

Parametric Output of Penetration Length in De-Laval Nozzle using Computational Fluid Dynamics

Rishab Kumar Agrawal, Sivaram Munagala



Abstract: A Contour shaped rocket nozzle, commonly known as Bell nozzle, is a more efficient form of a De-Laval nozzle useful for operation at different altitudes resulting in variable pressure ratio across the nozzle. Here the penetration length during the flow is calculated in a De-Laval nozzle using Computational fluid dynamics (CFD). The TOP nozzle¹ by G.V.R. Rao is modeled and simulated using CFD analysis by varying the divergent half angle of the bell nozzle. The geometry of different divergent angle of the bell nozzle is modeled using MATLAB code. The modeled geometry and mesh of the nozzle is simulated in ANSYS-FLUENT software. The simulation shows the variation in different flow parameter at different divergent half angles. The analysis so far for the same is done in a conical shaped rocket nozzle at the cost of varying the divergent length for changing the divergent half angle of the nozzle. This project simulates bell nozzle with constant divergent length. The simulation model is verified and validated using experimental and computational data at an NPR (Nozzle pressure ratio) of 1000. As the Rao Nozzle is used currently in rocket, missile, and satellite control systems worldwide. This research work to show how the penetration length effect the Mass-Weighted average Mach number in a particular designed bell nozzle with different divergent half angle. Based on the results obtained, the discussion is done about the parameters and the conclusion is given.

Keywords: Bell Nozzle, CFD, Divergent angle, De-Laval, NPR.

I. INTRODUCTION

The increasing demand and requirement of a highly efficient bell-shaped nozzle used in the space missions in the past decades has led to the crucial evaluation in rocket propulsion systems. This results in the advancement and alteration in the nozzle geometry. Today's rocket frequently uses a bell-shaped nozzle because of its higher efficiency. The prime focus of this project is to study De-Laval nozzle by varying its divergent angle using computational analysis. A method is established in order to model a dual-bell nozzle by developing a MATLAB code and carry out its computational analysis using ANSYS FLUENT Software, and study its flow parameters. The comparative study of the nozzle flow parameters is carried out by varying the divergent angle of the nozzle.

Nozzle being one of the major components of the rocket engine demands critical assessment for minimizing the losses and increasing the efficiency of the engine. The previous analysis done in conical nozzle with varying the angles resulted in changing the length of the nozzle, which diminishes the rocket engine performance.

The variation of angle in conical nozzle increases the weight, which is not preferable and demands for the change of contour from conical to bell nozzle.

The variation of half angle of the divergent section in the conical nozzle results in variation of the exit parameter that affects the performance whereas for a bell or contour nozzle the exit parameter variations are negligible up to a certain angle. This triggers us to work on varying the divergent half angle of the contour bell nozzle and study its variations. The literature review, to the best of our knowledge show that no analysis has been done yet by varying the divergent half angle in bell nozzle.

II. THEORY & METHODOLOGY

The present investigation is to study the behavior and flow parameters of a fluid in a bell nozzle when the initial wall angle of the nozzle is varied. The objective of the investigation is to model a bell-shaped nozzle (2D) by MATLAB code, carry out its computational analysis using ANSYS-FLUENT Software and to study its flow parameters with the comparative study by varying the divergent angle of the nozzle. Thus, the proper methodology is followed to develop a model for the bell nozzle with different divergent angles and fluent analyses of those nozzles are done in order to do a comparative study of the different flow parameters. Bell nozzle offers a wide range of advantage as compare to conical nozzle. Henceforth, it is widely used in the recent space applications. It has less weight compared to conical nozzle. It has a high angle expansion section otherwise known as initial wall angle. The focus of this project is given to the nozzle part of the rocket. Going further to the specific part, which is taken, into consideration is the bell shape of the nozzle. The parameters studied is the Mach number. The modelling of the bell nozzle is done by undertaking the Rao method^[1] of TOP or Thrust Optimized Parabolic approach that gives a perfect contour for a bell nozzle with help of three different curves. Generally, in order to obtain a bell contour using Rao method we should first consider the geometry of the throat. This design consists of three curves: a convergent curve, a divergent curve and a parabolic curve. The slope of the parabolic curve is tangent to the inflection angle, θ_n also known as initial wall angle of the parabola where the divergent curve circle and the parabolic curve intersect.

Manuscript received on February 10, 2020.

Revised Manuscript received on February 20, 2020.

Manuscript published on March 30, 2020.

* Correspondence Author

*Rishab Kumar Agrawal, Aerospace Engineer, Hindustan Institute of Technology & Science. Email: Rishabagrawal.100@gmail.com.

