

2D NOZZLE DESIGN TECHNIQUE IN MATLAB AND EXTENTION RESEARCH USING FDM

Kavita Gambhir¹, Dr. RK Gupta²

¹M. Tech, ²HOD, Mechanical Engineering, GITM, Bilaspur, MDU Rohtak

ABSTRACT: *In an era of manufacturing design work, it is very important work assignment for designing of nozzle. This device is designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber. A nozzle is often a pipe or tube of varying cross sectional area and it can be used to direct or modify the flow of a fluid. One of the challenging task in aerospace industries and mechanical industries is the design/analysis and realization of complex nozzle systems for solid rocket motors of satellite launchers, missiles, etc., Existing empirical relations have limitations to design such complex nozzle systems. The proposed technique gives the ideal nozzle geometry using the method of characteristics for a Quasi-2D Diverging Nozzle. Using 2D nozzle flow relations, an optimal throat area is found that will produce the max amount of thrust for the given ambient pressure and combustion chamber parameters. In proposed technique, nozzle exit area is found that produces the max amount of thrust. After simulation, the proposed technique finds the error Area, error Mach. Further, Shape of the said nozzle has been calculated using quasi 2D Diverging technique or that resembles the shape and length in terms of Mach Number P/P and Nozzle respectively.*

Keyword: Nozzle, Quasi-2D Diverging Technique, CFD, MATLAB

I. INTRODUCTION

1.1 Basic of nozzle design

A nozzle is a device designed to regulate the direction or characteristics of a fluid flow (especially to extend velocity) because it exits (or enters) an inside chamber or pipe. A nozzle is often a pipe or tube of variable cross sectional area, and it can modify the flow of a fluid (liquid or gas).[9]

1.2 Computational Fluid Dynamics (CFD) For Nozzle.

The goal of a nozzle is to increase the mechanical energy of the flowing medium at the expense of its pressure and internal energy. Nozzles can be delineated as merging (narrowing down from a good diameter to a smaller diameter within the direction of the flow) or divergent (expanding from a smaller diameter to a bigger one).

1.3 Quasi-2D Diverging Nozzle

This program gives the ideal nozzle design based on Computational fluid dynamics. Assume the gas is exhausting from a combustion chamber that has no mass flow rate. Using Quasi-2D Diverging nozzle flow relations, an optimum throat space is found that can manufacture the GHB

quantity of thrust for the given close pressure and combustion chamber parameters.

1.4 Use of nozzle

As nozzle is often a pipe or tube of varied cross sectional space, used to direct or modify the flow of a fluid (liquid or gas). Nozzles are often used to manage the speed of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. Nozzles are used to regulate the flow of fluid. The area unit is found at the tip of a hose or pipe from which liquid or gas substances are discharged. Different nozzles area unit are created for numerous applications.

1.5 Basics of Nozzle design & it's Technique

An aviation vehicle, whether an aviation plane, orbital dispatch vehicle or interplanetary shuttle, is quickened by an impetus framework to a speed managed by prerequisites particular to the vehicle's central goal. In all cases, a spout is utilized to remove the greatest push from high-weight debilitate gasses produced by the impetus framework. In case of supersonic nozzle, which is operating at pressure ratios well below its design point, a shock forms inside the nozzle and flow downstream of the shock separates from the nozzle walls. Even though this flow is very basic, it remains poorly understood. This is illustrated by the large discrepancy between predictions of quasi-one-dimensional in viscid theory and the actual behavior of the flow. [6]

II. DISCUSSION ON EXTENSION OF NOZZLE DESIGN

2.1 Fused Deposition Modeling

Fused deposition modelling (FDM) is a common additive manufacturing (AM) method for creating polymer parts. FDM builds parts additively by depositing a small bead of molten plastic through an extrusion head onto a work platform. The nozzle (or platform) moves via computer control to lay down a plastic pattern on the platform. Complete parts are built by laying down successive patterns one layer at a time. While the lead time to produce AM parts via FDM is relatively short, the per unit manufacturing time is much longer than conventional mass production techniques. This is partly due to the fact the extrusion head, which has a small diameter nozzle, has a low volumetric deposition rate. In conventional FDM the nozzle diameter is small to maximize the resolution of each layer. Three main options exist to increase the build rate: i) increase the speed of the horizontal movements; ii) minimize the material of the part; or iii) increase the diameter of the nozzle. This paper introduces a concept for adapting an existing AM system

which employs a variable diameter nozzle for the fused deposition of polymers. The variable nozzle allows the print resolution and the build speed to become independent variables which may be optimized.

