CONCEPTUAL DESIGN AND ANALYSIS OF TWO STAGE SOUNDING ROCKET

***Ankit Kumar Mishra, **Kshitij Gandhi, **Kuldeep Sharma, #Neerumalla Sumanth, $Y.Krishna Teja**

[*]*Department of Research and Development, Abyom Space tech and Defence Pvt. Ltd, Gorahkpur-274402, India.*
[**]*Department of Aerospace Engineering, School of Aeronautics, Rajasthan- 301705, India.*
[#]*Department of Aerospace Engineering, Hindustan Institute of Technology and Sciences, Tamil Nadu -521230, India.*

ABSTRACT

The objective of this paper is to design a two stage sounding rocket and its nozzles using fusion 360 and analysis of different properties using simulation on ANSYS software. The rocket is designed to reach maximum apogee to perform scientific experiments and can be recovered safely after use. Number of CFD simulations were done on structural design with different parameters to analyze the aerodynamic characteristics of the different stages of rocket and the verification of the nozzles.

***Keywords** – rocket, nozzle, ANSYS, CFD, fusion 360.*

INTRODUCTION

Sounding rocket- sounding rockets are specialized rockets designed for research and scientific purposes, at different orbits of earth. They can attain an apogee of 100km to 2000km. As orbital mission rockets are very costly, bigger in size and take a lot of time to design but for simple research purposes it will be extremely expensive, taking these causes in consideration, sounding rockets are built.

Nose cone- a nose cone is the forward section of the rocket vehicle which guides the rocket during the flight to outer space. It is conical in shape so that it can accumulate maximum speed and reducing drag to minimum achieving highest possible apogee during the flight. It can be used to carry satellite/payload for the mission as it gets detached after the required height is achieved. There are various nose cone such as 'Conical, Ellipsoid, Ogive, Parabolic, Power series, Haack series' designed and selected as per mission requirement.

Fins- fins are appendage attached to the structure of the rocket body, they play very important role to provide stability to the space craft during the flight that allows the rocket to maintain its flight path and orientation, the shape and design of the fins will decide how stable a vehicle will be. General stability factor falls under 1 to 2, under which it will be under-stable and above which it will be over-stable. Stability is calculated by using

stability $= \frac{C_g - C_p}{2}$ Where, C_g= location of center of gravity, C_p= location of center of pressure.

There are a lot of different type of fin shapes namely trapezoidal, clipped delta, rectangular and parallelogram fins, they are tested in wind tunnel experiment and chosen according to mission requirement. Number of fins attached to the body will decide stability of rocket, preferably 3 to 4 fins should be used to stabilize the rocket.

Rocket engine- rocket engine nozzle plays a vital role in space flight, they are used to expand and accelerate combustion of fuel and gases produced by burning propellants at high velocity out of the exit of the nozzle which produces thrust and lift the body from ground to reach orbit and outer space. Usually "de-Laval" type of rocket engine is used. To get optimum results it is preferred that the pressure at exit of nozzle to be equal to ambient pressure outside of nozzle, i.e. “Pe=Pa” below which it will be under expanded which will provide a weaker force and above which it will be over expanded increasing the probability of nozzle explosion. For vacuum use it is almost impossible to match exit pressure with ambient pressure, rather, nozzles with larger area ratios are more efficient.

Multistage rocket- multistage rocket generally termed as step rocket is a vehicle that uses two or more stages having its own engine and respective propellants. Multistage rocket provides multiple advantages over normal single stage rocket as it consumes less fuel and achieve greater height while reducing the weight of the rocket and propellant requirements. Every stage has its own purpose and after fulfilling it is detached from the main body lessening the overall weight of the rocket and providing extra impulse to the vehicle, for this paper two staged rocket is used in which the first stage which works as booster stage, which will up-lift the rocket from ground and will attain a desirable height after that the second stage which is sustainer stage will carry out its mission of taking the payload to required apogee, this process is called staging of rocket, it is performed until the desired velocity is achieved.

Multiple software are used to design and analyze the rocket described in the paper, for rocket structure and stability open rocket software is used, Rocket propulsion analysis(RPA) is used to calculate dimensions for nozzles required in the rocket. Both the rocket and nozzles design are then 3 dimensionally put up in fusion 360 CAD software. For analysis as well as simulation of rocket and nozzle bodies, ANSYS is used.

LITERATURE REVIEW

Lucas de Almeida Sabino Carvalho et.al: calculated and analysed the drag force due to different shapes of a nose cone of a rocket. A nose cone is an important component of a rocket structure and the drag forces over it are required to be measured to make it better for a rocket mission. The four discussed shaped are – Tangent Ogive, Parabolic Ogive, ellipsoidal ogive and conical shape. The designs and computational domain were made in SOLIDWORKS and the simulations were done in ANSYS FLUENT using the Shear Stress Transport k-ω turbulence model. The Mach no. range taken is about 0.05 to 0.62. The results show the conical shape has the most drag while other three having almost the same. Elliptical one has the least of three but it starts increasing suddenly as the Mach closes in to sonic. In conclusion the elliptical shaped nose cones are best for sub sonic flights while for supersonic- parabolic or tangent should be considered. [1]

LC Ji1 et.al: studies and compares the Wrap around fins (WAFs) with flat fins. The WAFs provide a rolling motion to the rocket. Various configurations are studied like, span to chord ratio, curvature radius and setting angles. The Spalart-Allmaras turbulence model is used. Data that was observed from the analysis is, drag is slightly lesser in M=4 than that in M=3, Flat fins have higher lift characteristics and pitching moments but WAFs have better stability in longitudinal axis. The self-induced moments and extra forces are due to the asymmetric shape. The extra moments and forces can be reduced at negative angles that can also lead to improved flight conditions at bigger angle of attacks. [2]

M. Abhinav et.al: analysed blunt nose cones with different fineness ratios in supersonic conditions. Fineness ratio is the ratio of nose cone length and the base diameter. The drag coefficient was found using ANSYS FLUENT. The observations showed that the drag coefficient reduces as we increase the fineness ratio but we cannot increase it infinitesimally as the skin friction drag will start becoming very prominent. It was also stated that wave drag contributes the most in supersonic speeds and the base drag is negligent. [3]

Girish Kumar et.al: analysed the flow over nose cones at transonic speeds. The author tried to find the aerodynamic heat over the different nose cones and which one will have the least drag coefficient. The designs were made in CATIA and simulated in ANSYS using SST k-omega model. It was observed that the conical nose shape experiences the normal shock and is reduced to subsonic speeds. Blunt nose cones experience the highest pressure. Ogive have least drag coefficient but the aerodynamic heating is higher. Flow separation is most prominent in blunt nose cones. [4]

A Sanjay Verma et.al: Comparison of various nose profile is carried out to know performance over existing nose profiles discussed in the paper. The objective of this paper is to identify the type of nose profile and with specific aerodynamic characteristics with minimum pressure coefficient and critical Mach number. Main purpose of this paper is to develop some prototype profiles with outstanding aerodynamic qualities and low cost for use in projects. The designs were made in ANSYS software. Flow observation in numerical simulation was done at Mach 0.8 for different nose profiles, and performance characteristics of selected profiles are presented. Von Karman Ogive nose profile give higher critical Mach number and minimum pressure coefficient which is desirable for the subsonic flow. [5]

Yong-Chao Chen et.al analysed the aerodynamic characteristics of a canard guided rocket. The author used both the mathematical and computational methods. ANSYS FLUENT was used to analyze the various configurations of the canard and its effect on the nose, fin and the canard itself. The results showed that the force in axial direction decreased with positive change in length of the Carmen curve. There was an increase in normal force coefficient, and betterment in the static stability. The change in span length led to increase in axial force. The stability and forces improved with a greater number of fins and reducing the number and increasing the aspect ratio led to better roll characteristics. [6]

Bogdan-Alexandru Belega et.al: analysed the flow inside a convergent-divergent rocket engine nozzle. A nozzle is used for producing kinetic energy from chemical energy. They are used for increasing the flow exhaust speed for greater thrust. The nozzle was design in GAMBIT software using conventional analysis and analysed in fluent using k-e turbulence model. It was observed that the there was an increase in velocity along the flow direction inside the nozzle. It was observed that there was sudden break in velocity due to a shock wave but after that there was normal increase in velocity. The authors concluded that the nozzle design worked as it should. [7]

Sreenath K R et.al: compared and analysed the flow through bell and dual bell nozzles. The nozzles were designed in gambit software and analysed in the fluent software using the SST k-omega model. Simulations were performed for velocity magnitude and static pressure. It was observed that dual bell has better performance at both lower and higher atmospheres and can save a lot of fuel at lower altitudes. Dual bells have better specific impulse too but the author concluded that it can be used for single stage orbital missions. It was also noted that bell shaped nozzle performs best at 1.5 Mach number. [8]

Manish Tripathi et.al: compared and analysed the effect of cross section and cascades on the rocket fins. The analysis was done by CFD. At lower angles of attack there is not much difference in lift from both flat plane cross-section and aerofoil cross section cascades. There is a positive change in lift for increasing the gap as the cascading effect is reduced. The aerofoil cross section has a faster and sharper stall angle. Drag is lesser for aerofoil wings. Aerofoil cascades have better aerodynamic properties and reduced drag. Flow separation is higher in aerofoil shapes at higher angles and pressure gradients. If the gap is reduced the angle for stalling can also be delayed. [9]

