

Design Conceptualization and Computational Analysis of Cryogenic Engine Nozzle-ABYOM

Masum Hossain^{1*}, AT. Pooja Srree², Kuldeep Sharma³,
Gokul Raghavendra Srinivasan⁴

Abstract

The main purpose of this paper is to design a cryogenic engine nozzle with specified dimensions using CATIA V5 software and analyze the nozzle using ANSYS software. The realizable $k-\epsilon$ viscous model was used for calculation and the Hydrogen (H_2) had retained as an ideal gas. The CFD simulations were done on the nozzle to find the pressure, density, velocity, and temperature. In the graph, there are 500 points streamlined to show the better result of the nozzle, and the density, velocity, and temperature have been compared with pressure. Experimental and numerical analysis have both been used to study the 3-D nozzle's performance. A twin bell nozzle created for an experiment moderately rocket engine application is subjected to numerical analysis to determine the impact of film injection on the nozzle's flows and transitional behaviour. Simulations of the start-up and higher manufacturing regimes of a twin bell nozzle are used to predict its actual flying characteristics. To examine how different parameters affect flow behaviour, coolant is injected into two separate places under varied operating conditions. With and without injection of coolant, modifications in shock interactions and shock patterns are examined during start-up flow.

Keywords: Nozzle, CFD, cryogenic engine, pressure, temperature, velocity, density.

INTRODUCTION

The fuel and oxidizer used in a cryogenic rocket motor are both liquefied gases that are kept at exceptionally low temperatures. The success of NASA's Saturn V rocket in reaching the Moon was largely due to these very effective engines, which were initially tested on the US Atlas-Centaur [1]. Here Cryogenic Engines use cryogenic fuels such as hydrogen, nitrogen, etc. in the form of liquid at

very low temperatures. Here we are using liquid hydrogen and liquid oxygen as fuel for our engine. At first, the gaseous hydrogen and oxygen are stored in large tanks and are cooled to liquids at the at most lower temperatures for high mass flow rate and efficiency of the rocket engines. Hydrogen liquefies at -253°C and at the same time, oxygen liquefies at -183°C [1]. If cryogenic fuels are burnt, they imply a greater thrust for the engine. Here Cryogenic engines use turbopumps. Generally, cryogenic fuel tanks are made of Aluminium-Lithium Alloy with a thickness of 25 mm [2].

Nozzles are designed in such a way to increase the velocity and decrease the pressure of a rocket engine. In rockets mostly we will use convergent-divergent nozzles for higher efficiency [1]. Heat exhaust from the combustion chamber diverges downward and converges at the minimum area, or

***Author for Correspondence**
Masum Hossain

¹Student, Department of Aeronautical Engineering, Nanchang Hangkong University, Nanchang, Jiangxi, China

²Student, Department of Aeronautical Engineering, Bharath University, Chennai, Tamil Nadu, India

³Student, Department of Aeronautical Engineering, School of Aeronautics, Rajasthan, India.

⁴Research Supervisor, Department of Research and Development, Abyom SpaceTech and Defence Pvt. Ltd., UP, India

Received Date: August 26, 2022

Accepted Date: September 04, 2022

Published Date: September 06, 2022

Citation: Masum Hossain, AT. Pooja Srree, Kuldeep Sharma, Gokul Raghavendra Srinivasan. Design Conceptualization and Computational Analysis of Cryogenic Engine Nozzle-ABYOM. Journal of Materials and Metallurgical Engineering. 2022; 12(2): 14–23p.

throat, of the Convergent-Divergent (C-D) nozzle. Normally nowadays we are seeing the latest advancements in cryogenics but, we are facing some major issues while using cryogenics as the fuel for rocket engines. The first and foremost difficulty is its storage, the cryogenics are to be stored at very lower temperatures and in larger area containers. These fuels are highly combustible and difficult to use. Here we are doing the steady-state simulation and numerical analysis for the cryogenic engine.

P. Gaillard, C. Le Touze, L. Matuszewski, and A. Murrone's (2016) research paper tries to show the importance of numerical simulation of cryogenic combustion to understand better the complex physics involved in the reactive flow of rockets engines and reduce the development cost of these engines. This study also focused on oxidizer injection and dispersion utilizing jet-dense core disintegration, atomization, and supercritical combining. To handle these physical phenomena, specific models have been implemented in ONERA's CFD code, CEDRE. The diffuser interface model used for the 'separated' two-phase flow (CHARME) is a multi-species compressible Navier Stoke system. And the Eulerian kinetic-based model is used for the dispersed phase (SPIREE), which is based on the mesoscopic description provided by the Williams Boltzmann kinetic equation. Also, primary atomization modeling is developed, which is used for the mass transfer between solvers accounting for primary atomization. The time evolution pressure, temperature, volume fraction, and penetration depth are calculated. The SPIREE solver acquired the spray volume fractions and the development of pressure and temperature obtained by the CHARME solver's resolution. A Large Eddy Simulation (LES) conclusion of the primary automatization between a diffuser interface 4-equation (1-velocity) and kinetic-based model is promising but needs to be converted [3].