2.2 Variable fused deposition modeling (vfdm)

2.2.1 Impacts on horizontal resolution

FDM nozzles utilize circular orifices to take advantage of the axisymmetric properties in the horizontal plane. The benefit of this is that the nozzle does not need to be kept normal to the extrusion path. The disadvantage is that all angles will have radii greater than or equal to the radius of the nozzle as shown in Figure 2.1.

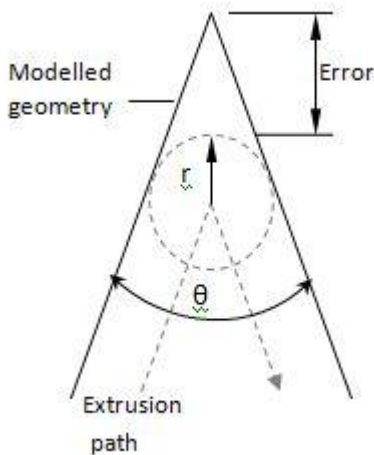


Figure 2.1: Geometric error due to circular nozzle
 The geometric error can be calculated using Equation 1 below:

$$Error = \frac{r}{\sin\left(\frac{\theta}{2}\right)} - r$$

Where r = radius of the extrusion orifice and θ = angle of modelled geometry.

Figure 2.2 shows the relationship between the geometric error and the intended angle for normalized nozzle radii. The bottom line represents a radius of one unit while the middle and top line represent radii two and three times larger respectively. As can be seen on the graph the error is small for large angles but grows significantly for angles less than 60°. For 60° angles the error is at least equal to the orifice radius

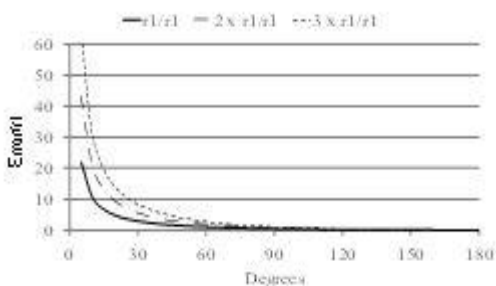


Figure 2.2: Graph showing minimum geometrical error of an angle printed with a circular nozzle

In practice, extrusion paths like the one shown in Figure 2.1 are prevented by the tool path generation software due to the overlapping projections of the nozzle path as it enters and exits the corner. To verify this statement, an isosceles triangle with a 15° vertex was printed in the horizontal plane on a Dimension 1200 BST FDM machine. The slicing and tool paths were generated using Stratasys' proprietary software Catalyst®EX. According to Equation 1, the calculated minimum error is 0.67 mm while the measured error was 3.87 mm. Therefore it is evident that the errors associated with printing high aspect ratio features have been highly underestimated and the need for small diameter nozzles is even greater. VFDM allows for a larger diameter orifice to be used for infill, where geometric error is less important, and smaller diameters for the exterior shell where high resolution is desirable. The authors believe that by using this process it is possible to improve the quality of FDM parts while increasing build speed.

2.3 Impacts on extrusion time

For large parts, with a high percentage of infill, the extrusion time during a build is a function of the nozzle diameter and the geometry of the part. For the purposes of an example, consider a 30, 60, 90° triangle which is 1 unit thick and has dimensions as defined in Figure 2.3.

