International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878, Volume-8 Issue-3, September 2019

Flow Analysis of Hybrid Rocket Nozzle Exhaust and Its Effects on Launch Pad and Guide Stand

Tejas Christopher, Prajwal Ashok Kumar, Siddalingappa P K, N K S Rajan

Abstract: ***The exhaust from the nozzle will be at very high temperature and the high temperature exhausts effect the launch pad and guide stand. In this research, the computational analysis is done on Hybrid Rocket Nozzle Exhaust and Its Effects on Launch Pad and Guide Stand. The flow characteristics such as pressure, temperature, velocity and Mach number at different time period have been studied for various exit geometries using ANSYS Fluent solver. Exhaust speeds vary, depending on the expansion proportion of the nozzle. The rocket nozzle along with guide stand is modelled. The aeroacoustics effects from the nozzle on Launch Pad and Guide Stand have been studied separately using acoustics model.***

Keywords: ***Hybrid rocket, nozzle, supersonic jet, expansion, Computational Fluid Dynamics***

I. INTRODUCTION

A rocket engine is essentially portrayed as a propulsion device which launches, by ejecting the mass stored to create thrust-capitalizing on the discussion of momentum depicted by Newton's Third law. The rocket engines are the best for propelling space vehicles, since they carry their own oxidizer which makes the propulsion independent of ambient atmosphere and it provides high energy system [1-5]. Hybrid rocket engines exhibit some undeniable and unobtrusive advantages over solid and liquid propellant engines. Some of the advantages are: -

- Mechanically simpler - has simple operation compared to liquid propellant engines.
- It can be controlled easily compared to solid propellant engines.
- It has higher specific impulse, as metal additives like aluminium; magnesium etc can be included in fuel grain.
- Unlike liquid propellant engine propellant pressurization is simple and easy in hybrid engines.

The remainder of the report deals with the basic study of the hybrid rockets and deals with flow outside the nozzle as follows.

The flow characteristics of under expanded jet from different nozzle such as square, circular, elliptical and rectangular are studied using eddy simulation at a pressure ratio of 5.60. The results show the jet penetrates fastest in the square nozzle and even though the turbulence transition is similar [6]-[7]. The effects of nozzle, jet temperature and heat transfer, distribution due to the jet on the smooth surface plate are studied.

The local wall temperature is measured using a thin metal foil technique. Influence of jet temperature on local heat transfer for different Reynolds number is studied. Nozzle such as circular, triangular and rectangular influences the local heater and it is studied for different Reynolds number of different jet to plate distances. Nusselt number is also measured for a circular nozzle in comparison with square and triangular nozzles based on equivalent diameter is the highest. Influence of jet temperature on the heat transfer on the smooth plate is studied [8]. Nozzle expands and accelerates combustion products, which gives thrust and momentum for aerospace vehicle. The nozzle design is very much important in producing the required jet. The nozzle is designed according to the requirement of the system design. One of the major nozzle configurations is defined by the specific impulse required. The software’s like fluent can be used to perform numerical solution and to understand the performance of different nozzle. Software like CFD is used to create a required mesh and selection of suitable turbulence model for computing nozzle jet. Spalart-Allmaras model and SST model were used for simulation. Results showed that scarfed nozzle had a relatively low Mach number, when compared to conical nozzle [9]. The under expanded free sonic jet flow from different nozzle like rectangular, elliptical and slot nozzles is studied. Aspect ratios of 1, 2, and 4 are described at pressure ratios of 2 and 3. The qualitative agreement between the experiment and numerical is satisfying. The incident shock system is formed by a complex system of shock waves in the case of rectangular jets [10]. The flow phenomenon was observed through experimental and numerical methods to highlight different nozzle operations in ref [11] and they concluded that the structural disintegration of the inner nozzle structure can happen when operating at high heat fluxes and pressure oscillations. Numerical method was proposed to capture the flow structure and understand the transition in rocket engine [12], here it is observed the side load which is acting on the actuator of the nozzle. The behavior of the combustion gas jet in a Laval nozzle flow is studied by numerical simulations and CFD analysis and observed the pressure and temperature for a rocket nozzle with four inlets [13]-[14].

