International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online)
Vol. 3, Issue 2, pp: (129-133), Month: October 2015 - March 2016, Available at: www.researchpublish.com

Case Study of Regenerative Cooling in Rocket Combustors Using Rectangular Cooling Channels

SARATH RAJ[1], SUJITH G[2]

[1,2] Assistant Professor, Department of Mechanical Engineering, SNIT, Adoor, India

Abstract: **In a cryogenic engine thrust chamber, the fuel is passed through the regenerative channels in order to take care of high energy combustion. However cryogenic cooling results in wide range of temperature distribution in the cross section and along the axis of the thrust chamber with respect to time. In order to have an understanding of temperature distribution and stable combustion condition an accurate prediction of heat transfer characteristics for the complete spectrum of thrust chamber with respect to time is necessary. The objective of the paper includes the effects of regenerative cooling and the companion conjugate heat transfer. GAMBIT and FLUENT software programs are used as grid generator and solver respectively in the solution. Analysis is done three dimensionally and fluid flow is considered to be transient turbulent flow. The standard k-ε turbulence model is employed. Transient temperature prediction of coolant using CFD simulation is the main study related to this work. The study includes the analysis of regeneratively cooled thrust chamber with rectangular channels with aspect ratio 2:1.**

Keywords: **Cryogenic engine, Regenerative Cooling, k-ε turbulence model, GAMBIT and FLUENT, CFD, Aspect Ratio.**

1. INTRODUCTION

All rocket engines have one problem in common; high energy released by combusted gases. This problem results in high combustion temperatures of around 2400 to 3600 K, a higher heat transfer rates of 0.8 to160 MW/m^2 in thrust chamber and it requires special cooling techniques for the engine. Cooling techniques developed to cope with this problem, either singly or in combination, comprises of regenerative cooling, film or transpiration cooling, radiation cooling, ablation, arid inert or endothermic heat sinks. To choose the proper cooling technique mission requirements, environmental requirements and operational requirements should be considered. Regenerative cooling is one of the most widely applied cooling techniques used in liquid propellant rocket engines. Regenerative cooling of a liquid propellant rocket engine consists of a balance between the energy rejected by the combustion gases and the heat energy absorbed by the coolant. The energy absorbed by the coolant is not wasted; it supplements the initial energy content of the propellant prior to injection, increasing the exhaust velocity slightly by 0.1 to 1.5%. Therefore thermal energy is recovered in the system. Basically there are three domains in a regeneratively cooled rocket engine; gas domain (combusted gases), liquid domain (coolant) and the solid domain (thrust chamber wall). An important issue in rocket combustor design is wall cooling. The very high energy release rates in the rocket engines place a large thermal load on the walls. In cryogenic engines, supercritical hydrogen is commonly used for regenerative cooling, and additional fuel is placed near the wall surface for protection on the hot side. As a coolant hydrogen is very effective, and at supercritical pressures that are required in the combustor, there is no issue of dry out or boiling. In the present paper computational fluid dynamics is used to investigate the rocket engine coolant characteristics in detail. The global scope of the investigation involves the regenerative fluid in the cooling passages, three dimensional channel walls, outer casing and purging gas through the combustor.

2. PROBLEM SOLVING WITH CFD

In solving fluid flow problems we must be aware that the underlying physics is complex and the results generated by a CFD code are at the best as good as the physics and chemistry embedded in it and at worst as its operator. Prior to setting up and running a simulation using CFD software there is a stage of identification and formulation of the problem in terms of its physical and chemical phenomena that need to be considered. Over 50% of time spent in industry on a CFD project is devoted for defining the domain geometry and for grid generation.

3. COMPUTATIONAL FLUID DYNAMICS SIMULATION

The design, scalability, and running of unit operations in chemical process industries rely heavily upon empiricism and correlations of overall variables for non-ideal or non-equilibrium conditions. Many equipment designs in use are based on the past experience of experts applying rules of thumb, resembling art more than science. Processes that are sensitive to the local phenomena and the reactant concentrations are often difficult to design or scale up, because the design correlations do not take local effects into account. Non-idealities introduced by scaling up of lab or pilot scale equipment are difficult, if not impossible to predict accurately.