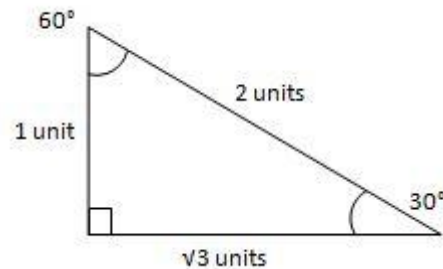


Figure 2.3: Plan view geometry of 30, 60, 90 triangle for extrusion time saving analysis

Assuming the part is 100% solid and the nozzle diameters, feed rate and layer thicknesses are known, it is possible to calculate the extrusion time as a function of part volume (Equation 2).

$$Extrusion\ time = \frac{volume}{D \times f \times l} \quad (2)$$

Where *volume* = the volume of the part (mm³), *D* = the nozzle diameter of the part (mm), *f* = nozzle feed rate (mm/s) and *l* = layer thickness.

If the nozzle is able to use a small diameter for the outer shell(s) and a larger diameter for the infill then Equation 2 becomes:

$$Extrusion\ time = \frac{volume\ e_{shell}(s)}{D1 \times f \times l} + \frac{volume\ e_{infill}}{D2 \times f \times l} \quad (3)$$

Where *volume_{shell(s)}* = volume of the outer layer(s) of the part, *volume_{infill}* = volume of the inside of the part, *D1* = small nozzle diameter, *D2* = large nozzle diameter.

Using the following parameters, *D1* = 0.5 mm, *D2* = 1.0 mm, *f* = 25 mm/s, and *l* = 0.5, a graph was created showing the

extrusion times for the 30, 60, 90° triangular prism over a range of volumes and variable nozzle diameter configurations. Configuration A (the steepest line) shows the extrusion time required if only a nozzle of diameter 0.5 mm is used. Configuration D (the shallowest line) shows the extrusion time for a fixed 1.0 mm nozzle. C and B show the extrusion times for parts with one and two layer thick shells respectively.

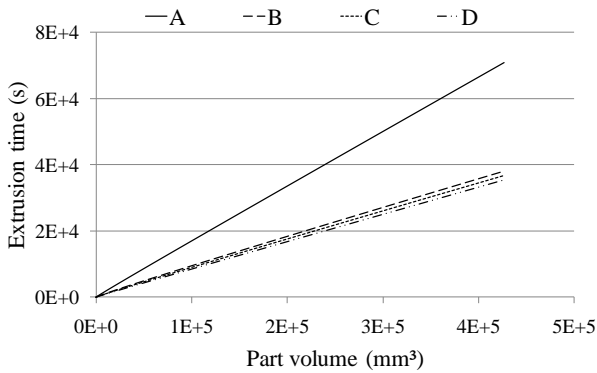


Figure 2.4: Graph showing the extrusion times as a function of part volume for different nozzle configurations

As can be seen by the small difference between configurations B, C and D, the amount of time required to add high resolution outer shells is insignificant for large volume builds. In the case of the 30° vertex, replacing the 1.0 mm nozzle with the 0.5 mm nozzle reduces the theoretical error from 1.43 mm to 0.72 mm. In practice, due to the way the tool paths are calculated, the improvements in resolution will be far greater. It should be noted here that the same extrusion time savings could be made by utilizing two separate extrusion heads with different fixed diameter nozzles. However, the problem with this solution is the time taken to switch extrusion heads and the extra room required on the positioning rack. This problem is magnified for machines with multiple material capabilities.

2.5 Concepts for VFDM

Possible concepts for VFDM nozzles can be placed into two main categories: continuously variable nozzles (CVNs) and discretely variable nozzles (DVNs). CVNs allow much more freedom in choosing the print resolution and build speed and allow for the possibility of size changes whilst printing. The main disadvantage of CVNs is the mechanical complexity and the necessary design compromises that result. Possible solutions reviewed by the authors include mechanical irises, sliding jaws and smart materials. All CVN solutions analyzed required that the orifice be an approximation of a circle. This increases the error relative to a circle of the same area and may result in orthotropic extrusion properties. For example think of a square moving in a direction parallel with its sides, and then imagine the square moving diagonally. Obviously the track will be wider for the diagonal movement. Another difficulty in designing a CVN is scaling down the actuating mechanisms to a size desirable for typical FDM applications. The design and build of CVNs is likely to be the topic of future research by the authors.

