Numerical Investigation of a Liquid Propellant Rocket (LH_2/LO_x) Combustion Chamber with Various Operational Condition and Micro-Gravity Effect by CFD

Pankaj Kumar Gupta[1], Vardan Singh Nayak[2], Arvind Gupta[3]

[1]M.Tech. Scholar Mechanical Engg. Dept. VIST, Bhopal
[2]Asst. Professor Mechanical Engg. Dept. VIST ,Bhopal
[3]Asst. Professor Electrical Engg. Dept. VIST, Bhopal

---***---

ABSTRACT - Cryogenic is a branch of physics, which deals with the production and behaviour of materials at very low temperatures. Cryogenic being a vast branch of physic has many applications. The application with which our study is concerned cryogenic rocket engine. Further explanation for cryogenic will be elaborated in this study. In present study we tend to investigate the Liquid rocket propellant Combustion Chamber system by changing the wall temperature of the combustion chamber and mass flow of the propellant through CFD simulation. Wall temperature is taking in regular intervals between 1000-3000K and mass flow from 41.8 to 61.8 kg/s for LH2 and 268.7kg/s for LOX. From the simulation results we tend to found that increasing the wall temperature increase the pressure and velocity and flow development is additional stable in nature that is needed during this style of engine in an economical manner. Equally after we increase the mass flow from 41.8kg/s to 51.8kg/s and 61.8kg/s the Static pressure is increased and velocity is additionally accrued by increasing turbulence kinetic energy. Flow pattern is additionally stable in nature and there's no damage in combustion chamber whereas increasing the wall temperature and mass flow at intervals this optimized limit. Finally we tend to conclude our study that increasing the wall temperature and mass flow can impact in final pressure and contour speed as a result of higher mixing of propellant and oxidizer and high reaction rate. By CFD results we can conclude that micro-gravitational effect on cryogenic combustion chamber is negligible because velocity of such type of engine is very high as compare to increased velocity value due to lower gravity but increased value of pressure is definitely the positive results in this study.

***Keywords*: Oxygen/Hydrogen, Combustion Device, Cryogenic Combustion, High Pressure, Diagnostic, Modelling, CFD, etc.**

1. INTRODUCTION

Thrust is the force which moves the rocket through the air, and through space. Thrust is generated by the propulsion system of the rocket through the application of Newton's third law of motion; for every action there is an equal and opposite re-action. In the propulsion system, an engine does work on a gas or liquid, called a working fluid, and accelerates the working fluid through the propulsion system. The re-action to the acceleration of the working fluid produces the thrust force on the engine. The working fluid is expelled from the engine in one direction and the thrust force is applied to the engine in the opposite direction. Forces are vector quantities having both a magnitude and a direction. When describing the action of forces, one must account for both the magnitude and the direction. The direction of the thrust is normally along the longitudinal axis of the rocket through the rocket centre of gravity. But on some rockets, the exhaust nozzle and the thrust direction can be rotated, or gimballed. The rocket can then be manoeuvred by using the torque about the centre of gravity. The magnitude of the thrust can be determined by the general thrust equation. The magnitude of the thrust depends on the mass flow rate of the working fluid through the engine and the exit velocity and pressure of the working fluid. The efficiency of the propulsion system is characterized by the specific impulse; the ratio of the amount of thrust produced to the weight flow of the propellants. All rocket engines produce thrust by accelerating a working fluid. But there are many different ways to produce the acceleration, and many different available working fluids. Let's look at some of the various types of rocket engines and how they produce thrust. The simplest rocket engine uses air as the working fluid, and pressure produced by a pump to accelerate the air. This is the type of "engine" used in a toy balloon or a stomp rocket. Because the weight flow of air is so small, this type of rocket engine does not produce much thrust. A bottle rocket uses water as the working fluid and pressurized air to accelerate the working fluid. Because water is much heavier than air, bottle rockets generate more thrust than stomp rockets. Model rockets, and most full scale rockets use chemical rocket engines. Chemical rocket engines use the combustion of propellants to produce exhaust gases as the working fluid. The high pressures and temperatures of combustion are used to accelerate the exhaust gases through a rocket nozzle to produce thrust. There are two important parts of a chemical rocket engine; the nozzle, and the propellants. The nozzle design determines the mass flow rate, exhaust velocity, and exit pressure for a given initial pressure and temperature. The initial pressure and temperature are determined by the chemical properties of the propellants. Propellants are composed of a fuel to be burned and an oxidizer, or source of oxygen, for combustion. Under normal temperature conditions, propellants do not burn, but require some source of heat, or igniters, to initiate combustion. Chemical rocket engines do not typically rely on the surrounding atmosphere as a source of oxygen. Therefore, chemical rocket engines can be used in space, where there is no atmosphere present.