Sivaram Munagala, Aeronautical Engineer Hindustan Institute of Technology & Science. Email: munagalasiva24@gmail.com

© The Authors. Published by Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP). This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>)

Parametric Output of Penetration Length in De-Laval Nozzle using Computational Fluid Dynamics

The Rao Nozzle [1] is used currently in rocket, missile, and satellite control systems worldwide.

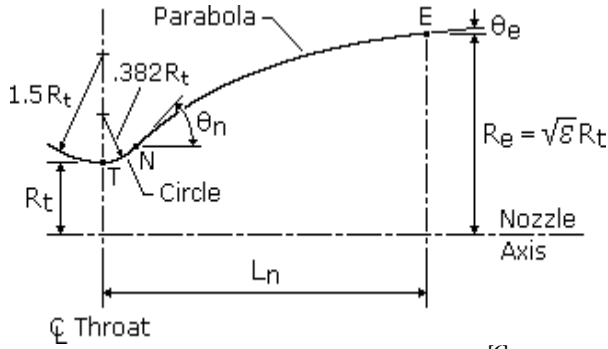


Fig 1- Rao nozzle Method [6]

The governing equations to model the nozzle coordinates for the aforementioned method is given as follows:

Length of Nozzle (Ln) [3] [6]:

$$L_n = \frac{k(\sqrt{\epsilon} - 1)R_t}{\tan \theta_e} \quad (1)$$

Convergent curve equations are [6]:

$$x_{first\ curve} = \cos(\theta_{first\ curve}) * 1.5 * R_t \quad (2)$$

$$y_{first\ curve} = \sin(\theta_{first\ curve}) * 1.5 * R_t + (1.5 * R_t + R_t) \quad (3)$$

Divergent curve equations are [6]:

$$x_{second\ curve} = \cos(\theta_{second\ curve}) * 0.382 * R_t \quad (4)$$

$$y_{second\ curve} = \sin(\theta_{second\ curve}) * 0.382 * R_t + (0.382 * R_t + R_t) \quad (5)$$

Parabolic curve equations are [6]:

$$\begin{bmatrix} a \\ b \\ c \end{bmatrix} = \begin{bmatrix} y_{second\ curve}^2 & y_{second\ curve} & 1 \\ Y_{exit}^2 & y_{exit} & 1 \\ 2y_{second\ curve}^2 & 1 & 0 \end{bmatrix}^{-1} \begin{bmatrix} x_{second\ curve} \\ x_{exit} \\ 1 \\ \frac{1}{\tan \theta_n} \end{bmatrix} \quad (6)$$

$$x_{third\ curve} = ay^2 + by + c \quad (7)$$

$$y_{third\ curve} = \sqrt{\epsilon}R_t \quad (8)$$

The software used for modelling of the nozzle for developing a code is MATLAB. This software is designed for numerical computing, matrix manipulation, implementing algorithms, plotting data etc. These formulas for Rao method of TOP contour, gives the coordinates for the convergent curve, divergent curve and for the parabolic curve through a systematic developed code. The scripting language of MATLAB is an efficient and user-friendly programming language which is case sensitive and reads the command in a sequential manner. The developed code demands for the variables as throat radius, exit area ratio, divergent half angle and exit angle for the variables to compute the different coordinates required to model the complete contour of the nozzle. A simple syntax to plot the functions and coordinates shows the complete contour for

the desired bell nozzle at different half divergent angles.

III. ANALYSIS & VALIDATION

The nozzles with different divergent half angle section are modelled. The length of nozzle with different divergent angles remain same in all the cases as the exit angle is defined in prior with a constant value of 16.5°. The exit area ratio undertaken for modelling the nozzle is 31.4754. The throat radius of the nozzle is 30.5 mm. The modelling of the nozzle is done within the divergent half angle limits of 20° to 50°. The angles taken for modelling of the nozzle are- 25°,30°,35°,40°,45°,50° with keeping there θ_e (exit angle) constant with 16.5°. This exit angle reveals that the length of the nozzle is approx. to 377 mm.

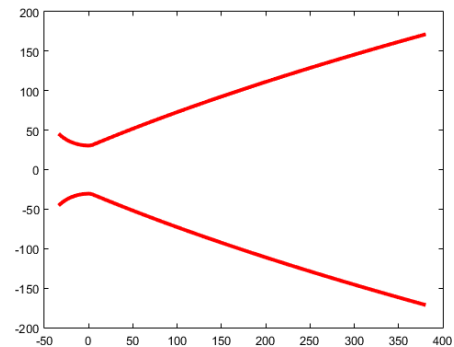


Fig 2- Modelled Nozzle at 25-degree divergent half angle.