DVNs are mechanically simpler than CVNs and are therefore relatively simple to manufacture at the sub-millimeter scales required. Possible solutions to DVNs are interchangeable nozzles and multiple stage nozzles. Interchangeable nozzles could sit in a carousel and automatically change via computer control. This concept was abandoned due to the difficulties associated with the nozzles into and out of the polymer flow stream. The multiple stage nozzle works by moving nested inserts inside the nozzle. The multiple stage nozzle was found to be relatively easy to implement while allowing circular orifices and uninterrupted polymer flow. The design for a two stage nozzle is presented in the next section.

2.6 Two stage nozzle design

The two stage nozzle consists of an inner and outer nozzle each with fixed orifice diameters. The inner nozzle is actuated by a solenoid, and in its lowest position is nested firmly against the inner surface of the outer nozzle, restricting the flow through the smaller orifice. When the inner nozzle is lifted a secondary melt chamber is formed which allows multiple polymer flow streams to coalesce before leaving the larger orifice as shown in Figure 2.5.

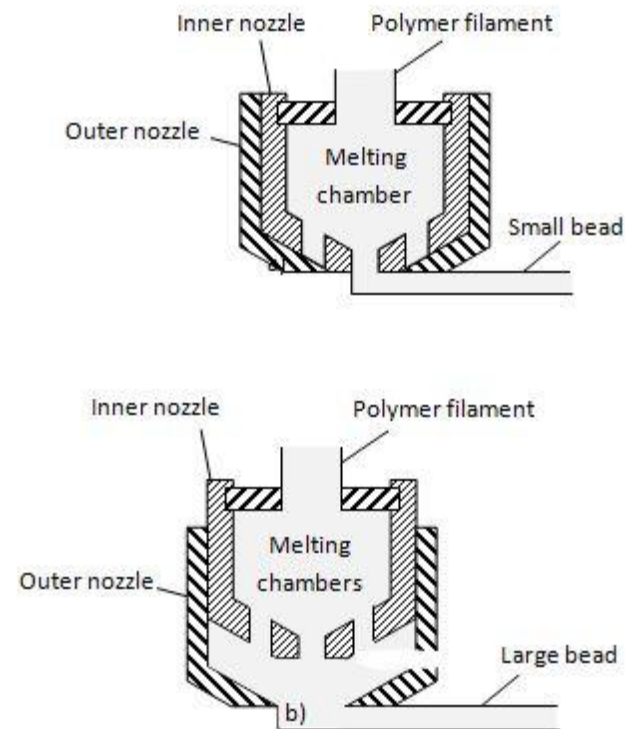


Figure 2.5: Two stage nozzle with a) small diameter orifice and b) large diameter orifice

This concept achieves many of the objectives of VFDM but also has a number of significant drawbacks. Assuming the outer shell of each layer is deposited first, there will be a small delay required before the infill can start as the secondary chamber must be filled with molten polymer. The second major drawback is that polymer in the secondary chamber is likely to be pushed out as the inner nozzle moves back into the lower position. This not only wastes material

but will add extra build time as the nozzle moves to a designated discharge point. Notwithstanding these problems it is expected that the two stage nozzle will reduce the build time for large solid high resolution parts. The following section describes how the nozzle is integrated into a RapMan 3.0 3D printer from Bits from Bytes (BfB).

2.7 Two Stage Nozzle Design

2.7.1 RapMan 3D printers

The RapMan 3.0 is a FDM machine based on the open source RepRap project and is shown in Figure 2.6. The low cost and open structure of the RapMan makes it highly suitable for research related modification.