Mukkarum Hussain, Iftikhar Ahmed, Ilyas Khan, Chu Anh My, Mirza Mehmood Baig, Afrasyab Khan, and Stanislav S. Makhanov (2021), have designed single shear coaxial injector to identify the dynamical behavior of liquid fuel combustion chamber. For the simulation of the combustion chamber, kerosene and oxygen are injected. The Navier Stokes equations along with the Soave Redlich Kwong equation model and ideal gas assumptions have been used for this paper, and the ANSYS Fluent Simulations were performed using the 16.02 code. The thermodynamic state typically shifts from supercritical to supercritical during ignition. The combustion chamber central line pressure distribution and chamber pressure ignition transient have been calculated [4].

Stefan Fechter, Tim Horchler, Sebastian Karl, and Klaus Hannemann (2019) did the numerical simulation of a high-pressure rocket combustion chamber for future space vehicles, and the main challenges faced during the completed these Thermodynamic effects are estimated frequently throughout a broad temperature range. This research is based on the simulation of turbulence effects in cryogenic rocket combustion chambers. The Navier-Stokes Averaged Reynolds (RANS), DES/LES for sub-scale CC, and DNS for periodic box models have been used to resolve the turbulence effects. In the mixing test scenario, a visual presentation of instantaneous flow topology and statistical evaluation of mean and root mean square (RMS) fluctuations at various cut plane sites in the fluid domain are presented. In the comparison of the BKC combustion chamber, we implemented a single experiment at the DLR test facility P8 in Lampoldshausen, the DLR subscale thrust chamber model "C" (designated BKC). For the Ruiz test case approaches, DES and the RANS code are good and the average test statistics fit very well but there are a few difficult 2D RANS results. In the main time, DES and LES provide unphysical results. The DES and RANS codes work well for the Ruiz test case techniques, and the average test statistics suit extremely well. Nevertheless, a few more troublesome 2D RANS produce physical turbulence values, whereas DES and LES generate unphysical results [5].

Gautam and A.K. (2008), high speed was used to identify the liquid Nitrogen jet and thermal imaging was used to examine the cooling effect of the liquid nitrogen jet. Under confined conditions, the liquid nitrogen jet was determined to expand sooner, resulting in less destabilization and mixedness with the surrounding gases. Confinement also lowers the entrainment of neighboring air and heat transmission from the surroundings. Changing the length of the recess in the injector had no

influence on the jet at the injector outlet. Some experimental jet destabilization frequencies and the fundamental instability frequency found in coaxial jets have been investigated [6].

Schmitt T., SafePlace G., Ducruix S., Groening S., Hardi J., & Oschwald M (2017) conducted large-eddy modeling to anticipate high-frequency combustion instabilities in cryogenic rocket engines (LES). DLR's sub-scale rocket combustor BKD features 42 coaxial injectors supplied with liquid oxygen and gaseous hydrogen and operates at full-scale engine conditions. The LES solver was initially validated on ONERA's Mascotte test bench. The experiment then simulates two BKD scenarios, corresponding to naturally steady and unsteady regimes. Depending on the injection characteristics, the BKD combustor is intrinsically unstable. According to current numerical simulations, case T57 is unstable, while case T131 is stable. This is the first time an LES solver has predicted combustion instability in a LOx/H2 rocket engine design [7].

Abdul Khalilullah, Sk. J. Basha & H.K. Rangavittal (2016) speaks about the advancements in the space sector. In this paper, we came across development in propellant tanks that can withstand 6 Ton class spacecraft with a total of 4 tanks of which 2 for oxygen and the other 2 for fuel. These four tanks are in axial mode [8].

Naraghi M.H. Dunn S. Coats (2006) generally tell us about the cooling system in liquid rocket engines. It deals with the throat area of the engine with higher temperatures as high as 7000°R. The engine is built with a material that can withstand higher temperatures. The coolant first enters the nozzle and moves towards the combustion chamber to decrease the temperatures [9].

Donald E. Jackle Jr (2007), in the new spacecraft era, propellant tank assembly plays a major role. They build that propulsion shell with 6AL-4V titanium alloy. These materials are ductile and provide enough strength to the tank to withstand its weight. This tank later went through a destructive burst pressure test [10].

METHODOLOGY

Mathematical Modeling

CATIA V5, developed by the French company Dassault system, has been used to produce the 3D CAD model. CATIA is a software package that may be used on several platforms for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), three-dimensional modeling, and product lifecycle management (PLM) [11]. After modeling in CATIA V5, the 3D model is examined in ANSYS 2022 R1. ANSYS is a CAE/Multiphysics engineering simulation program that may be used for product design, testing, and operation [12].

