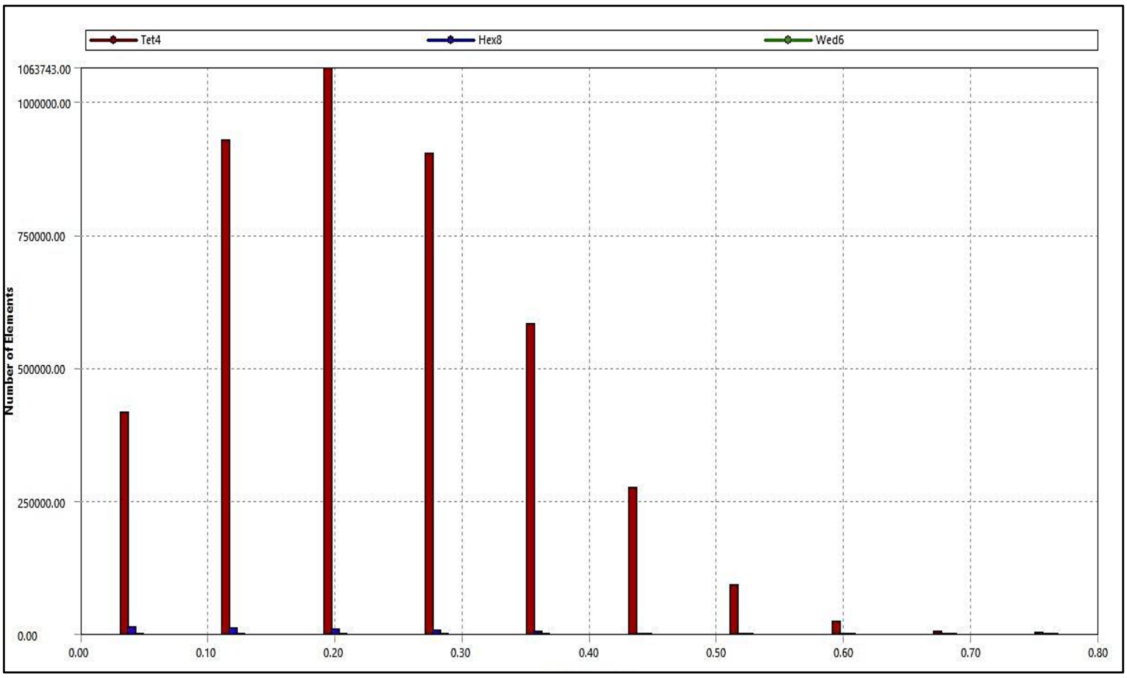


Fig 7.18 Skewness of the meshed element

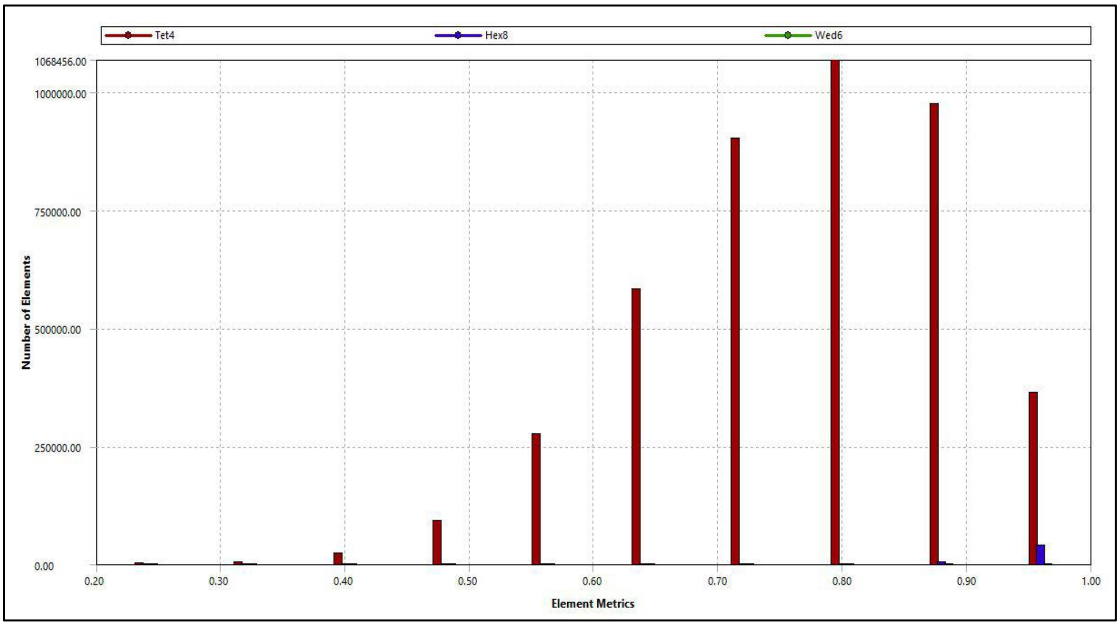


Fig 7.19 orthogonal quality of the meshed element

In CFD post we obtained temperature profile of the matrix fin effusion cooling model, **Fig 7.20** the shows the temperature contour of the model. It can be seen in the figure that the leading edge of the model is hot compare to the trailing edge it is because of the absence of the boundary layer on the hot side of the model