Figure 2. 6: Unmodified RapMan 3.0 from Bits for Bytes
The VFDM extruder is designed to replace the original RapMan extruder with minimal changes. The standard RapMan heating system will be changed from a fixed resistance wire and fire cement configuration to a removable aluminium block which contains two power resistors. One of the power resistors is connected to the RapMan control board along with a thermistor for temperature control. The other power resistor is connected to an external circuit to enable a base heating load giving greater temperature control via the thermostat, see Figure 2.7.

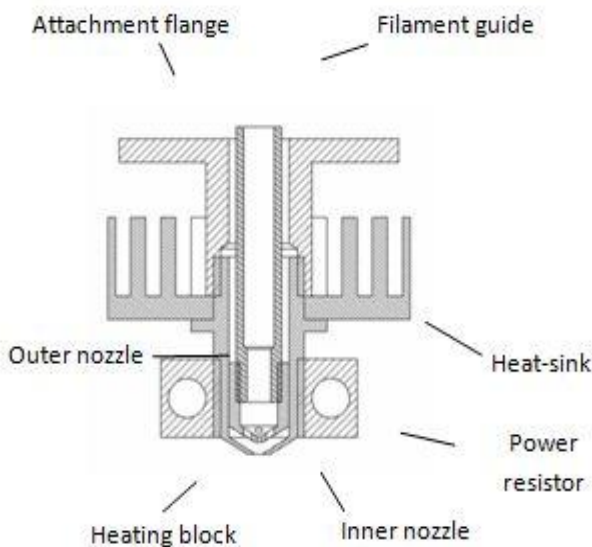


Figure 2.7. Two stage nozzle and heating assembly

III. TOOL PATH GENERATION

It is the authors' intention to develop an advanced tool path generation algorithm in parallel with future developments in VFDM. The tool path generation described in this paper is a more basic method which will be used for testing the two stage nozzle concept.

3.1 Standard printing procedure

A summarized standard procedure for printing parts on a RapMan machine is as follows:

- Generate an STL file from a CAD program.
- Upload the STL file into Bits from Bytes Axon (Skeinforge).
- Generate the G-code with the .bfb extension.
- Upload the .bfb file onto an SD card.
- Insert the SD card into the RapMan card reader and run the file.

The standard input file to most FDM systems is the stereolithography or STL file. STL files describe the surface geometry of parts by using a triangular mesh. The BfB Axon program is a user friendly front-end to the more complicated but versatile Skeinforge open source software. Skeinforge is a tool chain composed in Python script that allows the user to control many of the parameters that generated the tool path G-code. G-code is the common name given to the computer numeric control programming language adopted by many automated machine tools. While generic G-code generators exist, many manufacturing technologies use machine specific G-codes which require post processing in order to be used in other machines. The RapMan uses its own form of G-code which is, amongst other things, differentiated by the extension .bfb. The next section describes how the Skeinforge settings were adjusted for the two stage nozzle and how the BfB G-code was subsequently altered to allow for the nozzle mode changes.

IV. PROPOSED WORK

4.1 Solution Methodology: The exploration proposes and builds up the down to earth advancements for the configuration, testing, and investigation of the spouts utilized as a part of the QUASI-2D process.

- Develop a MATLAB programming skilled to create pragmatic diagrams for the QUASI-2D, prepare and produce the 2D suggested spout form, utilize the Computational Fluid Dynamics (CFD) .Further make a air flow to nozzle for test the shape of nozzle.

4.2 Source of Data and Methodologies: The investigative methodology to be embraced will fundamentally be guided by the points highlighted previously. These include:

- Gather QUASI-2D process data from exploration with the primary target concentrated on the spout outline,
- Use the compressible stream hypothesis to construct a brought together hypothesis for the computation of particles speed in the QUASI-2D process,
- Use the general hypothesis, called the 'Strategy for Characteristic' (MOC) to construct a model for creating a two dimensional 'least length' spout for various gas molecule extensions.
- Develop an effective MATLAB computational

programming that will handle all building counts and yearning parameters for the QUASI-2D spout, and utilize the Computational Fluid Dynamics (CFD).