The realizable k-ε model is defined by the following equations [13]:

The turbulence kinetic energy equation is given by:

$$\frac{D}{Dt}(\rho k) = \nabla \cdot (\rho Dk \nabla) + \rho G - \frac{2}{3}\rho(\nabla \cdot u)k - \rho \epsilon + S_k \quad (1)$$

And the dissipation rate by:

$$\frac{D}{Dt}(\rho \epsilon) = \nabla \cdot (\rho D \epsilon \nabla) + C_1 \rho |S| \epsilon - C_2 \rho \frac{\epsilon^2}{k + (\nu \epsilon)^{0.5}} + S_\epsilon \quad (2)$$

The turbulence viscosity is calculated using:

$$\nu_t = C_\mu \frac{k^2}{\epsilon} \quad (3)$$

Where the C_μ is given by:

$$C_\mu = \frac{1}{A_0 + A_s U_\infty^k} \quad (4)$$

For the theoretical calculation, there are used the isentropic flow equation to calculate flow thrust through the nozzle. The equations are:

The isentropic flow equations are [14]:

$$M = \frac{v}{a} \quad (5)$$

$$a = \sqrt{\gamma \frac{P}{\rho}} = \sqrt{\gamma R T} \quad (6)$$

$$\frac{P}{\rho^\gamma} = \text{Constant} = \frac{P_t}{\rho_t^\gamma} \quad (7)$$

$$\frac{P}{P_t} = \left(\frac{\rho}{\rho_t}\right)^\gamma = \left(\frac{T}{T_t}\right)^{\frac{\gamma}{\gamma+1}} \quad (8)$$

$$\frac{T}{T_t} = \left(1 + \frac{\gamma-1}{2} M^2\right)^{-1} \quad (9)$$

$$\frac{\rho}{\rho_t} = \left(1 + \frac{\gamma-1}{2} M^2\right)^{-\frac{1}{\gamma-1}} \quad (10)$$

$$\frac{A}{A^*} = \left(\frac{\gamma+1}{2}\right)^{\frac{-\gamma+1}{2(\gamma+1)}} \frac{\left(1 + \frac{\gamma-1}{2} M^2\right)^{\frac{\gamma+1}{2(\gamma+1)}}}{M} \quad (11)$$

Geometry Modelling

The nozzle is created using CATIA V5 software using its dimension as shown in Figures 1 and 2. The dimension of the inlet area is 20 mm, the throat area is 10 mm and the outlet area is 30 mm. The dimension of the longitudinal axis is 90 mm where the inlet to the throat is 30 mm and the throat to the outlet is 60 mm. The throat area is used at 10 mm, it is half of the inlet area. The nozzle domain is created to show the external flow of the nozzle as shown in Figure 2. The domain part is longitudinally 90 mm and vertically 40 mm. The Nozzle throat area is narrow compared to the inlet area and outlet area because the narrowing of the nozzle throat increases the pressure in the combustion chamber and the rate of propellant combustion, ultimately increasing the thrust of the rocket engine.

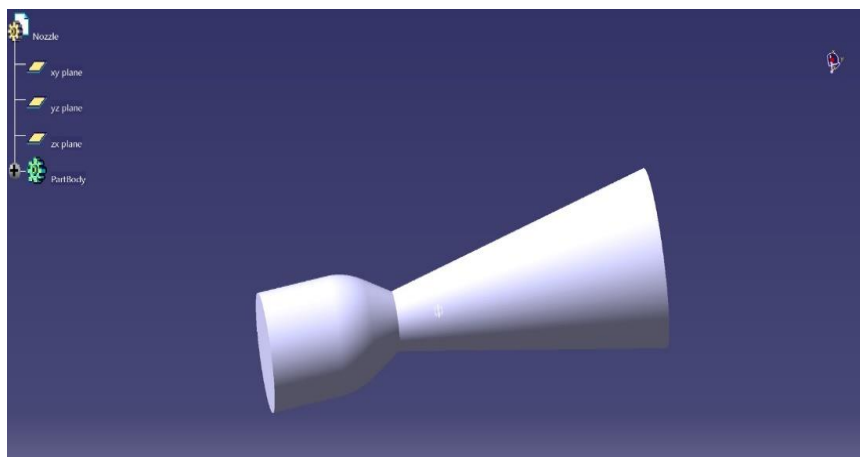


Figure 1. Nozzle design with domain.

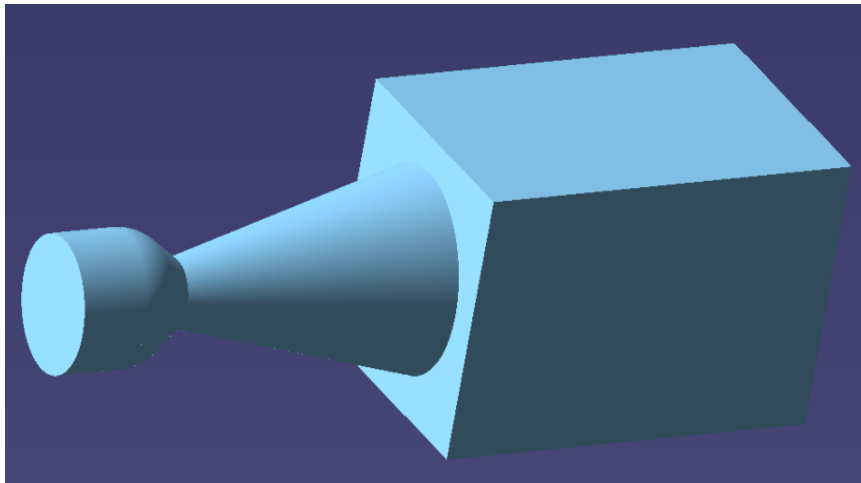


Figure 2. Nozzle design.