2. LITERATURE

1. M. Faisal Khan, Z. A. Quadri1, P. S. Kulkarni, U. Guven, S. P. Bhat, K. Sundarraj-A good liquid propellant with high specific impulse and a high speed of exhaust gases implies a high combustion pressure and temperature and small molecular weight. However there is another important factor i.e. density of the propellant leads to larger storage tanks, eventually increasing the weight of the launch vehicle. Also the storage of liquid propellants at cryogenic temperature as in case of military operations where there is no definite time to use these weapons (rockets, missiles etc) until there is war, is a tremendous challenge. Despite all these drawbacks, high efficiency of LH2/LOx makes these difficulties worth coping with, when reaction time and storability are not too critical. In the present work, the simulation of a combustion chamber is carried out for LH2/LOx fuel/oxidizer injected at cryogenic temperature in the combustion chamber and combustion takes place at high pressure (70 atm) and high temperature (5000-8000 K) and finally gases expended in a C-D nozzle resulting high velocity and high thrust. The geometry and meshing has been done on GAMBIT and commercial code Ansys FLUENT has been used for the CFD simulation. The present study has been carried out for geometries with varying number of inlets for better mixing and for varying nozzle designs for attaining the high exhaust velocity and impulse which is the main objective of designing of a launch vehicle. In the present work the combustion chamber and nozzle has been designed for LH2/LOx propellant rocket engine and the analysis of various flow parameters is carried out. The present design simulation for the chamber and nozzle profile is giving the output parameters i.e. exit velocity, exit pressure, temperature and the pattern of combustion as expected for the iso-entropic flow considering most expensive combustion model for the simulation (i.e. EDC) where rate of reaction is calculated based on detailed chemical reactions of fuel and oxidizer. The pressure decreases when the flow passes through the nozzle in the convergent section and continues decreasing through throat up-to the exit. The velocity starts increasing in the convergent section and continues increasing up-to 3500 m/s at the nozzle exit. The combustion modelling with the computationally expensive but with detailed chemical mechanism i.e. Eddy Dissipation Concept model has been analysed which is giving the results as expected. The present work has a tremendous scope in the future study. The effect of different pressure, temperature, mass flow rate, mixture ratio can be studied on the same design for optimization of modelling parameters. Apart from constant heated walls of chamber, a heat source can be located inside, plasma phase of the gases (ionized gas) can be considered in the Eddy Dissipation Concept model for combustion. Further the same approach can be tested on different geometrical design of chamber and nozzle to optimise the performance parameters of the space vehicle. Cooling of the chamber and heat transfer is another very important and critical aspect which is required to overcome for successful experimentation.

2. S. Senthilkumar, Dr. P. Maniiarasan, Christy Oomman Jacob, T. Vinitha -The basic concept of a rocket engine relays on the release of the internal energy of the propellant molecules in the combustion chamber, the acceleration of the reaction product and finally the release of the hot gases at the highest possible velocity in the convergent/divergent nozzle. Liquid rocket engines burn propellants, which undergo chemical reactions to convert the stored chemical energy to thermal energy which results in the generation of thrust. Thrust chamber of cryogenic engine is modeled at a chamber pressure of 40 bar and thrust of 50KN to reduce the high temperature and pressure in the combustion chamber. CFD analysis is done to show the pressure and temperature variation in the thrust chamber modeled for 50KN thrust and chamber pressure of 40 bar. The design of rocket engine should be such that it should withstand the high pressure and high temperature of the combustion chamber. Cryogenic propellants in liquid rocket engine provide high specific impulse which is suitable for use in rocket upper and booster stages. For the thrust chamber modeled at 50KN of thrust and chamber pressure of 40 bar with the propellant combination of LH2/LOX shows a very good consistency when compared with computational results. From the analysis result it is found that cryogenic rocket engine with the propellant combination of LH2/LOX is suitable for design of upper stage rocket. Cryogenic propellants in liquid rocket engine provide high specific impulse which is suitable for use in rocket upper and booster stages. For the thrust chamber modeled at 50KN of thrust and chamber pressure of 40 bar with the propellant combination of LH2/LOX shows a very good consistency when compared with computational results. From the analysis result it is found that cryogenic rocket engine with the propellant combination of LH2/LOX is suitable for design of upper stage rocket.

