Design of Minimum length nozzle by Method of Characteristics

Sudhanshu G Ogalapur
Lovely Professional University

Anuj kumar, 2020, 8:6
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

Design of Minimum length nozzle by Method of Characteristics

Anuj Kumar, Sudhanshu G. Ogalapur
Department of Aerospace Engineering,
Lovely professional University, Punjab, India

***Abstract*- In this research paper we will design a minimum length nozzle using method of characteristic. Method of Characteristic is a well-known method for designing Nozzle.Here, we will utilize method of characteristics (MOC) to develop a nozzle contour, by reducing length of expansion region of a nozzle. This method works with mathematical concept of Prandtl Meyer expansion fans to shrink expansion zone to a single point. The obtained nozzle would be known as a minimum length nozzle .In a minimum length nozzle, flow expansion will take place while passing through a Prandtl Meyer expansion fan originating from a sharp corner. Minimum length nozzle would be lighter and more efficient for operations. We will write a Matlab function to carryout calculations which cannot be done by hand. This matlab function will plot all the characteristic and contour wall points, the produced wall points would be written to a .dat file, which then can be used to create nozzle geometry in solid works or Ansys Space Claim. We will use Ansys Fluent (which a very successful commercial CFD package) to verify our results, by running a CFD simulation on nozzle. We will check whether the designed nozzle is working properly or not. We will also elucidate algorithm for the Matlab function in more simple words. This is going to be an iterative process so that we can have a better results. The obtained results might be inconsistent on a given inputs, so it is necessary verify the proper working of nozzle through a CFD study.**

Keywords: - Bell nozzle, Prandtl Meyer, expansion fan, CFD, minimum length nozzle.

I. INTRODUCTION

Rockets have now become a very common term. Rocket technology has evolved into something which isvery open source. Small startups, student organizations and hobbyist are now coming up with new innovation in the field of rocket science. The problem of designing a rocket is multidisciplinary, in this thesis we will restrict ourselves to nozzle only. Nozzle is one of the most important device in a rocket which should be designed separately, a nozzle uses thermal energy of exhaust gases to accelerate the flow to higher escape velocity. So ultimately nozzle decides the overall vehicle efficiency. There are different categories of nozzle available today, simplest one is De-Laval C-D nozzle which was devised by Karl Gustaf De Laval a Swedish engineer for use in steam turbines. A simple C-D nozzle is easy to manufacture but it is very inefficient due to various reasons. Various attempts were made to make C-D nozzles more efficient, it turns that bell nozzle are more efficient ,also effective mass of nozzle can be reduced by reducing nozzle length. In a simple C-D nozzle you will find a straight-line wall which won't have any kind of bulge, on the other hand Bell nozzles have bell shaped wall, which is very effective in turning the direction of flow parallel to the axis of nozzle with minimum losses. It is highly desirable to increase thrust efficiencies and total specific impulse of rocket motor to gain highest possible amount of performance from a rocket. We can do this either by optimizing our chamber conditions or nozzle's geometry. In a nozzle we have two section s:-

- Expansion region
- Straightening region

As the name depicts expansion region is the area where supersonic flow gradually expands to give flow which parallel to nozzle axis in straightening section. This expansion generally take place by the reflection of expansion waves from the nozzle wall. Exhaust is accelerated as it passes through this reflected waves.

To designsuch contour we need techniques like MOC, although various other techniques are available which can be used as an alternative to MOC.But MOC is most widely used method for designing supersonic nozzle with shock free - isentropic flow.

II. METHODOLOGY

Method of characteristic has to be implemented as a computer algorithm. For that we will first write a pseudo code, from which we will develop our algorithm. For writing algorithm we can choose any computer language, here we will use Matlab because it has rich set of predefined function for plotting and writing output files. Plotting functions can be used for visualizing the working of our computer algorithm.

Our matlab program will input exit Mach number, number of characteristics, specific heat capacity and output a .dat file containing x,y co-ordinates of nozzle wall ,dat file format is supported by most of the CAD environment like solid works, fusion 360,SpaceClaim.This .dat file will be used for making a 2d geometry for CFD analysis on Ansys Fluent for inspecting nozzle design.We will consider nozzle wall as a single curve only, the entire nozzle geometry can be generated using revolve tool.