Md Nizam Dahalan et.al: Multiple aerodynamics characteristics where tested on a curved fin rocket to analyze how it performs at different speed. Numerical and semi-empirical method is used to study the rocket. USAF DATCOM was used as a reference for semi empirical method and ANSYS fluent gives the data for numerical method. The design of curved fin rocket consists of conical nose cone, 4 curved fins symmetrically attached and a cylindrical body-tube. The study was conducted under subsonic and supersonic speed, Mach number used for subsonic speed were 0.15, 0.4, 0.6 and 0.8 as for supersonic speed 1.2, 1.4 and 2.0 Mach number were used at an angle of attack varying from 0 degree to 25 degree. CFD analysis was used because it is cost effective and fast process then wind tunnel testing and if can also calculate lift and drag coefficient as well. In the results it can be seen by comparing the graphs of CFD and other methods that they show similar pattern. It can be seen that CFD analysis for force and drag coefficient were comparable with wind tunnel testing method. [10]

Leonid Shabliy et.al. This paper shows results of a new design of rocket engine with a thrust of 25N compared to a prototype with chamber burnout defect which was fixed in this new design. Compared to the prototype this new engine has more pressure in the chamber with results in decreasing value of burnout absence. Multiple researches with CFD model of chamber burnout were done to completely remove its possibility. The workability and feature of the new engine were shown and proven with the numerical simulation done on ANSYS software. It is designed so that this CFD model can also be used for different purposes. [11]

Matteo Poli et.al: This paper describes how sounding rockets are being widely used and how useful they are in the future. However as there are quite expensive and take a lot of time to be made, the author and his team decided to develop reusable as well as cost effective rockets which can carry heavier payload. MATLAB and Simulink software were used to make a numerical model to observe different parameter of the rocket like apogee prediction and simulating the trajectory, in case of crash predicting the impact ellipse, analysis the landing area and many other aerodynamic factors were analysis. The flight simulation data was compared to the real flight data giving satisfactory results. The software helped in better understanding of the flight on rocket in real time. [12]

Blazej Marciniak et.al: Research in microgravity is important in rocket science field as it allows the scientists to do multiple researches on the upper layer of the atmosphere and use of sounding rocket is one such method to do research in easy and effective way. In the paper it is experimented on a rocket named ILR-33 Amber to make a sounding rocket which is recoverable as well as cost effective as sounding rockets are expensive and difficult to manufacture. The design was made from the scratch and then simulation of it was done to find the effect of aerodynamics factors on it. Different technologies were used including solid propellant boosters, separation mechanics, and recovery subsystems. Payload for microgravity experiment was placed bottom of the nose cone having volume of 10litres. The design was successful in staying in microgravity for 150 seconds with an apogee greater than 100km. multiple on ground testing such as wind tunnel research and motor ignition were done. [13]

Ankith Y.S et.al: This paper focusses on the analysis of thermal properties having different fin materials and varying thickness. In automobile components engine cylinder plays a major role so maintaining its efficiency and sustainability is important and removal of heat has to done quickly and very precisely. Material like Aluminum 6061, aluminum 356 and aluminum 204 are used during the heat analysis and observing the characteristics of heat dissipation. Model design is made on SOLIDWORK software and ANSYS workbench is used for meshing of the design. Parameters such as geometry, material, number, and size as well as air velocity of fins are observed during simulation of the fins. After comparing it is observed that heat dissipated through stepped fin is more than rectangular fin model. Fin length of 16mm gives better performance than 13mm. Aluminum alloy 6061 is much better than other materials. Design with 7 fins shows better results than 5 fins. And velocity from 35kmph to 85kmph resulted in more heat dissipation than others. [14]

C.P. Hoult et.al: In the field of sounding rocket roll lock in also known as catastrophic yaw is the trickiest phenomenon. Out of several causes only one is discussed in this paper which is contact between fore body vortices and tail fins which produce non-linear as well as high angle of attack roll moment. Both the roll moments show comparable magnitudes. During the roll lock-in to achieve the condition of steady state and to calculate its probability of happening, rigid body momentum equation is used. It is the method which precisely decrease the probability by adjusting the fin which are openly exposed. And with the help of mathematical calculations it is shown that value of static margin greater than two caliber heuristic rule can cause the difficulty. Analysis on fine taper ratio was also conducted and is have very minimal effect on the fins. If number of fins are kept above four then probability of roll lock-in can be drastically decreased. [15]

METHODOLOGY

The vehicle described in this paper is designed to be robust, effective, light in weight and cost effective and it can be safely recovered after use.

The amount of thrust produced by a rocket is given by

$$F = \dot{m} * v_e + (p_e - p_0) * A_e \tag{1}$$

Specific impulse of the rocket is:

$$I_{sp} = \frac{F}{\dot{m} * g_0} \tag{2}$$

The isentropic equations are:

$$\frac{T_0}{T} = [1 + \frac{(\gamma - 1)}{2} * M^2] \tag{3}$$

$$\frac{p_0}{p} = [1 + \frac{(\gamma - 1)}{2} * M^2]^{\frac{\gamma}{\gamma - 1}} \tag{4}$$

$$\frac{\rho 0}{\rho} = [1 + \frac{(\gamma - 1)}{2} * M^2]^{\frac{1}{\gamma - 1}} \tag{5}$$

The blueprint of rocket is firstly design on "open rocket "software to make it accurate and with required stability for safe and steady flight. A nose cone of 50 cm in length and 15 cm in base diameter was designed, after checking performance of different nose cone shape ogive nose cone is selected. A body tube of 200cm was attached to the nose cone carrying payload, flight computer, recovery system and propellant used. The upper stage was designed with a thickness of 0.5cm and 4 fins of trapezoidal shape having 0.3cm thickness with airfoil shape. A transition was added to the body tube to separate upper and lower stage of rocket having 15 cm and 30 cm as fore and aft diameter respectively. Another body tube of 300cm in length and 30cm in diameter containing booster stage propellant was continuing the rocket body after transition having a thickness of 1cm and 4 fins of trapezoidal airfoil shape of 0.5cm supporting the body of rocket.

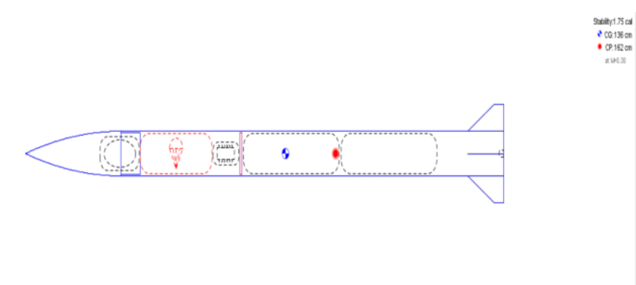


FIGURE 1. OPEN ROCKET DESIGN FOR UPEER STAGE

UPPER ROCKET PARAMETERS	VALUES
TOTAL MASS (KG)	45
PROPELLANT MASS (KG)	21
EMPTY/DRY MASS (KG)	24
PAYLOAD (KG)	10
DIAMETER (CM)	15
TOTAL LENGTH (CM)	250

TABLE 1. UPPER STAGE ROCKET PARAMTERS

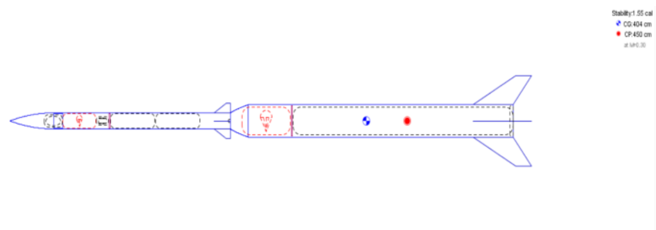


FIGURE 2. OPEN ROCKET DESIGN OF FULL ROCKET

FULL ROCKET PARAMETERS	VALUES
TOTAL MASS (KG)	321
PROPELLANT MASS (KG)	200
EMPTY/DRY MASS (KG)	121
DIAMETER (CM)	30
TOTAL LENGTH (CM)	591

TABLE 2. FULL ROCKET PARAMTERS

Rocket Propulsion Analysis (RPA) software was implemented for theoretical calculation of nozzles design of rocket body. Ammonium perchlorate with aluminum and HTPB was used as lower/booster stage propellant which will lift the rocket from ground providing thrust of 9500N at a chamber pressure of 320psi with area expansion ratio (Ae/At) of 20:1. Whereas, liquid oxygen and liquid methane was used as propellant for upper/sustainer stage to make the rocket achieve lower earth orbit applying thrust of 900N at 80psi chamber pressure with 30:1 area expansion ratio. The values obtained from the software are listed in the table below:

PARAMETERS	SUSTAINER STAGE	BOOSTER STAGE
CHAMBER PRESSURE(PSI)	80	320
THRUST(N)	900	9500
EXIT PRESSURE(PSI)	0.29	1.6679
PROPELLANT	LIQUID OXYGEN+LIQUID METHANE	AP/AL/HTPB
EXPANSION RATIO(Ae/At)	30	20
GAMMA	1.17	1.15
DENSITY	0.4215	2.3911
MACH NUMBER	3.7096	3.6841
O/F RATIO	3.153	4.077
REACTION EFFICIENCY(%)	95.9	97.02
NOZZLE EFFICIENCY(%)	97.75	97.7
OVERALL EFFICIENCY(%)	93.75	94.78
ISP(SEC)	335.27	161.4
MASS FLOW RATE(KG/S)	0.27373	0.81274
CHAMBER TEMPRETURE(K)	3200	2954.4
EXIT TEMPRETURE(k)	1969.45	1390.1
EXIT VELOCITY(M/S)	3306.85	2631.89

TABLE 3. NOZZLE DIMENSTION CALCULATIONS WITH RPA

After proper structural design of rocket body and its nozzles fusion 360 CAD software was used to create 3 dimensional structure of bodies, numerous functions provided by the software make the designing much easier and accurate.