3. Preclik, D., Wiedmann, D., Oechslein, W., and Kretschmer J -The CryoROC (Cryogenic Rocket Combustion) software, a sophisticated multi-phase Navier- Stokes code developed by Astrium (former DaimlerChrysler Aerospace AG - Dasa) is used to analyze complex flows in combustion chambers and nozzles of cryogenic hydrogen/oxygen rocket engines. This paper presents an introduction and overview of CryoROC's modeling philosophy and to discusses typical applications demonstrating the pole entail of the code to support the layout and optimization process of advanced rocket engines. It gives an introduction of industrial needs and environment while section 2 describes the code's main features. The importance of CryoROC's essential part. The

Lagrangian droplet tracking module, for accurate wall heat flux predictions.

3. OBJECTIVE

General design issues of liquid propellant engines are addressed, demonstrating a clear need for a new approach in engine development taking into account the complex chemical and physical processes in rocket engines. A brief description about the organization of the research programs is followed by a short presentation of the test facilities and the experimental setup's in use. Examples of experimental as well as numerical achievements applying sophisticated optical diagnostic tools are discussed and finally, the paper summarizes the results achieved so far and addresses the questions still open.

Our main objective of the present study can be expressed in following steps-

1. The aim of this investigation is the implementation by user defined routines of thermodynamically consistent real gas mixing equation of state into the commercially available CFD-code ANSYS Fluent and the validation using experimental data.

2. CFD simulations will also perform by using different approaches: the Soave Redlich-Kwong real gas model and Peng Robinson have been implemented to model the physical properties of the species.

3. Study of the LOX/H2 injection, mixing and combustion in liquid rocket engines with shear coaxial injectors, at supercritical conditions by using one-step reaction mechanism and detailed CHEMKIN Mechanism.

4. Super-critical mixing and combustion phenomena involve a large spectrum of interconnected physical processes which determine complexity in the problem formulation. Near critical or trans-critical condition the reacting mixture exhibits large variation in thermodynamic and transport properties, which affect drastically the mixing and combustion processes. An optimization process is also required for further case study which can be evaluate the super-critical mixing effect on the accuracy of the simulative results.

5. Effect of mass flow rate, wall temperature with varying operating conditions of combustion chamber will be investigated for the thrust effect on a system.

6. Finally we are to present work is to investigate the microgravity effect on the performance of the current system and stability phenomenon on the chamber for various gravity mode at the supersonic flow condition of the cryogenic engine.

4. METHODOLOGY

METHOD APPLIED IN PRESENT WORK

STEP 1 CAD GENERATION

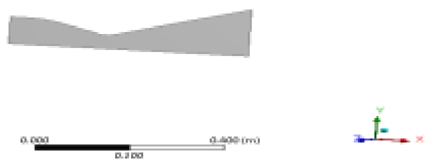

Figure 4.1 CAD MODEL (Axissymmetry)

STEP 2 MESH GENERATION

Mesh Type: grid meshing
Element Edge Length =3.21E-03m
No. of Nodes = 64507
No. of Element = 51203

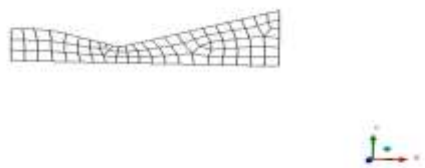

Figure 4.2 MESH MODEL

- Fluent setup: After mesh generation define the following setup in the Ansys fluent.
- Problem Type **:** 2D axissymmetric
- Type of Solver: Pressure-based solver.
- Physical model: Viscous: K-epsilon two equation turbulence model.
- Material (Fluid**)-** LH_2 and LO_2

Boundary Condition- The boundary conditions for test simulation has been taken as properties of liquid hydrogen and liquid oxygen injection

Properties	**LH2**	**LOx**
Velocity (m/s)	300	120
Droplet diameter (μm)	100	100
Flow Rate (kg/s)	41.8	268.7
Density (kg/m3)	70.85	1142
Specific Heat (j/kg-K)	9772.2	1699
Vaporization Temperature(k)	14	55

Study-1
WALL TEMPERATURE: – Tw= 1000K, 2000K, and 3000K
Study-2
MASS FLOW RATE:- 41.8kg/s, 51.8kg/s and 61.8kg/s
Study 3
MICRO GRAVITY
Results will be compared with different gravity like 0.3 g, 0.6 g and 0.9g
Solution Method- Second order
Solution Initialization- Initialized the solution to get the initial solution for the problem.
Run Solution: Run the solution by giving 2000 no of iterations.