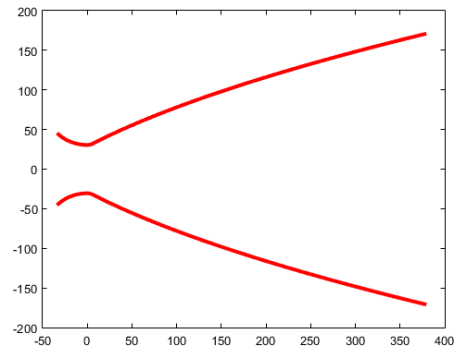


Fig 3- Modelled Nozzle at 30-degree divergent half angle

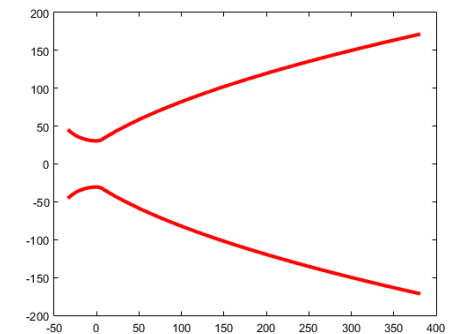


Fig 4- Modelled Nozzle at 35-degree divergent half angle

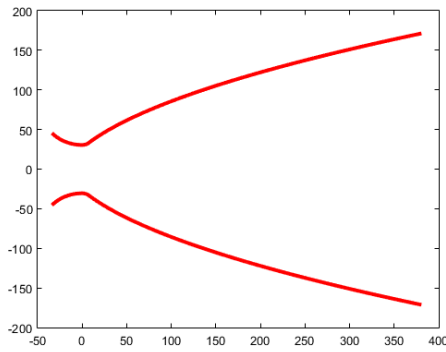


Fig 5- Modelled Nozzle 40-degree divergent half angle

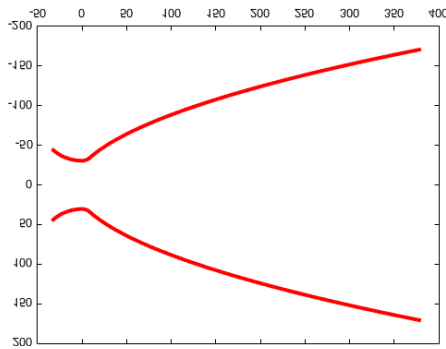


Fig 6- Modelled Nozzle at 45-degree divergent half angle

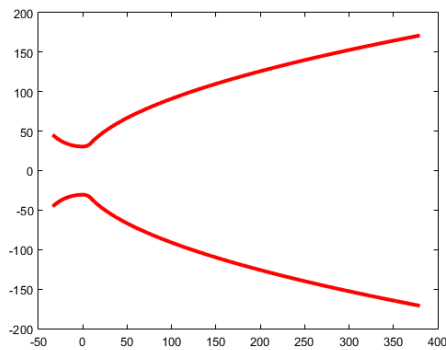


Fig 7- Modelled Nozzle at 50-degree divergent half angle

The above figures clearly show that how changing the divergent angles of the nozzle completely changes the geometry for the nozzle where the flow needs to flow around the contour. The 3D curve was formed by importing the notepad file in which the coordinates were saved. Thus, the closed figure of the nozzle is formed. The Surface from edges option is selected and all the four edges were selected, and surface was generated. The surface was mirrored to ZX plane and complete nozzle was modeled by Body transformation. Boolean Transformation was used to join the two surfaces as a single surface. The workbench was thus saved.

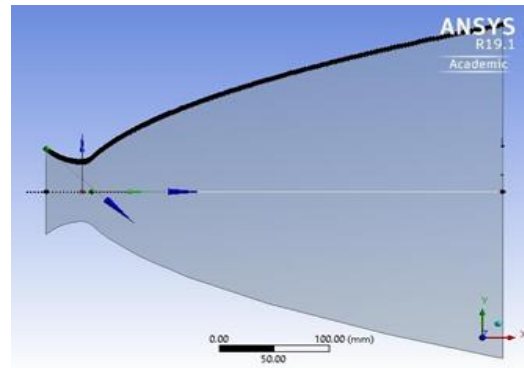


Fig 8- Surface geometry in ANSYS- Design Modeler

The geometry formed then undergoes for meshing into ANSYS-Meshing software so that the perfect number of nodes can be generated in order to get precise results in FLUENT analysis. The meshing of any surface is a critical part as the parametric values of the flow are recorded only at the nodes thus proper value of the element size is mandatory in order to get a repeated and precise values in the given constant flow condition. The meshing of the surface body of the nozzle begins by changing the physical preference to CFD thus changing the mesh compatibility with FLUENT. The automatic mesh de-featuring is turned off in order to get complete structured mesh near the proximity of the wall. The capture proximity function with cells gap is kept at 10 with curvature normal angle of 9°. Thus, saving the default mesh parameters to generate a structured mesh. In order to refine the mesh for better analysis of the nozzle flow Mapped Face Meshing method is chosen that generates a fine all structured mesh inside the nozzle. The generated mesh is made finer with Face Sizing of the nozzle in the structured all quads mesh type with an element size chosen through the grid independent test. The generated mesh then is optimal for conducting a smooth simulation with no effects on increasing nodes for the mesh. The complete structured mesh geometry is then given their edges name for a future reference in FLUENT. The mesh model is then exported out separately with FLUENT compatible '. msh' extension for future consideration.