FIGURE 3.CAD DESIGN OF 2 STAGE SOUNDING ROCKET- FUSION 360

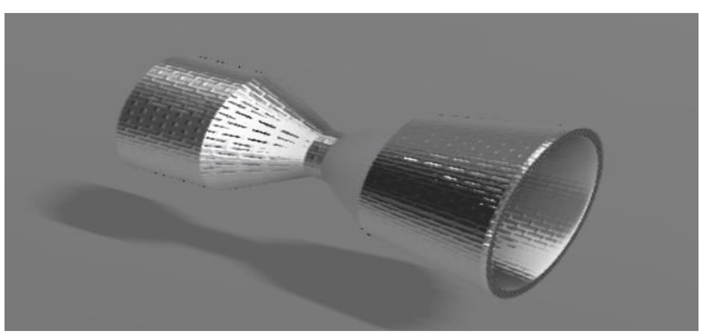


FIGURE 4. SUSTAINER STAGE NOZZLE DESIGN USING FUSION 360

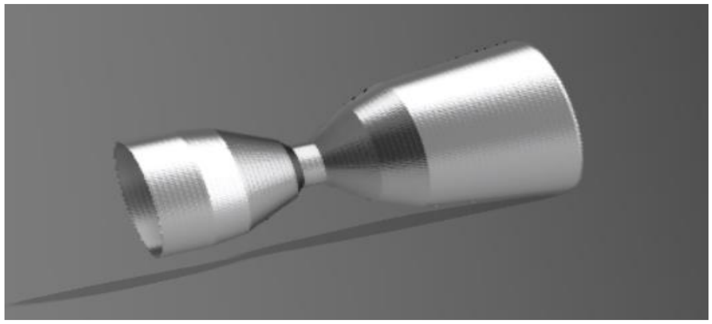


FIGURE 5. BOOSTER STAGE NOZZLE DESIGN USING FUSION 360

ANSYS software was used for analysis and simulation of the structural rocket and nozzles of different stage. The analysis was done on the complete rocket and the upper stage to compare and check the performance of vehicle at different situations. The analysis is done at different Mach numbers and operating conditions. Multiple graphs were plotted to compare the results and analyze the performance of the vehicle.

MESHING

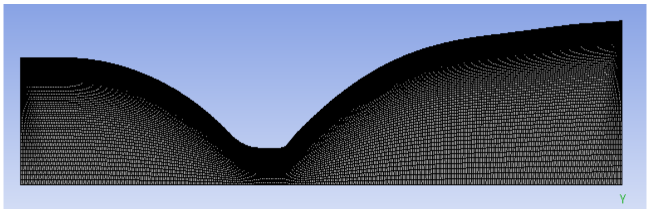
The meshing of the nozzles was done on Ansys Meshing software. Face meshing was applied to make a structured mesh and edge sizing was applied with required bias to capture the flow and boundary layers correctly during the simulation. Only half geometry of the nozzle was used due to the symmetry. For the rocket full body and rocket upper stage, proper domains were made in the Space claim software to capture the flow correctly. Since the rocket bodies have quarter symmetry so only that much of the body was used to save computational time. The meshing was done with fluent meshing software using *watertight* meshing. The polyhexa core type of volume mesh was used to provide the best quality of mesh.

Geometry	Nodes	Elements
Upper nozzle	16362	16080
Lower nozzle	28684	28300

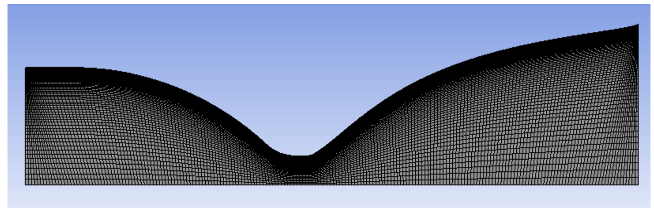
TABLE 5. MESH VALIDATION FOR NOZZLE

Geometry	Cells	Faces	Nodes
Upper Stage	779692	3107542	1634641
Full rocket	1000115	4663377	2846566

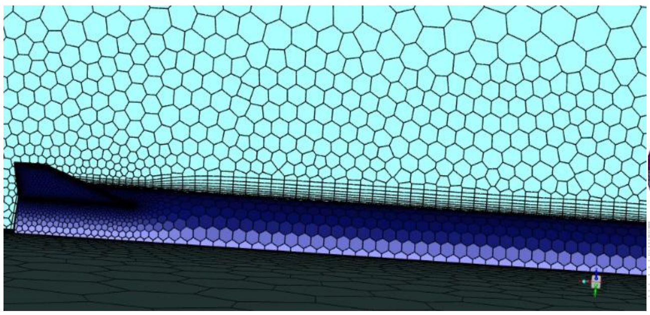
TABLE 6. MESH VALIDATION FOR ROCKET



MESH GENERATION – LOWER NOZZLE [FIGURE- 6]

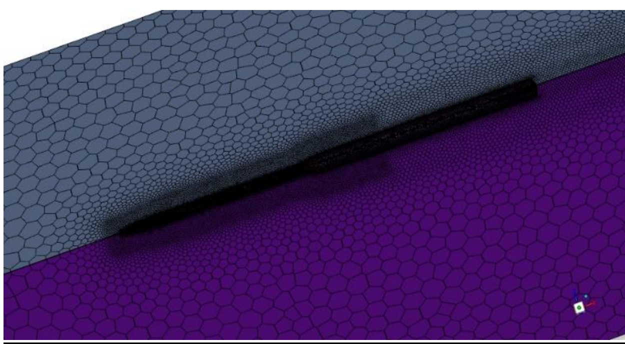


MESH GENERATION – UPPER NOZZLE [FIGURE- 7]

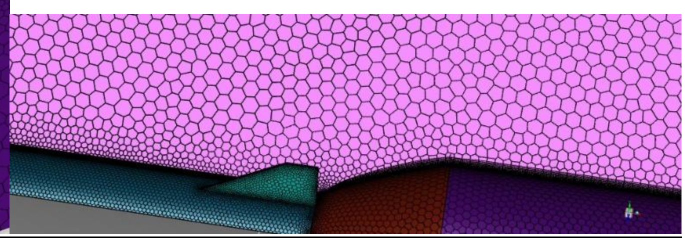


MESH GENERATION – UPPER ROCKET BODY [FIGURE- 8]

MESH GENERATION – UPPER ROCKET BODY [FIGURE-8.1]



MESH GENERATION – FULL ROCKET BODY [FIGURE-8.2]



MESH GENERATION – FULL ROCKET BODY [FIGURE-8.3]

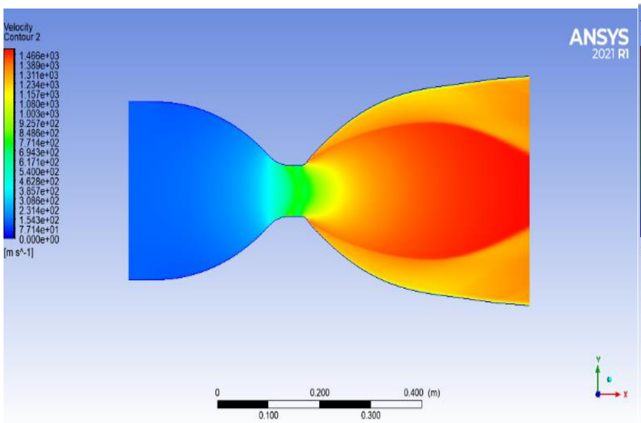
PRE-CONDITIONS

For the nozzles the density-based solver is used in Ansys fluent. SST k-omega turbulence model is used and air is kept as ideal gas to capture the change in density. A pressure inlet and outlet are used. After initialization enough iterations are run till the solution converges. For the lower nozzle, at the inlet gauge pressure is 5Mpa and temperature is 1500kelvin and at the outlet, gauge pressure is 0. The sea level operating conditions are used. For the upper nozzle, the gauge pressure at the inlet is 2Mpa and temperature is 1500 k and at the outlet the gauge pressure is 0. The atmospheric conditions at 50km height are used. For the whole rocket and upper stage, the pressure far field condition is used on the domain. The pressure-based solver is used. The turbulence model is SST k-omega as it captures the wall functions accurately and is mostly preferred for cases like these. The air is kept as ideal gas and for accuracy the Sutherland model of viscosity is used. Second order upwind methods are used with high order term relaxation. The full rocket and just the upper stage are simulated at different Mach Numbers and at different heights having respective atmospheric conditions to make the approach a bit realistic. The full rocket is simulated at Mach -> 0.6, 1 and 2 at the height of 5km, 15km and 25km respectively and the upper stage is simulated at Mach -> 3 and 4, at the height of 40km and 50km respectively.