ANALYSIS

Meshing

The nozzle is meshed using Ansys meshing software. Ansys Meshing is a step in the engineering simulation process that divides intricate geometries into simple elements that may be utilized as discrete local approximations of the broader domain. The mesh has an impact on the simulation's accuracy, convergence, and speed. A C-type flow domain mesh is generated and refined in this step. First, there used facing mesh to make a structured mesh in the nozzle and used inflation to get a better result (Figure 3). The inflation layer is intended to provide thin components capable of capturing the normal gradient with few elements [15]. After complete meshing, their nodes have 97744 and elements have 468381. Their elements sized have used 0.8 mm.

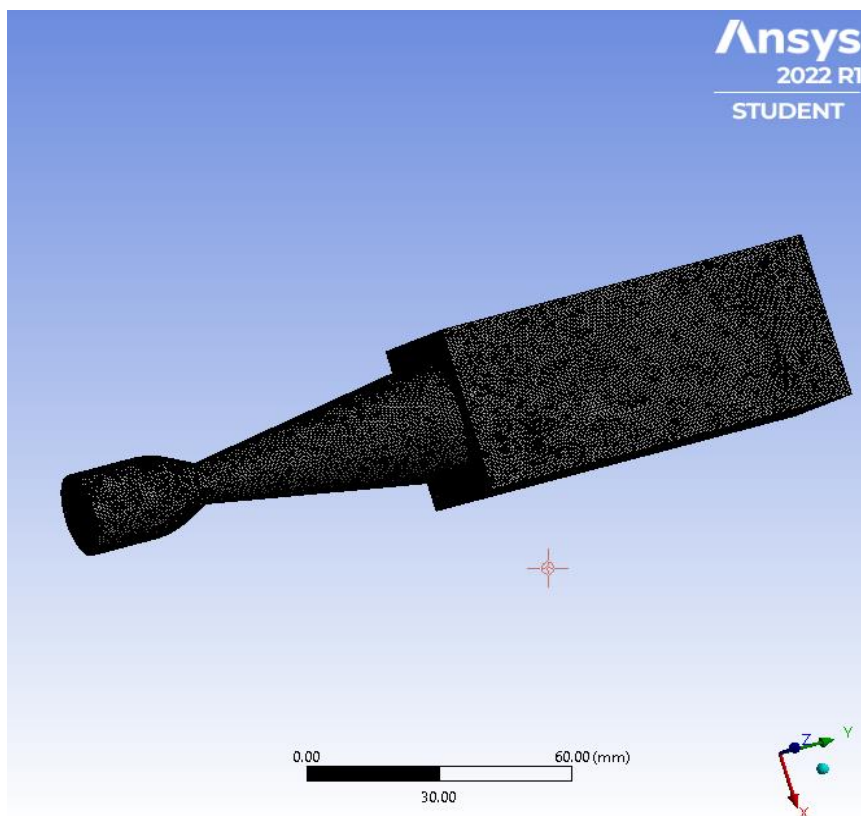


Figure 3. Generated mesh of the nozzle.

Fluent (CFD)

For simulation, ANSYS 22 R1-FLUENT is utilized. Fluent is a fluid simulation tool that can model fluid flow, heat and mass movement, chemical reactions, and other phenomena. Fluent provides cutting-edge turbulence models, multiphase flows, heat transfer, combustion, form optimization, Multiphysics, and other complex physics modelling tools [16]. In Ansys fluent, the Pressure-based solver is used for the nozzles. The energy equation was activated in all cases for boundary conditions. For the computation, the realizable k-model is employed. The k-model investigates the processes that determine turbulent kinetic energy. Hydrogen (H_2) is preserved as an ideal gas to capture density change, and the Sutherland model of viscosity is employed for accuracy. The nozzle has a pressure intake and a pressure exit. Following initialization, enough iterations are performed until the solution converges. For the nozzle, the inlet gauge pressure is 2468000 Pa and the temperature is 2500 K, the outlet gauge pressure is 21671 Pa, and the temperature is 644.32 K.

RESULTS AND DISCUSSION

Results

Basis of the analysis, the contour plots and graphs of the different nozzles are derived.