5. RESULTS

5.1 CFD Simulation Results for All Cases

CASE 1- Temp. 1000K and Mass Flow Rate – 41.8 Kg/sec and Normal gravity

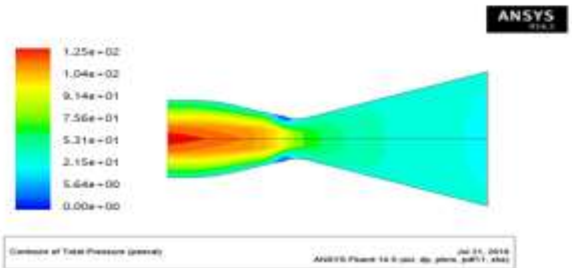

Figure 5.1 Total Pressure at 1000K and MFR-41.8Kg/sec@ Normal gravity

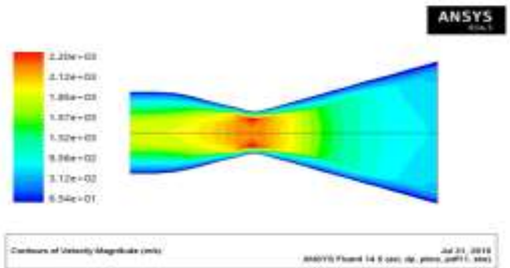

Figure 5.2 Velocity (m/sec) at 1000K and MFR-41.8Kg/sec@ Normal gravity

CASE 2- Temp. 1500K and Mass Flow Rate – 41.8 Kg/sec@ Normal gravity

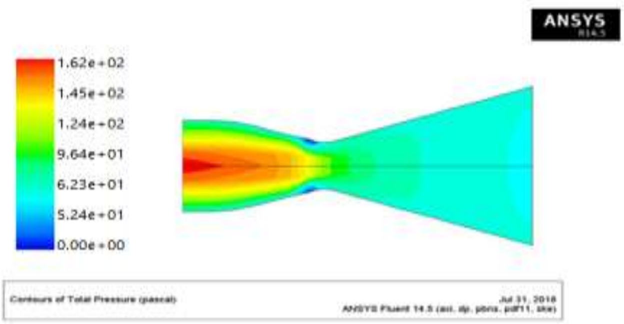

Figure 5.3 Total Pressure at 1500K and MFR-41.8Kg/sec@ Normal gravity

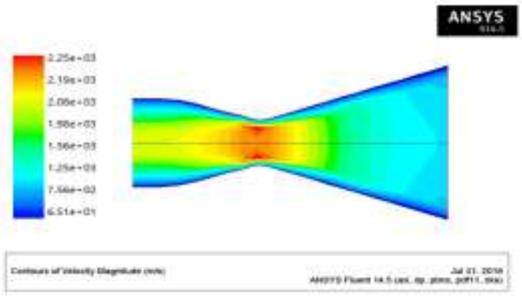

Figure 5.4 Total Velocity (m/sec) at1500K and MFR-41.8Kg/sec@ Normal gravity

CASE 3- Temp. 2000K and Mass Flow Rate – 41.8 Kg/sec@ Normal gravity

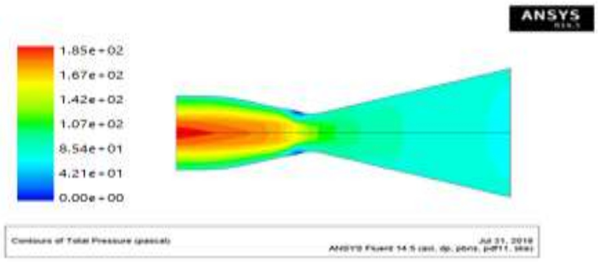

Figure5.5 Total Pressure at 2000K and MFR-41.8Kg/sec@ Normal gravity

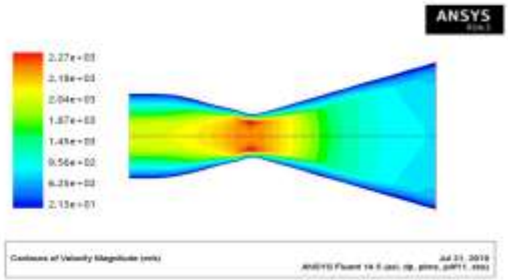

Figure 5.6 Total Velocity (m/sec) at 2000K and MFR-41.8Kg/sec@ Normal gravity

CASE 4- Temp. 2500K and Mass Flow Rate – 41.8 Kg/sec@ Normal gravity

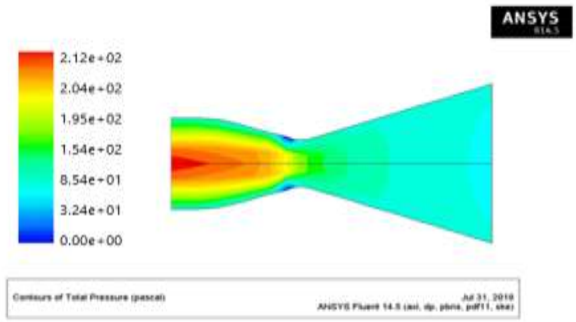

Figure 5.7 Total Pressure at 2500K and MFR-41.8Kg/sec@ Normal gravity

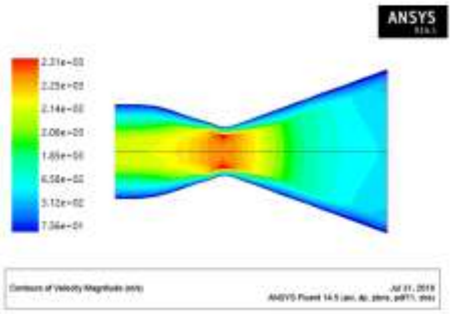

Figure 5.8 Total Velocity (m/sec) at 2500K and MFR-41.8Kg/sec@ Normal gravity

CASE 5- Temp. 3000K and Mass Flow Rate – 41.8 Kg/sec@ Normal gravity

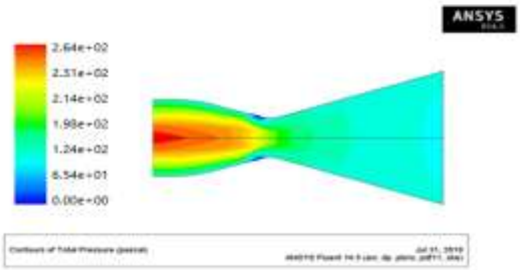