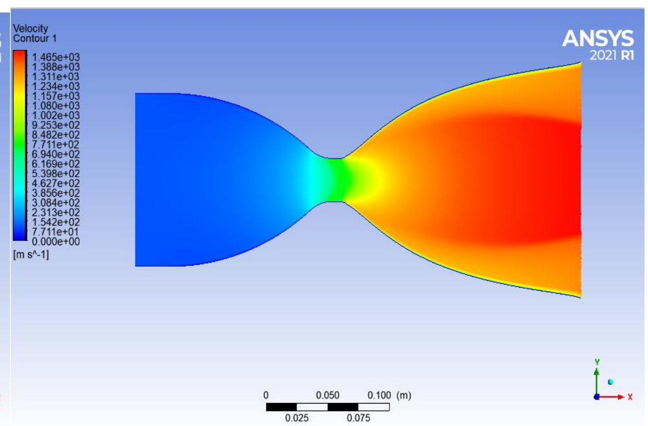
RESULTS

After analysis the following contour plots and graphs of respective nozzles are obtained:

Velocity contour: The velocity is minimum at inlet and keep on increasing as we move forward toward the exit. At the throat of the nozzle Mach is at 1 which is known at choked flow condition.



VELOCITY MAGNITUDE CONTOUR (m/s)
[FIGURE – 9]



VELOCITY MAGNITUDE CONTOUR (m/s)
[FIGURE – 9.1]

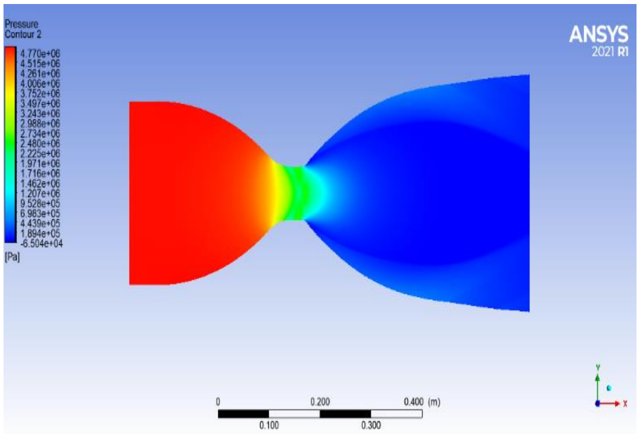
Velocity at exit of booster stage nozzle is 1383.16m/s at Mach number 3.05

The exit velocity of sustainer stage nozzle is 1427.7 m/s at Mach number 3.34

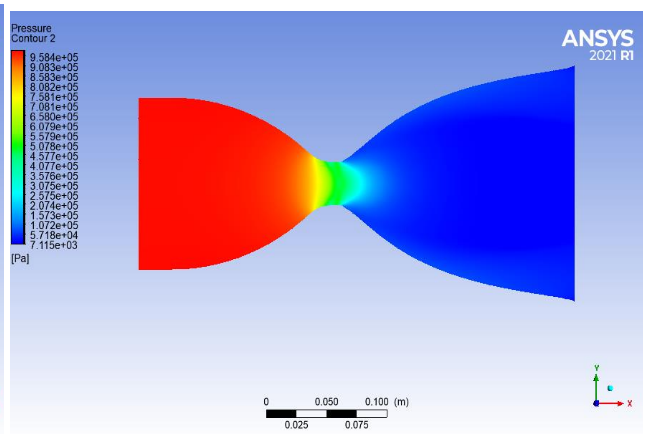
At the walls of the nozzles after the throat a visible boundary layer is also observed.

Pressure contour: The pressure value is maximum at inlet and keeps on decreasing as we move forward to nozzle exit. The pressure will suddenly decrease after throat because of the formation of shock waves.

Presented at International Conference on Innovations in Multidisciplinary Research (ICIMR-2021)
Held on 23rd & 24th August, 2021



STATIC PRESSURE CONTOUR (PASCAL)
[FIGURE – 10]

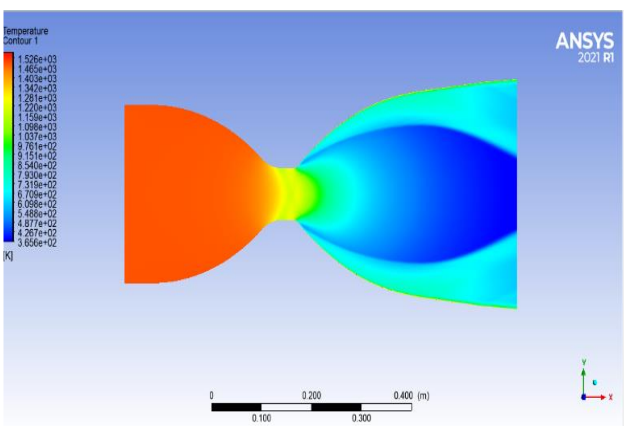


STATIC PRESSURE CONTOUR (PASCAL)
[FIGURE – 10.1]

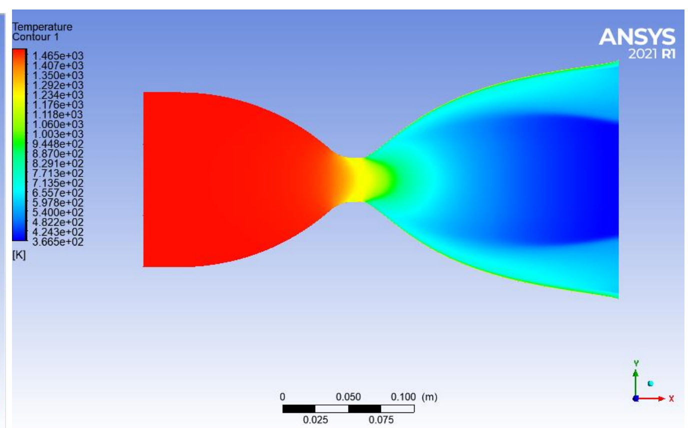
Static pressure at exit of the booster stage nozzle is 49151 Pa

Static pressure at exit of the sustainer stage nozzle is 20730.6 Pa

Temperature contour: The temperature is maximum at the inlet of the nozzle and keep on decreasing towards the exit of nozzle.



STATIC TEMPERATURE CONTOUR (K)
[FIGURE – 11]



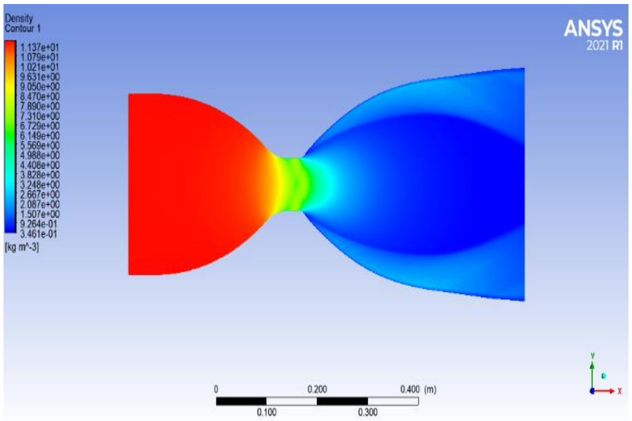
STATIC TEMPERATURE CONTOUR (K)
[FIGURE – 11.1]

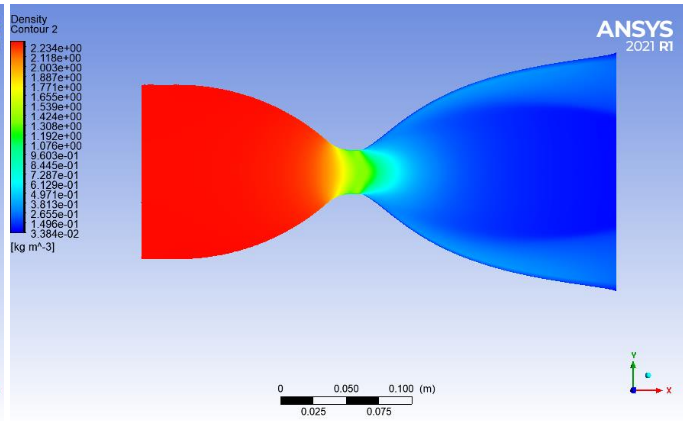
Temperature of exit of booster stage nozzle is 547.39K

Temperature at the exit of sustainer stage nozzle is 483 K

Density contour: The density is maximum at the inlet of the nozzle and keeps on decreasing as we move forward towards the exit of the nozzle.

Presented at International Conference on Innovations in Multidisciplinary Research (ICIMR-2021)
Held on 23rd & 24th August, 2021





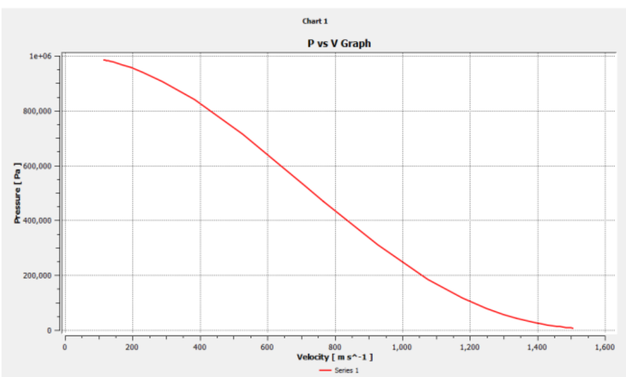
DENSITY CONTOUR (kg/m3)
[FIGURE – 12]

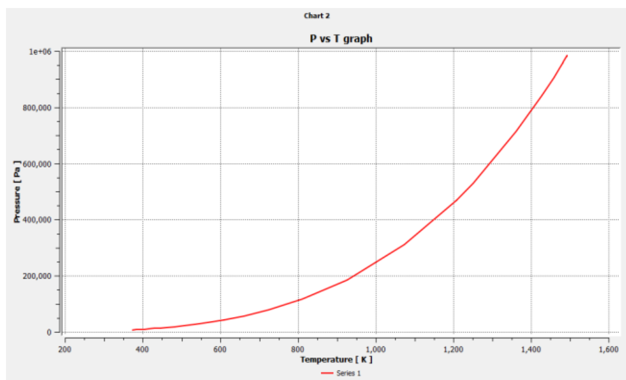
DENSITY CONTOUR (kg/m3)
[FIGURE – 12.1]

Density magnitude at the exit of booster stage nozzle is 0.87 kg/m^3.