Pressure Contour

As we see the Figure 4, the pressure value is greatest at the intake and gradually decreases as we approach the nozzle exit. Because of the creation of shock waves, the pressure will abruptly drop after the throat. The nozzle's minimum pressure is 170.56 Pa. The maximum pressure of the nozzle is 2439640 Pa.

Temperature Contour

According to the Figure 5, the temperature is highest near the nozzle's entrance and gradually decreases towards the nozzle's exit. The Nozzle minimum temperature is 364.85 K. The maximum temperature of the nozzle is 2491.81 K.

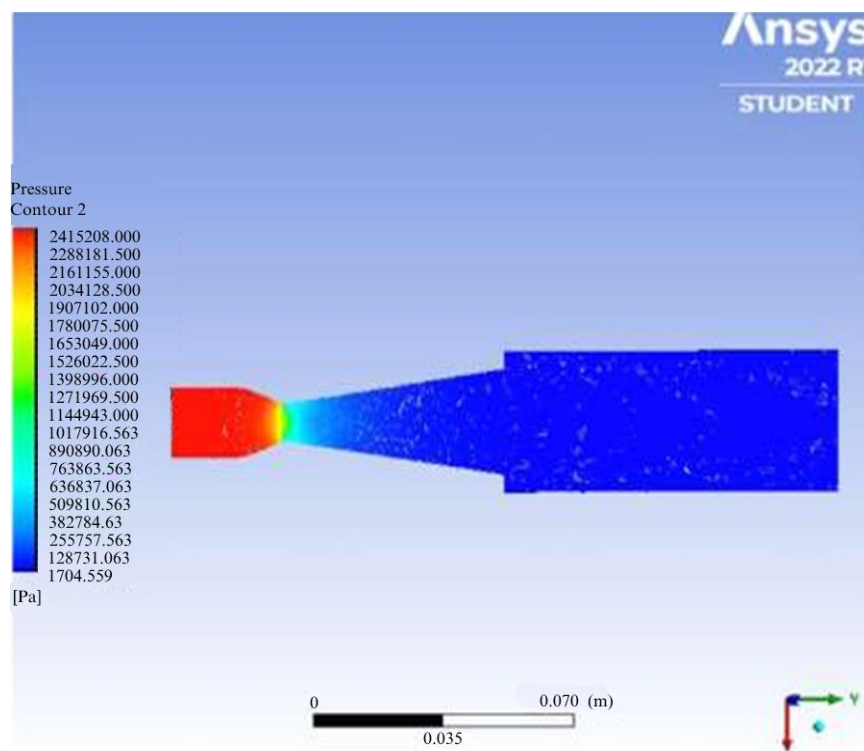


Figure 4. Pressure contour.

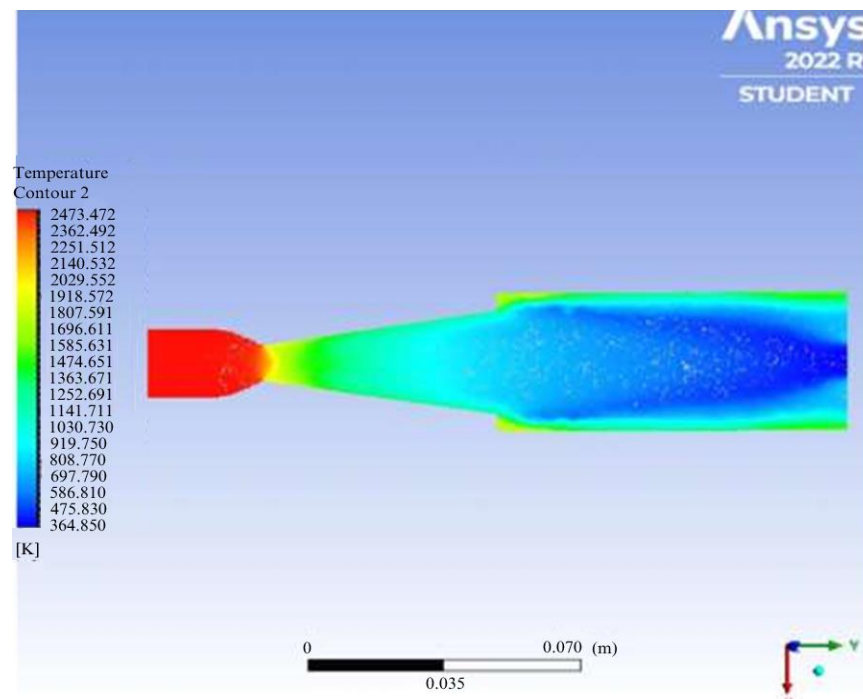


Figure 5. Temperature contour.

Density Contour

The density is greatest at the nozzle entrance and gradually decreases as we go towards the nozzle's exit. The minimum density of the nozzle is $0.00035138 \text{ Kg/ m}^3$. The maximum density of the nozzle is 0.237107 Kg/ m^3 (Figure 6).

