PERFORMANCE ANALYSIS OF CONICAL SCARFED NOZZLES USING CFD

Harsha Koganti¹, Dr. Manoj T. Nair²
¹Department of Aerospace Engineering
Indian Institute of Space Science and Technology
Thiruvanathapuram, Kerala- 695547
Email: harshakoganti98@gmail.com
²Associate Professor
Department of Aerospace Engineering
Indian Institute of Space Science and Technology
Thiruvanathapuram, Kerala- 695547
Email: manojtnair.iist@gmail.com

ABSTRACT
Scarfed nozzles consist of a basic nozzle followed by an extension nozzle that is sectioned at an angle. Performance variation of these nozzles with respect to geometric parameters was analysed using specific impulse as a measure. Both viscid and inviscid analyses were carried out according to the requirement in order achieve accurate solution using minimum computational time. Results for variation in specific impulse with change in length of nozzle, sectioning angle and expansion ratio were presented. Also, set of viscid analyses were carried out to analyse variation with nozzle pressure ratio.
Keywords: Specific impulse, mass flow rate, scarfed nozzle, nozzle pressure ratio, canted nozzle

NOMENCLATURE
β  Scarf angle
α  Basic nozzle half cone angle
ε  Area ratio
Xp  Nondimensionalised length
δ  Nozzle extension half cone angle

INTRODUCTION
Nozzles find their application in areas where a fluid needs to be accelerated using the pressure energy possessed by the fluid. Rockets, missiles, crew modules are some of them where nozzles are used extensively. Space vehicles are subject to various obstacles during their flight which might cause a great deviation in the trajectory followed. Thus, guidance and control systems are used to ensure that the desired trajectory is followed. Guidance system occupies the initial portion of vehicle followed by propulsion system and nozzle. For a better control, nozzles were canted and placed close to the guidance and control system. Also, in some cases such as the crew modules, off axis nozzles were demanded due to the vehicle configuration. Nozzle part protruding out of the vehicle was flushed along the vehicles skin in order to reduce drag which has resulted in scarfed nozzles. Scarfed nozzles consist of a basic axisymmetric nozzle followed by a scarfed extension. Thrust produced by scarfed nozzles acts at an angle to the vehicle axis. This property can also be used in effective thrust vector adjustment. Moments due to slight difference in thrust produced by nozzles can be avoided using scarfed nozzles by aligning the thrust vector such that it passes through centre of gravity of the vehicle. Thrust generated by the nozzle depends on many factors such as, basic nozzle contour, expansion ratio, scarfed nozzle geometry, motor pressure, ambient pressure etc. These nozzles result in forces in two directions, both contribut-
ing to the thrust along vehicle axis. Due to canting of nozzle, only part of components of forces act along the vehicle axis resulting in thrust loss. It is evident that thrust generated could be controlled using the mentioned parameters. Depending on the application, range of the nozzle operation and requirement of thrust, the geometry of nozzle can be designed. Variation of performance of the nozzle with geometric parameters is also important to be known so as to choose an appropriate design given specifications of performance. This work majorly focuses on implementing a two-dimensional flow model without considering it to be axisymmetric. Variation in performance with some of the geometric parameters such as scarf angle, area ratio was found. CFD++ was used for the corresponding work to perform the analyses.

**LITERATURE**

Literature based on scarfed nozzles performance has been included in this section. Experimental as well as numerical methods were used to understand and compare the performance of scarfed nozzles. This section also includes literature based on flow models, geometries of scarfed nozzles considered. Nozzles occur in various contours and shapes depending on their application. Classification of nozzles was presented by Jon Ostlund(2002). Conical nozzles are the most commonly used nozzles. Though the flow through a conical nozzle is subjected to divergence losses, it is advantageous to use them for lower half cone angles. Ease of manufacturing is the greatest advantage of using conical nozzles[4]. In order to overcome the divergence losses due to conical nozzles, extensive research was done to find an ideal contoured nozzle that results in a uniform exit flow.