Density magnitude at the exit of sustainer stage nozzle is 0.316 kg/m^3.

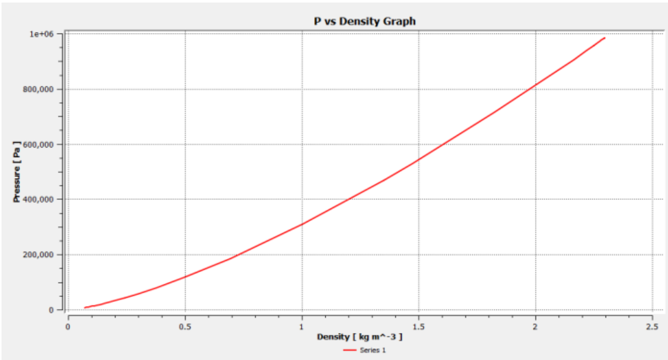
Graphical representation of sustainer stage nozzle performance:





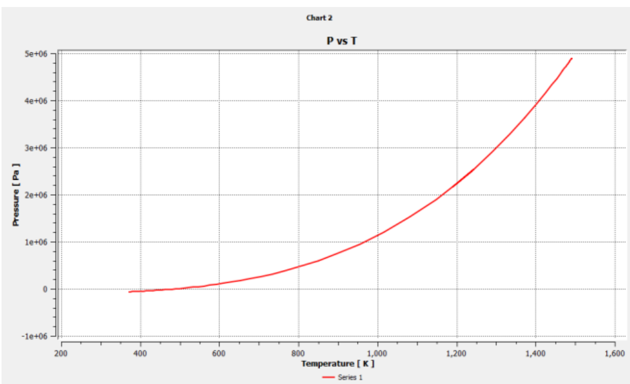
PRESSURE vs VELOCITY
[GRAPH – 1]

PRESSURE vs TEMPERATURE
[GRAPH – 2]

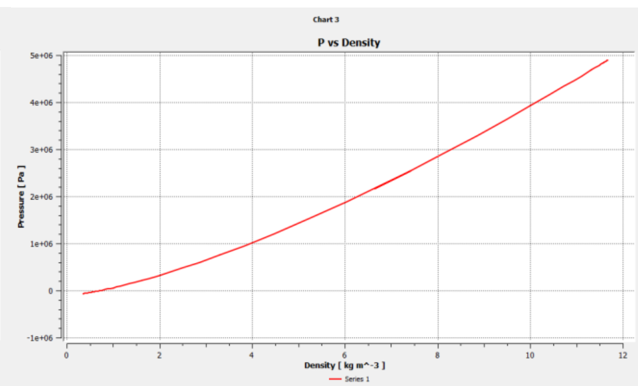


PRESSURE VS DENSITY
[GRAPH – 3]

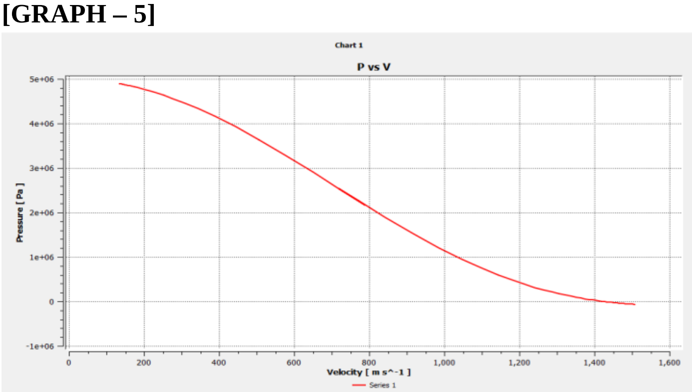
Graphical representation of booster stage nozzle performance



PRESSURE vs TEMPERATURE
[GRAPH – 5]



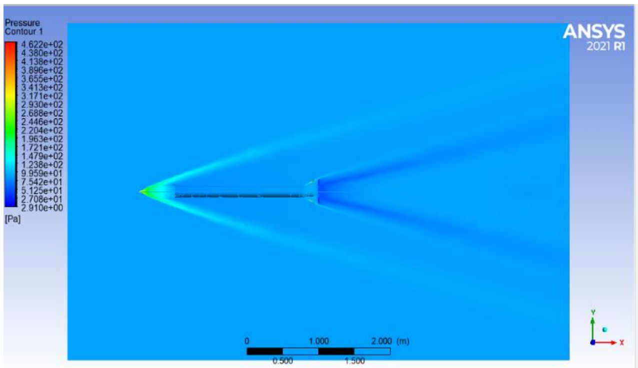
PRESSURE VS DENSITY
[GRAPH – 6]



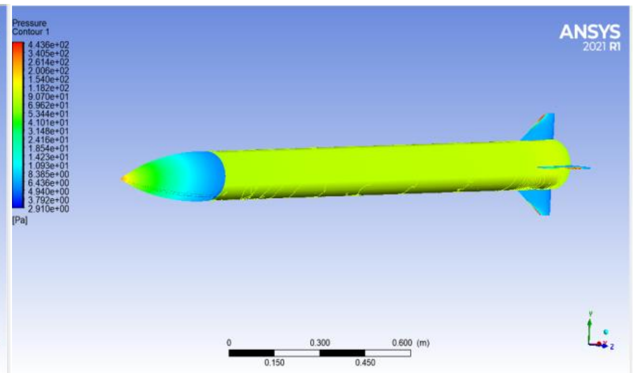
PRESSURE vs VELOCITY
[GRAPH – 4]

CFD ANALYSIS OF ROCKET

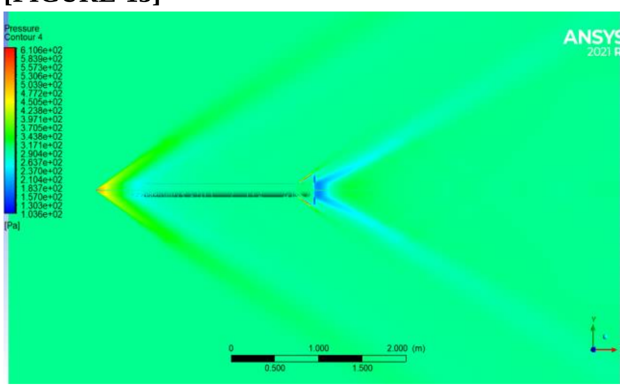
PRESSURE



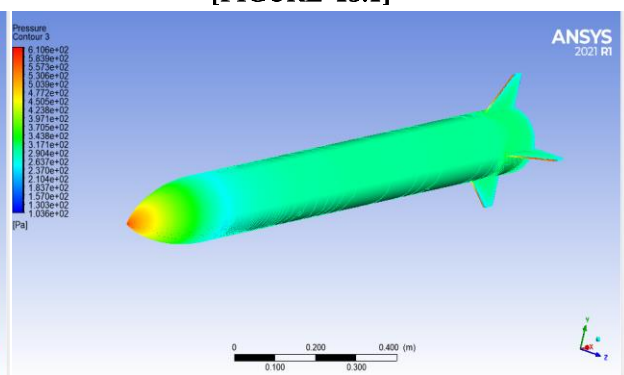
Upper body at Mach 4
[FIGURE-13]



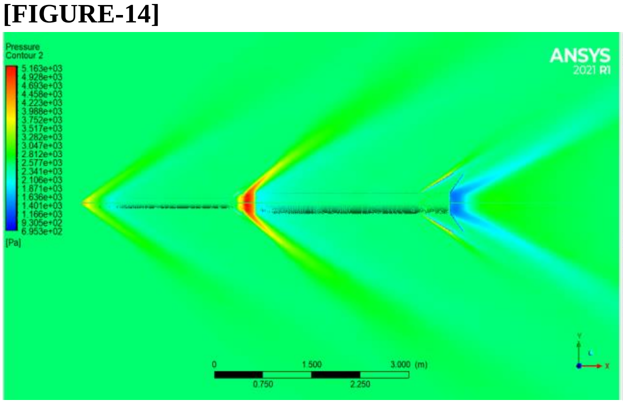
Upper body at Mach 4
[FIGURE-13.1]