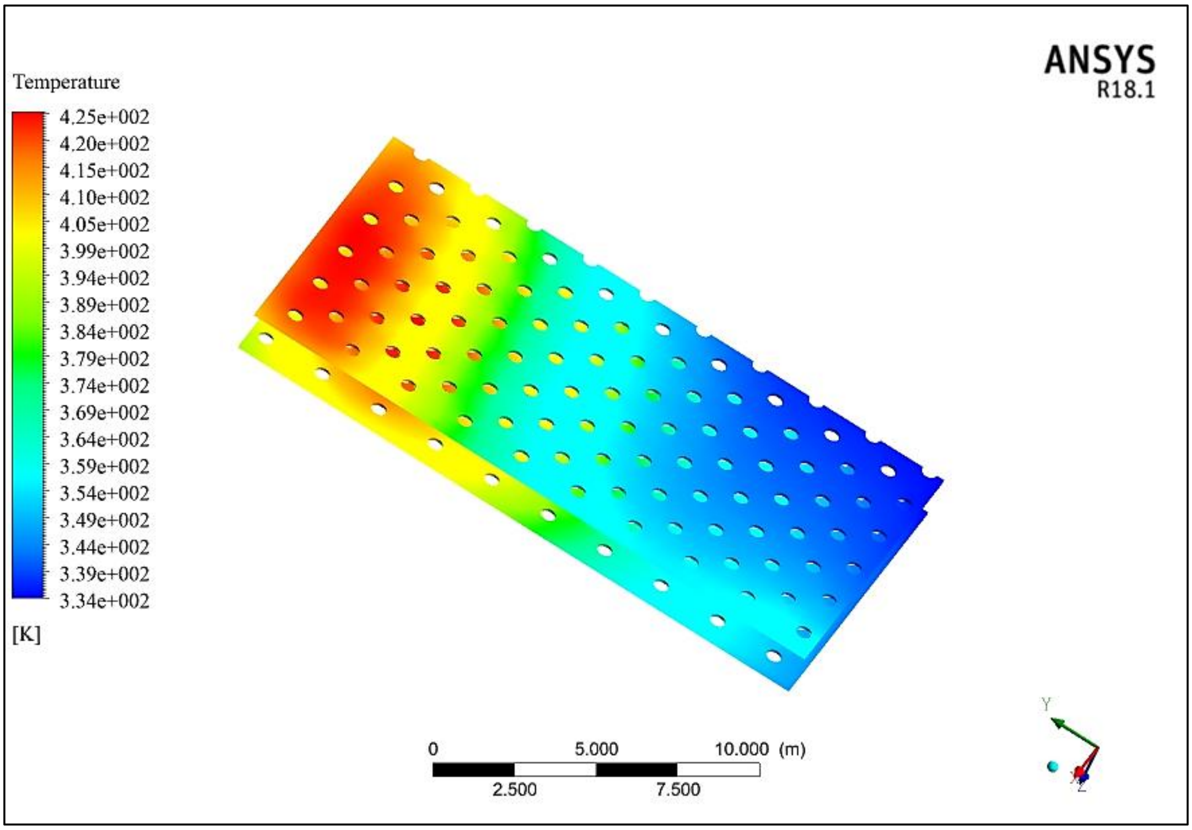


Fig 7.20 Temperature profile of the model

Since the temperature is varying along the length of the model average temperature of the model is used to calculate the cooling effectiveness. Using the functional calculator, we are able to get the maximum temperature equal to 425.135 K, minimum temperature equal to 334.105 K, and average temperature equal to 384.966 K. We can see form the analysis that the maximum temperature is 425.135 K which is more than the maximum temperature of the model with the matrix fin within the thickness. From this higher temperature we can infer that the absence of the matrix fin really effects the internal convection of the liner and the absence of the turbulence not really helping the convection heat transfer from the liner. Even though there is formation of the boundary layer **Fig 7.21** on the hot side of the surface the absence of the internal convection is really affecting the increase in the temperature of the liner temperature