Figure 5.9 Total Pressure at 3000K and MFR-41.8Kg/sec@ Normal gravity

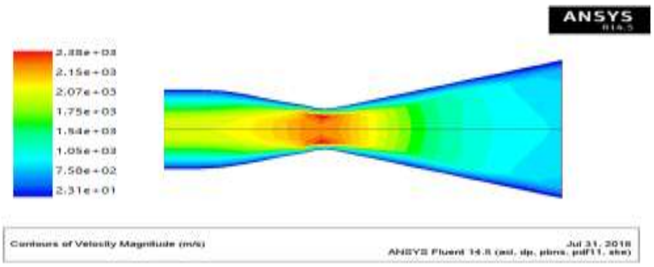

Figure 5.10 Total Velocity at (m/sec) 3000K and MFR-41.8Kg/sec@ Normal gravity

CASE 6- Temp. 3000K and Mass Flow Rate – 51.8 Kg/sec@ Normal gravity

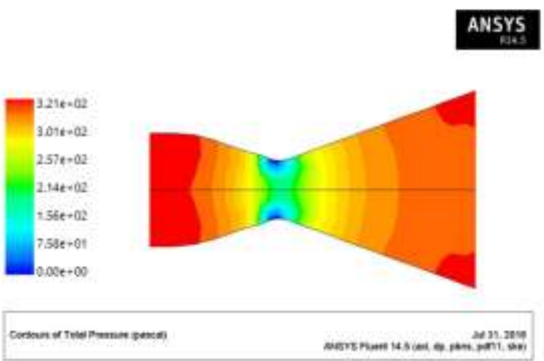

Figure 5.11 Total Pressure at 3000K and MFR-51.8Kg/sec@ Normal gravity

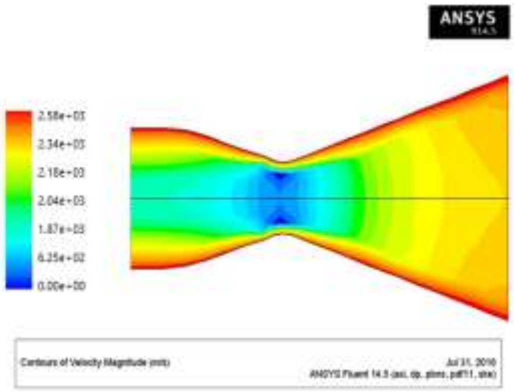

Figure 5.12 Total Velocity (m/sec) at 3000K and MFR-51.8Kg/sec@ Normal gravity

CASE 7- Temp. 3000K and Mass Flow Rate – 61.8 Kg/sec@ Normal gravity

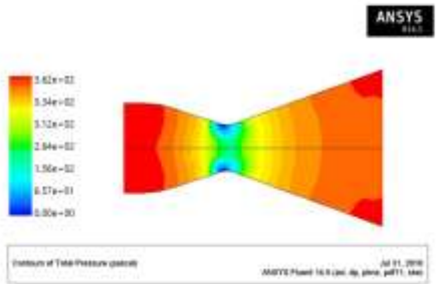

Figure 5.13 Total Pressure at 3000K and MFR-61.8Kg/sec@ Normal gravity

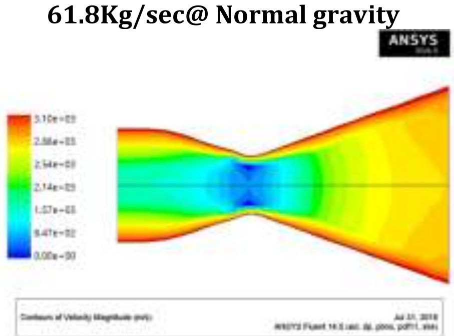

Figure 5.14 Total Velocity (m/sec) at 3000K and MFR-61.8Kg/sec@ Normal gravity

CASE 8- Temp. 3000K and Mass Flow Rate – 61.8 Kg/sec@ 0.3g

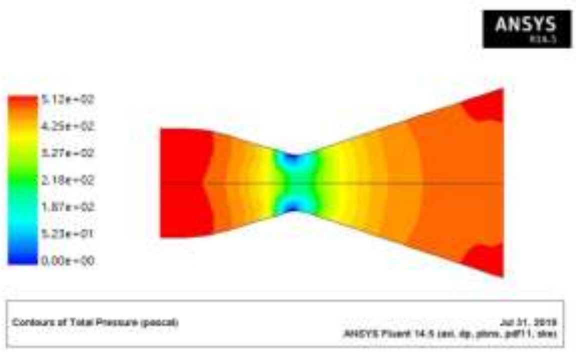

Figure 5.15 Total Pressure at 3000K and MFR-61.8Kg/sec@ 0.3g

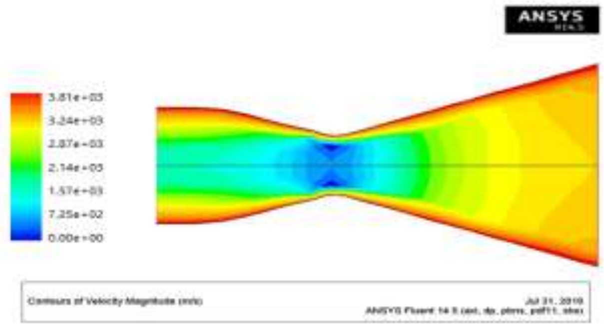

Figure 5.16 Total Velocity (m/sec) at 3000K and MFR-61.8Kg/sec@ 0.3g

CASE 9- Temp. 3000K and Mass Flow Rate – 61.8 Kg/sec@ 0.6g

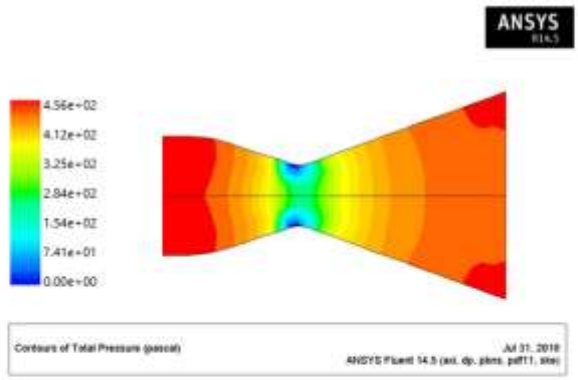

Figure 5.17 Total Pressure at 3000K and MFR-61.8Kg/sec@ 0.6g

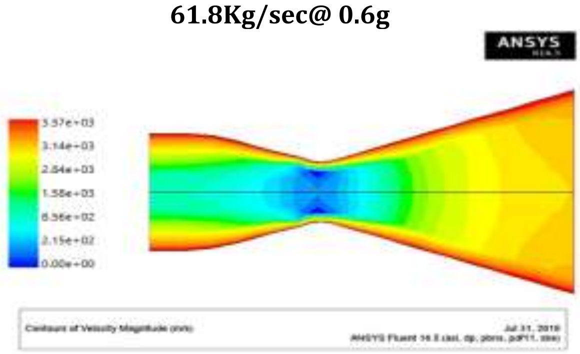

Figure 5.18 Total Velocity (m/sec) at 3000K and MFR-61.8Kg/sec@ 0.6g