4.3 MacCormack Plan: The unequivocal MacCormack calculation (MacCormack, 1984) utilizes two stages to progress as a part of time, one is the indicator and the other is the corrector. It is a arrangement in both space and time. In the indicator step the flux vector at a specific face is figured utilizing the properties of the forward cell, though in the corrector step the property estimations to be utilized are the ones in respect to the retrogressive cell.

4.4 Objectives: Mat lab Simulation based on Quasi-2D Diverging Technique.

- Solve for the nozzle exit area that produces the max amount of thrust.
- Use the method of characteristics to solve for the shape of the said nozzle.
- Automatically make curvilinear mesh for simulating the nozzle
- Solves the Euler equations on the mesh using the Mac Cormack finite volume method.
- Plots the result

The filament guide also acts as the connection between the solenoid and the inner nozzle. The filament feed mechanism will be moved slightly to allow the solenoid to be positioned directly above the inner nozzle. The RapMan uses open source software Skein forge, to generate the required G-code.

V. SIMULATION AND RESULTS

5.1 Simulation Tools

Many specialists and researchers overall use MATLAB to dissect and outline the frameworks and items changing our reality. MATLAB is in car dynamic wellbeing frameworks, interplanetary shuttle, wellbeing observing gadgets, brilliant force matrices, and LTE cell systems. It is utilized for machine learning, signal handling, picture preparing, PC vision, interchanges, computational money, and control outline, apply autonomy, and considerably more.

5.2 Simulation Parameters

- $T_c = 2000$;
% Temperature in the combustion chamber (K)
- $P_c = 1.2e6$;
% Pressure in the combustion chamber (Pa)
- $P_{amb} = 101e3$;
% ambient pressure (Pa)
- $T_{amb} = 300$;
% ambient temperature (K)
- $\Gamma = 1.25$;
% Ratio of Specific Heats C_p/C_v (Gamma)
- $W = 25.4$;
% Molecular weight of gas (kg/kmol)
- Width = .1;
% Nozzle width (meters)
- $H_{th} = .025$;
% Throat height (meters)

5.3 Simulation Steps

First, we initiate with nozzle's file from the current folder and run the simulation.

From this simulation, we get value of error area & error mach.

After that Fig 5.2, 5.3 & 5.4 will show up.

Second, Click on Noz_mesh from the current folder and run the Simulation.

From this simulation, we get the ans= mesh done.

Third, click on the noz_cfd from the current folder and run the simulation.

Then, we see that some values are running from K=1 to K=499. When its reach to K=500, some graph will show again. (Fig 5.5 & 5.6).