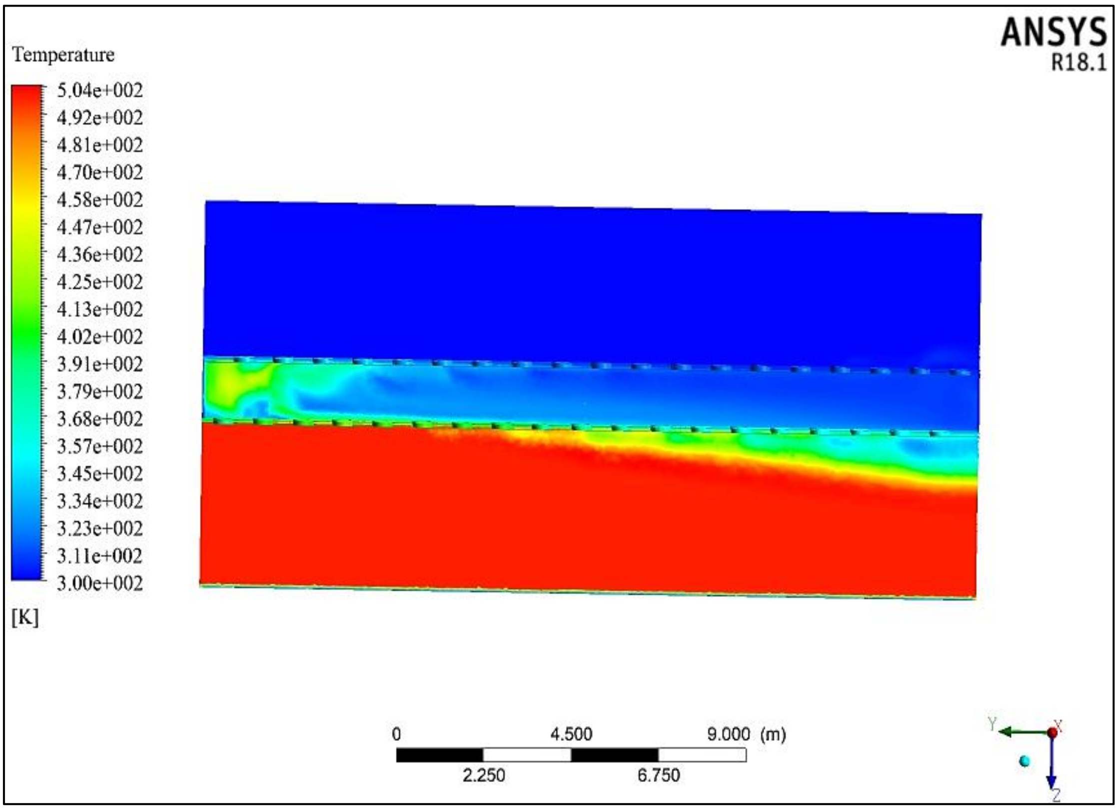


Fig 7.21 Boundary layer formation

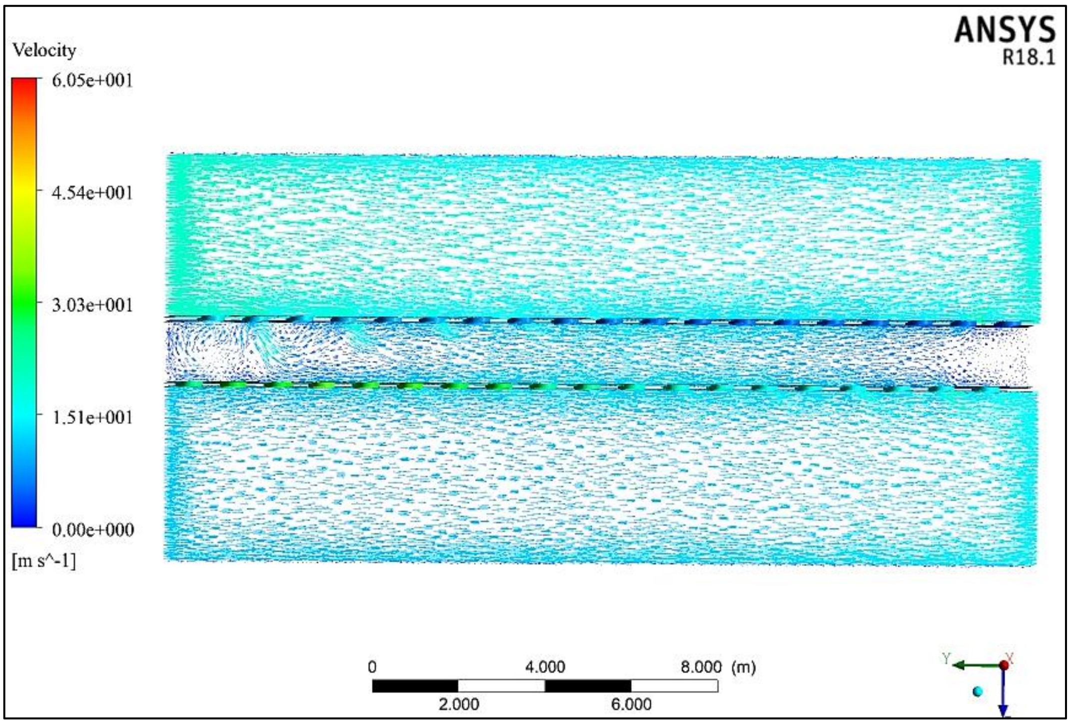


Fig 7.22 Velocity vectors in fluid domain

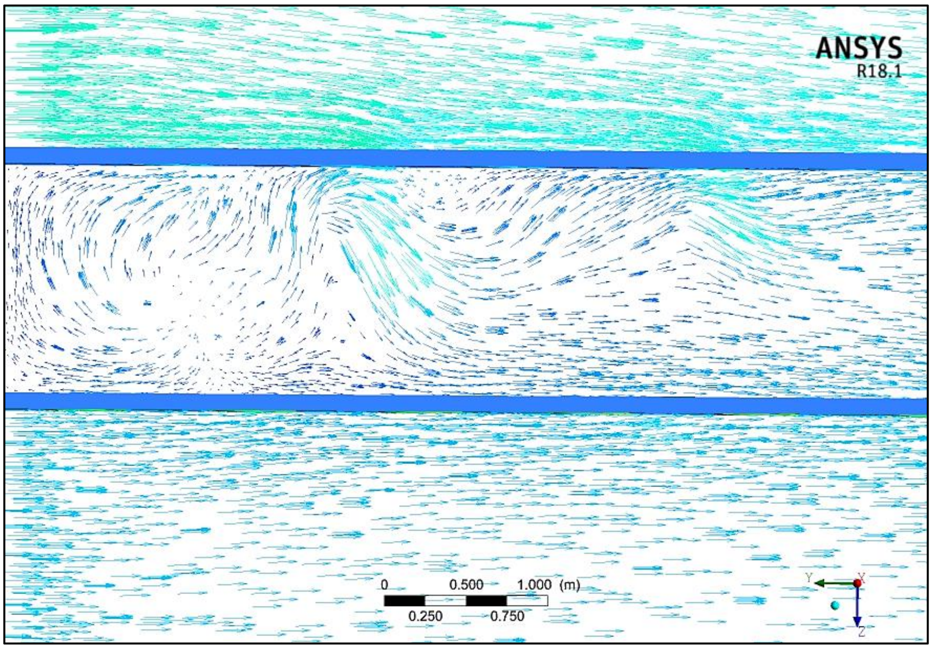


Fig 7.23 Velocity vectors of the flow in the fin

From the analysis we are able to obtain the temperature profile and get the average value of the temperature of the model, the obtained temperature is then used to calculate the cooling effectiveness using the formula

Average temperature = 384.966 K

Maximum temperature = 425.135 K

Minimum temperature = 334.105 K

Cooling effectiveness $= (500 - 384.966)/(500 - 300)$

= 0.5752

7.3 For Hot Gas Inlet Temperature 850K

To see the effect of the temperature of hot fluid we did the analysis on the model with matrix fin using the hot fluid temperature as 850K.The optimised model of matrix fin effusion cooling system which was designed in the Solidworks 2016 **Fig 7.24** is imported in to the ANSYS Fluent geometry. Inlet velocity of the hot fluid 11.3m/s with temperature 850K(A) is used as hot fluid inlet boundary condition, inlet velocity of cold fluid 20m/s with temperature 300K(B) is used as inlet boundary of cold fluid **Fig 7.25**