CASE 10- Temp. 3000K and Mass Flow Rate – 61.8 Kg/sec@ 0.9g

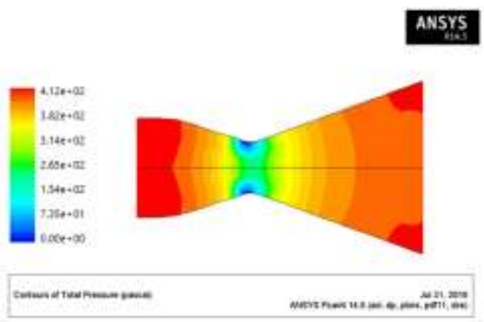

Figure5.19 Total Pressure at 3000K and MFR-61.8Kg/sec@ 0.9g

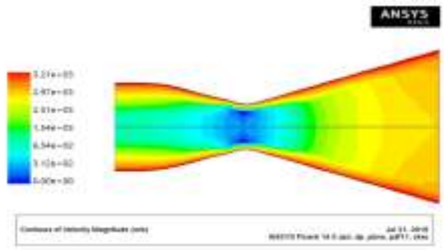

Figure 5.20 Total Velocity (m/sec) at 3000K and MFR-61.8Kg/sec@ 0.9g

6. CONCLUSION

In current study we analyzed the model for different wall temperature, mass flow rate and various gravity cases at constant thrust.

In current research we investigate the effect of different microgravity through numerical investigation by using gravity mechanism model in Fluent.
Simulation results predict the flow phenomenon and turbulence model of a system. In first part of the study we investigate by the CFD results that as we increase the wall temperature pressure and velocity is also increases and flow phenomenon is more stable in nature which is required in this type of engine in an efficient manner.
Similarly when we increase the mass flow rate from 41.8kg/s to 51.8kg/s and 61.8kg/s the Static pressure is increased and velocity is also increased by increasing turbulence kinetic energy. Flow pattern is also stable in nature and there is no harm in combustion chamber while increasing the wall temperature and mass flow rate within this optimised limit. Finally we conclude our study that increasing the wall temperature and mass flow rate will effect in final pressure and streamline velocity due to better mixing of propellant and oxidiser and high reaction rate.

