International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Heat Transfer Analysis of Coolant Channel in Liquid Rocket Engine

[1]Sruthi S. Kumar, [2]Dr. R. R. Neela Rajan

[1,2]Department of Aeronautical Engineering, Noorul Islam University, Tamil Nadu, India

A B S T R A C T

The primary objective of cooling in rocket is to prevent the chamber and nozzle walls from becoming too hot, so they will no longer be able to withstand the imposed stresses, thus causing the chamber or nozzle wall to fail. High temperature is essential due to the requirement of high efficiency but the materials will loss their strength when the rocket engines operate at temperatures about 3500K (3,200 °C; 5,800 °F). Any change in temperature brings significant change in the tensile strength, percentage elongation, and elastic modulus of the propellant. When air is heated, the air molecules move further apart and so the air becomes less dense. Thus we would expect our rocket to go higher on a hotter day than on a colder one, assuming all other factors such as humidity are the same. Coolant channels are slots provided in the combustion chamber where the coolant pass through to reduce the temperature of thrust chamber walls. In latest technology, coolant channels with different shapes, aspect ratio & most efficient temperature resistant materials, are manufactured. There are different patterns are used for the coolant channels, because of the variation of improvement in properties. This project work is related to optimize the efficient pattern & shape of the coolant channel in rocket engine and how the variation of heat flux distribution differ with coolant channel patterns and shapes. Steady state thermal analysis has been carried out in Ansys software and the prototype of rocket engine thrust chamber is created using the Catia, RS-25 rocket engine have considered to analyse the performance parameters which affect the cooling of thrust chamber and the exhaust nozzle thus the entire rocket engine components.

***Keywords**: Rocket engine cooling, Coolant channels, Heat transfer, Convection, Heat flux distribution, Regenerative cooling, High temperature.*

1. INTRODUCTION

Thrust chamber designs are generally categorized or identified by the hot gas wall cooling method or the configuration of the coolant passages, where the coolant pressure inside may be as high as 500 atm. The high combustion temperatures (2,500 to 3,500° K) and the high heat transfer rates (about16 kJ/cm²-s) encountered in a combustion chamber present a formidable challenge to the designer. To meet this challenge, several chamber cooling techniques have been utilized successfully. Selection of the optimum cooling method for a thrust chamber depends on many considerations, such as type of propellant, chamber pressure, available coolant pressure, combustion chamber configuration, and combustion chamber material. The efficiency of rockets is about 0.678. Firstly, feels bad, but thermodynamics has something called "Carnot efficiency" which offers maximum technical efficiency for machines making work by heat. Due to the higher propellent efficiency rocket engine is efficient. The more heat a rocket engine produces, cavitation will occur and the more you will have to cool your engine so it doesn't melt, which is not ideal. This means that rocket engines have a fuel to oxidizer ratio slightly off from stoichiometric. The heat flux through the chamber wall is very high; 1-20 MW/m^2 is not uncommon. The amount of heat that can flow into the coolant is controlled by many factors including the temperature difference between the chamber and the coolant, the heat transfer coefficient, the thermal conductivity of the chamber wall, the velocity in the coolant channels and the velocity of the gas flow in the chamber or the nozzle.

1.1. IMPORTANCE OF COOLING IN ROCKET ENGINE

Heat transfer in the thrust chamber is of great importance in the design of liquid propellant rocket engines. Regenerative cooling is an advanced method which can ensure not only the proper running but also higher performance of a rocket engine. Pressure fed engine is cooled using regenerative cooling technique. To compensate for the pressure drops in the piping when fuel flows, the chilled propellant makes it. heavy in density and helps to maintain greater injection pressure. In jet engines, due to large surface area it is impossible to provide the proper cooling of components, thus at lower temperatures efficiency will be low. Rockets has none of these limitations, So the temperatures reached the rocket engines often exceeds the melting points of nozzle and the combustion chamber materials, led to many challenges in designing field. We should keep the rocket engine without melt, vaporize or combust.

1.2. REGENERATIVE COOLING

Regenerative cooling is a widely used cooling method, where the high velocity coolant flows through the coolant walls through the backside of the chamber, After cooling the thrust chamber this coolant will return back and will be utilized as the propellant. It is a cryogenic method in which the rapid expansion of a portion of a gas to be liquefied is utilized to lower the temperature of the remainder. Nanofluids have found a broad range of applications in the field of heat transfer because of the unique characteristic as a special coolant. Many studies have discovered that the addition of the base fluid and thus change its heat transfer performance. Due to Joule-Thomson effect, a portion of gas cooled and it tends to cool the coming gas, thus the cooling effect may be increased many times and this process is called the regenerative cooling. When the initial temperature is low Joule-Thomson cooling effect may increases.

