CFD ANALYSIS OF CONVERGENT-DIVERGENT AND CONTOUR NOZZLE

Madhu B P*
Assistant Professor, School of Mechanical Engineering, REVA University, Bangalore, India

Syed Sameer, Kalyana Kumar M
School of Mechanical Engineering, REVA University, Bengaluru, India

Mahendra Mani G
Associate Professor, Department of Mechanical Engineering, Government Engineering College, Ramanagaram, Karnataka, India

ABSTRACT

Expansion in C-D nozzle has been studied and analyzed experimentally as well as numerically by various researchers with an objective to optimize the overall performance under given conditions. In the present work supersonic flow through the rocket nozzle has been simulated using numerical method. The parameters like Mach number, static pressure and shocks are observed for conical and contour nozzles using axi-symmetric model in ANSYS FLUENT 14® software. The occurrences of shocks for the conical nozzles were observed along with the other parameters for various divergent angles. The parameters under observation are compared with that of contour nozzle for respective divergent angles by maintaining the inlet, outlet and throat diameter and lengths of convergent and divergent portions as same. The convergent portion and throat diameter are kept constant across the cases. The phenomenon of shock was visualized and the results showed close resemblance in formation of Mach disk and its reflection patterns as reported in various experimental studies on expansion in conical C-D nozzles with lower divergent angles. No occurrence of shocks is observed with higher divergent angles. Results depicted higher exit velocity and higher degree of flow separation with contour nozzles compared to that of with corresponding conical nozzles.

Key words: C-D nozzle, ANSYS FLUENT 14®, rocket nozzle, Mach disk.

http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=8&IType=8
1. INTRODUCTION
Rocket engines are reaction engines that obtain thrust in accordance to Newton’s third law of motion. The majority rocket engines are in the group of internal combustion engines. Rocket engines have the lightest, maximum exhaust velocities, and are the least energy efficient of all types of jet engines.

The fluid exhaust is passed through a supersonic propelling nozzle which converts pressure energy into useful kinetic energy which gives maximum outlet velocity and from Newton’s law the reaction to this pushes the engine in the opposite direction as shown in figure 1.

1.1. Rocket Engine Nozzle
A device used to control fluid flow out of a chamber. In rockets, specifically the nozzle is used to maximize the thrust. Nozzle design constitutes an important phase of rocket development. The performance of a rocket depends heavily on its nozzle’s effectiveness in converting thermal energy to kinetic energy.

2. LITERATURE REVIEW
BijuKuttan P et al Conducted numerical analysis to determine an optimum divergent angle for the nozzle which would give the maximum outlet velocity and meet the thrust requirements. The inlet dimensions and the boundary conditions are kept constant and the divergent angles are varied in order to understand how the variation in divergent angle affects the flow pattern through the nozzle. Among the various models available in Fluent, the k-ε model was selected for their work. A two-dimensional axi-symmetric geometrical model of the nozzle was used for the analysis purpose. Divergence angles chosen were 4°, 7°, 10°, 13° and 15°[1].

C A Hunter conducted experimental, theoretical, and computational study of separated nozzle flows. Experimental testing was performed at the NASA Langley 16-Foot Transonic Tunnel Complex. As part of a comprehensive static recital examination, moment, force and pressure measurements were made and Schlieren flow visualization was obtained for a sub-scale, non axi-symmetric, two-dimensional, C-D nozzle. In addition, two-dimensional numerical simulations were run using the computational fluid dynamics code PAB3D with two-equation turbulence closure and algebraic Reynolds stress modeling. For reference, experimental and computational results were compared with theoretical predictions based on one-dimensional gas dynamics and an approximate integral momentum boundary layer method [2].

Nazar Muneam Mahmood simulated steady flow of a gas through a C-D nozzle which has a varying cross sectional area and showed that the nature of the flow can be explained by considering how the flow and its characteristics in the nozzle changes as the back pressure decreases. The characteristics of gas flow were simulated using the ANSYS Fluent 12.1
software to solve the quasi-one dimensional nozzle flow. According to the study, the reduction in the back pressure cannot affect conditions upstream of the throat, hence the nozzle is choked. The shock wave increases the pressure, density and temperature and reduces the velocity and Mach number to a subsonic value and as back pressure is further reduced to a certain value, the extent of the supersonic flow region increases, the shock wave moving further down the divergent portion of the nozzle towards the exit plane [4].