We investigate the system by simulation results in Fluent for different microgravity like 0.3g, 0.6g and 0.9g and contour results clearly shows that when gravity is reduced velocity and pressure is increased due to lower gravitational force but increased value is not very significant. So we can conclude that micro-gravitational effect on cryogenic combustion chamber is negligible because velocity of such type of engine is very high as compare to increased velocity value due to lower gravity but increased value of pressure is definitely the positive results in this study.

7. REFERENCES

[1]. M. Faisal Khan, Z. A. Quadri1, P. S. Kulkarni, U. Guven, S. P. Bhat, K. Sundarraj CFD Simulation Of A Liquid Rocket Propellant (LH2/Lox) Combustion Chamber. 15th Annual CFD Symposium, Indian Institute of Science, 2013, Bangalore

[2]. S. Senthilkumar , Dr. P. Maniiarasan ,Christy Oomman Jacob , T. Vinitha, "Design And Analysis Of Thrust

Chamber Of A Cryogenic Rocket Engine", Department of Mechanical Engineering, Anna University, Chennai, Department of Aeronautical Engineering , Nehru Institute of Engineering & Technology, Coimbatore 2009

[3]. Preclik, D., Wiedmann, D., Oechslein, W., and Kretschmer, J., "Cryogenic Rocket Calorimeter Chamber Experiments and Heat Transfer Simulations", AIAA 98-3440, 1998.