Manuscript published on 30 September 2019
***** Correspondence Author

Tejas Christopher, Department of Aeronautical Engineering, Nitte Meenakshi Institute of Technology, Bangalore, India
Prajwal Ashok Kumar, Department of Aeronautical Engineering, Nitte Meenakshi Institute of Technology, Bangalore, India
Siddalingappa P K*, Department of Aeronautical Engineering, Nitte Meenakshi Institute of Technology, Bangalore, India.
N K S Rajan, CGP Laboratory, Department of Aerospace Engineering, Indian Institute of Science, Bangalore, India

A. Basics of Hybrid Rocket

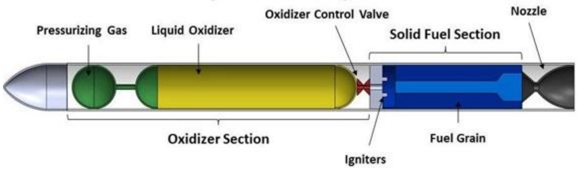

Fig. 1. Hybrid Rocket Configuration [15]

A hybrid rocket uses solid and liquid or gaseous propellants in two non-identical phases is as show in figure 1. Hybrid rockets have the benefit above both liquid and solid rockets. A simple hybrid rocket holds the liquid propellant (oxidizer) in pressure container and solid propellant in combustion compartment, and a valve isolating both the propellants. The valve is opened and ignited in combustion compartment, when the thrust is required. The liquid propellant is supplied to the combustion compartment, where it gets gasified and it reacts with the solid propellant in a thermochemical process.

B. Hybrid Rocket Nozzle

A rocket engine nozzle expands and accelerates the combustion gases produced from blazing propellants, so at the nozzle exit supersonic/ hypersonic velocities are obtained. Depending on the nozzle expansion ratio, the exhaust conditions vary. Rockets run with combustion temperatures that can reach up to 3400 K. The temperature of the exhaust from the nozzle is very high. The rocket engine nozzle should be optimal size, so that it can be used within the atmosphere and is achieved when the exit pressure almost equals to ambient pressure. For rockets travelling beyond Earth atmosphere, the nozzle is working optimally at specific designed altitude, and it loses efficiency at other altitudes. Every bit the throat passes, the gas pressure is higher, and it should be reduced among the throat and the nozzle exit. If the exhaust jet pressure at the nozzle exit is above ambient pressure, then a nozzle is said to be "under expanded"; if it is below ambient pressure, then it is called as "over expanded".There is a slight reduction in the efficiency for over expansion nozzle condition, but otherwise there is no harm. Nevertheless, if the exit pressure is less than approximately 40% that of ambient, then "flow separation" occurs. This can cause instability of the jet, which causes damage to the nozzle or it can cause difficulty in controlling the thrust production.

II. METHODOLOGY

A. Nozzle Design and Geometry

In order to solve the flow at the nozzle exhaust, the nozzle is designed with the dimensions given in the figure 2 (All dimensions are in mm). Which ease our process in the fluid flow analysis of the rocket nozzle. With the consideration of these dimensions, we have considered a boundary for the flow analysis of the exhaust and designed the model using ICEM CFD, wherein we can analyze the flow characteristics coming out of the nozzle. The boundary considered is the distance from the exit of the nozzle to the point of any obstruction, which is ground or a flat plate.

B. Mathematical Model

To solve this exhaust flow at the converging-diverging nozzle, few governing equations are used in the calculation in order to find the flow characteristics at the nozzle exhaust.