J.S. Lilley(1986) considered a particular flow field model assuming it to be axisymmetric within the scarfed extension as well for an analysis on performance of scarfed nozzles. It was based on an assumption that much angle of cone at the exit of scarfed extension was less than the corresponding scarf angle of nozzle. It was also mentioned that in such a case, external flow cannot influence the flow emanating from the scarfied extension. Performance of nozzles can be determined by considering various parameters such as mass flow rate, thrust vector and moment vector as mentioned by J.S. Lilley[1]. Total axial force acting on the nozzle was calculated using separate force calculation in three regions- initial value line, basic nozzle and scarfed extension. Summation of these three forces results in the total axial force acting on the nozzle. Also, the nozzle is assumed to be canted where it is flushed along the skin of the vehicle. Axial force acting on the vehicle was found using corresponding transformation of axes equations. Moment was not considered in the performance since a symmetrical arrangement of the nozzles result in zero moment.

<table>
<thead>
<tr>
<th>Geometric parameter</th>
<th>Values chosen</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\beta$</td>
<td>20$^\circ$</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>10, 20, 30</td>
</tr>
<tr>
<td>$\delta$</td>
<td>0$^\circ$</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>20$^\circ$</td>
</tr>
</tbody>
</table>

**VALIDATION WITH LITERATURE**

This section involves validation of CFD model before using the model for study on scarfed nozzles. Comparison was done with results from a paper by Jay S. Lilley(1986) on performance of scarfed nozzles. Following was the methodology used by Jay S. Lilley(1986). Analysis was done on scarfed nozzles with conical basic nozzle and cylindrical scarfed extension. Flow field was chosen to be axisymmetric in the scarfed extension and flow through the nozzle was solved using method of characteristics. Steady axisymmetric transonic potential flow, supersonic rotational flow, oblique shock equations and rotational flow equations were used as the main governing equations to obtain the flow through the nozzle. Performance parameter was chosen to be specific impulse in the analysis and its variation with geometric parameters was estimated. Similarly, analysis was done using CFD++ software, Metacomp Technologies to validate the results with the analyses performed by Jay S. Lilley(1986).

Nozzle geometric model consisted of a conical scarfed nozzle with a cylindrical scarfed extension as mentioned above. Nozzle in the paper by Jay S. Lilley was assumed to be canted at an angle equal to the scarf angle so that the nozzle remains flushed against the surface of vehicle. Range of values for each parameter was selected and the analysis was performed. Comparison with results from literature was done for three cases where $\beta$ was taken to be 20$^\circ$, $\alpha$ to be 20$^\circ$ and $\epsilon$ was varied by 10$^\circ$ for three cases. Table 1 summarizes the values of each parameter considered from literature.

Jay S. Lilley(1986) used specific impulse as a measure of the performance of scarfed nozzles in order to understand the variation in performance of nozzle with geometric parameters. Similarly, specific impulse was used as the performance parameter in order to validate the results obtained using CFD++.

Analyses in the present study were carried out for two-dimensional axisymmetric flow field, non-axisymmetric inviscid and viscous flow field models. Obtained results were then compared to validate the models chosen.

Thus, from Table 2 and Table 3 it was observed that two-dimensional flow field without axisymmetry produced a result 16% less than the result by Jay S. Lilley(1986). Also, it was observed that inviscid analysis produced a result almost equal to
TABLE 2. Comparison of specific impulse for inviscid and viscid axisymmetric case with result from analysis by Jay S. Lilley (1986) for one nozzle configuration with area ratio 10 for NPR= 68.092

<table>
<thead>
<tr>
<th></th>
<th>Viscid</th>
<th>Inviscid</th>
<th>Literature</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>214.5089 Ns/kg</td>
<td>215.6016 Ns/kg</td>
<td>234.6 Ns/kg</td>
</tr>
</tbody>
</table>

TABLE 3. Comparison of specific impulse for inviscid 2D axisymmetric and only 2D case with result from analysis by Jay S. Lilly (1986) for a nozzle geometric configuration with area ratio 10 for NPR=68.092