A. Grid Independent Study

The grid independence test carried out in order to ensure that the output is independent of the element size and nodes generated in the meshing of the constructed geometry. The meshing details for the independent study are shown below in the table.

Table I- Data of Grid Independent Study

Element Size (mm)	No. of elements	Mach no.
5	2511	3.26
4	3808	3.25
3	7104	3.246
2	14405	3.24
1	54016	3.23
0.9	64000	3.24
0.8	88704	3.24

The data shows that change in the element size from 5mm to 0.9 mm creates a change in the flow parameters as it shows the error corresponding to the number of nodes present.

Parametric Output of Penetration Length in De-Laval Nozzle using Computational Fluid Dynamics

The flow parameter Mach number stops altering when the element size is 0.9mm or 0.8mm generating nodes greater than 64000. The graphical representation of the data is shown below which depicts that with increasing the number of nodes values the error in Mach number reduces and attains a constant non-deviated value.

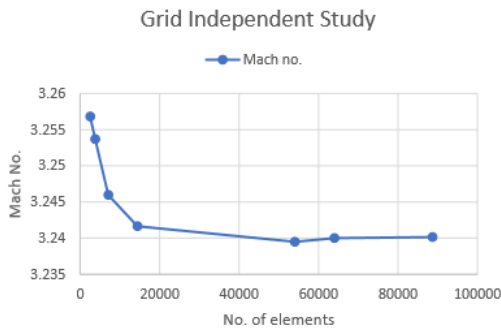


Fig 9- Graphical representation of Grid Independence Study

Thus, the grid independent test reveals that the element size of the sizing plays a major role in generating results without error. The element size chosen for the meshing of all the different geometry created is 0.9mm. The meshing nodes generated in all the geometry is greater than 60000. Thereby, a structured mesh is generated in all the geometry with a high-quality meshing of 99 percent nearly in all the different divergent angle geometry.

B. FLUENT Validation [3][2]

The validation of the results and procedure is done through the references [2] and [3] the results obtained by the author at 50° angle reveals the exit Mach number of 4.11 whereas our exit Mach number reaching 4.01. The same procedure is followed for all the different geometries for recording a complete comparative study of the different parameters obtained by the FLUENT analysis. Recently calculated Mach number profile along the centerline of the CD nozzle is in agreement with the experimental and computational datasets given for validation. Below is the plot between Mach number along the length of the nozzle. The experimental data shows the Mach number increases along the length of the nozzle and so the analyzed computational data. The divergence percentage error in the validation of the result is 2.4%.

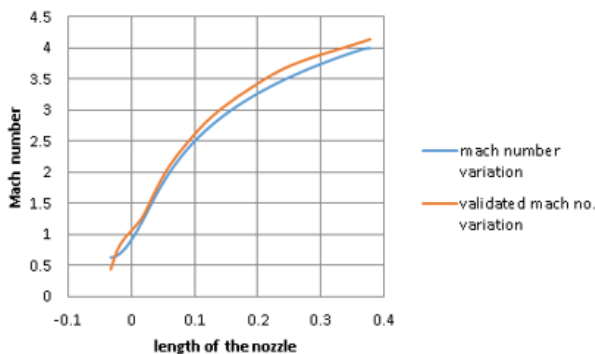


Fig 10- Parameter Validation.