The equations are as follows,

Inlet Mach number: $\frac{A_i}{\mathrm{A}*} = \frac{1}{M}\left[\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M^2\right)\right]^{\frac{\gamma+1}{2(\gamma-1)}}$

At inlet,

Pressure ratio: $\frac{\mathrm{P}_i}{\mathrm{P}'} = \left[1+\left(\frac{\gamma+1}{2}\right)M^2\right]^{\frac{\gamma}{(\gamma-1)}}$

Temperature ratio: $\frac{\mathrm{T}_i}{\mathrm{T}'} = \left[1+\left(\frac{\gamma-1}{2}\right)M^2\right]$

At throat, Mach number = 1

At exit,

Area ratio: $\frac{A_o}{\mathrm{A}*} = \frac{1}{M}\left[\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M^2\right)\right]^{\frac{\gamma+1}{2(\gamma-1)}}$

C. Pre-Processing

The far field or the flow field is the place where the analysis of the flow coming out of the nozzle, generally the flow field is four times the actual nozzle geometry. The flow domain is created using the coordinate system of ICEM CFD. The flow domain for the given geometry is as shown in figure 3.

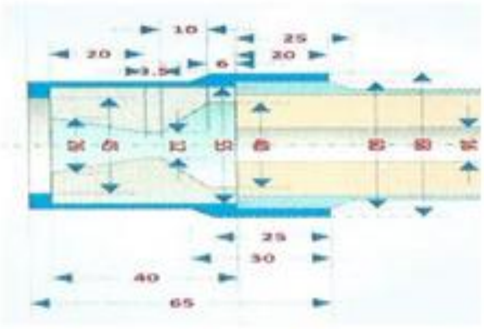

Fig 2. Nozal

The geometry indicates Inlet, Outlet, External boundary, Nozzle wall and Launch pad. The inlet indicates the flow entry where the flow starts and exits at the outlet; the geometry has been designed to have two exits as shown in the above figure. This geometry is useful in determining the type of flow.In order to examine the flow over the guide stand the above geometry has been modified to have one outlet and a guide stand as shown in the figure 4. The inlet indicates the flow entry where the flow starts and exits at the outlet. This geometry is useful in determining the type of flow and helpful in determining the temperature effects.

Retrieval Number C5538098319/2019©BEIESP
DOI: 10.35940/ijrte.C5538.098319
Journal Website: www.ijrte.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

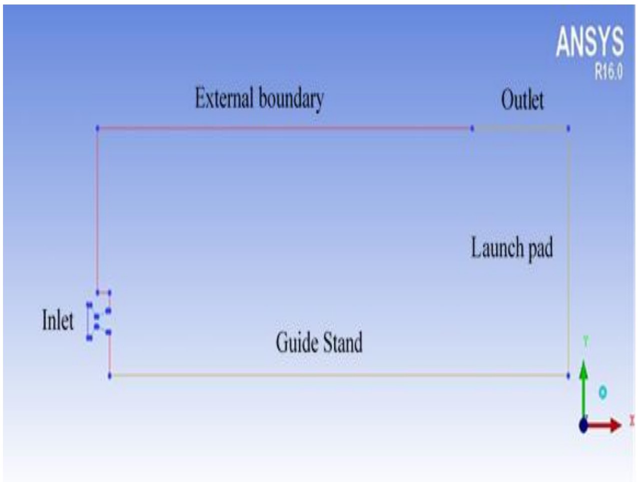

Figure 4: Far Field Geometry

D. Mesh Generation and its Quality

In order to achieve the solution, the geometries have been meshed using the ANSYS ICEM CFD as shown in figure 5.

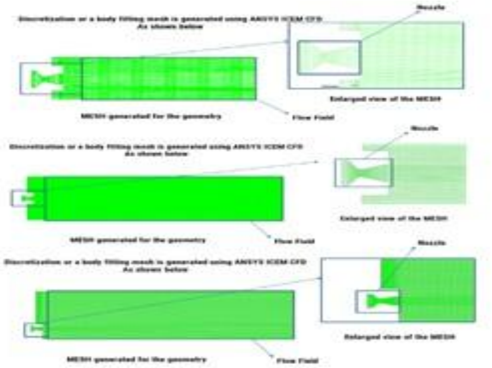

Fig. 5. Geometry creation and body fitting mesh.

E. Computational Approaches

Boundary Conditions

The solver used is a density-based solver in a steady state condition. The models used are energy equation, viscous model which is a standard k-epsilon radiation model: P1 model and acoustics model a broadband noise. The fluid selected is air and the cell zone conditions are a fluid. The following boundary conditions were used in the research work.