III. METHOD OF CHARACTERISTICS

MOC can be used to deal with any hyperbolic partial differential equation.it devises a curve also called characteristics curve along which the PDE would behave as a simple ODE(Ordinary differential equation),the result is obtained by solving this ODE along the characteristic curve and then it is substituted with the PDE to derive the solution. As we know that governing equation for steady supersonic flow fields are hyperbolic and method of characteristics is applicable to solve these kinds of differential equations. If we want to calculate fluid properties at a point a, which can be assumed as the intersection point of characteristics lines or Mach waves originating from two point's b, c the properties at b,c should be known prior to calculation.

By making use of different compatibility equation which can be used to relate direction and magnitude of velocity at point a with the velocity at point b, c. The problem can be solved for entire fluid flow field. But here we are only interested in finding contour points of nozzle wall.

Form physical point of view a characteristic curve can be defined as:-

- A characteristic is defined as the path along which disturbance s are propagated in a supersonic flow at local speed of sound.
- A characteristic is a curve along which the derivatives of a physical quantity may be discontinuous while the property may itself remains continuous.
- A line along which PDEs representing a physical quantity or phenomenon can be treated as Odes. Which can later be solved easily.

In Method of Characteristics we treat expansion fan or Machwaves produced in a nozzle as characteristics lines, along these characteristic lines disturbances are propagated at the local speed of sound.

We can design a nozzle having minimum length by plotting intersection points of these characteristics lines after finding their orientation with respect to nozzle axis by using Prandtl Meyer compatibility equation. Expansion waves exist in a supersonic flow, can be defined as a collection of Mach waves, each of these Mach waves has a different Mach number, and the orientation of these Mach waves is governed by Prandtl Meyer equations. To explain expansion fans, consider supersonic flow at the corners of a duct.

IV. GOVERNING PRINCIPLES

1. Isentropic Turns from Infinitesimal Shocks:

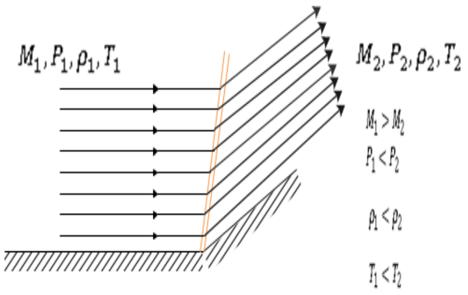

Fig 1. Expansion Fan.

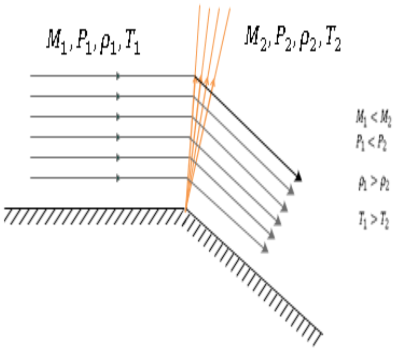

Fig 2. oblique shockwave (red).

Flow has been marked by blue lines, black lines show the boundary of the duct.

You can observe the behavior supersonic flow when it encounters an obstacle, there are two corners one is a concave corner another one is a convex corner, In a concave corner a oblique shock wave would be produced which will direct the flow away from the duct wall, while in a convex type corner an expansion fan would be generated, demonstrating a completely different behavior [5],the flow will expand ,parameters including pressure, Mach number would increase while the density would decrease downstream.

It should not be confused with a normal shock waves, because velocity generally decreases, this can be validated by using equation of continuity ($\rho A v$). These types of flows are called Prandtl Meyer flow, as we are dealing with adiabatic flows isentropic process, the loses are reduced to zero, which makes this process completely reversible. At a smooth concave turn we will have Prandtl Meyer compression and at a convex corner we will have Prandtl Meyer expansion.