Researchers, equipment designers and chemical process engineers are increasingly using computational fluid dynamics (CFD) for analysing the flow characteristics and performance of process equipment, such as chemical reactors, stirred tanks, fluidized beds, cyclones, spray dryers, pipeline arrays, combustion systems, heat exchangers and other equipments. CFD allows for in depth analysis of the fluid mechanics, local effects, and chemistry in these types of equipment such as turbulence and combustion. CFD can be used when design correlations or experimental data are not available. It provides comprehensive data that are not easily obtainable from experimental tests. It highlights the root causes, and the effect in a short time. This method reduces scale-up problems, because the models are based on the fundamental physics and are scale-independent.CFD is basically the science of predicting heat transfer, chemical reactions, mass transfer, fluid flow characteristics and related phenomena by solving the mathematical equations that govern these processes using numerical algorithm. It is the combination of the classical branches of theoretical and experimental science, with the infusion of modern element of numerical computation. The results of CFD analysis are the relevant engineering data that are used in conceptual studies of new designs, product development, and redesign. In many cases, CFD results in better insight, better reliability, improved performance, more confident scale-up, improved product stability, and higher plant productivity. The progress of CFD during the last fifty years has been extraordinary. Much of this progress has been driven by the outstanding increases in digital computing speed. The continual and exponential increase in computing power, also improved the physical models in many CFD codes, and better user interfaces now enables non-experts to use CFD as a design tool on day-to-day basis. As a consequence, CFD has progressed from the domain of mainframe to the high-end engineering workstation and even to laptop PCs.

3.1 Working of a CFD code:

CFD codes are organized based on numerical algorithms that can tackle fluid flow problems. In order to provide easy access for solving all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. All codes contain 3 main elements.

(i) Pre-Processor (ii) Solver (iii) Post-Processor

Pre-processing include the input of a flow problem to a CFD program by means of an operator friendly interface and the successive transformation of this input into a form suitable for use by the solver. There are three variants of numerical solution techniques like finite difference, finite element and spectral methods. In outline the numerical methods that form the foundation of the solver perform the following steps. i.e. approximation of the unknown flow variables, discretization by substitution of the approximations into the governing flow equations and successive mathematical manipulations and the solution of the algebraic equations. The main differences between the three streams are related with the method in which the flow variables are approximated and with the discretization processes. As in pre-processing a huge amount of development work has been recently taken place in the post-processing field. Due to the increased popularity of engineering work stations, many of which have outstanding graphics capabilities, the leading CFD packages are now available with versatile data visualization tools. These include domain geometry and grid display, line and shaded contour plots, vector plots, 2D and 3D surface plots, particle tracking, View manipulation, colour postscript output etc.

3.2 Problem Solving With CFD:

In solving fluid flow problems we need to know about that the underlying physics is complex and the results generated by a CFD code are as good as the physics and chemistry embedded in it and at worst as its operator. Prior to setting up and running a CFD simulation problem there is a stage of identification and formulation of the flow problem in terms of the physical and chemical phenomena that are to be considered. Over 50% of time spent in industry on a CFD project is devoted to the definition of the domain geometry and grid generation.

3.3 Computational Fluid Dynamics Simulation:

Many equipment designs in use are based on the past experience of experts applying rules of thumb, resembling art more than science. Processes that are very much sensitive to local phenomena and reactant concentrations are often difficult to design, because the design correlations do not take local effects into account. Researchers, equipment designers, and the process engineers are increasingly using CFD to analyze the flow and performance of process equipment, such as chemical reactors, fluidized beds, cyclones, combustion systems, stirred tanks, spray dryers, pipeline arrays, heat exchangers and other equipment. CFD allows for in depth analysis of the fluid mechanics, local effects, and chemistry in these types of equipment such as turbulence and combustion. CFD can be used when design correlations or experimental data are not available. It provides comprehensive data that are not easily obtainable from experimental tests. It highlights the root cause, not just the effect and many 'what if' scenarios can often be analyzed in a short time. This method reduces scale-up problems, because the models are based on the fundamental physics and are scale-independent.