- Inlet: pressure inlet
- Supersonic inlet pressure: 5bar
- Inlet temperature: 3000K
- Outlet: pressure outlet
- Outlet gauge pressure: 0bar
- Outlet temperature: 300K
- Wall: stationary wall
- Nozzle wall: stationary wall

With the inlet pressure as 5 bar and operating pressure as relatively 0 bar.

III. RESULTS AND DISCUSSIONS

After setting up the problem and solving it with much iteration following results are obtained. While analyzing the problem we have followed a step by step approach in order to solve the problem. The problem is discretized into three parts and solved accordingly and discussed below.

A. Shock Structures from the Exhaust

The first step towards solving the problem was determining the shock structures from the exit of the rocket nozzle. The figure 6 shows the development of the shock near nozzle exit. From the velocity contours obtained, under expanded jet from the nozzle along with the intercepting shock and the reflected shocks are visualized.

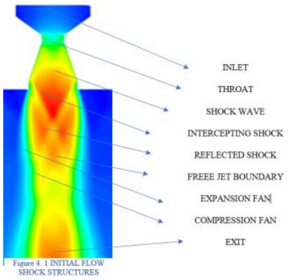

Fig. 6. Shock structure at nozzle exit

B. Complete Development of the Shock Structure

With the initial flow results of the under expanded jet, along with iteration or to say increase the time of flow, it resulted in a better shock structure. The newly formed or completely developed shock structure is obtained as shown in figure 7.

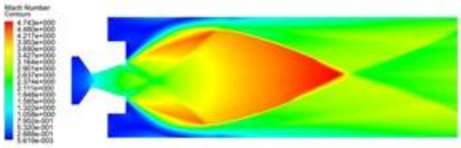

Fig. 7. Mach number for fully developed jet

The above picture dictates the result obtained during the fully developed flow out of the nozzle and gives a clear understanding of how the exhaust jet will behave when the flow expelled out from the nozzle. Here the shock structures are properly formed indicating proper intercepting shock and the reflected shock.The shape of exhaust jet and the shocks in it are studied and further considered for the study of jet and launch pad interactions.

C. Nozzle Exhaust with Launch Pad

To begin with this interaction, study a wall is created which will be facing the jet coming out of nozzle. The wall geometry and respective body fitting mesh is created. The boundary conditions for the study are as follows.

- Inlet: pressure inlet
- Supersonic inlet pressure: 5bar
- Inlet temperature: 3000K
- Outlet: pressure outlet
- Outlet gauge pressure: 0bar
- Outlet temperature: 300K
- Wall: stationary wall
- Nozzle wall: stationary wall

Resultant Contours: The resultant contours depict the results obtained by solving the problem for around 80000 iterations, the resultant graphs are as shown below.

Pressure Contour: The pressure contour obtained as shown below consists of shock structure at the exit of the nozzle and keeps on decreasing to maintain a constant static pressure, nearing to end where the plate is placed a normal shock is obtained as shown in figure 8.

Figure 8: Pressure contours for with launch pad case

Velocity Contour: The velocity contour depicts the shock structure or the shock diamonds exiting with a velocity of around 180 m/s and progressively decreasing with the distance and causing a normal shock to appear towards the end where the plate is placed and is as shown in figure 9.

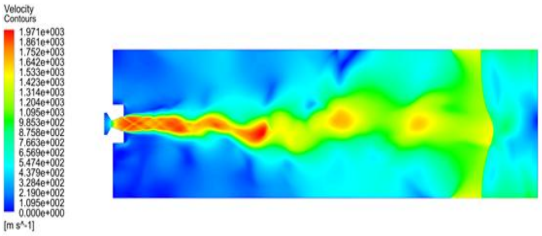

Fig. 9. Velocity contours for with launch pad case

Mach number Contour: The Mach number contour depicts the shock structure or the shock diamonds which are expelling out of the nozzle with a Mach number of around 2.8, and progressively decreasing with the distance away from the nozzle. This causes the normal shock to appear near the plate which acts as a launch pad is shown in figure 10.