<table>
<thead>
<tr>
<th></th>
<th>2D</th>
<th>2D axisymmetric</th>
<th>Literature</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>194.7857 Ns/kg</td>
<td>215.6016 Ns/kg</td>
<td>234.6 Ns/kg</td>
</tr>
</tbody>
</table>

FIGURE 1. Nozzle geometric model that was used by Jay S. Lilley (1986)

that obtained using viscid analysis. Thus, two-dimensional non-axisymmetric flow field model was used with further simulations carried out using inviscid analyses in order to reduce the computational time.

SETUP AND PROCEDURE

Simulations were carried out using CFD++, Metacomp Technologies software. Validation was done by comparing with results generated by Jay S. Lilley (1986). Based on the validation, procedure and various models were adopted for the analyses which are discussed below:

Nozzle geometric model

Figure 1 represents the nozzle geometric model that was used by Jay S. Lilley (1986) which was selected for the current study. Geometric parameters which can be referred to in Fig. 1 were varied in order to analyse the variation in performance of the nozzle. Table 4 can be referred to for the values that were chosen for current study.

TABLE 4. Geometric parameter values adopted from Jay S. Lilley, 1986

<table>
<thead>
<tr>
<th>Geometric parameter</th>
<th>Values chosen</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\beta$</td>
<td>8° - 48°</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>10, 20, 30</td>
</tr>
<tr>
<td>$\delta$</td>
<td>0°</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>20°</td>
</tr>
<tr>
<td>$\rho_{tu}$</td>
<td>25.4mm</td>
</tr>
<tr>
<td>$\rho_{td}$</td>
<td>0.254mm</td>
</tr>
<tr>
<td>$\gamma_t$</td>
<td>25.4mm</td>
</tr>
</tbody>
</table>

Analysis type

2D inviscid and viscid analyses were carried out for various cases. Viscous analyses were carried out for variation in nozzle pressure ratio below the design pressure ratio while inviscid analysis was carried out for remaining set of analyses where the geometric parameters were varied.

Performance parameter

Mass flow rate, thrust vector and moment vector can be used as measure for nozzles in order to estimate their performance. Also, uniform flow at exit can be one of the measures to estimate performance of a nozzle. Basic nozzle chosen to be a conical one eliminates the possibility of uniform exit flow or can be achieved using a cylindrical extension. Therefore, major concern is to achieve better thrust with minimum losses from available possibilities of nozzle geometries. Specific impulse is one parameter that gives a direct measure of performance of the nozzle. $I_{sp}$ was selected as the performance parameter for the current study. Axial force/thrust acting on the vehicle is dependent on the stagnation pressure of propellant gases at the inlet[2]. Also, mass flow rate is dependent on on stagnation pressure which results in specific impulse that remains independent of stagnation pressure. Ambient pressure acts normal to the surface of vehicle which has no influence on the axial thrust. Thus, specific impulse remains independent of stagnation as well as ambient pressure. This parameter suits well for the purpose of understanding variation in performance of nozzle with change in geometric quantities of the nozzle. Specific impulse being independent of the pressures, is solely dependent on the geometric parameters of scarfed nozzles as mentioned by Jay S. Lilley (1986). Table 5 can be referred to for the nozzle operating conditions and gas thermodynamic model used for simulations. Also, angle at which flow exits from the nozzle was calculated for every simulation to
observe the variation in thrust angle.

\[ F_x = \dot{m}_e u_e + P_e A_e \cos \beta - P_a A_e \cos \beta \]

\[ F_y = \dot{m}_e v_e + P_e A_e \sin \beta - P_a A_e \sin \beta \]

\[ m_e = \frac{P_a^* (\frac{2}{\gamma + 1})^{\frac{1}{\gamma - 1}}}{\sqrt{R T_t}} \]

\[ F_X = F_x \cos \beta - F_y \sin \beta \]

\[ I = \frac{F_X}{m_e} \]

\[ I_{sp} = \frac{F_X}{\dot{m}} \]