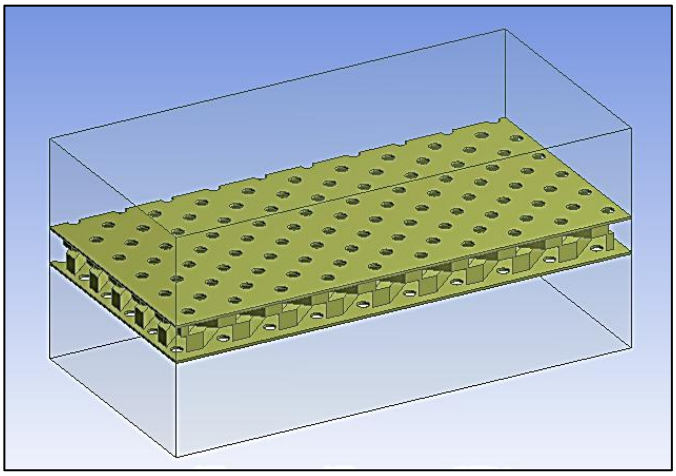


Fig 7.24 Analysis setup of the effusion cooling system

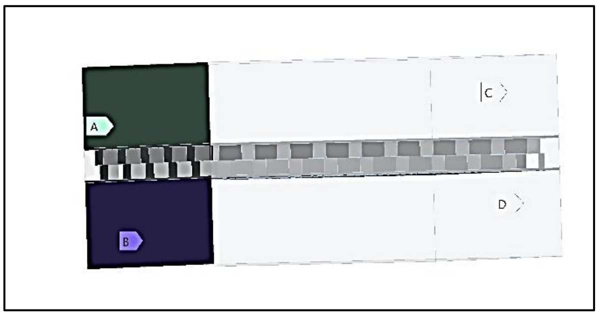


Fig 7.25 Fluid domains with inlet and outlet boundary conditions

Fig 7.26 Meshing of the model

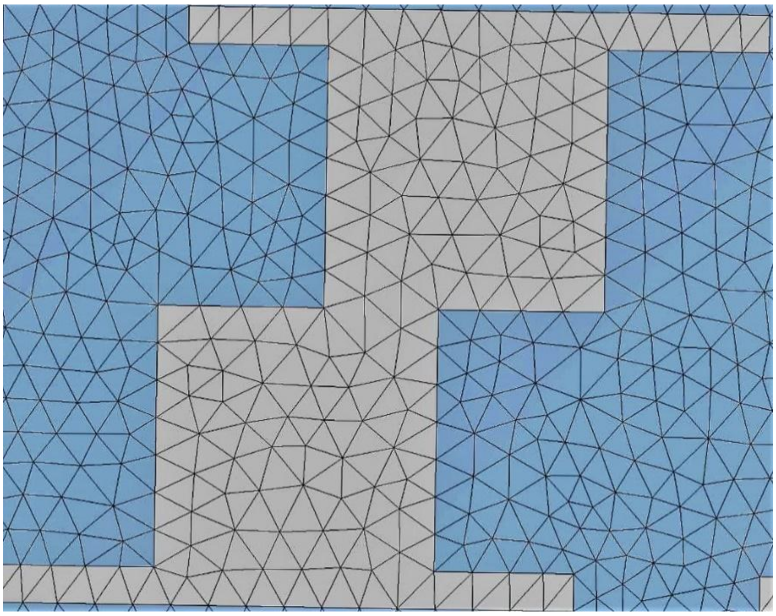


Fig 7.27 Zoomed image of the mesh element

In CFD post we obtained temperature profile of the matrix fin effusion cooling model; the **Fig 7.28** shows the temperature contour of the model. It can be seen in the figure that the leading edge of the model is hot compare to the trailing edge it is because of the absence of the boundary layer on the hot side of the model