CFD is basically the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving equations that govern these processes using numerical algorithm. It is the merger of the classical branches of experimental and theoretical science, with the infusion of the modern element of numerical computation. The result of CFD simulations are the relevant engineering data used in conceptual studies of new designs, detailed product development, troubleshooting, and redesign. In many cases, CFD results in better insight, improved performance, more confident scale-up, improved product consistency, better reliability and higher plant productivity. The progress of CFD during the last fifty years has been extraordinary. Much of this progress has been driven by the phenomenal increases in digital computing speed. The continual and exponential increase in computing power, improved physical models, and improved user interfaces now enables non-experts to use CFD as a design tool on day-to-day basis. Thus CFD has progressed from the domain of mainframe to the high-end engineering workstation and even to laptop PCs. This power of digital computing has transformed engineering and research especially in fluid flow analysis, just as it has in virtually all fields of human endeavours.

4. MODELLING

The overall length is 100mm, height 6mm and width 3 mm.The cooling channel is made up of copper and water is taken as coolant. Figure 1 shows the 3D modelling of cooling channel and Figure 2 shows it's meshing.

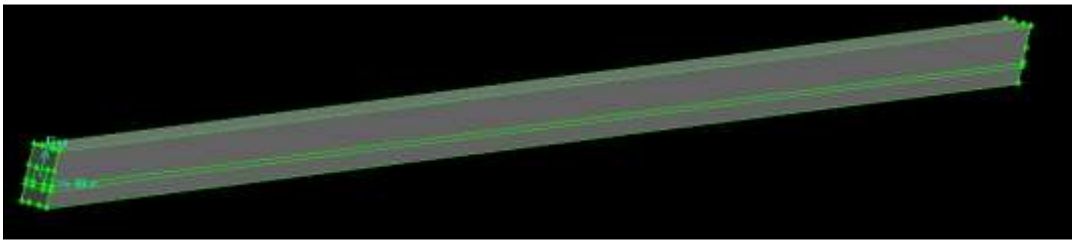

Figure 1: 3D Model of cooling channel.

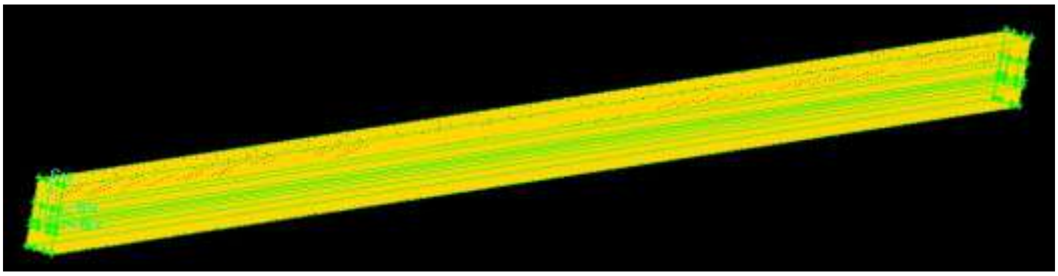

Figure 2: Meshed model

Analysis is done using k-ε turbulence model using FLUENT. Boundary conditions for the Computation involve temperature of the entering water is taken as 300K and its velocity as 10 *m/s*. An adiabatic boundary condition is applied on the outside surface of the combustor. Upstream conditions were taken as 3000K and 500 m/s for the combustion gases as well as on the intermediate surface.