1.3. COOLANT CHANNELS

The walls of thrust chambers in modern liquid rocket engines encounter very high pressures and temperatures. In hydrogen/oxygen liquid rocket engines, the walls are cooled by hydrogen flowing at high flow rate through rectangular micro-channels. Wall cooling effectiveness is of paramount importance to the thrust chamber's life, which can be nearly doubled if wall temperature is reduced by 50-100°C. High cooling performance can be achieved by using a large number of closely spaced thin walls, provided flow in the channels is not severely restricted. A high channel height-to-width ratio both enables maximum utilization of the wall's fin effect and helps prevent high pressure drop in the cooling channels. Depending on type of coolant and operating conditions of the rocket engine, flow boiling may or may not be desired. Flow boiling does offer the advantage of greatly enhancing cooling performance and reducing wall temperature. However, critical heat flux [CHF] inside the cooling channel is a primary concern and must be ascertained and prevented. Flow boiling CHF in this application is complicated by the channel's rectangular shape, high aspect ratio and, most importantly, curvature. Flow boiling is a circulation effect caused by the natural buoyancy or pump, there are several flow patterns will occur and the flow regimes exists according to the thermophysical properties of the fluid. The fuel flow through the coolant channel will cool the thrust chamber and it will get injected into the combustion chamber for combustion. Coolant channels are the slots in the combustion chamber of the rocket engine through which the coolant flows and this coolant helps to provide cooling for thrust chamber and thus the temperature effects reduces. The coolant channel lies between the chamber jacket and the chamber liner, the wall which separates the slots are coolant channel wall. Liquid hydrogen is used as a rocket propellant because it has the highest efficiency relative to the amount used, over any other known propellant. In combination with an oxidizer like liquid oxygen, it is light and able to burn powerful at 3000degree Celsius. Hydrogen has the lowest molecular weight of any known substance, making it ideal for keeping the weight of a rocket relatively small. When combined with liquid oxygen, hydrogen creates the most efficient thrust of any rocket propellant. However its density is low and takes less space for the fuel storage, it can be stored in pressurized and thermally insulated containers after liquification. It possess high specific energy and low volumetric energy density but prone to explosion hazards. The coolant will flow in the opposite direction of the hot gas flow in the chamber. The coolant will flow in the channels with a temperature of 700K and the coolant injection will be at almost 30K. The thrust chamber liner will experience a temperature of 2800K because the coolant with low temperature flown through the slots and thus the temperature will decease near the thrust chamber wall. The inlet temperature of the combustion chamber is about 3500K and the throat temperature is 3200K, the hot gas will eject at a temperature of 2800K.

Different types of coolant channel patterns are opted by the manufactures with different shapes such as rectangular, oval circular, parallelogram, truncated channels and so on. These variety patterns are contributing the cooling of thrust chamber in several ways and some channels built different sections notably 'I' sections in the channel wall which separates the slots. Researches shows that high aspect ratio rectangular channels provide better effects than low aspect ratio slots, they are trying the variations in the edges and corners of these rectangular channels and make new designs with positive results.

Aspect ratio of rectangular slot = Height of coolant channel/Width of coolant channel

In the latest studies other than the rectangular channels, channels with different shapes including the rhombus are designing in the sector.

2. METHODOLOGY

2.1 STEADY STATE THERMAL ANALYSIS

Steady-state thermal analysis is evaluating the equilibrium system of constant heat loads and environmental conditions. A steady- state heat transfer analysis is used to determine the steady-state temperature distribution and heat flow. It is a time independent calculation, the temperature at every point of the prototype will not change with respect to time. Heat flux (Φ) can be defined as the rate of heat energy transfer through a given surface (W) and heat flux density (φ) is defined as the heat flux per unit area (Wm^2). There are three modes of heat transfer conduction, convection and radiation, heat flux depends on temperature difference and heat transfer coefficient. Convection is the primary mode of heat transfer in liquids and gas, heat energy will always transfer from a region of higher temperature to lower temperature. When a fluid is forced to flow by an internal source, then the process is forced convection. The heat transfer coefficient or film coefficient , in thermodynamics is the proportionality constant between the heat flux and the thermodynamic driving force for the flow of heat. Convection cooling can be described by Newton's law of cooling, it states that the rate of heat loss of a body is proportional to the difference in temperatures between the body and its surroundings. The constant of proportionality is the heat transfer coefficient. The law applies when the coefficient is independent, of the temperature difference between object and environment. For the study the three coolant channel patterns were opted, Rectangular shape coolant channel (AR:4), Oval shaped channel and Tubular channels.

