

2D NOZZLE DESIGN USING QUASI AND MACCORMACK STRATEGY TECHNIQUE IN MATLAB

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 Computataional 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]

1.6 MacCormack strategy

In computational liquid elements, the MacCormack strategy is a broadly utilized as discretization plan for the numerical arrangement of hyperbolic incomplete differential conditions. This second-arrange limited contrast technique was presented by Robert W. MacCormack in 1969. The MacCormack strategy is exquisite and straightforward and program. The MacCormack strategy is appropriate for nonlinear conditions (Inviscid Burgers condition, Euler conditions, and so forth.)[8]

II. LITERATURE SURVEY

C. Xuyi, Sun Yat-sen proposed in the year of 2015 that a thin band model to research the cooling and radiation of a vectoring spout and, specifically, to figure the gas phantom trademark in infrared band. The radiative warmth exchange between the hot gas and the divider is considered with a fenced in area model. The computation of film cooling is performed through a cooling adequacy technique. [1] H. R. Castro K. G. Kostov; V. Prysiazhnyi proposed in the year of

1015 for the impact of spout geometry on the properties of barometrical weight Ar plasma plane was explored. Three diverse spout sorts were tried: 1) a decreased spout (to get a dainty plasma tuft); 2) a straight spout (utilized as a part of most cases); and 3) an amplified spout (to build the dynamic plasma region).[2] H. T. Lim ; H. Tune ; Y. S. Kim ; K. D. Bae ; C. H. Cho ; C. S. Lee ; J. W. Shin ; S. J. Shin ; K. Kuk ; Y. S. Goodness proposed in the year of 2003 for A novel warm inkjet print head with solidly created nickel spout plate on SOI wafer has been proposed interestingly. A chamber and a restrictor are actualized on the 40/spl mu/m thick top-silicon layer, and a spout plate covering warmer layers are solidly created on them.[3] Bing Liu; Dingbiao Wang proposed in the year of 2009 that a solitary spout, double nozzles constrained cooling of billet warmth exchange coefficient test. By testing diverse sorts of nozzles to get the warmth move coefficient in various weights, and the distinctive impacts between single-nozzle and twofold spout. The aftereffects of the investigation are shoen that the test of the spout warm execution give confirmation to spout design of auxiliary cooling and improving optional water cooling framework.[4] Yu Hai-long Zhang Chao ; Dong Xiang-yuan ; Guo Shu-qing proposed a novel Coal Water Slurry (CWS) spout for gasification was produced, and its atomizing execution was tentatively concentrated on. The impacts of the spout structure upon the atomized molecule dispersion, the Sauter Mean Diameter (SMD) and the spout atomizing point were examined. [5]

III. EARLIER WORK

F.J. Salvador, J. Martinez-Lopez, J. V. Romero, M.D. Roselloone of the principle elements influencing spout release coefficient and the energy of the infused fuel plane is cavitation in the spout gap. This is particularly the case in straight round gaps with a sharp edged passageway. Geometric induced cavitation happens when the neighborhood weight drops underneath the vapor weight and a vapor stage frames. At the point when cavitation happens in the spout opening the stream gets to be gagged. It is the first occasion when that another Arbitrary Length Nozzle plan technique (ALN strategy) is connected to the spout outline of a gas dynamic laser. For examination, ALN spout and MLN spout (Minimum Length Nozzle) are outlined with the same Mach number at spout leave, the same tallness of the spout throat and the same length of the spout. 2-D gas dynamic laser Small Signal Gain (SSG) field recreation is done to the spouts and the reenactment results show that ALN spout with appropriate outline parameters can acquire shorter spout length and greater SSG than the MLN spouts. It is demonstrated that the ALN technique is a more propelled outline strategy on the spout configuration of a gas dynamic laser. [7]

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. This exploration intends

to:

- 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) technique to recreate the gas stream in a MOC spout.

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. The discrete estimate of all the fluxes crossing the surface of the control volume for the indicator step:

$$T_p^{n+1}(Q_{i,j}) = \left[\left(\bar{P}_{i+1,j} \cdot \bar{S}_{i+\frac{1}{2},j} \right) - \left(\bar{P}_{i,j} \cdot \bar{S}_{i-\frac{1}{2},j} \right) + \left(\bar{P}_{i,j+1} \cdot \bar{S}_{i,j+\frac{1}{2}} \right) - \left(\bar{P}_{i,j} \cdot \bar{S}_{i,j-\frac{1}{2}} \right) \right] \quad (1)$$

And for the corrector it is:

$$T_c^{n+1}(Q_{i,j}) = \left[\left(\bar{P}_{i,j}^{n+1} \cdot \bar{S}_{i+\frac{1}{2},j} \right) - \left(\bar{P}_{i-1,j}^{n+1} \cdot \bar{S}_{i-\frac{1}{2},j} \right) + \left(\bar{P}_{i,j}^{n+1} \cdot \bar{S}_{i,j+\frac{1}{2}} \right) - \left(\bar{P}_{i,j-1}^{n+1} \cdot \bar{S}_{i,j-\frac{1}{2}} \right) \right] \quad (2)$$

Considering the forward and backward discretization the scheme is as follows.