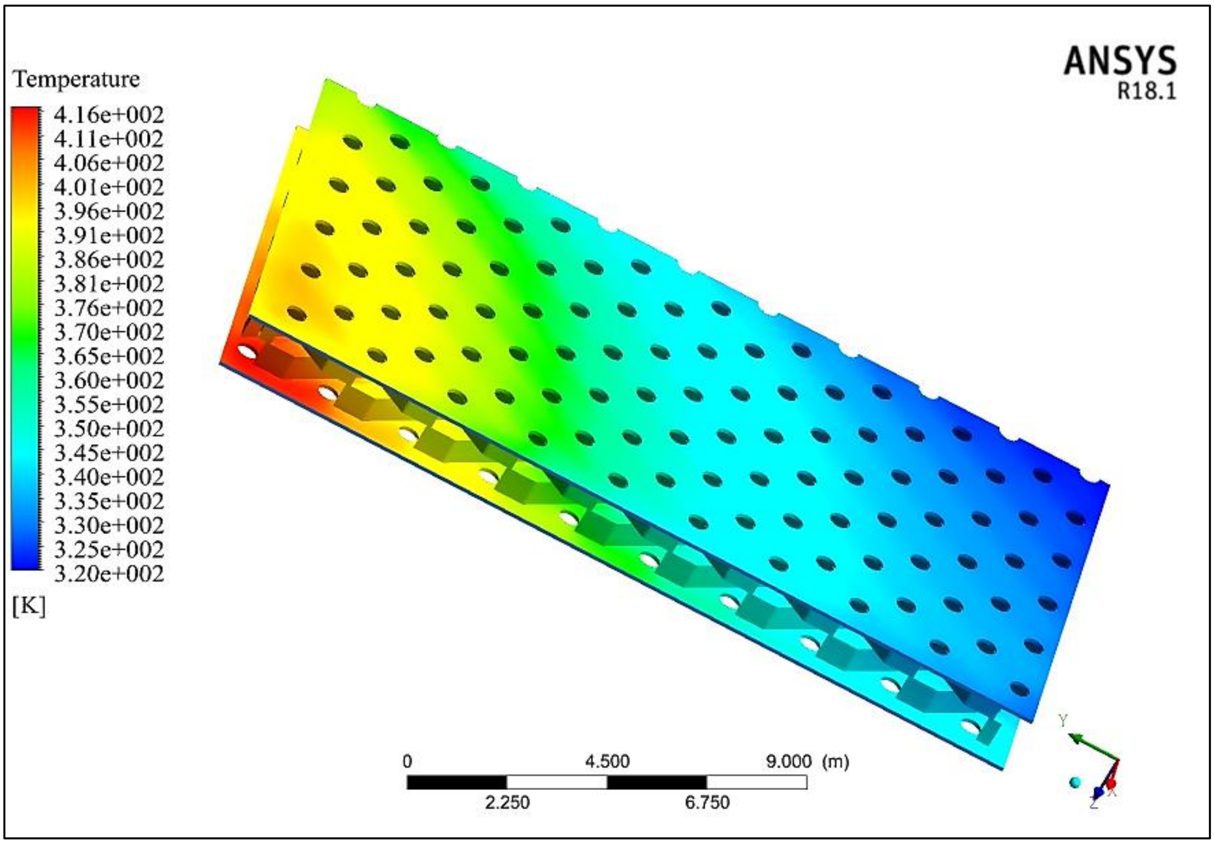


Fig 7.28 Temperature profile of the model

Since the temperature is varying along the length of the model average temperature of the model is used to calculate the cooling effectiveness. Using the functional calculator, we are able to get the maximum temperature equal to 417.14K, minimum temperature equal to 319.829K, and average temperature equal to 357.612 K. We can see form the analysis that the maximum temperature is 417.14K that is way below the temperature of the hot combustion environment this is due to the formation of the boundary layer of the cold fluid on the solid surface which is facing the combustion zone. These boundary formations can be seen in the **Fig 7.28** where the cold fluid enters through the effusion holes from the top and pass through the matrix fin and forms the cold fluid boundary layer on the hot side of the plate such that it prevents the hot fluid of the combustion coming in direct contact with the combustion liner.

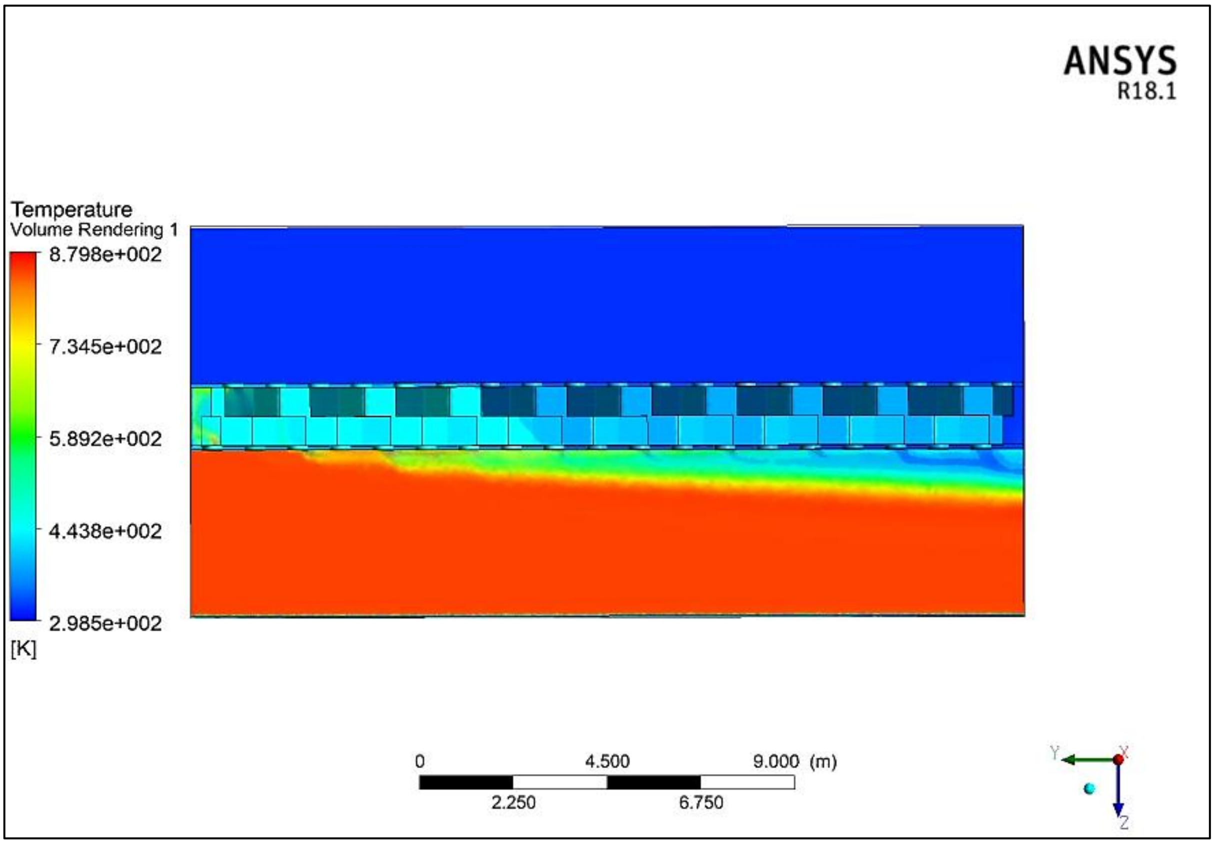


Fig 7.29 Boundary layer formation

The continuity of the boundary layer can be seen in the **Fig 7.29** where two perpendicular planes are created to see how the boundary layer is attached to the surface along longitudinal and lateral direction of the model. In the **Fig 7.30** we can see that there is a uniform growth in the boundary layer of the cold fluid which then mixes with the combusted air in the secondary zone and enters the turbine with hot air. In lateral direction the depth of the cold fluid is maximum in the region where the holes are present and have grown to cover the whole surface with the cold air boundary layer.