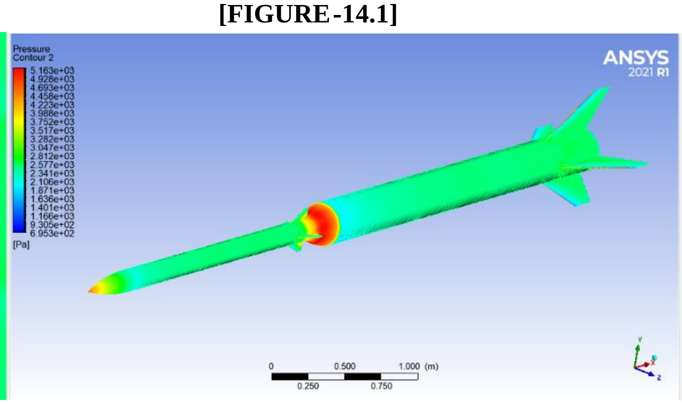
Upper body at Mach 3
[FIGURE-14]



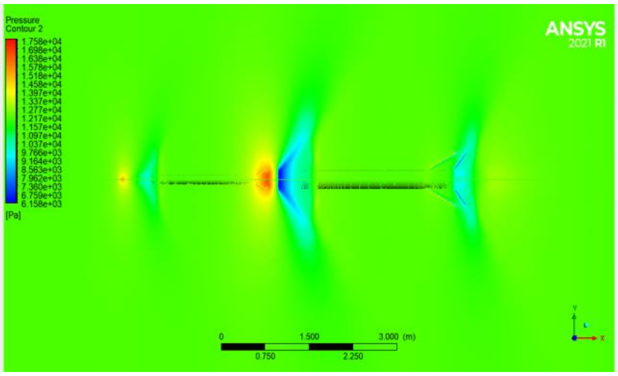
Upper body at Mach 3
[FIGURE-14.1]



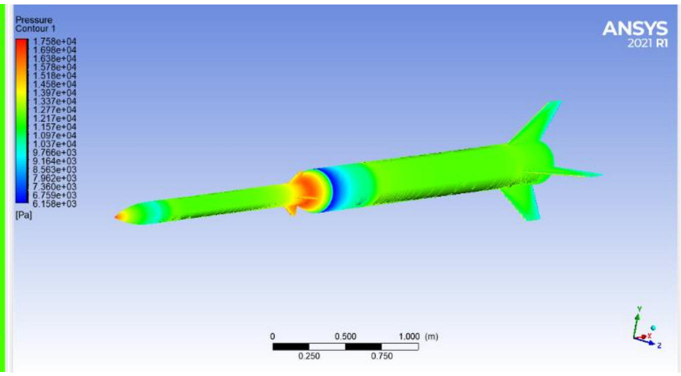
Full body at Mach 2
[FIGURE – 15]



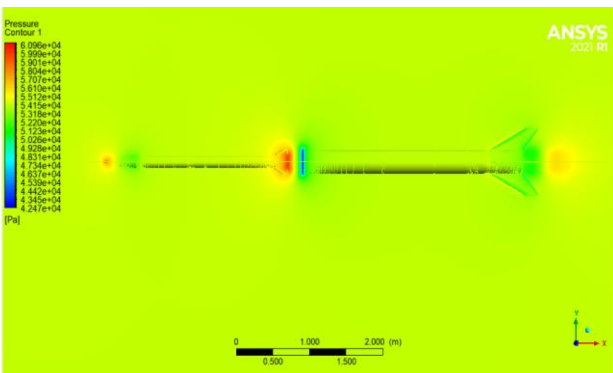
Full body at Mach 2
[FIGURE – 15.1]



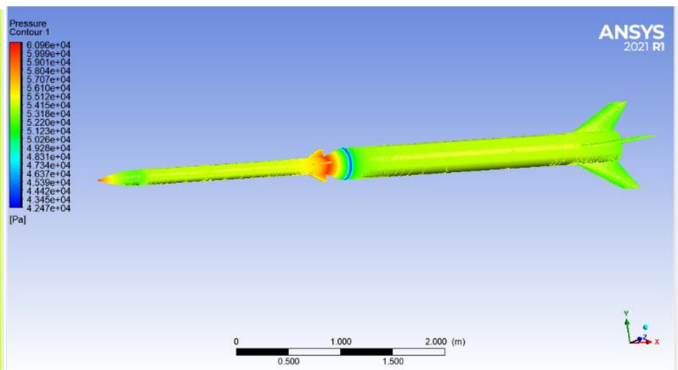
Full body at Mach 1
[FIGURE – 16]



Full body at Mach 1
[FIGURE – 16.1]



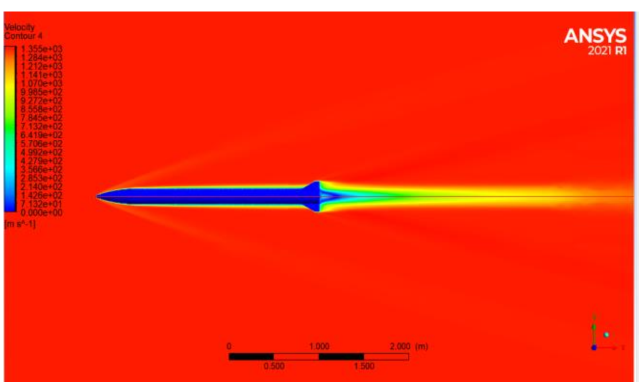
Full body at Mach 0.6
[FIGURE – 17]



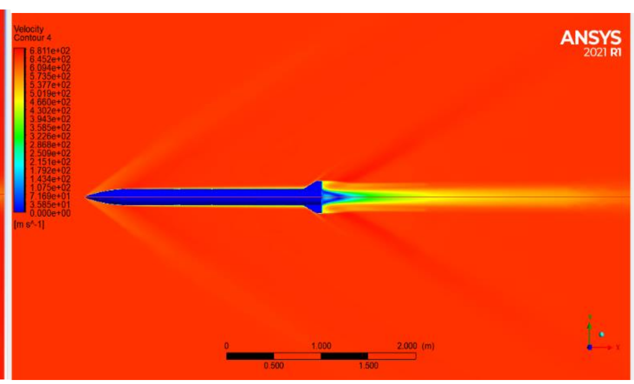
Full body at Mach 0.6
[FIGURE – 17.1]

Various contours of pressure have been shown above. In figure [13] & [14] we observe a bow shock at the front of the rocket and a delayed bow shock at the end. In figure [15] we observe similar bow shock at the front of rocket and the transition a big shock formation can be seen which is clearly due to increase in surface area of the booster stage. In figure [16] & [17] we observe pressure considerations at the tip, transition and back fin.

VELOCITY-

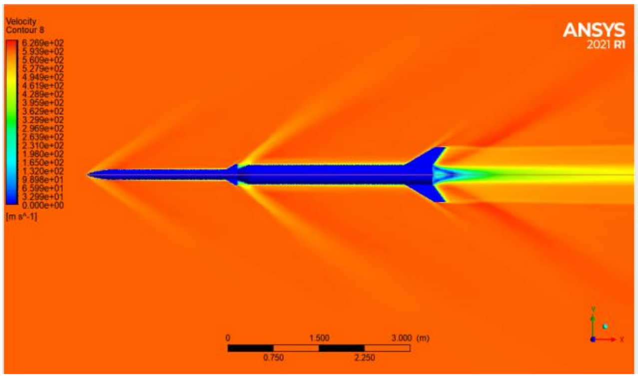


Upper body at Mach 4



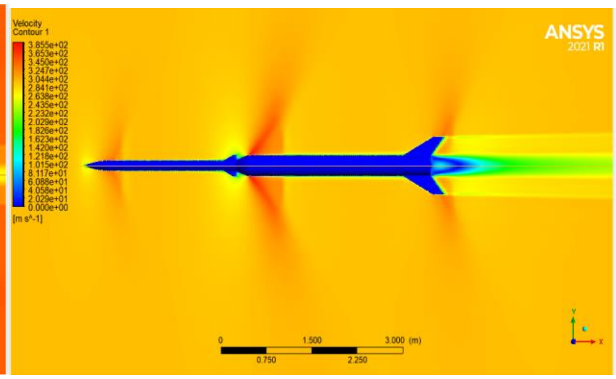
Upper body at Mach 3

[FIGURE –18]



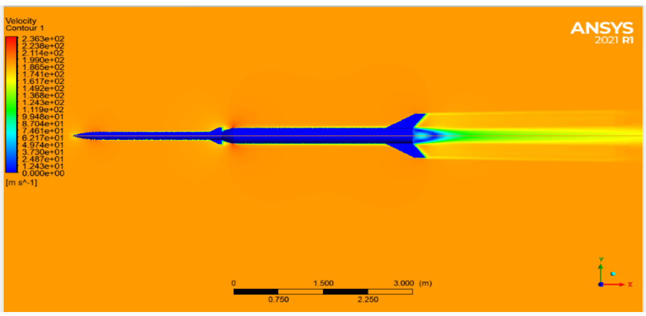
Full body at Mach 2

[FIGURE –19]



Full body at Mach 1

[FIGURE -20]

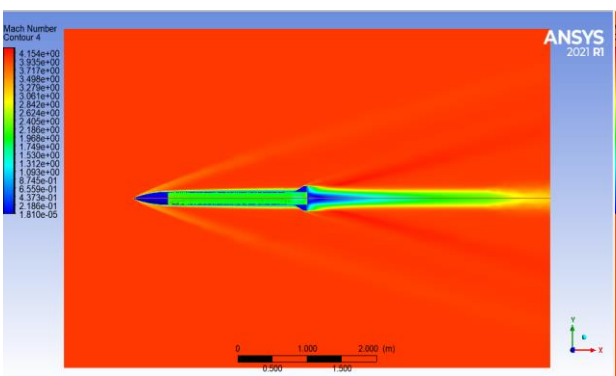


Full body at Mach 0.6

[FIGURE –21]