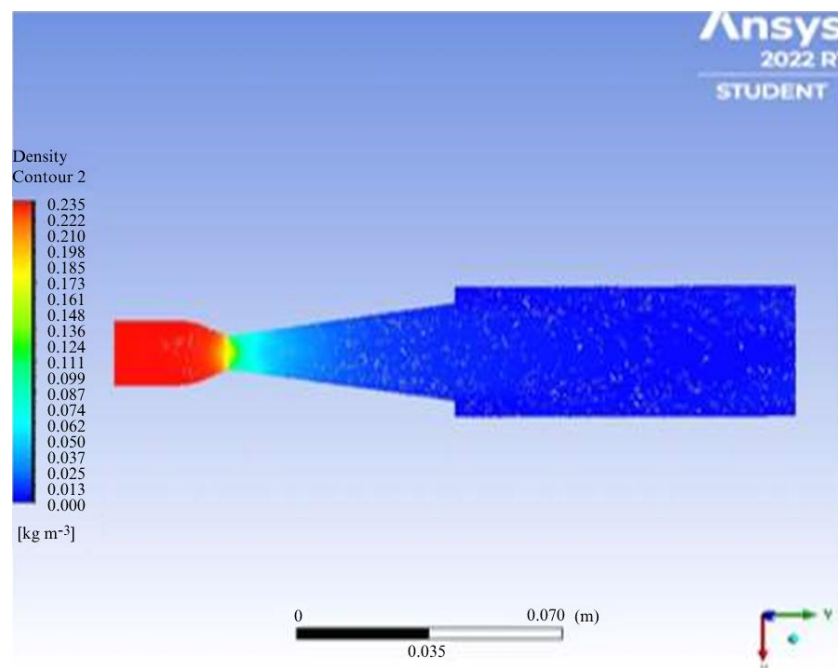


Figure 6. Density contour.

Velocity Contour

The velocity is lowest at the inlet and increases as we move towards the exit as we see the Figure 7. At the nozzle's throat, Mach is equal to one, indicating a choked flow condition. Choked flow is a

fluid dynamic state caused by the venturi effect [17]. The minimum velocity of the nozzle is 0 m/s. The Nozzle maximum velocity is 8245.54 m/s.

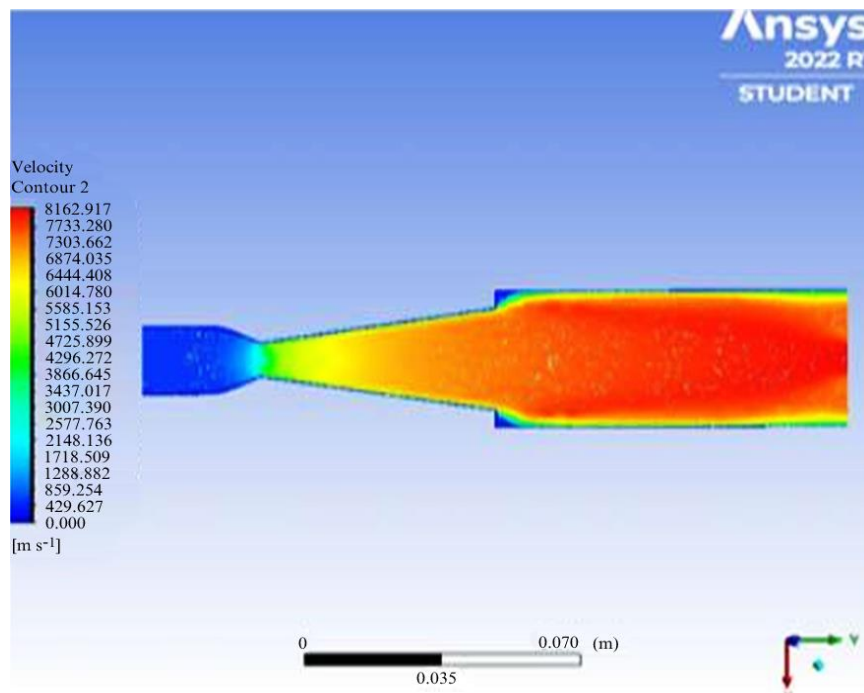
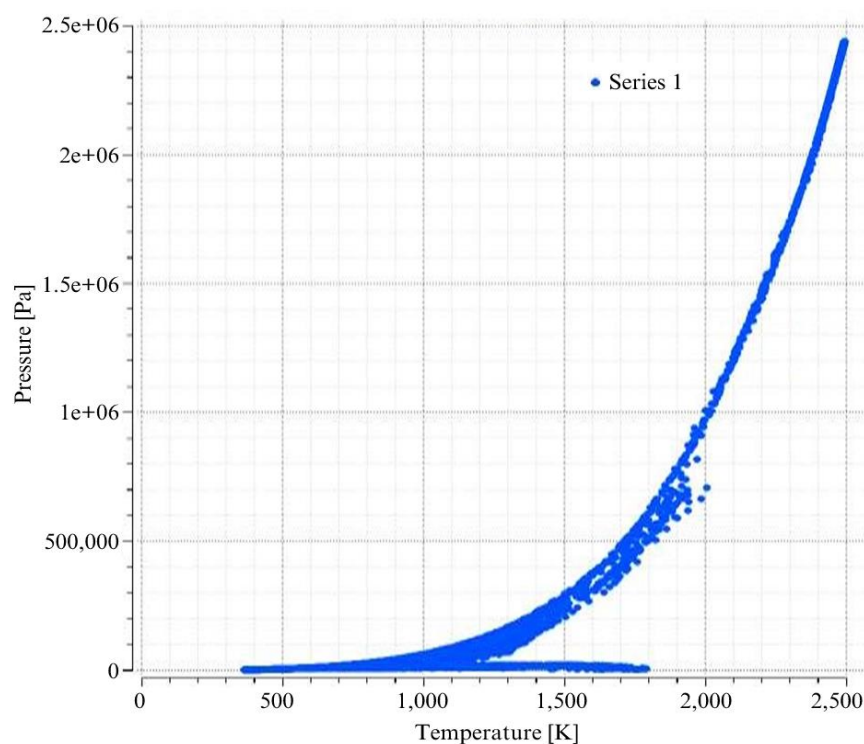