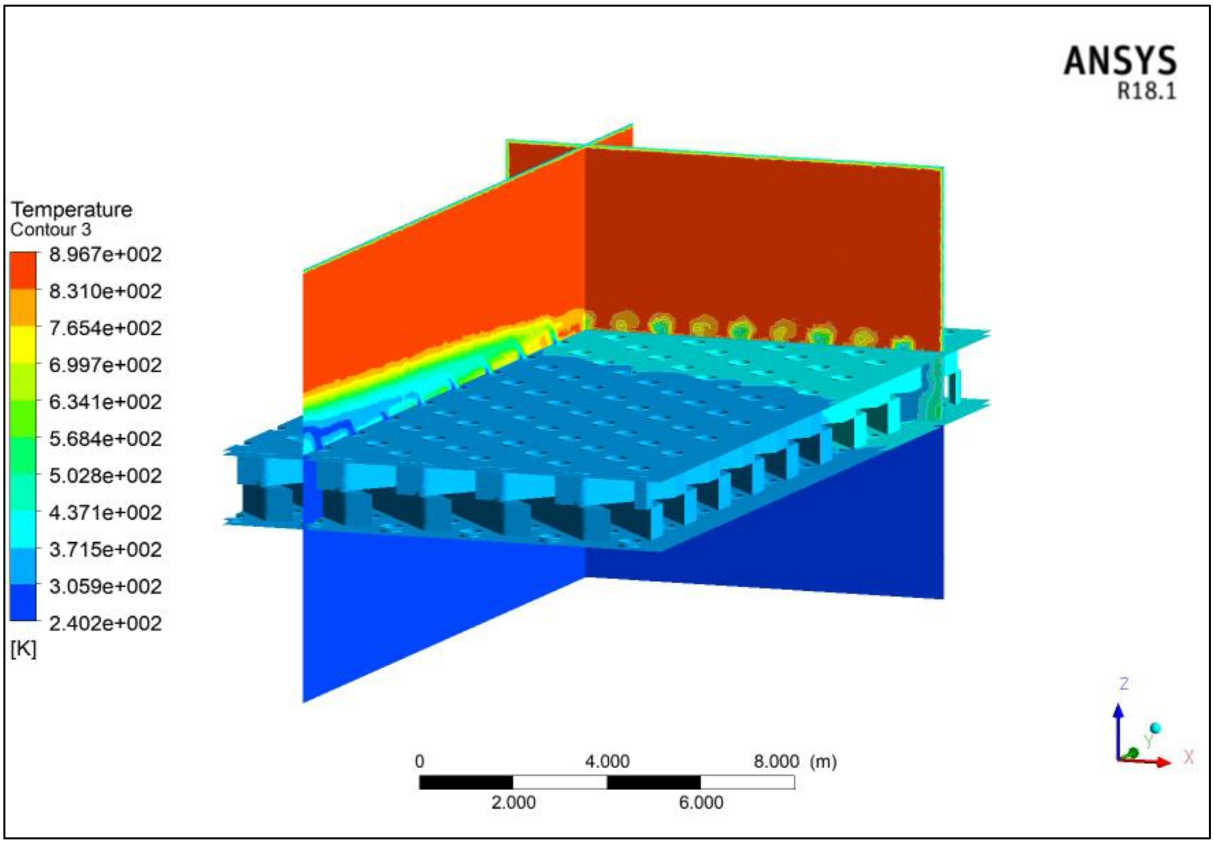


Fig 7.30 Growth of the boundary layer in the lateral direction

From the analysis we are able to obtain the temperature profile and get the average value of the temperature of the model, the obtained temperature is then used to calculate the cooling effectiveness using the formula

Average temperature = 357.612 K

Maximum temperature = 417.14 K

Minimum temperature = 319.829 K

Cooling effectiveness $= (850 - 357.612)/(850 - 300)$

$= 0.8952$

CHAPTER 8

CONCLUSION

8.1 Conclusions

There are multiple parameters whose values influence the performance of the matrix fins. To find the optimisation combination of the parametric values, it is important to use the proper mathematical models which are applicable for the desired operating conditions. MATLAB is used to obtain the optimum parameters of the fin geometry, and the graphs have shown that the change in the velocity of the fluid has no effect on the optimum point. Design of the model is done using these parameters and analysis is done using ANSYS proving that the new matrix fin transply effusion cooling to be more effective than the current transply effusion cooling system. An increase in cooling effectiveness by 14.47% for hot fluid temperature of 500K and 32% for hot fluid temperature of 850K (keeping cold fluid temperature at a constant value of 300K) was obtained by applying the matrix fin in to the effusion cooling system which is a significant improvement. Furthermore, when the combustion chamber temperature increased from 500K to 850K, there was only a 1.6K rise in the average temperature of the wall which implies that it is possible to increase the temperature of the combustion chamber above the metallurgical limitations.