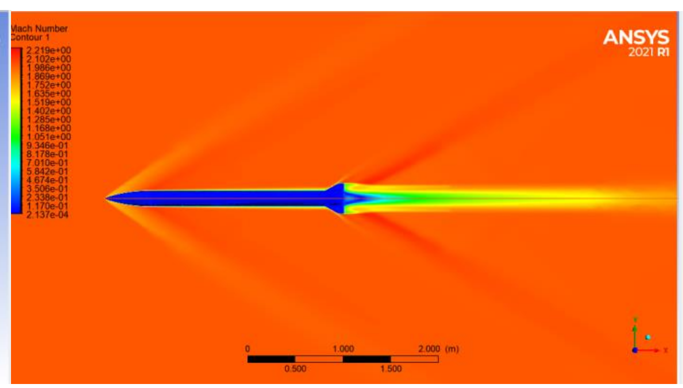
[FIGURE –22]

MACH-



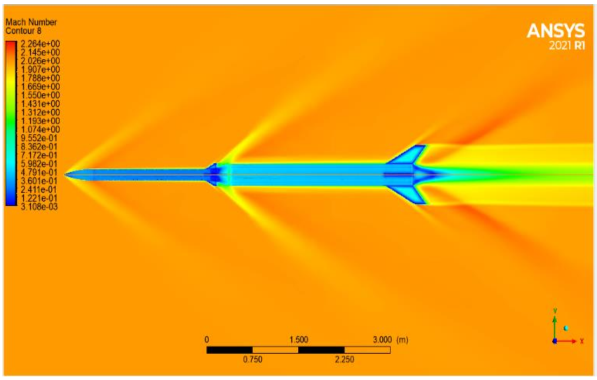
Upper body at Mach 4

[FIGURE –23]

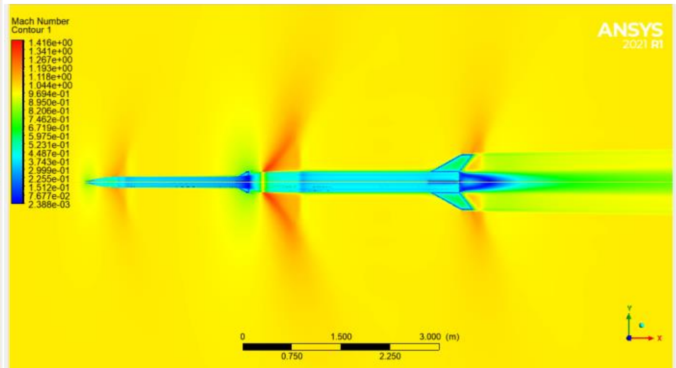


Upper body at Mach 3

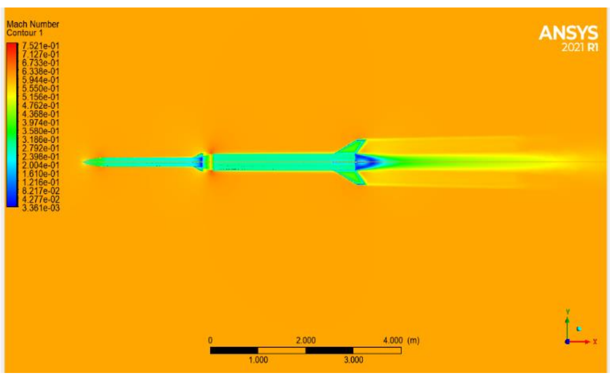
[FIGURE –24]



Full body at Mach 2
[FIGURE –25]



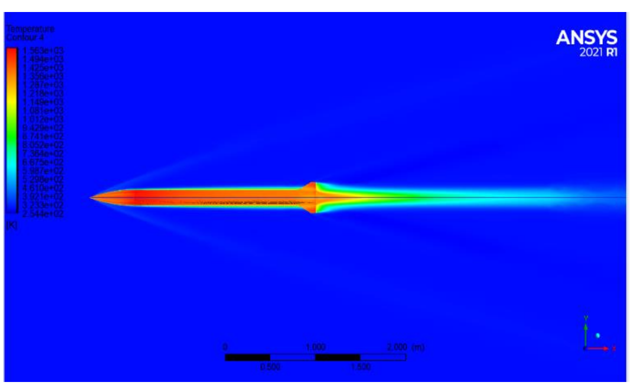
Full body at Mach 1
[FIGURE –26]



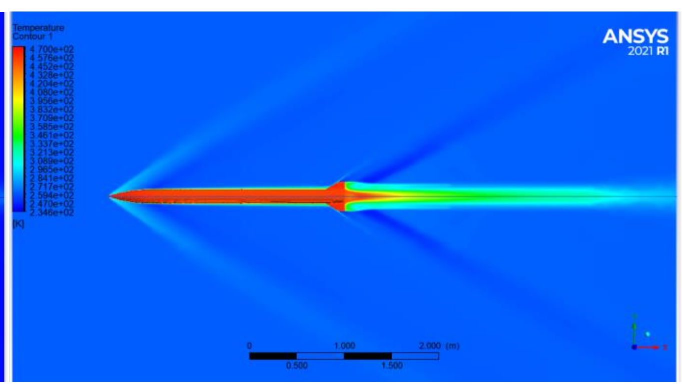
Full body at Mach 0.6
[FIGURE –27]

In the Mach number and velocity contours we observe at the boundary wall the speed of air is almost zero, which showcases the correct representation of boundary layer and in the full rocket body contour we observe a drop in velocity between the forward fin and transition which is due to formation of vortices and hence, forming high pressure region.

TEMPERATURE-

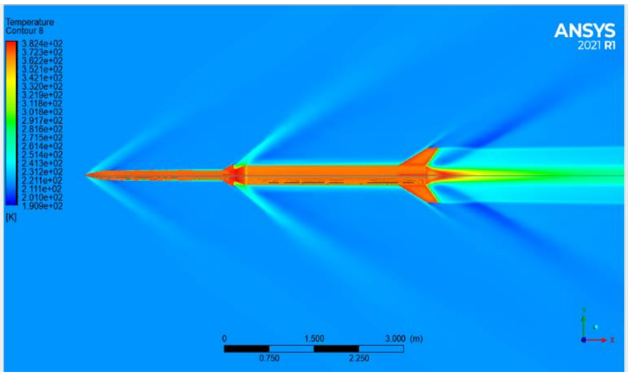


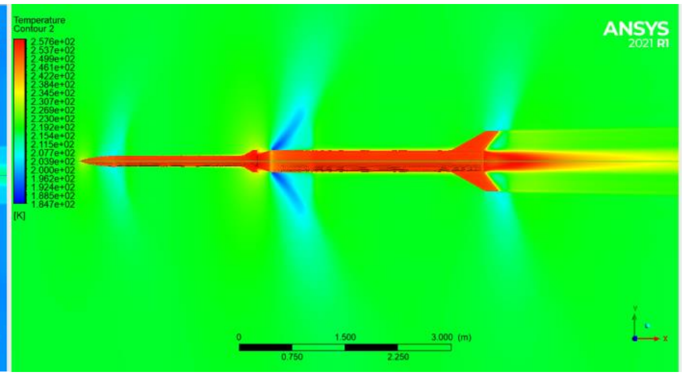
Upper body at Mach 4
[FIGURE-28]



Upper body at Mach 3
[FIGURE-29]

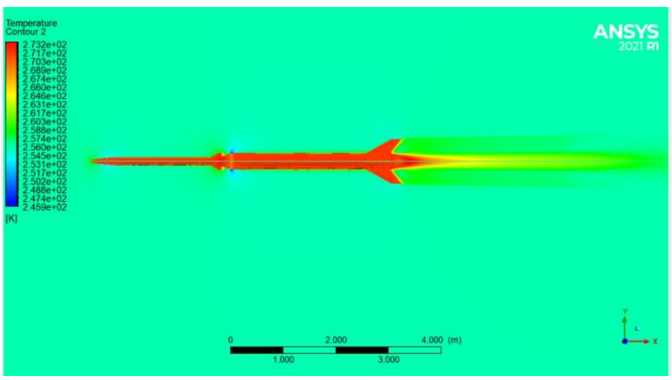
Presented at International Conference on Innovations in Multidisciplinary Research (ICIMR-2021)
Held on 23rd & 24th August, 2021





Full body at Mach 2
[FIGURE-30]

Full body at Mach 1
[FIGURE-31]



Full body at Mach 0.6
[FIGURE-32]

In temperature contours we observe temperature over the body is the highest and at the walls of the rocket we observe boundary layers where small change in temperature can be observed.