Figure 7. Velocity contour.

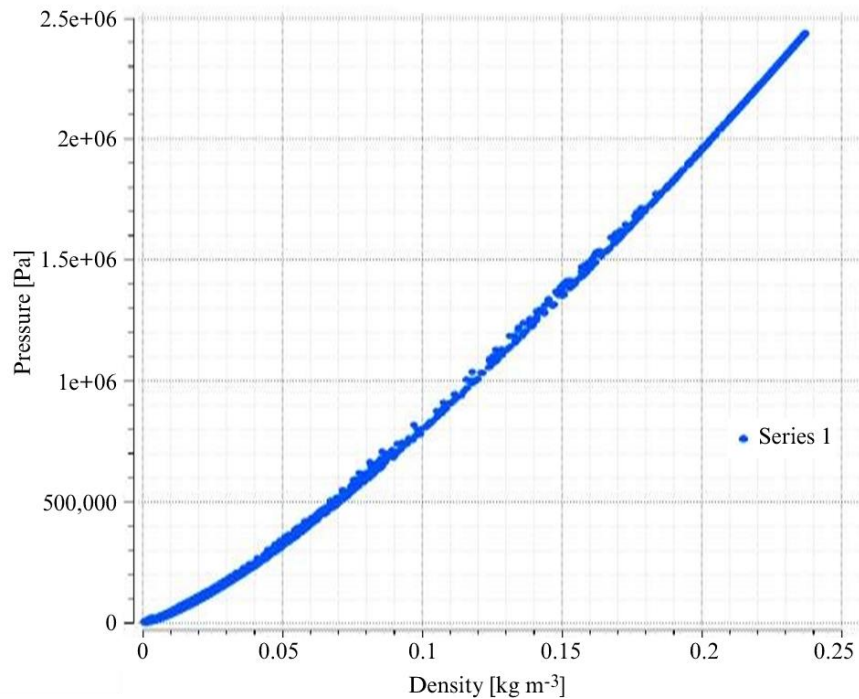
Graph

The Following is a Graphical Illustration of Nozzle Performance

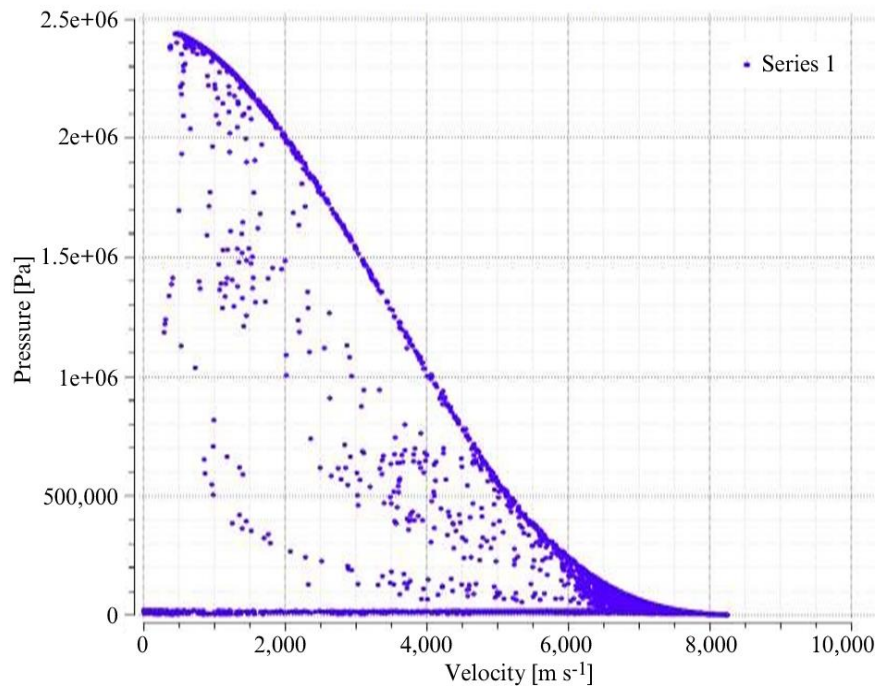
In the (Graphs 1–3), there are used 500 points of streamline to show the better result of the nozzle.