IV. RESULTS AND DISCUSSIONS

The simulation of different geometry nozzle is completed in ANSYS-FLUENT and the post-processing work is done using the contours and plot functions of fluent. Comparative plots are drawn in excel by extracting the points or data in a notepad file. Contour plots are typical iso-lines that represent lines having a constant magnitude for a definite variable in the domain. They help in analyzing the transition of the flow parameters from one definite value to other as the flow proceeds in the nozzle. There are variety of contour parameters that are plotted in FLUENT depicting the flow in the nozzle. The levels of contour in the transition shows that how there is a gradual change in flow parameters in the flow domain Axis plots are different types of plots that create a graphical plot through the individual values of the parameters with different planes and axis variables. These plots can be extracted in the excel or MATLAB in order to create a comparative plot study of all the different simulation parameters present within the flow domain. The study of the flow parameters at a Nozzle Pressure Ratio (NPR) of 1000 reveals the formation of shock and separation near the wall surface thus changing the flow conditions and parameters in complete nozzle flow. The comparative graphs drawn for the penetration length and mass-weighted average Mach number of the different angle reveals the optimized divergent angle in the conventional bell nozzle. This also explains the Flow Separation Phenomenon that describes the free shock separation (FSS) and restricted shock separation (RSS) formed due to separation in the nozzle. Free Shock Separation is formed when the flow separates from the wall and moves towards the centerline of the nozzle. This separated shear layer flows as a free jet and is known as free shock separation [7]. In reference [7] the simulation is done for a conical nozzle where was the same methodology is used here for the simulation of the bell nozzle.

A. Mach Contours

The Mach contours for the different divergent angle's analysis are plotted:

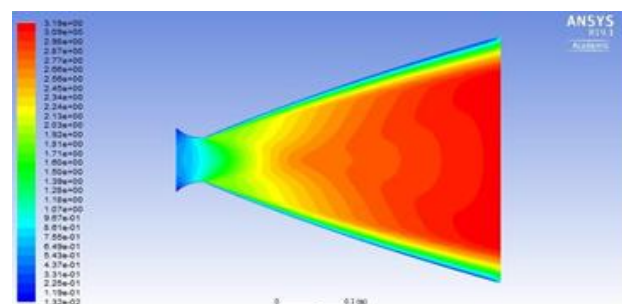


Fig 11- Mach contour for Divergent Half Angle 25°

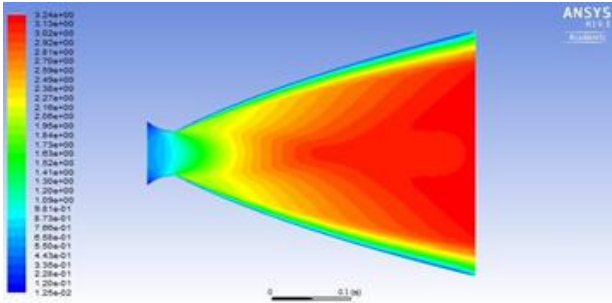


Fig 12- Mach contour for Divergent Half Angle 30°

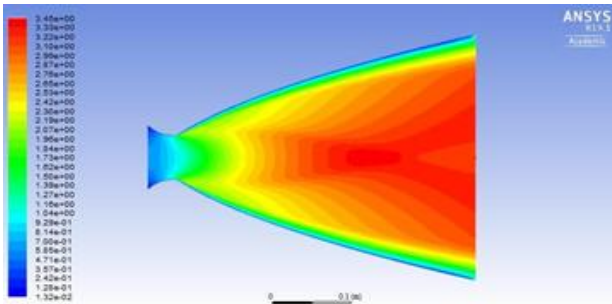


Fig 13- Mach contour for Divergent Half Angle 35°

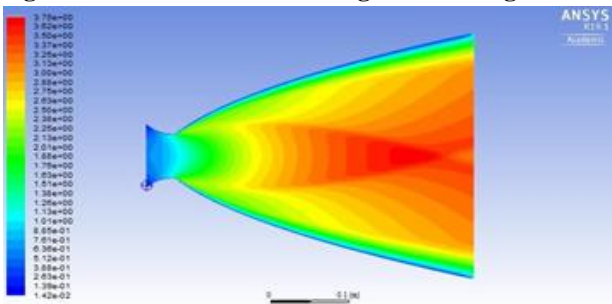


Fig 14- Mach contour for Divergent Half Angle 40°

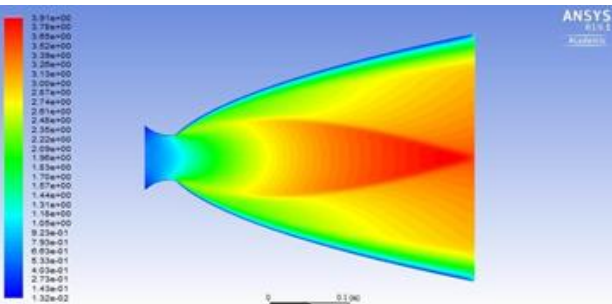


Fig 15- Mach contour for Divergent Half Angle 45°

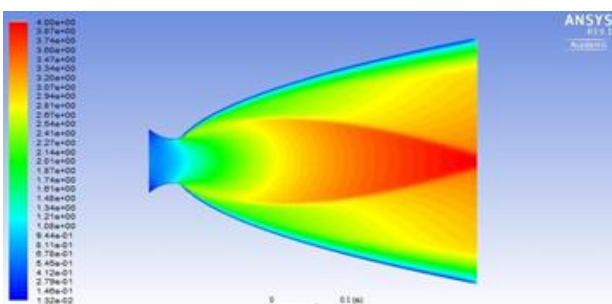


Fig 16- Mach contour for Divergent Half Angle 50°

The above figures give the contour plots of Mach number with varying the divergent angle. We can clearly see that