5.4 Simulation results from MATLAB

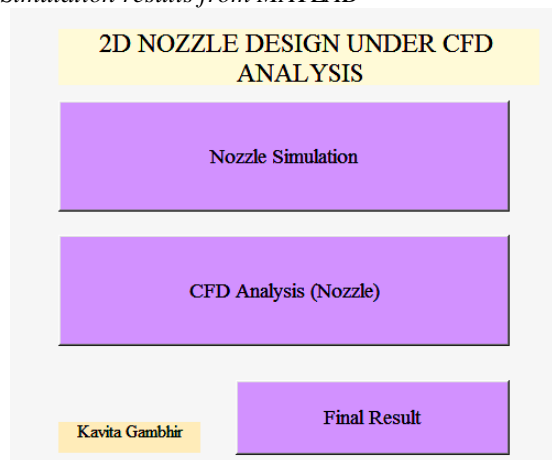


Fig 5.1 First GUI

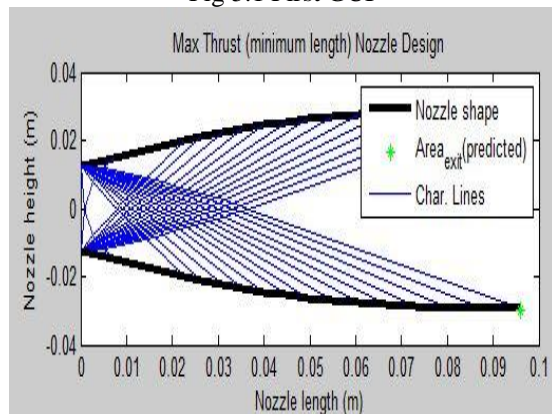


Fig 5.2 Maximum Thrust Nozzle Design

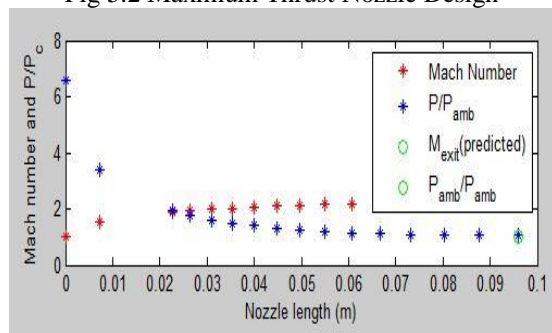


Fig 5.3 Mach number and P/P vs Nozzle Length

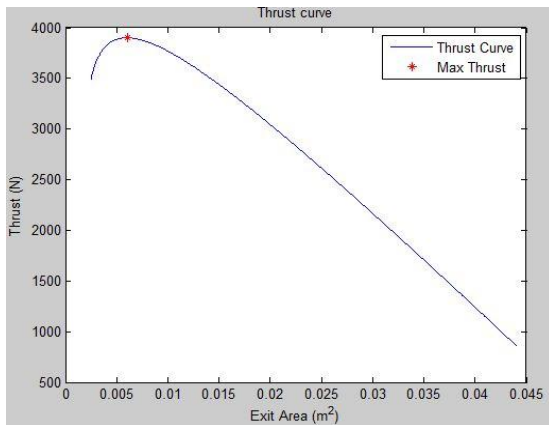


Fig 5.4 Static thrust vs. Exit Area

Program to extrapolate the data points from nozzle design and make a uniform grid spacing in the x-direction (Change nothing). Simply run this script. Find the minimum spacing given by the method of characteristics and set as the dx value.

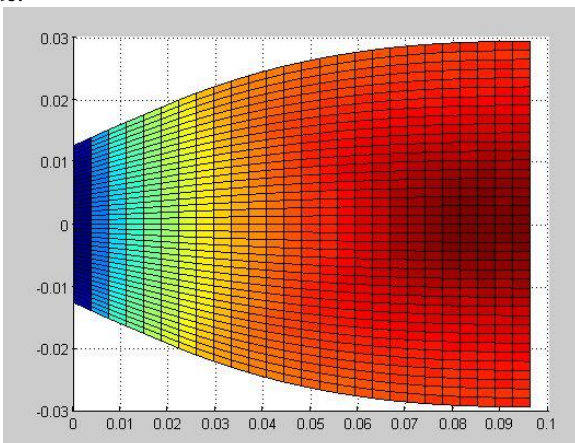


Fig 5.5 Nozzle Design and Plots of Mach Number (Pressure vs Length)

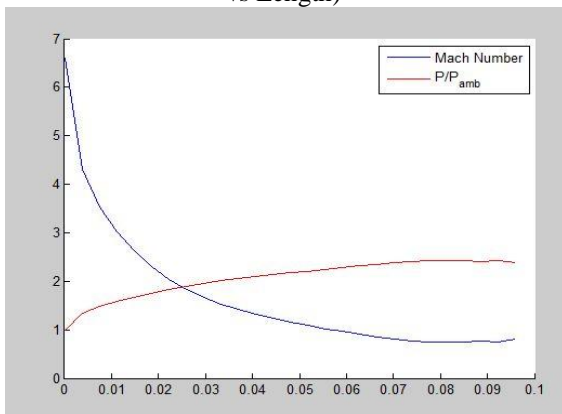


Fig 5.6 CFD Simulation of Designed nozzle

5.5 RESULT

The proposed technique gives the ideal nozzle geometry using the method of characteristics for a Quasi-2D Diverging Nozzle. Using 2D nozzle flow relations, an optimal throat area is found that will produce the max amount of thrust for the given ambient pressure and combustion chamber

parameters. After Simulation, This technique finds a nozzle exit area that produces the max amount of thrust as shown in fig 5.2. After run the simulation, this technique finds the error_Area= -3.1045, error_Mach= -4.2719e-004. Further, Shape of the said nozzle has been calculated using quasi 2D Diverging technique or that resemble the shape and length in terms of Mach Number P/P and Nozzle respectively.