Graph 1. Pressure vs temperature.



Graph 2. Pressure vs density.



Graph 3. Pressure vs velocity.

CONCLUSIONS

A cryogenic rocket thrust chamber was designed using hydrogen and oxygen as fuel and oxidizer respectively using the CATIA V5 CAD software and its analytical simulations were carried out on Ansys fluent. In the paper, the analytical and theoretical data were compared and found to be in the acceptable range. Various contour representations showed the effective working of the nozzle and the graph represents the data extracted. The maximum pressure reached inside the nozzle is 2439640 Pa. The maximum temperature attained by the nozzle is 2491.81 K. The Nozzle maximum velocity

reached at the exit is 8245.54 m/s. Finally, more optimized configurations can be approached in future designs by gap location and nozzle type.

REFERENCES

1. Wikipedia contributors. Cryogenic rocket engine. Wikipedia, The Free Encyclopaedia 2022.
2. K. T. Venkateswara Rao, Weikang Yu R. O. Ritchie. Cryogenic toughness of commercial aluminium-lithium alloys: Role of delamination toughening. *Metallurgical Transactions A*. 1985; 20(3), 485–497. <https://doi.org/10.1007/bf02653929>.
3. P. Gaillard, C. Le Touze, L. Matuszewski, A. Murrone. Numerical Simulation of Cryogenic Injection in Rocket Engine Combustion Chambers. Aerospace Lab, Alain Appriou, 2016; <https://doi.org/10.12762/2016.AL11-16>.
4. Mukkarum Hussain, Iftikhar Ahmed, Ilyas Khan, Chu Anh My, Mirza Mehmood Baig, Afrasyab Khan, Stanislav S. Makhanov. Simulation of liquid fuel combustion start-up dynamical behavior. *Case Studies in Thermal Engineering*, 2021; <https://doi.org/10.1016/j.csite.2021.101025>.
5. Fechter, Stefan, and Horschler, Tim and Karl, Sebastian and Hannemann, Klaus. Investigation of RANS turbulence models for cryogenic rocket combustion chambers. 8th European Conference for Aeronautics and Space Sciences (EUCASS) 2019; <https://doi.org/10.13009/EUCASS2019-311>.
6. Gautam, V. Gupta, A. K. Simulation of flow and mixing from a cryogenic rocket injector. *Journal of Propulsion and Power*, 2007; 23(1), 123–130. <https://doi.org/10.2514/1.19731>.
7. Schmitt T.Staffelbach G. Ducruix S. Groening S. Hardi J. Oschwald M. Large-Eddy Simulations of a sub-scale liquid rocket combustor: influence of fuel injection temperature on thermo-acoustic stability. 7TH EUROPEAN CONFERENCE FOR AERONAUTICS AND AEROSPACE SCIENCES (EUCASS) 2017; <https://doi.org/10.13009/EUCASS2017-352>.
8. Abdul Khalilullah, Sk. J. Basha & H. K. Rangavittal “Design and Analysis of Propellant Tanks Support Structure for an Advanced Spacecraft” *journal of applied research in mechanical engineering* – 2011.
9. Naraghi M. Dunn S. Coats D. Dual regenerative cooling circuits for liquid rocket engines. 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. 2006; <https://doi.org/10.2514/6.2006-4367>.
10. Donald E. Jackle Jr. Design and Manufacturing of propellant tank assembly, *journal of AIAA*. 2007.
11. Wikipedia.org. Retrieved.2022.from <https://en.wikipedia.org/wiki/CATIA>.
12. Wikipedia.org. Retrieved 2022. <https://en.wikipedia.org/wiki/Ansys>.
13. Sharma K. Manohar, P.M.S.S., & Mishra, A.I. Analysis and optimization of Dual secondary fuel injector for Thrust vector control. *Journal of Physics. Conference Series*, (2022) 2272(1), 012002. <https://doi.org/10.1088/1742-6596/2272/1/012002>.
14. Nasa.gov. Retrieved, from <https://www.grc.nasa.gov/www/k-12/rocket/isentrop.html>.
15. Lamendola M. Top 5 Do's & Dont's for CFD. SimuTech Group. 2020; <https://simutechgroup.com/top-5-dos-donts-for-cfd>
16. Rajarajan S. Manikandan P.Lokesh Kumar K Surya PL Syed Masood T S Karthikeyan M. Computational flow analysis over an aircraft wing by incorporating turbulent flow generators. *IRJET Journal*. 2020; 7 (10).
17. Wikipedia.org. Retrieved 2022; from https://en.wikipedia.org/wiki/Choked_flow.