**Grid generation**

Scarfed nozzles with mentioned geometric parameters and with variation in area ratio were generated along with grid in Pointwise Inc. software. Coordinates of various segments were used in generation of scarfed nozzle model in the software. As mentioned earlier, inviscid analysis was done in order to reduce computational time. Also, both 2D and 2D axisymmetric analyses were done in order to make the conditions more realistic. Wall spacing of 0.0004m was used with 650 \times 200 grid points in the diverging section. Also, the scarfed extension comprised of 650 \times 200 grid points owing to its importance in present study where flow through the scarfed extension is of utmost importance. Domain around the nozzle was considered to be 900y_t \times 120y_t where throat radius was specified to be 25.4mm. Structured grid was generated with the mentioned parameters. Initially, viscous analysis was done for one of the nozzle configurations and then compared with inviscid analysis case. Specific impulse value in both cases were almost same and thus analysis was continued with inviscid case so as to reduce computational time. For the viscous case, \( y^+ \) spacing was considered to be 1 with wall spacing equal to 3.456e-7m in order to capture the boundary layer effectively. Also, better wall spacing was ensured near the regions of formation of shocks.

**CFD Solver**

Viscous analysis was performed by solving the compressible Navier-Stokes equations and using SST k-\( \omega \) turbulence model which was found to be appropriate as mentioned by R. Stark et al.(2007)[5]. Inviscid analysis was done by solving the compressible Euler equations. Thermodynamic properties of the gas were used as mentioned by Lilley(1986). Initial condition was defined to be at atmospheric condition while boundary conditions for each boundary were defined as follows: stagnation temperature and pressure were defined at inlet, inviscid surface tangency boundary condition was defined for all walls, symmetry boundary condition was defined for the axis, back pressure imposition was given to the outlet and finally, static temperature and pressure using inside velocity BC was defined for boundaries indicating atmosphere. For the viscous case, adiabatic viscous wall condition was defined for all the walls. 2D axisymmetric, implicit steady state analysis was performed for all the nozzle configurations. For the 2D inviscid analysis case, boundary conditions used were same except for the symmetry BC which was not included in this particular case.

**Approach used**

On comparison of CFD analysis results obtained for three nozzle configurations with literature results by Jay S. Lilley(1986), an approach was followed for rest of the simulations in order to check for the variation in performance parameter with change geometric parameters. 2D inviscid analyses were carried out by varying \( \beta, \epsilon \) and \( \delta \). A geometric parameter defining the projected length was used in the analyses \( (X_p) \). Normalised projected length is considered to be the ratio of projected length to throat radius. Table 6 refers to the different cases for which simulations were carried out. Also, all the above cases of simulations were carried out for NPR= 68.05 while for one configuration, simulations were carried out for NPR= 7 varying the scarf angle.

---

**TABLE 5. Gas thermodynamic model and nozzle operating conditions adopted from Jay S. Lilley, 1986**

<table>
<thead>
<tr>
<th>( T_t )</th>
<th>( P_t )</th>
<th>( P_a )</th>
<th>( \gamma )</th>
<th>( R )</th>
</tr>
</thead>
<tbody>
<tr>
<td>2777.778 K</td>
<td>6.895 MPa</td>
<td>101352.9 Pa</td>
<td>1.2</td>
<td>349.86 J/kgK</td>
</tr>
</tbody>
</table>

**TABLE 6. Different cases considered for the variation in geometric parameters**

<table>
<thead>
<tr>
<th>S.no</th>
<th>( \delta )</th>
<th>( \epsilon )</th>
<th>( \beta )</th>
<th>( X_p )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0°</td>
<td>10, 20, 30</td>
<td>8° - 48°</td>
<td>variable</td>
</tr>
<tr>
<td>2</td>
<td>0°</td>
<td>10, 20</td>
<td>8° - 48°</td>
<td>constant</td>
</tr>
<tr>
<td>3</td>
<td>0°</td>
<td>10</td>
<td>20°</td>
<td>20-40</td>
</tr>
</tbody>
</table>

**FIGURE 2. Grid generated (a) axisymmetric case (b) non-axisymmetric case (c) complete domain around the nozzle**
FIGURE 3. Illustration of the possibilities considered to perform analysis (a) variable $\beta$ and $X_p$ (b) variable $\beta$ and constant $X_p$ (c) constant $\beta$ and variable $X_p$.