5. RESULTS AND DISCUSSIONS

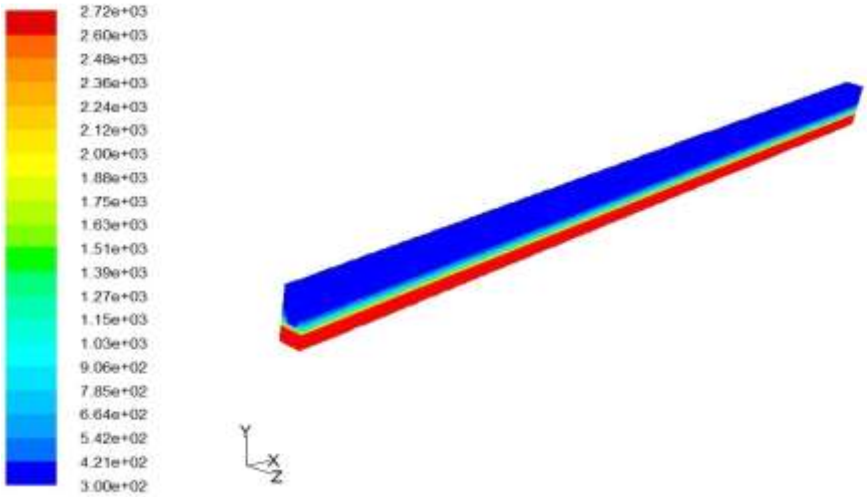

Figure 3:Three-dimensional schematic of computational domain

A three-dimensional schematic of the computational domain and the converged temperature profiles is shown in Fig. 3. In this figure, the flow goes from left to right through the passage. The lower red area represents the hot gas, while the upper area which is blue represents the copper wall. The wall heating by the gas is clearly seen at the bottom of the copper wall.

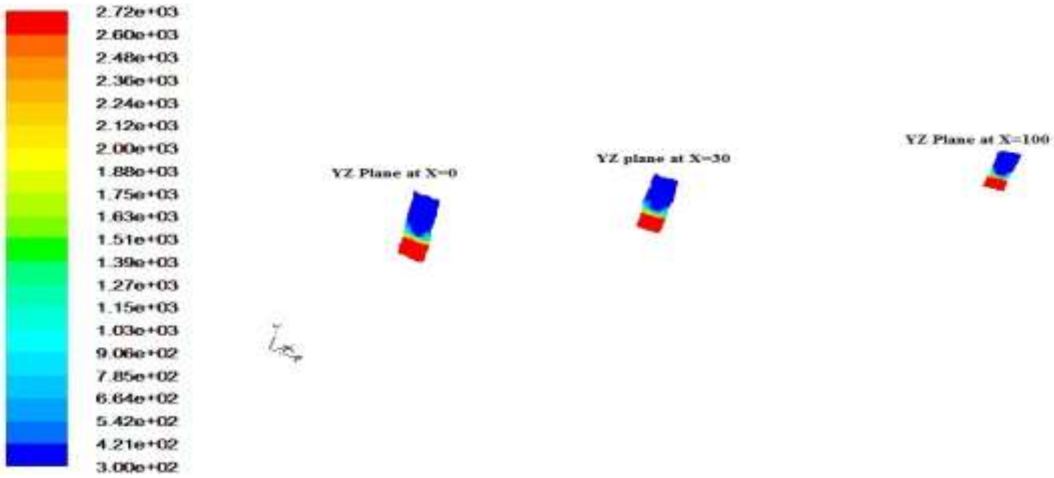

Figure 4: The temperature contours in a series of three slices across the domain at right angles to the passage

The plane on the left shows the temperature contours at the upstream end, x = 0. The succeeding plots from left to right correspond to x = *30*, and *100 mm* downstream. Because of the relatively short length of the tube, the effects of heating are not immense, but several interesting phenomena appear. At the upstream end, the temperature at the copper wall increases rapidly with the maximum temperature being at the symmetry plane between the channels. With distance downstream, it is observed that the temperature at this symmetry plane first decreases slightly before again starting to increase. This decrease in the temperature value is a direct reflection of the high conductivity of the copper compared to the conductivity of the cooling fluid that conducts heat rapidly away from the copper/hot-gas interface. After this initial drop in the value, the fluid temperature near the copper walls again starts to increase. The temperature value of the thin layer of copper between the coolant and the hot gas also reduces with axial distance as the boundary layer of the gas thickens. It is emphasized that these results represent conditions at the entrance to the tube, and that on increasing distance downstream will continue to show increased heating as the temperature at the copper wall and the coolant continue to increase.