Predictor:

$$Q^{(1)} = Q^{(0)} - \frac{\Delta t}{V_{i,j}} \left[T_p(Q^{(0)}) - Da(Q^{(0)}) \right], \quad (3)$$

Corrector:

$$Q^{(2)} = Q^{(0)} - \frac{\Delta t}{V_{i,j}} \left[T_c(Q^{(1)}) - Da(Q^{(1)}) \right], \quad (4)$$

Update:

$$Q^{(n+1)} = \frac{1}{2} \left[Q^{(1)} + Q^{(2)} \right] \quad (5)$$

The explicit MacCormack scheme, even though it is an

upwind method, does not implicitly introduce a sufficient amount of artificial dissipation. Thus, it is necessary to explicitly add it. Without these terms the code diverges. [8]

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

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

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.[9]

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

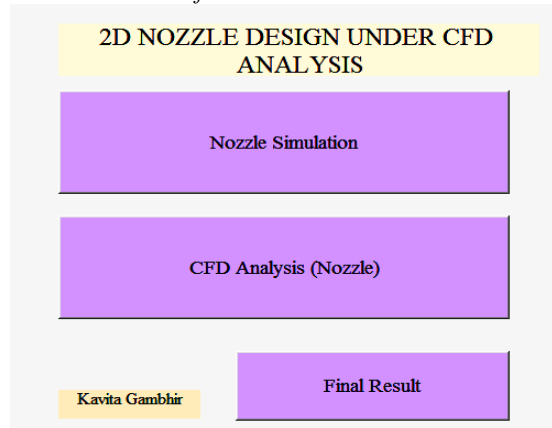


Fig 5.1 First GUI

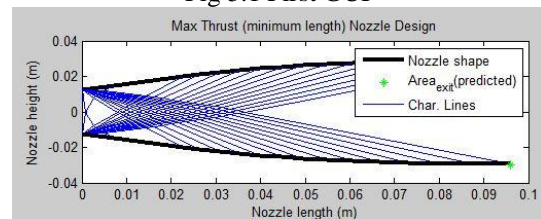


Fig 5.2 Maximum Thrust Nozzle Design

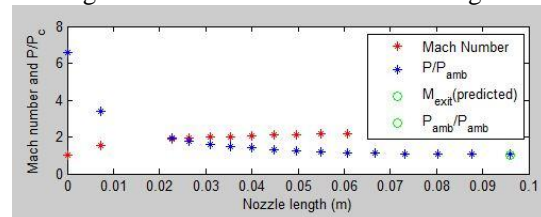


Fig 5.3 Mach number and P/P vs Nozzle Length

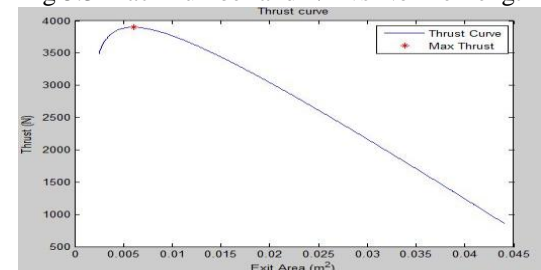


Fig 5.4 Static thrust vs Exit Area

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.

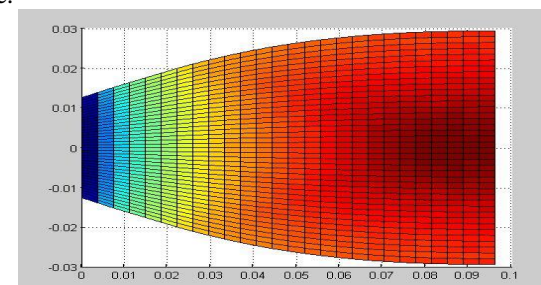


Fig 5.5 Nozzle Design and Plots of Mach Number (Pressure vs Length)

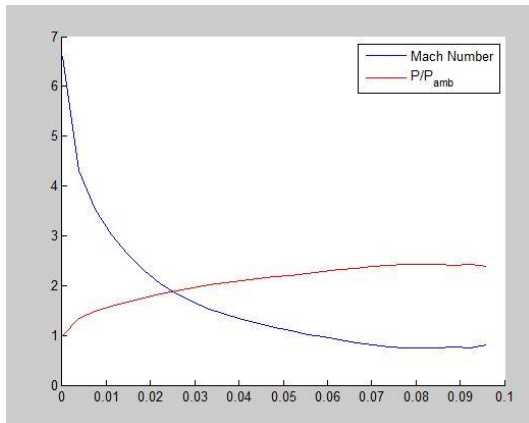


Fig 5.6 CFD Simulation of Designed nozzle

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. Another advancement, will be carried out in constructing 3-D simulation of nozzle in MATLAB.

REFERENCES

- [1] C. Xuyi ; Sun Yat-sen University, No.135, Xinggang Xi Road, Guangzhou, 510275, P. R. China X. Zhang “Coupled analysis of the film-cooling and infrared characteristics of an axisymmetric vectored 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. Prysiaznyi “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 – 4562 978-1-4244-9436-112266840Hohhot10.1109/MACE.2011.5988022IE EE.
- [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] Sibendu Som, 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/