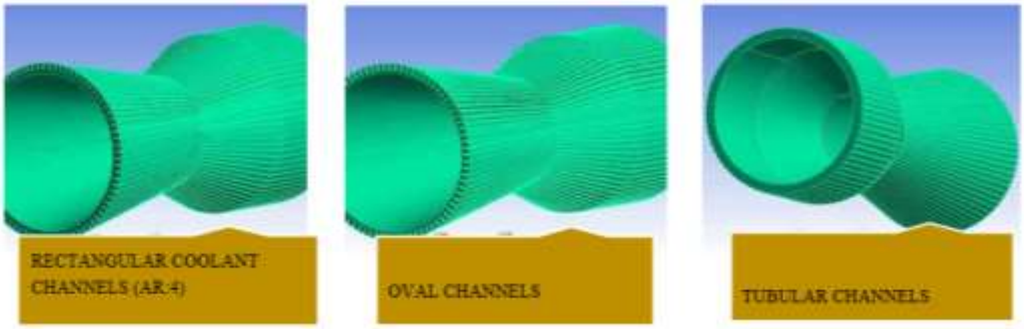


Fig 2.1 Patterns of coolant channels

2.2 BOUNDARY CONDITIONS APPLIED FOR ANALYSIS

After conducting the mesh study, 15mm is chosen as the appropriate mesh size. As we know the mode of heat transfer is forced convection and the heat transfer co-efficient is 650 W/m^2K.The coolant channel temperature is 700K and the liquid hydrogen injection takes place at almost 30K. Meshing is one of the key components to obtaining accurate results from an FEA model. ANSYS meshing technologies provide physics preferences that help to automate the meshing process. Choosing a perfect mesh size is inevitable for the analysis, The inlet temperature of a thrust chamber is about 3500K and the throat temperature is 3200, the hot gas will eject at a temperature of 2800K.

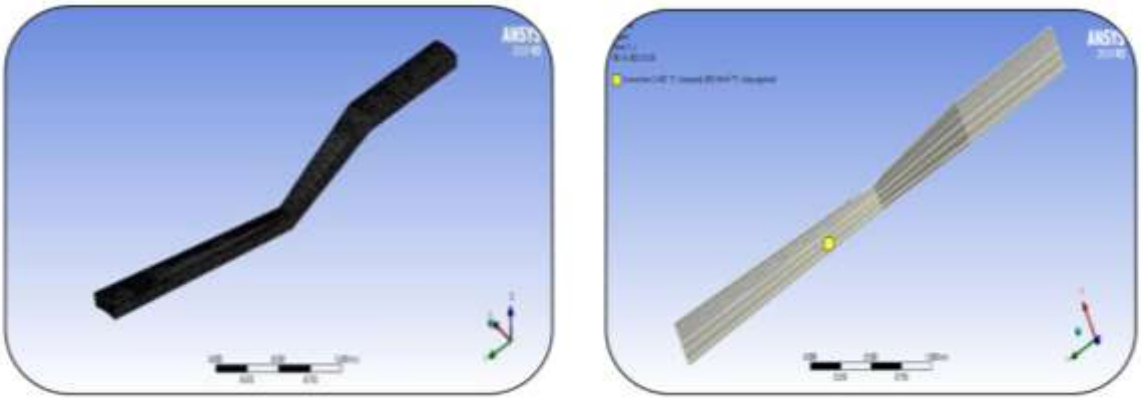
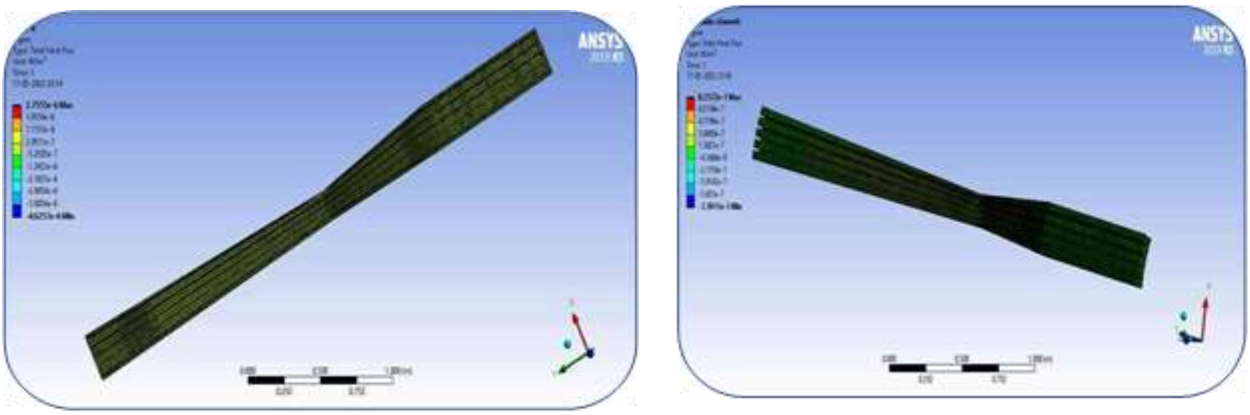