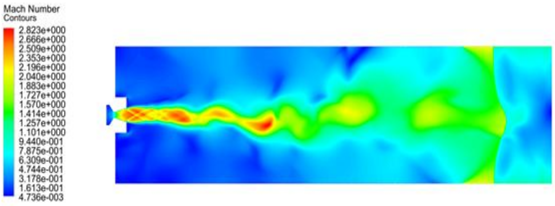

Fig. 10. Mach number contours for with launch pad case

Temperature Contour: Temperature contour consists of a cold region in the middle or at the exit of the nozzle compared to the adjacent to it. The temperature progressively decreases with the distance causing a normal shock towards the end which is reflected from the plate as shown in the figure 11.

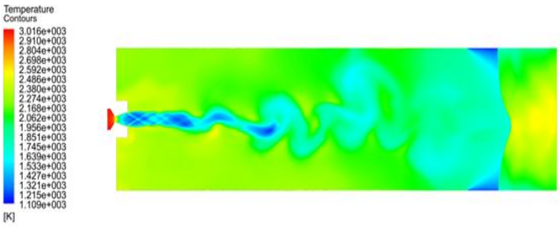

Fig. 11. Temperature contours for with launch pad case

D. Nozzle Exhaust Jet with Launch Pad and Guide Stand

A guide stand is a supporting structure to the rocket before take-off, the structure provides mechanical support to the rocket and houses other complimentary systems which help in successful take-off of the rocket.

The launch pad and the guide stand are the main structures which are affected by the nozzle exhaust, temperature, the plume from the nozzle exhaust heats up the surface causing thermal damage to the structure.

In order to simulate the launch pad and the guide stand effects the geometry needs to be modified from that of the previous one. With the mesh generated setting up the fluent is an important task, the boundary conditions used in this case are as follows

- Inlet: pressure inlet
- Supersonic inlet pressure: 5bar
- Inlet temperature: 3000K
- Outlet: pressure outlet
- Outlet gauge pressure: 0bar
- Outlet temperature: 300K
- Wall: stationary wall
- Nozzle wall: stationary wall

E. Resultant Contours

Pressure Contour: The pressure contour obtained shown in figure 12, shows the proper shock structure with the intercepting shock. Here the shock diamond is slightly bent towards the wall rather than the usual straight shock wave produced.

Fig. 12. Pressure contours for with launch pad and guide stand case

Velocity Contour: The velocity contour shown in figure 13. indicates a bent shock structure in resemblance to the first phase of our problem. Here the exit velocity of the nozzle is around 180 m/s and it follows increase and a decrease path wherein the velocity at the exit decreases, meanwhile there is a normal shock created at the corner of the two plates indicating very less velocity in that region.

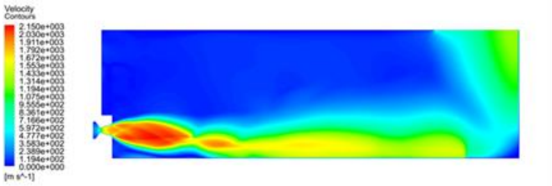

Fig. 13. Velocity contours for with launch pad and guide stand case

Mach number Contour: The Mach number contour obtained indicates a shock structure that is more likely bent towards the wall, in resemblance to the first phase of our problem. Here the exit Mach number from the nozzle is around 2.8 as shown, and it follows an increase and followed by a decrease in Mach number wherein the velocity at the exit decreases, meanwhile there is a normal shock created at the corner of the two plates indicating very less Mach number in that region. It is as shown in figure 14.

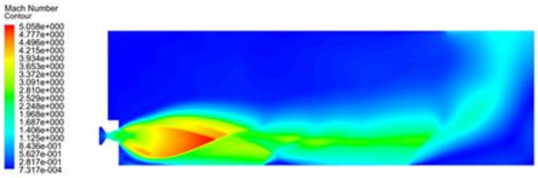

Fig. 14. Mach number relation for with Launch pad and guide stand case

Temperature Contour: In the temperature contours it can be seen that the temperature entering the nozzle is 3000K and temperature at the exit of the nozzle is around 1100K and the temperature existing in the flow field is around 2000K and also there is a formation of normal shock at the corner of the plate where the temperature is around 2800K. Temperature contours obtained are as shown in figure 15.