the contour plot and Mach changes with change in geometry the pressure gradient from the Navier Stokes Equation results in creating a flow regime in the nozzle, which changes the velocity in the Y-momentum direction of the nozzle and helps in studying a shear layer at the wall. As there is an increment in divergent angle of the nozzle, the flow expansion inside the nozzle completes earlier than required. The formation of wall-separated shock in the flow regime shows the variation of the Mach inside the flow of the nozzle generating different regimes of Mach transition taking place in the flow regime. The various mass-weighted average value of the Mach number at the exit of the nozzle is shown below:

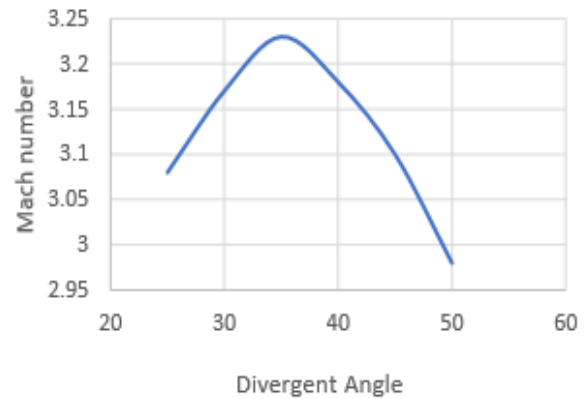


Fig-17 Mass-Weighted Average Mach at different divergent angle.

Table II- Mass-Weighted Average Mach at different divergent angle.

Angle	Mach Number
25°	3.08
30°	3.17
35°	3.23
40°	3.18
45°	3.1
50°	2.98

The above data shows that the exit Mach number of the nozzle at different divergent angle is varying with increasing from 25° to 35° and then decreasing with increment in angle. The maximum value of mass-weighted average of the Mach number is attained at an angle of 35°. The higher the exit Mach number corresponds to higher velocity at the exit thus producing greater amount of thrust at the exit plane. Since the mass flow rate remains a constant of 0.139 kg/s. The contour plot shows a great variation of flow as there is change in the geometry of the nozzle. This variation is caused due to the shear stresses developed in the nozzle flow and with the increase in angle shows the formation of free separation shock towards the centerline resulting in decreasing the overall exit Mach number irrespective of the extra expansion area ratio at higher divergent angles.

B. Penetration Length

Penetration length is defined as the length along the nozzle up-to where the Mach number increases predominantly at the centerline without being affected by the free separation shock. This penetration length defines the flow in the nozzle to expand optimally before it leaves the nozzle. The below graph shows the variation of Mach number along the centerline of the different modelled nozzle:

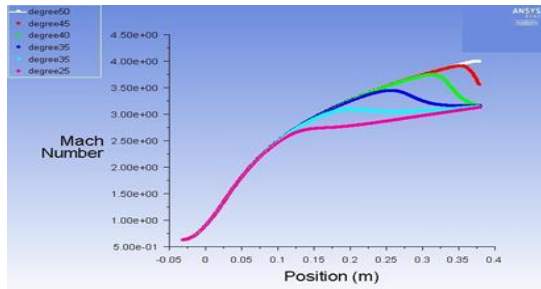


Fig 18- Centerline Mach Number variation

From fig 18 the variation of the Mach number along the axis [4] shows that the Mach increases predominantly in the region of favorable pressure gradient as the geometry of the nozzle changes the region of low pressure to high pressure thereby increasing the Mach number.

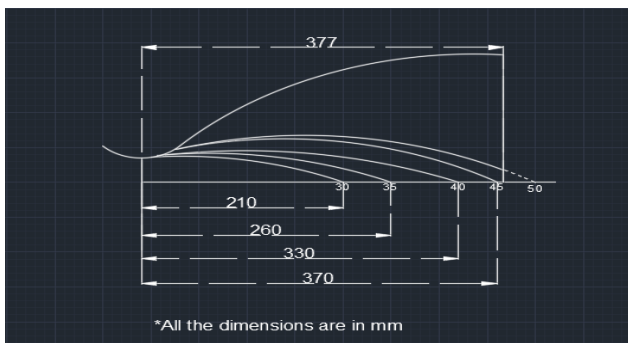


Fig 19- Penetrating length schematic at different angles different angles [4]

From fig 20 The exit Mach number of the nozzle with 25° to 40° is same with a near value of 3.2 Mach. The maximum Mach achieved inside the flow nozzle before exit is linearly increment of 0.2 Mach. This increased Mach number then tends to decrease on arriving at the penetrating length, as there is loss of energy by the FSS and RSS acting through the centerline.