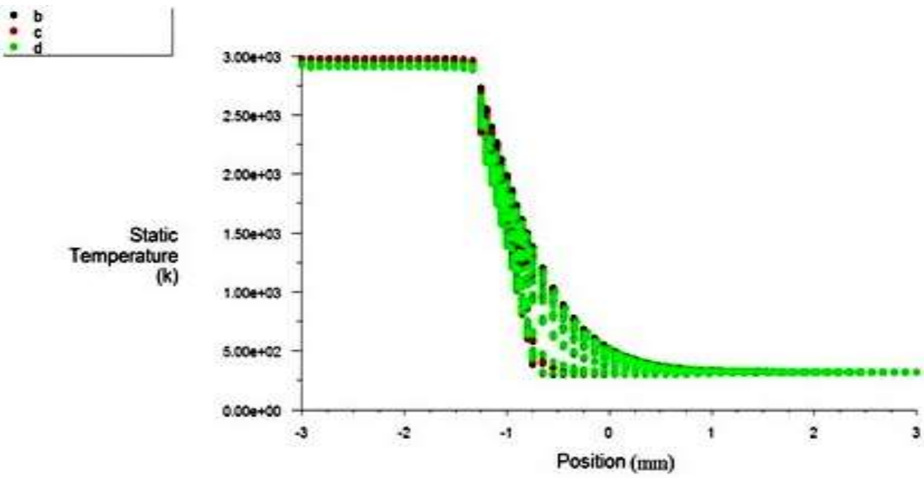

Figure 5: Temperature profiles across combustor wall at, *x* = 0

From figure 5 it is clear that at the downstream locations, the temperature distribution is qualitatively similar to that at $x = 0$, but it contains important quantitative differences. The boundary layer near the hot gas can be easily seen in these two distributions. The temperature of the copper section between the hot gas and the wall has nearly disappeared as this wall section is nearly at a constant temperature of 400K which is colder than at the entrance plane. The boundary layer on the hot side of the coolant remains very thin and it indicates a very high Reynolds number in the channel and the entire coolant stream remains at 300*K*. There remains a thin boundary layer on the outer side of the coolant passage and the copper and its temperature remains constant at the same level as at $x = 0$.

6. SUMMARY AND CONCLUSIONS

The detailed characteristics of the contours of temperature and heat transfer characteristics of a regeneratively cooled combustion chamber wall have been studied by means of CFD simulation. The results are based on a complete turbulent flow analysis of the 3D flow inside the channel, the heat conduction inside the surrounding copper walls, and a perfect gas analysis of the flow of hot gas inside the combustion chamber. The results show the local characteristics of the coupled solution in the region near to the coolant entrance. In general, the temperature contours are considerably different from that obtained by computing the hot-gas side, the coolant side or the heat conduction in the solid region separately. The results show that the metal temperatures is high at the inlet plane, then decrease slightly as part of an entry length effect, and increase slowly with downstream distance. The corresponding heat transfer is highly three dimensional in nature and there are significant tangential variations in the temperature inside the combustor wall.

REFERENCES

[1] Mary F. Wadel, “Heat Transfer in Rocket Combustion chambers”, N 94-20041, SECA.Vol.2,pp 99-108.

[2] C. L. Merkle, D. Li and V. Sankaran, “Analysis of Regen Cooling in combustion Chambers.”, NASA Space Act Agreement, NCC8-200.

[3] Daniel K.Parris and D. Brian Landrum, “Regenerative Cooling for Liquid rocket Engines”, Journal of Thermal Science Vol.4, No.1, pp. 54-59.

[4] Micheal L.Meyer, “Advanced Rocket Engines”,AIAA-2004,NATO OTAN- 74239.

AUTHOR BIOGRAPHY

SARATH RAJ – Completed M.Tech in Industrial Refrigeration and Cryogenic Engineering from TKM College of Engineering, Kollam and B.Tech from Travancore Engineering College, Kollam. Research interests include 1. Refrigeration and Cryogenics. 2) Combustion studies in rocket fuels.3) Cooling systems in rockets.4) Automobile Engineering

SUJITH G – Completed M.Tech in Industrial Refrigeration and Cryogenic Engineering from TKM College of Engineering, Kollam and B.Tech from Saintgits College of Engineering, Kottayam. Area of interest is in “Heat Transfer and Fluid Flow Analysis”.