Fig 2.2 Meshed model and forced convection

2.3 TOTAL HEAT FLUX DISTRIBUTION

Due to the complexity of structure for analysis, a portion of thrust chamber is considered for analysis by applying the temperature and other conditions. The molar mass of liquid hydrogen is 2.016 $g{\cdot}mol^{-1}$ and the density is 70.85 g/L.



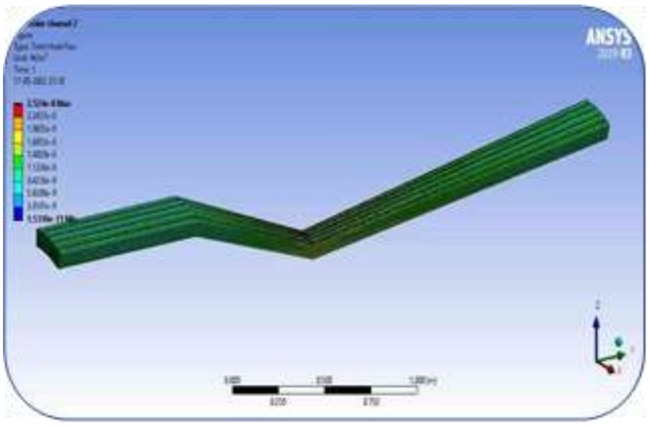


Fig 2.3 a) Rectangular cooling channels b) Oval shaped cooling channels c) Tubular cooling channels.