FIGURE 4. Variation of density along wall for five cases of grid size.

GRID CONVERGENCE STUDY

Grid convergence study is important in order to choose the optimum number of grid points in order to generate accurate results provided computational time is reasonably low. Thus, this section is dedicated to the results obtained on performing a grid convergence study. Nozzle geometric configuration with area ratio 20, scarf angle $15^\circ$, $\delta$ of $0^\circ$, and $X_p = 42.913$ was used with initial grid size of 650 points along the length of divergence section and the scarfed extension. Later, the grid points were varied from 350 to 1250 for 5 cases. 2D inviscid analyses were carried out for all five cases. Density distribution along wall and pressure, mach number distribution along centerline were compared for all five cases. Figure 4 represents variation in density along the wall for five grid conditions.

RESULTS AND DISCUSSION

Analyses were performed in order to observe the variation in specific impulse with geometric parameters such as scarf angle, expansion ratio and length of the nozzle. Specific impulse was found to increase with decrease in scarf angle and also with increase in expansion ratio. Analyses were carried out for two cases one in which scarving was done into the basic nozzle and outside the basic nozzle. Also, decrease in specific impulse was observed in case of nozzles which were scarfed into the basic nozzle when compared to nozzles with scarving done outside the basic nozzle. Specific impulse was found to increase with increase in non-dimensional length for a given scarf angle until scarving reached a point outside the basic nozzle and then it was observed to remain constant. All the above analyses were carried out for nozzle pressure ratio of 68.05 which results in an underexpanded nozzle. Few analyses were carried out to understand the variation in specific impulse with change in nozzle pressure ratio. Therefore, analyses were carried out for NPR= 7 and varying scarf angle from $8^\circ$ to $48^\circ$. Defined variation of specific impulse with scarf angle could not be defined due to its dependence on ambient pressure. Thrust angle was observed to increase with decrease in area ratio and initially increases and the decreases with increase in scarf angle. Thrust angle was observed to decrease continuously with increase in scarf angle when length of the nozzle was constrained. On varying the length of nozzle given a scarf angle, thrust angle was found to decrease and then remain constant on reaching a certain value of the non-dimensionalised length.

CONCLUSION AND FUTURE WORK

As discussed in the earlier section, specific impulse variation with geometric parameters like scarf angle, length of nozzle and area ratio was observed for a nozzle pressure ratio of 68.029. Analyses were performed using CFD++ by applying parameters as mentioned in section 3. Thus, specific impulse was found to increase with increase in area ratio and decrease with increase in scarf angle while it was found to increase and then remain constant with increase in non-dimensional length of nozzle.

Future work can be carried out in varying the pressure ratio to find the variation in flow and analysing its variation with geometric parameters. Also, performance parameter becomes dependent on various factors when the flow is not two dimensional.
Optimum geometry of scarfed nozzle can be found for certain ranges of pressure ratio when flow does not turn three dimensional.

ACKNOWLEDGMENT

I sincerely thank my guide Dr. Manoj T. Nair for giving me an opportunity to work in the area of scarfed nozzles under his constant guidance and supervision. I would also like to thank him for his guidance in carrying out simulations in CFD. I sincerely...
thank Dr. B. R. Vinoth for his guidance on analysing various aspects in the field of nozzles. I also thank him for his valuable suggestions which helped in the completion of the study. I thank my senior Chintan Panigrahi for his support and suggestions which helped in completion of the project.

REFERENCES