This behavior is effectively used to turn the exhaust gaseous flow when the flow passes through the throat, making it parallel to nozzle axis with minimum losses.

$$dv = \sqrt{\frac{(M^2-1)dV}{V}}$$ .....equation 1

Where,

dv : the angle through which the flow is turned

dV: change in velocity when flow passes through expansion fan

V: is flow velocity before it enters expansion fan

M: initial flow Mach number

Equation 1 relates changes in Mach number with the flow turning angle. To simplify equation 1 further we will try to express d V/V in terms of flow Mach number[6]

Which is

$$\frac{dV}{V} = \frac{1}{1+[\frac{y-1}{2}]M^2}\frac{dM}{M}$$

Put this in equation 1, we will get

$$dv = \frac{\sqrt{(M^2-1)}}{1+[\frac{y-1}{2}]M^2}\frac{dM}{M}$$ ....equation 2

On integrating equation 2, we will have value of v in terms of Mach number and γ

$$v(M) = \sqrt{\frac{y+1}{y-1}}tan^{-1}\sqrt{\frac{(y-1)(M^2-1)}{y+1}} - tan^{-1}\sqrt{M^2-1}$$

Which reveals that d v = f (M,γ)

Hence we have derived Prandtl Meyerfunction[6]

2. Compatibility equations:

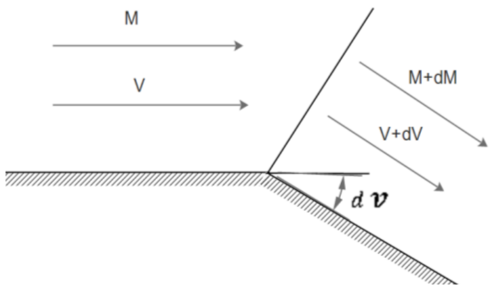

Fig 3. Expansion Fan.

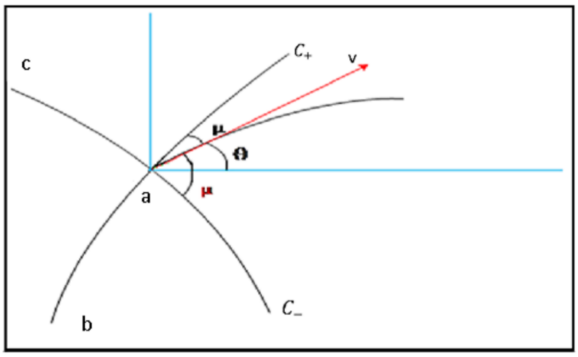

Fig 4. Expansion Wave.

$$\frac{dy}{dx} = tan\ (\theta \pm \mu)\ \ v_x = v\cos\theta$$

$$\sin\mu = \frac{1}{M}\ \ v_y = v\sin\theta$$

$\theta + v(M) = C\_$ Forleft running characteristics $C\_$ [2]

$\theta - v(M) = C_+$ For right running characteristics C_+ [2] Vector v (marked as red) defines the direction of velocity of fluid flow.

These equations are used to relate orientation of characteristics with flow parameters.

- θ: the absolute value of angle of stream line with the coordinate axis(the angle between velocity vector and nozzle's axis)
- K: Constant along each characteristics (called Riemanninvariants).
- ν(M): Prandtl-Meyer function, M is the mach number of the flow given as;

$$v(M) = \sqrt{\frac{y+1}{y-1}} tan^{-1}\sqrt{\frac{(y-1)(M^2-1)}{y+1}} - tan^{-1}\sqrt{M^2-1}$$

Let's assume that flow makes angle θ_a at point a, θ_b at point b, θ_c at point cand v_a, v_b, v_c be Prandtl Meyerangle at point a,b,c.

We can use above equation to evaluate these parameters for point c.

$\theta + v(M) = C\_$ **(**Along left running characteristic**)** [2]

$\theta - v(M) = C_+$ **(**Along right running characteristic**)** [2]

Now at point c we will take v_c,θ_c along $K\_$ and K_+**,** in order to have two linear algebraic equations

$$\theta_c + v_c = C\_$$
$$\theta_c - v_c = C_+$$

Now we can solve them by simple elimination method to get the valuev_c,θ_c-

$$\theta_a + v_a = \theta_c + v_c$$
$$\theta_b - v_b = \theta_c - v_c$$

We will get :-

$$\theta_c = \frac{[(\theta_a + v_a) + (\theta_b - v_b)]}{2} = \frac{[(C\_)_1 + (C_+)_2]}{2}$$

$$v_c = \frac{[(\theta_a + v_a) - (\theta_b - v_b)]}{2} = \frac{[(C\_)_1 - (C_+)_2]}{2}$$