VI. CONCLUSION AND FUTURE WORK

One of the challenging task in aerospace industries and mechanical industries is the design/analysis and realization of complex nozzle systems for solid rocket motors of satellite launchers, missiles, etc., Existing empirical relations have limitations to design such complex nozzle systems. The proposed technique gives the ideal nozzle geometry using the method of characteristics for a Quasi-2D Diverging Nozzle. Using 2D nozzle flow relations, an optimal throat area is found that will produce the max amount of thrust for the given ambient pressure and combustion chamber parameters. Future work is directed towards the qualification testing of the nozzle system under various environmental conditions like temperature, humidity and cyclic loading. Advancement, will be carried out in constructing 3-D simulation of nozzle in MATLAB.

REFERENCES

- [1] C. Xuyi; Sun Yat-sen University, No.135, Xinggong Xi Road, Guangzhou, 510275, P. R. China X. Zhang “Coupled analysis of the film-cooling and infrared characteristics of an axisymmetric vectoring exhaust nozzle” 978-1-4799-5379-0 10.1109/AERO.2015.7119191 IEEE7-14 March 2015.
- [2] H. R. Castro ; Dept. of Phys. & Chem., Sao Paulo State Univ., Guaratingueta, Brazil ; K. G. Kostov ; V. Prisyazhnyi “Influence of Nozzle Shape on the Performance of Low-Power Ar Plasma Jet” IEEE Transactions on Plasma Science (Volume:43 , Issue: 9), 2015.
- [3] H. T. Lim ; H. Song ; Y. S. Kim ; K. D. Bae ; C. H. Cho ; C. S. Lee ; J. W. Shin ; S. J. Shin ; K. Kuk ; Y. S. Oh “T-jet: a novel thermal inkjet printhead with monolithically fabricated nozzle plate on SOI wafer” TRANSDUCERS, Solid-State Sensors, Actuators and Microsystems, 12th International Conference on, 2003 (Volume:1) IEEE.
- [4] Bing Liu; Zhengzhou University, School of Chemical Engineering and Energy, Zhengzhou, China; Dingbiao Wang. Mechanic Automation and Control Engineering (MACE), 2011 Second International Conference on 15-17 July 2011 4559,4562978-1-4244-9436.
- [5] Yu Hai-long ; School of Energy & Environment, Zhongyuan University of Technology, China ; Zhang Chao ; Dong Xiang-yuan ; Guo Shu-qing “Experimental study of the effect of nozzle structure on atomizing performance of a novel coal water slurry nozzle” 2009 4th IEEE Conference on

Industrial Electronics and Applications.

- [6] SibenduSom, Anita I. Ramirez, Douglas E. Longman, Suresh K. Aggarwal. "Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions" Elsevier Ltd.Fuel 90, vol. 90, 1267–1276; 2011.
- [7] F.J. Salvador, J. Martinez-Lopez, J. V. Romero, M.D. Rosello. "Computational; study of the cavitations phenomenon and its interaction with the turbulence developed in diesel injector nozzles by Large Eddy Simulation (LES)"Mathematical and Computer Modeling; Vol. 57, 1656– 1662., April 2013.
- [8] R MacCormack , "The effect of viscosity in hypervelocity impact cratering"Journal of spacecraft and rockets, 2003 - arc.aiaa.org Cited by 1629, ,aaaAIAA ,Journals of space and rocket, volume 40, No-5, 2003.
- [9] www.mathworks.com/products/simulink/