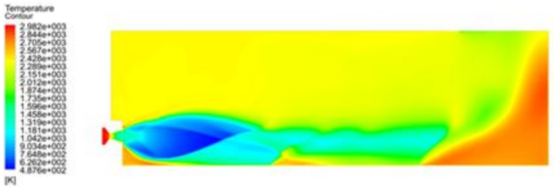

Fig. 15. Temperature contours for launch pad and guide stand case

F. Nozzle Exhaust Jet with Launch Pad and Guide Stand at 300[K]

From the above study it can be said that the temperature of the flow exiting the nozzle is very high to cause material fail. For this reason, it is required that a coolant system be provided that can maintain the temperature at 300K. For our consideration we are simulating the plate to be maintained at 300K at all the time during the flow.A guide stand is a supporting structure to the rocket before take-off, the structure provides mechanical support to the rocket and houses other complimentary systems which help in successful take-off of the rocket. Resultant Contours

The resultant contours depict the results obtained by solving the problem for around 80000 iterations, the resultant graphs are as shown below. Pressure Contour: The pressure contour obtained shows the proper shock structure with the intercepting shock is as shown in figure 16. Here the shock diamond is slightly bent towards the wall rather than the usual straight shock wave produced.

Fig. 16. Pressure contours for launch pad and guide stand case

Mach number Contour: The Mach number contour obtained indicates a shock structure that is more likely bent towards the wall, in resemblance to the first phase of our problem. Here the exit Mach number from the nozzle is around 2.8 as shown in figure 17, and it follows an increase and followed by a decrease in Mach number wherein the velocity at the exit decreases, meanwhile there is a normal shock created at the corner of the two plates indicating very less Mach number in that region

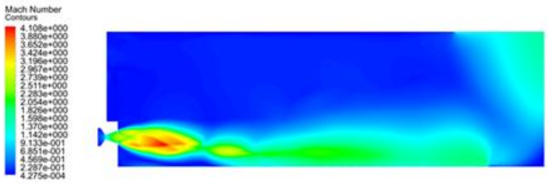

Fig. 17. Mach number contours for launch pad and guide stand case

Temperature Contour: The temperature contours shows a cold region in the flow that is in the central portion of the shock compared to other regions of flow, the temperature entering the nozzle is 3000K, and temperature just exiting the nozzle is around 1100K and the temperature exiting the flow field is around 2000K and also there is a formation of normal shock at the corner of the plate where the temperature is around 2800K. Here the plate is maintained at 300K whose temperature is not exceeding more than 420K as shown in the figure 18.

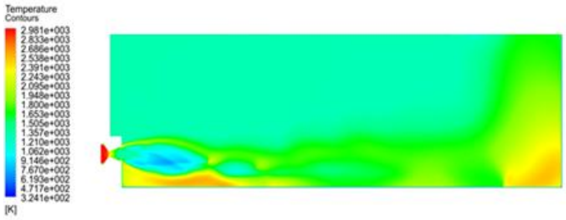

Fig. 18. Temperature number contours for launch pad and guide stand case

G. Observation

The above results are obtained when a Mild steel plate acting as a launch pad and the guide stand is placed and is being maintained at 300K. The flow coming out of the nozzle is like the flow wherein no wall is maintained at any temperature. Here the velocity is increased, and the temperature is slightly reduced in the flow region as shown in above contours

H. Comparative Results

For a mild steel plate maintained at 300K and a plate without any temperature range.

The comparative study is difficult to make, hence hard points are considered. The data points are obtained by plotting temperature versus the distance of the launch pad and the guide stand for each iteration. The iterations are the indicators of flow time. Temperature graphs for both the cases at increasing time step is plotted and the results are shown in figure 19.