Now we will apply this concept to our nozzle to find value of $\theta_{\max}$ which is the maximum wall angle it decides wall geometry [3]

consider two characteristic AC and CB

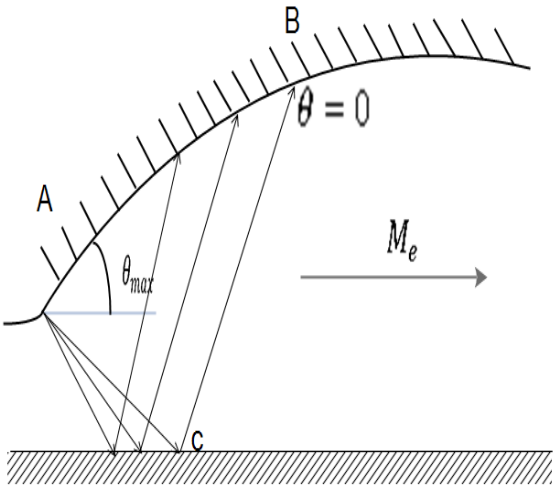

Fig 5. illustration of expansion fans in nozzle.

On CA, we will have following compatibility equation

$$\theta_C - v_C = \theta_B - v_B$$

$\theta_B, \theta_c = 0$ (Since flow is parallel to nozzle axis, so this depicts that

$v_B = v_c = v_{Me}$ (M_e Exit Mach number)

On CB, we will have –

$$\theta_a + v_a = \theta_c + v_c$$

$0+v_c=\theta_{max}+v_A$

3. Prandtl Meyer angle:

$\theta_A=v(M_2)-v(M_1)$, where $v(M_1)=0$,because value of Prandtl Meyer function is 0 at mach 1.

$\theta_A=v_A-0$

So, $\theta_A=v_A=\theta_{max}$

Finally we will have,

$$\theta_{max}=\frac{v_{Me}}{2}$$ [2]

V. NUMERICAL ALGORITHM

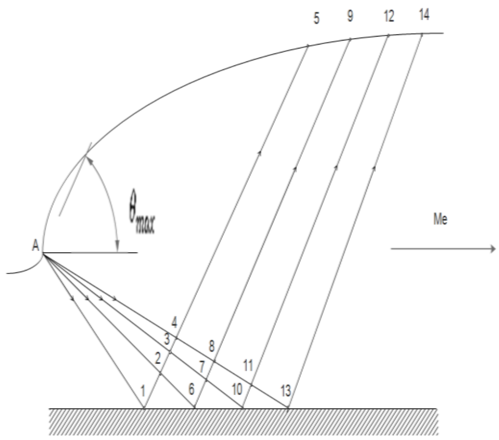

Fig 6. illustration of expansion fans in nozzle with reflection along axis.

We will develop a matlab script to calculate contour points of our nozzle wall. Those points will then be exported to a .dat file which can imported in most of the cad software used today,atlast we will run a CFD simulation to check the effectiveness of our nozzle.

Algorithm-

- Calculate total number of nodes by using n(input from user, number of characteristics),total nodes = $\frac{n(n+2)}{2}$
- Calculate the value θ, v, C_-, C_+ for each node
- Calculate x,y co-ordinates of each single node starting from 1
- Connect the outermost point located on the wall by using average slope formulae.
- Avg, slope between point A and 5 is, Slope (m) = $\frac{\theta_A+\theta_5}{2}$

1. For nodes located on first right running characteristics nodes [1,2,3,4,5]

- Divide θ_{max} into subdivisions $\Delta\theta$,such that θ_{max} = n$\Delta\theta$ θ_i =i$\Delta\theta$ For node 5, $\theta_4=\theta_5$
- $v_i=\theta_i$ (at each nodes)
- $(C_-)_i=v_i+\theta_i$
- $(C_+)_i=v_i-\theta_i$
- i ∈ [1,2,3,4,5]