8.2 Scope for future work

1. Applying the matrix fin design as microchannel for compact heat exchanger
2. Doing the analysis by considering the radiation heat transfer.
3. Checking its feasibility in power electronics, high heat-load optical instruments, laser diodes, X-ray medical devices and components where area that needs to be cooled is limited.

REFERENCES

1. Ing. Marco Pievaroli, Matrix Cooling Systems for Gas Turbine Nozzles and Blades: Experimental Investigations and Design Correlation, Department of Industrial Engineering of Florence, 2014.
2. Tomas Gronstedt, Development of methods for analysis and optimization of complex jet engine systems, Department of Thermal and Fluid Dynamics Chalmers University of Technology SE-412 96 Goteborg, Sweden, 2000.
3. M.R. Hajmohammadi, M. Ahmadian, S.S. Nourazar, Introducing highly conductive materials into a fin for heat transfer enhancement, International Journal of Mechanical Sciences 150 (2019) 420–426.
4. Metzger, D. E., Berry, R. A., and Bronson, J. P. Developing heat transfer in rectangular ducts with staggered arrays of short pin fins, ASME J. Heat Transfer, 104:700–706, 1983.
5. Dr. Jassim, M. Abdulkarim Jaff, Mahmood H. Tahir, Yad F. Tahir Shangar S. Sleman, Hero B. Abdullah, Contribution in Development of Design and Performance of Turbine Jet Engine, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 12, Issue 6 Ver.III (Nov. - Dec. 2015), PP 14-23.
6. Rolls-Royce - The Jet Engine-Technical Publications Department, Rolls-Royce plc. Derby, England (1996).
7. R. Shah and D. Sekulic´, Fundamentals of heat exchanger de- sign, 1st ed. Hoboken, NJ: John Wiley & Sons, 2003, pp. 283, 475, 482, 483.
8. A. B. Wassell, J. K. Bhangu, The Development of Application of Improved Combustor Wall Cooling Techniques, ASME 80-GT-66, 1980.
9. Aspi R. Wadia, Advanced Combustor Liner Cooling Technology for Gas Turbines, Def Sci J, Vol 38, No.4, October 1988, pp. 363-380.
10. Performance of Aluminium Silicon Carbide Metal Matrix Composite as a Fin Material, International Journal of Scientific and Engineering Research Vol.5, June 2014, pp. 935-938.
11. Devendra J. Waghulde, Prof. V H Patil, Prof. T. A. Koli, Effect of Geometry Material and Thickness of Fin on Engine Cylinder Fins, IRJET, Vol 4, Issue 7, July 2017, pp. 2192-2195.

12. Mei-Ni Su, Ben Young, Material properties of normal and high strength aluminium alloys at elevated temperatures, Thin-Walled Structures 137 (2019), pp. 463–471.
13. Yu Liu, Hongbo Liu, Zhihua Chen, Post-fire mechanical properties of aluminum alloy 6082-T6, Construction and Building Materials 196 (2019), pp. 256–266.
14. J. Oñoro, M.D. Salvadorb, L.E.G. Cambronero, High-temperature mechanical properties of aluminium alloys reinforced with boron carbide particles, Mater. Sci. Eng. A 499 (2009) pp. 421–426.
15. J.G. Kaufman, Properties of Aluminum Alloys: Tensile, Creep, and Fatigue Data at High and Low Temperatures, ASM International, Material Part, OH, 1999.
16. Ji-Hua Zhu, Ben Young, Numerical investigation and design of aluminum alloy circular hollow section columns, Thin-Wall. Struct. 46 (12) (2008), pp. 1437–1449.
17. K. Budinski and M. Budinski, Engineering materials - properties and selection, 8th Ed. New Delhi: Prentice-Hall India, 2007, pp. 653.
18. Meena KL, Manna A, Banwait SS, Jaswanti (2003), An analysis of mechanical properties of the developed Al/SiC mmc, American journal of Mechanical engineering 1(1): pp. 14-19.
19. L.T. Li, Y.C. Lin, H.M. Zhou, Y.Q. Jiang, Modeling the high-temperature creep behaviors of 7075 and 2124 aluminum alloys by continuum damage mechanics model, Compute. Mater. Sci. (73) (2003) pp. 72–78.
20. Yonghin Ji, Bing Ge, Zhongran Chi, Overall cooling effectiveness of effusion cooled annular combustion chamber liner at reacting flow conditions, Applied Thermal Engineering (2018), pp. 877-888.
21. Zhong Ren, Sneha Reddy Vanga, Nathan Rogers, Internal and external cooling of a full coverage effusion cooling plate: Effects of double wall configuration and condition, International Journal of Thermal Sciences (2018), pp. 36-49.
22. Kyung Min Kim, Hokyu Moon, Jun Su Park, Optimal design of impinging jets in an impingement/effusion cooling system, Energy (2014), pp. 839-848.
23. Jenny Sundberg, Heat Transfer Correlations for Gas Turbines for Gas Turbine Cooling, pp.37-63.