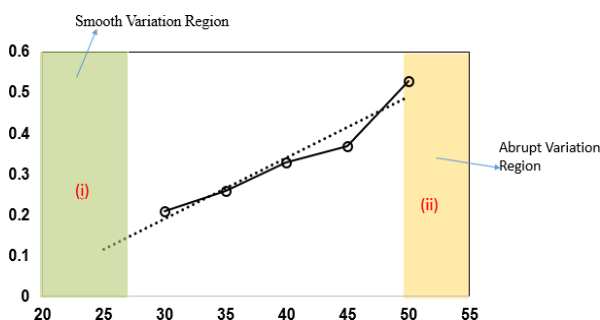


Fig 20- Penetrating length (m) at different angles.

The penetrating length for angle 25 degree does not go valid as it lies towards the smooth variation region with increment of angle to 30 degrees, we see that penetration length is being developed. The trend line shows that this length keeps on increasing as we increase the divergent angle of the nozzle. At an angle of 35 degrees, the divergent length is reached up-to 3/4th length of the nozzle. The increased divergent angle of 45 degrees notices that the penetrating length increases and reaches to the nozzle exit therefore creating more losses in the entire nozzle flow domain. The divergent half angle of 50 degrees witnesses that the penetration length crosses the nozzle length thus having a blunt end at the exit plane. This blunt surface reveals that the losses in the nozzle flow is maximum at the divergent half angle of 50 degrees which is the result of it experiencing the lowest mass-weighted average Mach.

V. CONCLUSION

The parametric study of the different modeled nozzle reveals that changing the divergent angle of the nozzle while keeping the divergent length of the nozzle constant at an NPR of 1000 results in asymmetric change in the flow parameters in the flow regime of the nozzle.

The studied parameters affect the performance of the nozzle. The exit Mach number experienced by the nozzle is greater at divergent angles of 45° and 50° at the centerline but from the Mass-Weighted average Mach number we see that the maximum Mach number attained in the modeled nozzle with minimal losses is at an angle of 35°.

The increase in percentage of Mass-Weighted average Mach number from divergent angle 25° to 35° is 4.87%. Whereas the increase from 30° to 35° is 1.89%. As we can see from the table 2 that the Mass-Weighted average Mach number decreases after the divergent angle of 35°. So, the decrease in percentage of it from 35° to 40° is 1.55%; from 35° to 45° its 4.02% and from 35° to 50° the decrease in percentage of Mass-Weighted average Mach number is 7.74%.

If the penetration length is of the order of 75% of the total nozzle length, the flow expands optimally to give the maximum thrust output at a defined pressure ratio across the nozzle.

The thrust obtained in the nozzle is directly proportional to the exit velocity of the flow. Thus, the exit Mach number for 35° divergent half angles is concluded to generate the maximum thrust.

FUTURE WORK

The current work is done at an NPR of 1000, which reveals the divergent half angle of 35° gives the perfect exit parameters for the rocket nozzle to perform. The rocket nozzle works at a varying pressure ratio from the sea level to a target altitude [5]. Further work at different pressure ratio and changed exit angle in order to alter the length of the nozzle might unleash better performance results for the nozzle. The analysis in a different turbulent model could tell how the viscous effects alter the parameters of the nozzle.

This real-life modeling of the nozzle at a relevant pressure ratio gives the industry to choose the perfect combination of the half angle and exit angle with the desired application.