2. For nodes located on remaining right running characteristics

2.1 For nodes located on nozzle centerline or central axis [6,10,13]

- $\theta_i=0$
- Value of C_- would be equal to the value of C_- coming from adjacent point
- $(C_-)_i-\theta_i=v_-$
- $(C_+)_i=v_i-\theta_i$
- i ∈ [6,10,13]

2.2 For nodes located between nozzle centerline and nozzle wall

- C_+ is taken equal to the value C_+ taken on centerline points
- Value of C_- would be equal to the value of C_- originating from point A
- $\theta_i=\frac{(C_+)_i+(C_-)_i}{2}$
- $v_i=\frac{(C_+)_i+(C_-)_i}{2}$

2.3 For nodes located on nozzle wall

- $\theta_i=\theta_{i-1}$
- $(C_-)_i=(C_-)_{i-1}$
- $v_i=v_{i-1}$
- $(C_+)_i=(C_+)_{i-1}$
- i ∈ [9,12,14]

Now we have calculated value of θ, v, C_-, C_+ for each node, now we will calculate mach μ angle for each node by using inverse Prandtl Meyer function. [4]

$$\mu = \frac{1 + A.y + B.y^2 + C.y^3}{1 + D.y + E.y^2},$$

Where,
A=1.3604, B=0.0962, C=-0.5127, D=-0.6722, E=-0.3278

$$y = {}^{2/3}\sqrt{\frac{v}{V_o}},$$

$$V_o = \frac{\pi\left(\sqrt{6}-1\right)}{2},$$ where v is input Prandtl Meyer angle.

After this the only thing remaining is to calculate x, y coordinates of each node and then plotting them to find out nozzle contour. We will use slope formulae for a straight line to formulate equation which will give out coordinates of each point.

$\frac{y_2 - y_1}{x_2 - x_1}$ =slope (m),

Average slope angle for a left running characteristics [2]

is given as, $m_+ = \frac{((\theta_b + \theta_a) + (\mu_b + \mu_a))}{2}$

Average slope angle for a left running characteristic[2] is

given as, $m_- = \frac{((\theta_c + \theta_a) - (\mu_c + \mu_a))}{2}$

We have used matlab to apply above algorithm, with Me (exit mach number), n(number of characteristics to be used for calculation),R_{throat} (throat radius),γ(specific heat ratio) defined by user. The generated points were exported to a .dat file.

VI. RESULT

These are the result obtained from the formulated algorithm. Dimension were taken in mm.Subsequently .dat file was obtained, this files was then imported in solid works to create 2d geometry for CFD analysis. Here is table that denotes output at given inputs, however it was not possible to show wall points in this research paper, so we have shown exit radius ,max wall angle along with area ratio. Depending on the inputs sometimes the output may become irrelevant, so it is advisable to verify given point by using a CFD analysis.

Table 1. Using matlab all the points were plotted against nozzle axis.

Inputs				Outputs		
Me	n	γ	R_{throat}	R_{exit}	θ_{max}	Area ratio
2.5	100	1.4	35mm	92.3078	19.5618	6.9557
3	100	1.4	35mm	148.2647	24.8787	17.9448
3.5	100	1.4	35mm	237.7480	29.2649	46.1421

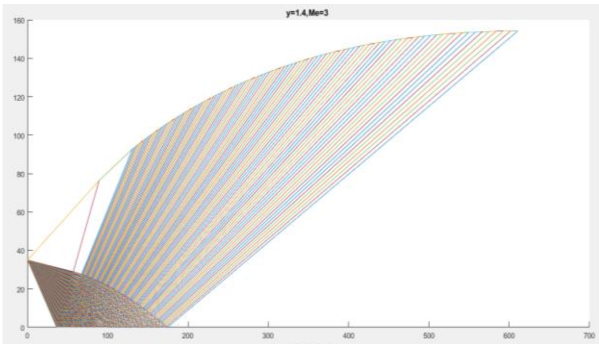

Fig 7. Plot of different characteristics in MATLAB.