DRAG-

D $= \frac{1}{2}\rho v^2 C_d A,$

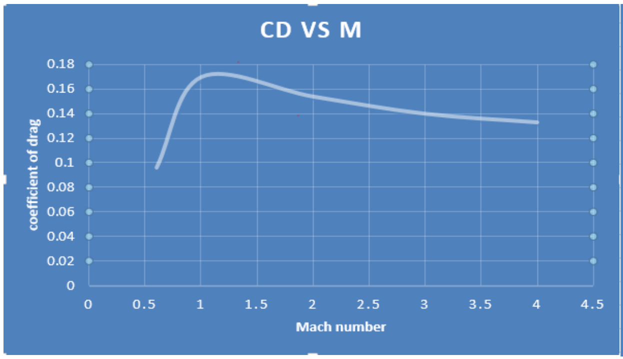
Where, D = drag force (N)

ρ= density (kg/m^3)

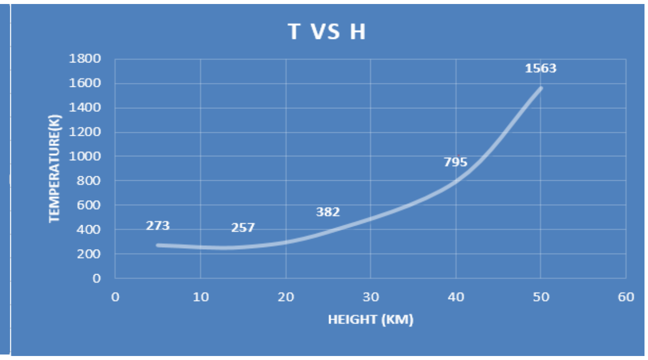
v= velocity of object (m/s)

C_d= coefficient of drag

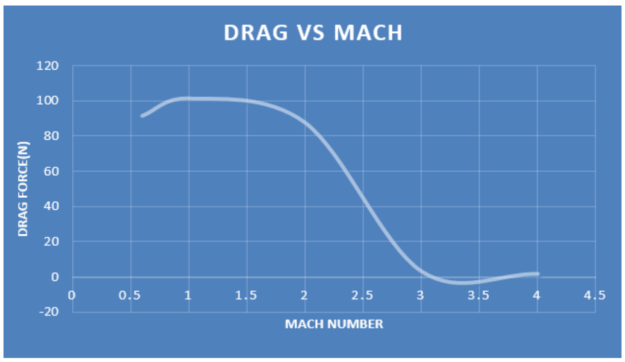
A = cross-sectional area (m^2)



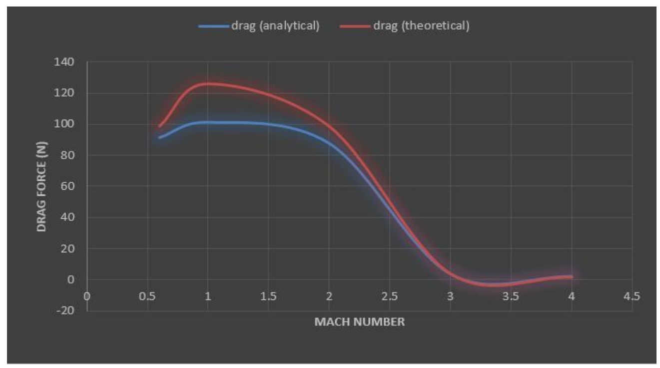
GRAPH-1



GRAPH- 2



[GRAPH – 3]



[GRAPH – 4]

CONCLUSION

The design is made on open rocket software taking consideration the fineness ratio of rocket body, after observing different nose cone designs, ogive shape is selected. The nozzles are designed using RPA software with inlet conditions of 80 & 320 psi chamber pressure on upper and lower stage nozzle respectively. Both the rocket body and nozzle design are three dimensionally made using fusion 360 software. The nozzles designed in accordance to chamber conditions and ambient conditions have been simulated and they show desirable results. In the full rocket simulations we observe small pressure considerations at the tip, fin and transition at subsonic speed which is within the expected results, whereas, at sonic and supersonic speeds a much larger concentration of pressure is observed that the transition which needs to be further analyzed and worked on. In the upper stage only small bow shocks are observed which do not hinder the flow of rocket. The temperature over the body of the rocket is observed to be maximum at Mach 4 at a height of 50 km. So, we can conclude that we need to use some kind of thermal coating in the upper stage of the rocket to keep the payload and flight computer safe from high temperature. The coefficient of drag value is highest at Mach number 1 which is acceptable as the greatest shocks are formed at that speed, the theoretical and analytical are almost similar which further validate the CFD analysis.

REFERENCES

[1] Lucas de Almeida Sabino Carvalho, Geovanio Claudino Cavalcante Filho, CFD analysis of drag force for different nose cone design, Conference proceedings, 2019.

[2] GQ Zhang, LC Ji, Y Xu, J Schlu¨ ter, Parametric study of different fins for a rocket at supersonic flow, Proc IMechE Part C: J Mechanical Engineering Science 0(0) 1–13, 2015.

[3] M. Abhinav, V. Narasimha Reddy, Computational Investigation of the Effects of Leading-Edge Bluntness on Drag at Supersonic Speeds, International Journal of Engineering Research & Technology (IJERT), Vol. 5 Issue 06, June-2016, 2278-0181.

[4] Girish Kumar, Dr. Pravin V Honguntikar, CFD Analysis of Transonic Flow over the Nose Cone of Aerial Vehicle, IJSRD - International Journal for Scientific Research & Development, Vol. 3, Issue 07, 2015, 2321-0613

[5]. A Sanjay Varma, G Sai Satyanarayana, Sandeep "CFD ANALYSIS OF VARIOUS NOSE PROFILES", International Journal of Aerospace and Mechanical Engineering, ISSN (O): 2393-8609, Volume 3 – No.3, June 2016.

[6] Yong-Chao Chen, Xin-Bao Gao, Min Gao, Hui-Miao Lv, Aerodynamic characteristic of a canard guided rocket, International Journal of Modeling, Simulation, and Scientific Computing Vol. 8, No. 1 (2017) 1750001

[7] Bogdan-Alexandru Belega, Trung Duc Nguyen, analysis of flow in convergent-divergent rocket engine nozzle using computational fluid dynamics, international conference of scientific paper afases 2015.

[8] Sreenath K R, Mubarak A K, Design and analysis of contour bell nozzle and comparison with dual bell nozzle, IJRE, Vol. 03 No. 06, June 2016, 2348-7852.

[9] Tripathi, M., Misra, A., & Sucheendran, M. M. (2018). Effect of Planar Member Cross-Section on Cascade Fin Aerodynamics. Journal of Spacecraft and Rockets, 1–17. Doi: 10.2514/1. a34238

[10]. Md Nizam Dahalan, Ahmad Fitri Suni, Iskandar Shah Ishak, Nik Ahmad Ridhwan Nik Mohd and Shabudin Mat "Aerodynamic Study of Air Flow over A Curved Fin Rocket", Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, volume 40, pages 46-58 December 2017.

[11]. Vasiliy Zubanov, Vitaliy Egorychev, Leonid Shabliy "Design of Rocket Engine for Spacecraft Using CFD-Modelling" 2015.

[12]. Filippo Trevisi, Matteo Poli, Mattia Pezzato, Eugenio Di Iorio, Alberto Madonna, Nicola Bressanin and Stefano Debei "1 Simulation of a sounding rocket flight's dynamic" 2017.

[13]. Blazej Marciniak, Adam Okninski, Bartosz Bartkowiak, Michal Pakosz, Kamil Sobczak, Wojciech Florczuk, Damian Kaniewski, Jan Matyszewski, Pawel Nowakowski, Dawid Cieslinski, Grzegorz Rarata, Pawel Surmacz, Dominik Kublik, Damian Rysak, Jaromir Smetek and Piotr Wolanski "Development of the ILR-33 "Amber" sounding rocket for microgravity experimentation" November 2017.

[14]. Ankith Y.S, Jobin A George and prof. Vipin Raveendran "Design of fins to maximize the heat transfer rate from the engine cylinder" INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH IN TECHNOLOGY, volume 6, June 2019.

[15]. C.P. Hoult and Hien Tran "Sounding Rocket Fin Design to Mitigate Roll Lock-In" IEEE Aerospace Conference, march 2015.

[16] Nikhil D. Deshpande, Suyash S. Vidwans, Pratik R. Mahale, Rutuja S. Joshi, K. R. Jagtap, "theoretical & CFD analysis of de Laval nozzle", International Journal of Mechanical And Production Engineering, April -2014.

[17] Military Technical Academy, Bucharest, Romania, Paul Sabatier University, Toulouse, France, “analysis of flow in convergent-divergent rocket engine nozzle using computational fluid dynamics”, International Conference of Scientific paper, May 2015.

[18] Mohan Kumar G., Dominic Xavier Fernando, R. Muthu Kumar, - Design and Optimization of De Lavel Nozzle to Prevent Shock Induced Flow Separation - Advances in Aerospace Science and Applications. ISSN 2277-3223 Volume 3, Number 2 (2013), (pg. 119-124).

[19] John D. Anderson, “Introduction to Flight”, Mc Graw Hill Education, sixth edition.

[20] John D. Anderson, “Fundamental of Aerodynamics”, Mc Graw Hill Education, sixth edition.

[21] B.N. Suresh, K. Sivan, “Integrated design for Space Transportation System”, springer (pg. 231-282, 329-386).

[22] Cornelisse Schoyer Wakker, “Rocket Propulsion and Space Dynamics”, pitman publishing limited, (pg. 85-98, 262-281).

[23] Capt. Bertrand R. Brinley, “Rocket Manual for Amateurs”, Ballantine books, (pg. 54-100).

[24] K.Ramamurthi, “Rocket Propulsion”, Macmillan publishers India, 2012 edition, (pg. 54-89).

[25] George P. Sutton and Oscar Biblarz, “Rocket Propulsion Elements”, A. Wiley- Interscience Publication, Ninth Edition, 2001, (pg. 1-99).