2.4 DIRECTIONAL HEAT FLUX DISTRIBUTION

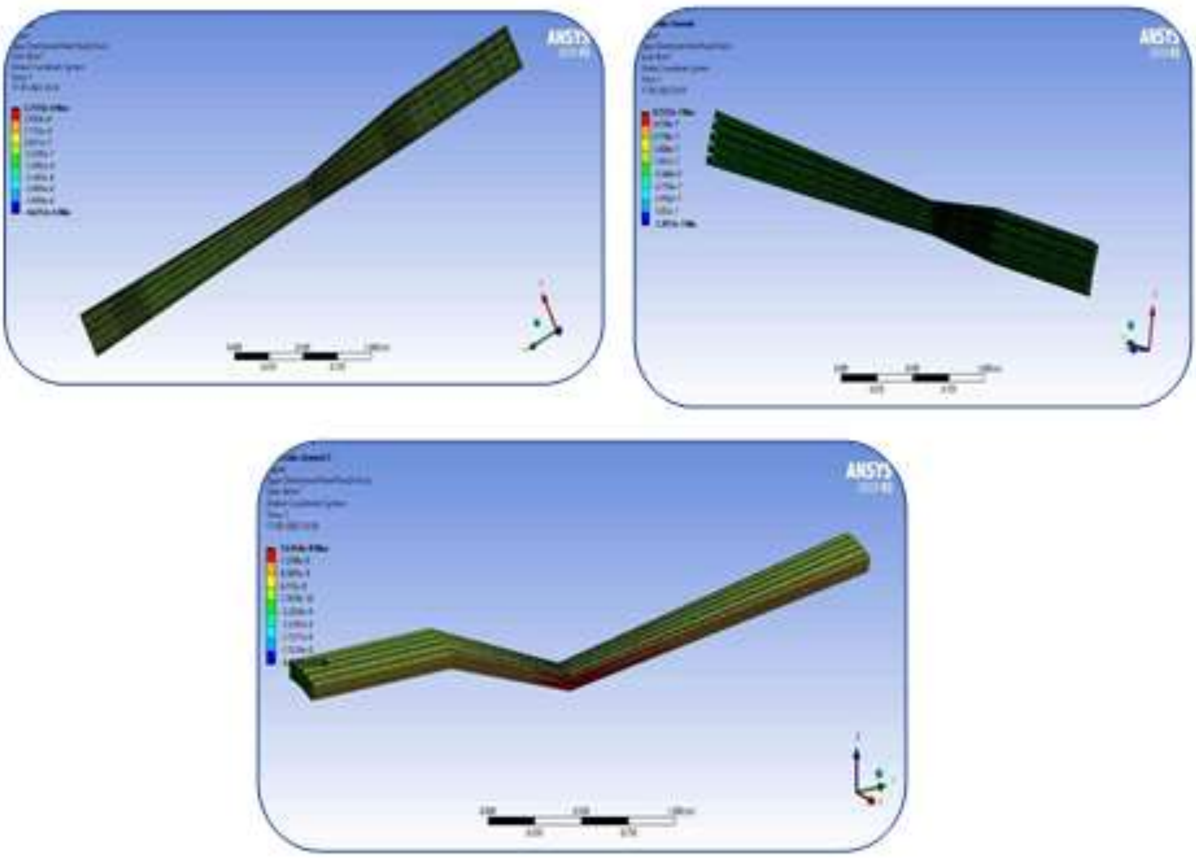


Fig 2.4 a) Rectangular cooling channels b) Oval shaped cooling channels c) Tubular cooling channels.

3.RESULTS

SL NO	SHAPE OF COOLANT CHANNEL	TOTAL HEAT FLUX	DIRECTIONAL HEAT FLUX
1	RECTANGULAR SHAPE (AR:4)	2.755e-6	2.755e-6

2	OVAL SHAPED CHANNELS	8.2572e-7	8.2572e-7
3	TUBULAR CHANNELS	2.524e-8	1.6764e-8

Table 3.1 Heat flux distribution

4.CONCLUSION

The tubular coolant channels with circular cross section have the lowest heat flux distribution and it is the most efficient configuration. The rectangular cross section will cause the distribution of higher heat flux because it has the sharp corners which allows the flow of large amount of heat & the excess temperature. So for the design of coolant channels the aerospace industries prefer the curved shapes especially oval or circular which can offer the better thermal properties than the rectangular cross section .By using the curved /oval/tubular coolant channels the flow pattern which can be disturbed by the injection of the coolant can also be prevented. After the analysis of three types of coolant channels, it is concluded that the tubular channels (with circular cross section) are the most efficient coolant channels which are preferred by the aerospace industry. The curved portion of the coolant channels can offer better performance than the stagnant portions. Thus the flow disturbances caused by the sharp corners can be prevented.

5.REFERENCE

[1] Cooling Channel Analysis of a LOX/LCH4 Rocket Engine Demonstrator, Author: Marco Pizzarelli et.al.

https://www.researchgate.net/publication/266673294

[2] Optimization of Regeneratively Cooled Rocket Engines Cooling Channel Dimensions, Author: Ehsan Atefi et.al.,

https://www.researchgate.net/publication/335213441

[3] Comparison of High Aspect Ratio Cooling Channel Designs for a Rocket Combustion Chamber With Development of an Optimized Design-Mary F. Wade et.al., NASATM--2008-20631

[4] Prediction of Wall Heat Fluxes in a Rocket Engine with Conjugate Heat Transfer Based on Large-Eddy Simulation, Author: Luc Potier et.al.,

https://doi.org/10.3390/e24020256

[5] Numerical simulation and optimization on heat transfer and fluid flow in cooling channels of liquid rocket engine thrust chamber, Author: Quiwang et.al., https://www.researchgate.net/publication/235297215

[6] Thermal structural analysis of regeneratively cooled thrust chamber wall in reusable LOX/Methane rocket engines, Author: Jiawen song et.al., Chinese Journal of Aeronautics, (2017), 30(3):1043-1053

[7] Numerical Analysis of Regenerative cooling in liquid propellant rocket engines, Author: A.Ullas et.al., 1270-9638 doi: 10.1016/j.ast.2011.11.006

[8] Heat transfer aspects and analysis of regenerative cooling in semi-cryogenic thrust chamber with fixed and variable aspect ratio coolant channels, Author: Thomas Santhosh et.al., IOP Conf. series: Material science and engineering: 502 (2019) 012066

[9] Review of film cooling on liquid rocket engines, Author: S.R Shine et.al., https://doi.org/10.1016/ j.jppr.2018.01.004

[10] Thermal analysis on regenerative cooled liquid propellant rocket thrust chamber using ANSYS, Author: Pon Maheshkumar, IOP Conf.series: Material science and engineering doi:10.1088/1757/899X/1059/1/012068