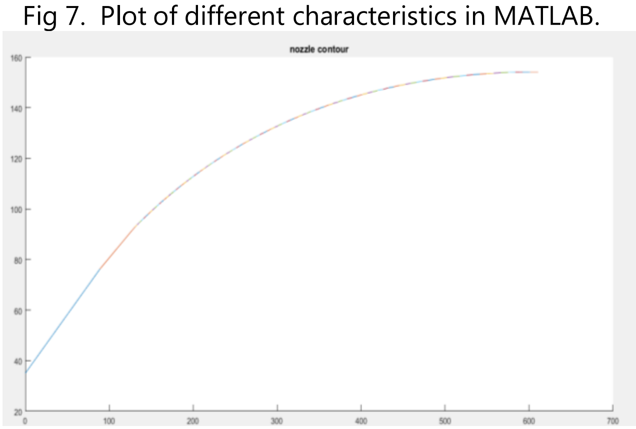

Fig 8. Plot of wall points.

Those points were later used to make a 2d domain for CFD analysis.

Table 2. Boundary Condition

Viscous model	k-epsilon Realizable
Chamber pressure	2.5MPa
Ambient pressure	38365Pa
Chamber temperature	1200K
Ambient temperature	243K

After meshing and preprocessing the results were obtained, these result verified that the nozzle is working properly and efficiently.

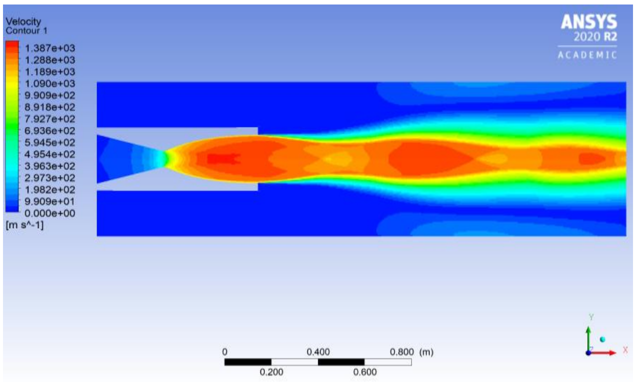

Fig 9. Velocity contour of designed nozzle.

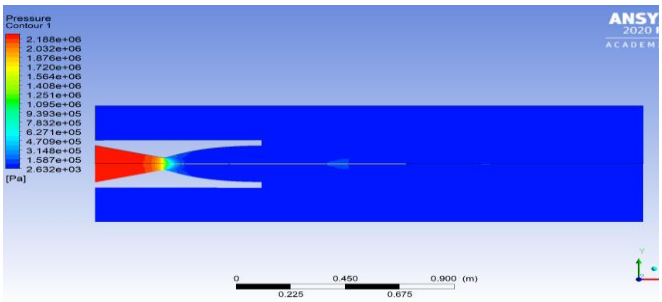

Fig 10. Pressure contour of designed nozzle.

VII. CONCLUSION

We have successfully applied method of characteristics to generate nozzle contour for a minimum length nozzle. The obtained nozzle points can be imported to any CAD environment, for further refinement and manufacturing processes.it is highly recommended to use this method with an iterative design process to obtain best results, in accordance with the input specification. Further you have to select an optimum number of characteristics, number should be large enough to produce a curvy contour (bell shaped) by the use straight lines. The overall effectiveness of MOC can be increased by running multiple iteration our algorithm, with small variations in the input parameters.

REFERENCES

[1] Shapiro, A. H., "Nozzles for Supersonic Flow Without Shock Fronts," Journal of Applied Mechanics.

[2] Anderson, J. D. (2007). "Fundamentals of Aerodynamics", Fourth Edition. New York: McGraw-Hill Education.

[3] Design of a Supersonic Nozzle using Method of Characteristics by Akhtar Khan, Sanjay Kumar Sardiwal, M. Sharath, D. Harika Chowdary , Published 2013.

[4] Analytical Solutions for Prandtl-Meyer Wave – Oblique Shock Overtaking Interaction by Acta Astronautica.

[5] Prandtl-Meyer Expansion Wave by Maciej Matyka.

[6] Robert D. Zucker, Oscar Biblarz - Fundamentals of gas dynamics (2002, J. Wiley).