3. COMPUTATIONAL FLUID DYNAMICS
Computational fluid dynamics is a science that, with the help of digital computers, produces quantitative predictions of fluid flow phenomena based on the conservation laws (mass, momentum and energy) governing fluid flow. These predictions normally occurs under those conditions defined in terms of flow geometry, the physical properties of a fluid, and boundary and initial conditions of a flow.

4. OBJECTIVE OF WORK
The aim of present work is to numerically simulate expansion through convergent-divergent and Contour rocket nozzle using FLUENT to understand various physical phenomena such as occurrence of shocks, formation of Mach diamonds and flow separation associated with the supersonic expansion.

5. METHODOLOGY
Extensive study of existing literature on experimental and numerical analysis of expansion through convergent–divergent nozzles has been done. Nozzle geometry, expansion ratios, flow and boundary conditions have been drawn on the basis of the literature survey. Creation of two-dimensional geometry of the nozzle and meshing is done on Workbench of ANSYS 14®. Simulation of expansion is carried on ANSYS FLUENT 14® software using different levels grid refinement and turbulent models. Grid independence test is carried for each of the cases studied to select appropriate number of elements for the computational domain.

Table 1 Co-ordinates of contour nozzle

<table>
<thead>
<tr>
<th>Coefficients / Divergent Angle</th>
<th>1st</th>
<th>2nd</th>
<th>3rd</th>
<th>4th</th>
</tr>
</thead>
<tbody>
<tr>
<td>4°</td>
<td>1.804</td>
<td>-3.045</td>
<td>0.113</td>
<td>1.001</td>
</tr>
<tr>
<td>13°</td>
<td>-0.030</td>
<td>0.122</td>
<td>0.044</td>
<td>0.251</td>
</tr>
</tbody>
</table>

Validation of simulated results with that of experimental is done and the numerical approach, which resulted in acceptably proximate predictions, is adopted for further study of the cases taken in this work. Post processing features of ANSYS FLUENT are used to generate static pressure, total pressure and velocity contours and Mach number plots for all cases. Parameters that are essential for detailed analysis of expansion like flow exit velocity, pressure at exit and position of occurrence of shocks are obtained from post processor generated contours and plots.

5.1. Geometrical Description
A two-dimensional modeling of the nozzle has been formed using Workbench ANSYS 14. Diameters at inlet, throat and exit have been maintained same for all cases considered in this work. Dimensions of the conical nozzle for 15° of divergent angle are given in the table 2.
Table 2 Dimensions of the conical nozzle for 150 of divergent angle

<table>
<thead>
<tr>
<th>Particulars</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet width (Diameter) (m)</td>
<td>1.000</td>
</tr>
<tr>
<td>Throat width (Diameter) (m)</td>
<td>0.304</td>
</tr>
<tr>
<td>Exit width (Diameter) (m)</td>
<td>0.861</td>
</tr>
<tr>
<td>Throat radius of curvature (m)</td>
<td>0.228</td>
</tr>
<tr>
<td>Convergent Length (m)</td>
<td>0.640</td>
</tr>
<tr>
<td>Convergent angle (deg)</td>
<td>30</td>
</tr>
<tr>
<td>Divergent angle (deg)</td>
<td>15</td>
</tr>
</tbody>
</table>

Table 3 Meshed Descriptions for Different Divergence Angles

<table>
<thead>
<tr>
<th>Div angle</th>
<th>Type of element</th>
<th>Number of nodes</th>
<th>Number of element</th>
</tr>
</thead>
<tbody>
<tr>
<td>4˚</td>
<td>Quad</td>
<td>19113</td>
<td>18776</td>
</tr>
<tr>
<td>13˚</td>
<td>Quad</td>
<td>180005</td>
<td>179401</td>
</tr>
</tbody>
</table>

5.2. Boundary Conditions

Inlet - Mass flow inlet boundary conditions are used to define the flow fluid along with all relevant scalar properties of the flows, at flow inlets. Mass flow rate for all the cases chosen has 200 kg/sec and temperature at the inlet is 3400 K. Outlet - Pressure outlet boundary conditions are applied to the outlet. Gauge pressure at the outlet becomes zero.