APPENDIX

MATLAB CODE

```
clear all
close all
clc

Rt = 30.5;
area_ratio = 31.4754;
theta = 50;
theta_exit = 16.5;
theta_n = theta*pi/180;
k = 0.8;

Ln = k*(sqrt(area_ratio)-1)*Rt/tand(theta_exit);

% FIRST CURVE (FC)

% Angles for First Curve
Angle_FC = -(pi+(45*pi/180));
FC_step = (-pi/2-Angle_FC)/100;
theta_FC = (-3*pi/4):FC_step:(-pi/2);

% Coordinates for First Curve
x_FC = cos(theta_FC)*1.5*Rt;
y_FC = sin(theta_FC)*1.5*Rt+(1.5*Rt+Rt);
x_FC1 = x_FC';
y_FC1 = y_FC';

% SECOND CURVE (SC)

% Angle for Second Curve
Angle_SC = -pi/2;
SC_step = ((theta_n-pi/2)-Angle_SC)/100;
theta_SC = -pi/2:SC_step:(theta_n-pi/2);

% Coordinates for Second Curve
x_SC = cos(theta_SC)*0.382*Rt;
y_SC = sin(theta_SC)*0.382*Rt+(0.382*Rt+Rt);
x_SC1 = x_SC';
y_SC1 = y_SC';

% THIRD CURVE (TC)

% Coordinates for Third Curve
x_TC = cos(theta_n-pi/2)*0.382*Rt;
y_TC = sin(theta_n-pi/2)*0.382*Rt+(0.382*Rt+Rt);

% Exit Coordinates
y_exit = sqrt(area_ratio)*Rt;
matrix_y = [y_TC^2 y_TC 1; y_exit^2 y_exit 1; 2*y_TC 1 0];
matrix_x = [x_TC; Ln ; 1/tan(theta_n)];
x_exit = matrix_y^-1*matrix_x;
a = x_exit(1,1);
b = x_exit(2,1);
```

```
c = x_exit(3,1);
y = y_TC*0.8:y_exit;
x = a*y.^2+b*y+c;

% Complete Coordinate
Rao_x = [x_FC x_SC x];
Rao_y = [y_FC y_SC y];
% Lower Geometry
Rao_y1 = -1*Rao_y;

%Coordinates
R_X = Rao_x';
R_Y = Rao_y';
Rao = [R_X R_Y];

% Figures
figure(1)

plot (Rao_x,Rao_y,'r','LineWidth',3)
hold on
plot (Rao_x,Rao_y1,'r','LineWidth',3)
hold off
```

REFERENCES

1. G. V R. Rao. "Exhaust Nozzle Contour for Optimum Thrust", *Journal of Jet Propulsion*, Vol. 28, No. 6(1958), pp. 377-382.
2. Jéger, C. and Veress, Árpád (2019) "Novell Application of CFD for Rocket Engine Nozzle Optimization", *Periodica Polytechnica Transportation Engineering*, 47(2), pp. 131-135.
3. Kate Davis, Elizabeth Fortner, Michael Heard, Hannah McCallum, and Hunter Putzke. "Experimental and Computational Investigation of a Dual-Bell Nozzle", *53rd AIAA Aerospace Sciences Meeting, AIAA SciTech Forum*, (AIAA 2015-0377)
4. Biju Kuttan and M Sajesh (2013) "Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics", *The International Journal Of Engineering And Science (Ijes)*, Volume-2, Issue 2 Pages 196-207.
5. C. Génin, R. Stark, O. Haidn, K. Quering and M. Frey (2013), "Experimental and numerical study of dual bell nozzle flow", *Progress in Flight Physics 5 Pages 363-376*
6. Sarah L. Kulhanek (2012), "Design, Analysis and Simulation of Rocket Propulsion System" a thesis by a student (Master in Science), *University of Kansas, Lawrence, Kansas, U.S.*
7. M.H.M. Noh, A.H.A. Hamid, R. Atan and H. Rashid (2011), "Numerical Investigation of Chocked Converging-Diverging Nozzles for Thruster Application", *IJUM Engineering Journal, Special Issue, Mechanical Engineering*

AUTHORS PROFILE



Rishab Kumar Agrawal, was born in Odisha, India, in 1996. He received the B. Tech degree in Aerospace engineering from Hindustan Institute of Technology & Science, Chennai, India, in 2019. In 2018, he worked as an Intern for Engenius Aerospace Ltd., Ukraine as a Jr. Design Engineer in the field of Rockets & Missile. He is in the core team of National Space Society (USA)-Mumbai Chapter, working as a Lead Researcher. He is also the Technical Director of Cosmagen Space Organization of Tamil Nadu region, Chief Technical Officer of a startup named JHATAYU; in which he received INDIA BOOK OF RECORD & ASIA BOOK OF RECORD for manufacturing Asia's first flying bike. His research interest includes Solid Rocket Propulsion, Rocket motor, Dark matter. Email: Rishabagrawal.100@gmail.com.



Parametric Output of Penetration Length in De-Laval Nozzle using Computational Fluid Dynamics



Sivaram Munagala, is pursuing B. Tech degree in Aeronautical engineering at Hindustan Institute of Technology & Science, Chennai, India. He is a final year student. He has published research papers on “CFD Analysis of Rocket Nozzle with various angle of divergence using Ansys Fluent Software” & “Design and Flow analysis of various Rocket

Nozzles by using computational fluid dynamics (CFD)” in JETIR & IJRAR journals respectively. He is also a member of National Space Society (USA)-Mumbai Chapter. His area of interest are Structure and Propulsion. Email: munagalasiva24@gmail.com