[4]. O. Knab, M. Frey, J. Görgen, C. Maeding, K. Quering, and D. Wiedmann Astrium GmbH, "Progress in Combustion and Heat Transfer Modelling in Rocket Thrust Chamber Applied Engineering", Space Transportation, Launcher Propulsion, System Analysis, Munich, 81663, Germany 2009

[5]. Preclik, D., Knab, O., Estublier, D. and Wennerberg, D., "Simulation and Analysis of Thrust Chamber Flow fields: Storable Propellant Rockets, Liquid Rocket Thrust Chambers: Aspects of Modelling, Analysis, and Design", AIAA Progress in Astronautics and Aeronautics, Vol. 200, 2004

[6]. Vinitha, S. Senthilkumar, K. Manikandan, "Thermal Design And Analysis Of Regenerative Cooled Thrust Chamber Of Cryogenic Rocket Engine", Department of Aeronautical Engineering, Nehru institute of engineering and technology

[7]. Ten-See Wang and Van Luong., "Design Support for Advanced Storable Propellant Engines by ROCFLAM Analyses," AIAA, 1999.

[8]. Naoki Tani, Shin-ichi Tsuda, Nobuhiro Yamanishi, Yoshiki Yoshida, "Development And Validation Of New Cryogenic Cavitation Model For Rocket", Proceedings of the 7th International Symposium on Cavitation, 2009, Ann Arbor, Michigan, USA

[9]. O.J. Haidn, M. Habiballah, "Research on High Pressure Cryogenic Combustion", Aerospace Science and Technology , (2003)

[10]. Yolanda Mack , "Development And Validation Of New Cryogenic Cavitation Model For Rocket, CFD-Based Surrogate Modeling Of Liquid Rocket Engine Components Via Design Space Refinement And Sensitivity Assessment", UNIVERSITY OF FLORIDA, 2007

[11]. Maria Grazia De Giorgi , Laura Tarantino , Antonio Ficarella, Domenico Laforgia "Numerical Modelling Of High-Pressure Cryogenic Sprays" University of Salento –Dep. Engineering for Innovation , Lecce, Italy

[12]. Vishnu Narayan, Aswathy V G, Sathis Kumar," CFD Modeling And Analysis Of Radial Dynamic Seal For Cryogenic System", International Journal of Science, Engineering and Technology Research (IJSETR), Volume 4, 2015

[13]. Maeding, C., Wiedmann, D., Quering, K., and Knab, O., "Improved Heat Transfer Prediction Engineering Capabilities for Rocket Thrust Chamber Layout", 3rd European Conference of Aerospace Sciences, EUCASS 2009-90, Versailles, 2009

[14]. Stephen Gen-Ken Chuech, Andrzej J. Przekwa, and Chih-Yuan Wang "Numerical Modelling For Atomization Of Coaxial Liquid/Gas Jets "Journal of Marine Science and Technology, Vol. 12, No. 4, (2004)

[15]. Langel, G., Obermaier, G., Taubenberger, G., and Pierro, R., "Technology Demonstration for Advanced Storable Propellant Turbo pump Engine", AIAA, 1998.

[16]. M. Thirupathi, N. Madhavi, K. Simhachalam Naidu, "Design and Analysis of a Fuel Injector of a Liquid Rocket Engine", International Journal of Engineering and Advanced Technology (IJEAT), Volume-4 Issue-5, June 2015

[17]. Vivek Gautam, "Flow And Atomization Characteristics Of Cryogenic Fluid From A Coaxial Rocket Injector", University of Maryland, 2007

[18]. T. Kaltz, M. Glogowski, and M. M. Micci "Shear Coaxial Injector Instability Mechanisms" Aerospace Engineering Department & Propulsion Engineering Research Center ,The Pennsylvania State University, University Park

[19]. A.m. Ashwini, M. Prabhakaran (2015) "Cryogenic technology in rocket engines" B.E. Final year, Dept of Aeronautical Engineering, Dr. Pauls Engineering College.

[20]. M. Poschner, M. Pfitzner (2009) "CFD-Simulation of supercritical LOX/GH2 combustion considering consistent real gas thermodynamics" Thermodynamics Institute, Faculty for Aerospace Engineering, University of the Federal German Armed Forces , Munich

[21]. J.-P. Hickey AND M. Ihme(2013) "Supercritical mixing and combustion in rocket propulsion ",Center for Turbulence Research Annual Research Briefs.

[22]. Oschwald, M., Smith, J. J., Branam, R., Hussong, J., Schik, A., German Aerospace Center (DLR), Institute of Space Propulsion, Lampoldshausen, Germany Chehroudi, Engineering Research Consultants, Lancaster, Pennsylvania, USAB. and Talley, D U.S. Air Force Research Laboratory (AFRL),Propulsion Directorate, Edwards Air Force Base, California, USA(2000) "Injection of Fluids into Supercritical Environments", Combustion Science and Technology, Vol. 178,