Axis - Axis boundary condition Wall - Wall boundary condition –No slip condition.

5.3. Validation

![Figure 2 Comparison of numerical data with experimental using k-ε model](image)

Before simulating expansion process through nozzles for different cases developed with various geometrical configurations for the analysis, numerical approach for simulation has been validated with well accepted experimental results. Various turbulent models were tried with various grid densities to tune up the approach that would result in most proximate predictions with the experimental results.

Experimental results cited by C.A. Hunter et al on AIAA [2] have been used to validate the numerical approach adopted in this work. Experiments were conducted on conical nozzles.
for different pressure ratios. Plots depicting the variation of non-dimensional pressure against non-dimensional distance on the axis from the inlet of the nozzle have been used for the validation. Expansion had been simulated for same NPR for conical nozzle as of experimental adopting both k-ω and k-ε turbulent models.

6. RESULTS AND DISCUSSION

6.1. Case1: Divergent angle=4° -Conical Nozzle

![Mach Number Contours](image)

Figure 3 Mach number for conical nozzle with divergent angle 4°

Figure 3 shows the Mach contour of the CD nozzle for divergence angle 4°. From the figure it is obvious that shock is observed at the throat area. Also it is observed that velocity increase from inlet to outlet continuously and valued of Mach number is at the inlet 5.34e-02 Mach at the throat is 1.00e+00 Mach and at the exit it is found the value of 2.16e+00 Mach.

6.2. Static Pressure Contour

![Static Pressure Contours](image)

Figure 4 Static Pressure for conical nozzle with divergent angle 4°

The static pressure contour shows a reduction in the static pressure throughout the nozzle. At the inlet, the static pressure is found to be 8.42e+05Pa. At the throat it has reduced to
5.73e+05Pa. This value again reduces to a value of -9.07e+03Pa and remains constant till the exit section.

**Figure 5** Plot of Mach number v/s Position for conical nozzle with divergent angle 4°

The Mach number V/s position plot also shows a continuous increase in the velocity from inlet section to the outlet section. And there is no sudden drop in velocity which shows that no shock is occurring in the nozzle. The exit Mach number of the nozzle is 2.16e+00 Mach at the axis.

### 6.3. Case 2: Divergent angle=4° -Contour Nozzle

**Figure 6** Mach number for Contour Nozzle with Divergent angle 4°

The variation in the Mach contour with increase in the divergent angle from 4° to 7° can be observed from figure 6. Here it is noticeable that only one shock has occurred inside the divergent section. The inlet section has a velocity of 4.64e-02 Mach. At the throat the velocity varies from 9.23e-01 and 1.07e+00Mach. across the shock the velocity drops from 2.53e+00 Mach to 1.51e+00 Mach. The velocity again increases towards the exit of the nozzle. The exit velocity is found to be 2.68e+00 Mach along the axis of the nozzle. The variations in velocity along the walls of the nozzle are due to the viscosity effects.
Figure 7 Static Pressure for contour nozzle with divergent angle 4°

From Figure 7 the pressure dropped from $9.71 \times 10^5$ to $6.97 \times 10^5$ Pa at the throat section. Up to the position of shock pressure decreases to $9.42 \times 10^4$ and suddenly increases to $2.59 \times 10^5$ pa. Further it reduces to $-1.54 \times 10^4$ at the exit.

Figure 8 Plot of Mach number v/s Position for Contour Nozzle with Divergent angle 4°

The position of shock can be found the Mach plot as in figure 8. It is observed that the shock occurs at 1.50 m from the inlet. It is thus found that the shock has displaced by about 0.3 m as the divergent angle increased from 4° to 7°.

7. CONCLUSIONS

- Validation of numerical results show that even by restricting the computational domain to exit of the nozzle acceptable results can be obtained.
- From the Mach number graph it is shows that velocity increases from inlet to outlet.

REFERENCES


http://www.iaeme.com/IJMET/index.asp editor@iaeme.com