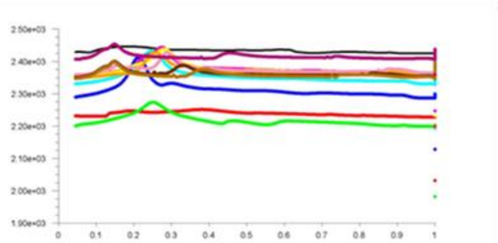

Fig. 19. Temperature (K) Vs. Position (m) graph without Mild Steel Plate

The above graph is obtained by collecting data points for temperature versus the distance of the plate, here the temperature lies within the range of 2200K-2500K, which is a very high range of temperature for any material to withstand.The figure 20 represents the temperature variation along the mild steel plate for different flow time, here initially the temperature was around 300K to start with and kept on increasing up to 420K as shown in the graph.

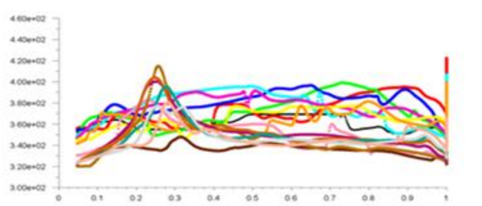

Fig. 20. Temperature (K) Vs Position (m) graph with Mild Steel Plate

In this graph the temperature reaches a peak value around 250mm and then returns to normal condition, the peak value represents highest temperature on the steel plate that is the place where the maximum temperature from the nozzle is impinged on the steel plate, or the place where the shock formation bends a little towards the plate.

With the temperature graphs obtained we are now able to have a comparative result with and without the steel plate as shown above. Without a steel plate the temperature ranges around 2200K -2500K whereas by placing the mild steel plate it ranges only around 300K-420K respectively

The study helps in determining the temperature ranges and helps in selection of a suitable material and a suitable coolant to maintain the temperature by referring the above graphs

IV. CONCLUSION

In this study simulation of under expanded nozzle and the temperature effects on the launch pad and the guide stand are studied using the fluent solver. Three different geometries with different outlet locations is used to determine the flow behavior of the jet and the temperature effects of it. The mathematical models along with various boundary conditions are employed to investigate the flow behavior of the jet. Based on the results, following concluding remarks can be made.

- The evolution of flow and the development of flow at different time period is simulated for various exit geometries indicating the gradual increase in the velocity, Mach number and various other parameters
- The obtained values from the contours of velocity, Mach number, pressure and temperature at the inlet, throat and the exit of the nozzle is obtained at the inlet, throat and the exit of the nozzle.
- The flow features of the three different jets obtained for three different geometries are like each other in terms of formation of the shock structure. And all the three jets show under expansion type of nozzle properly.
- The predicted exiting shock structure from the nozzle and the resulting shock structure properly indicated the various parts of the shock diamond.
- The velocity of the gas at the entry of the nozzle is very less and increases gradually at the throat and exit of the nozzle and thereby maintaining a near steady exit as shown in the contours.
- The jet temperatures exiting from the nozzle are studied in detailed and observed that the temperature reduces as it leaves the nozzle and causes an impact on the surrounding equipment's of the rocket.
- The temperature plots with respect to the distance at different flow time helps us to study the increment of temperature in the flow for different time periods. These plots help to determine the flow temperatures on various affected geometries and predict suitable measures to prevent it. The temperature graphs help in determining the type of material that can be used to withstand this high temperature.
- Lastly the aeroacoustics effects from the nozzle is the power of sound coming out of the nozzle helps us in guiding the amount of noise generated by the expanding nozzle and predicting the preventive causes for it.

REFERENCES

1. "Hybrid Rocket Propulsion Overview". Space Propulsion Group, Inc.
2. HIH Saravanamuttoo, GFC Rogers, and Cohen, H, "Gas Turbine Theory", ISBN-13: 978-0132224376, ISBN-10: 0132224372
3. V Babu. "One dimensional flows" from Fundamentals of Gas dynamics"
4. V Babu. "Normal shock waves and Oblique shock waves" from Fundamentals of Gas dynamics
5. Yunus A Cengel and John M Cimbala, 'Fluid Mechanics" ISBN-13: 978-0071249348, ISBN-10: 0071249346
6. Xiaoping Li, Rui Zhou, Wei Yao, Xuejun Fan, "Flow characteristic of highly under expanded jets from various nozzle geometries" Applied Thermal Engineering, July 2017, DOI: 10.1016/j.applthermaleng.2017.07.002
7. Finley, P. 'The flow of a jet from a body opposing a supersonic free stream". Journal of Fluid Mechanics, 26(2), 337-368. doi:10.1017/S0022112066001277

8. Ravish Vinze, S Chandel, M.D Limaye and S.V Prabhu. "Influence of jet temperature and nozzle shape on the heat transfer, distribution a smooth plate and impinging air jets", International Journal of Thermal Sciences, Volume 99, January 2016, https://doi.org/10.1016/j.ijthermalsci.2015.08.009
9. L. Sushma, A. Udaya Deepak, Sathish Kumar Sunnam, Dr. M Madhavi, "CFD Investigation for different nozzle jets", ICAAMM-2016 Volume 4, Issue 8, https://doi.org/10.1016/j.matpr.2017.07.263
10. Menon, N. & Skews, B.W. Shock Waves (2010) 20: 175. https://doi.org/10.1007/s00193-010-0257-z
11. Gerald Hagemann, Hans Immich, Thong Van Nguyen, Gennady E. Dumnov. "Advanced Rocket Nozzles", Journal of Propulsion and Power Vol.14, No.5, September –October 1998
12. Taro Shimaizu, Mastoshi Kodera and Nobyuki Tsuboi "Internal and External Flow of Rocket Nozzle" Journal of earth simulator, Vol 09, March 2008.
13. Zhao-Xin Gong, Chuan-Jing Lu, and Jia-Yi Cao "The Gas Jet Behaviour in Submerged Jet Nozzle", Journal of Hydrodynamics, Ser. B, Volume 29, Issue 6, December 2017, https://doi.org/10.1016/S1001-6058(16)60817-X
14. K M Pandey, "CFD Analysis of a Rocket Nozzle with Four Inlets at Mach 2.1" International Journal of Chemical Engineering and Applications, Vol. 1, No. 4, December 2010, ISSN: 2010-0221
15. http://www.spaceflightinsider.com/missions/commercial/rocket-crafters-notes-safety-hybrid-rockets-spacex-disaster/

AUTHORS PROFILE

Mr. Tejas Christopher: is currently pursuing his MBA in Aviation Business Management at Jain CMS Business School, Bangalore, India. His fields of interests are in aerodynamics and rocket propulsion. He is a graduate in Aeronautical engineering from Nitte Meenakshi Institute of Technology, Bangalore, India in the year 2018.

Mr. Prajwal Ashok Kumar: is currently a post-graduate student at Cranfield University, United Kingdom. His study research focuses mainly on aircraft vehicle design with specific attention to the structural analysis. He is a graduate in Aeronautical engineering from Nitte Meenakshi Institute of Technology, Bangalore, India in the year 2018.

Mr. Siddalingappa P K: is Assistant Professor in the Department of Aeronautical Engineering Nitte Meenakshi Institute of Technology, Bangalore. His fields of interests are Aerodynamics and Computational Fluid dynamics. He has got degree M.Tech degree in Aeronautical Engineering in 2014 from VTU Belagavi, Karnataka, India.

Prof. N K S Rajan: is a Chief Research Scientist in Combustion, Gasification & Propulsion Laboratory (CGPL), Department of Aerospace Engineering, Indian Institute of Science, Bangalore, India. completed his Ph.D. in Aerospace Engineering, from IISc Bangalore in 1990. He has than 25 years of Academic Research and R&D in sponsored projects in Government Sponsored & Industrial Consultancy and expert member of several committees of the Institute and outside in Governmental and Autonomous Institutions. His fields of research are Combustion; Heat Transfer; Fluid Dynamics (Experimental and Computational); Biomass Gasification; Renewable Energy; Advanced CFD Simulations, Instrumentation.