Flow Investigation of an Axisymmetric Single and Multi Jets

Padmalochini. S¹, Sankar.V², Karthikeyan.L.M³
¹(Aeronautical Department, Nehru Institute of Engineering and Technology, Coimbatore, India, lochin20@gmail.com)
²(Head of the Department, Aeronautical Engineering, Nehru Institute of Engineering and Technology, Coimbatore, India)
³(Department of Aeronautical Engineering, Techno Global University, India)

Abstract—Computational studies have been made for axisymmetric free jets exhausting from sonic and supersonic nozzles into still air. When considered for still air atmosphere, the jet plume characteristics such as effects of jet Mach number, jet wavelength and shape and curvature of the jet boundary are analysed for varying nozzle divergent angle (5, 10 degree) and jet pressure ratio. The primary variables considered in jet plume structure for jet exhausting into still air are initial inclination of jet boundary, location of first normal shock or Riemann shock wave and its diameter, jet Mach number. For computational calculation of jet plume structure, Gambit and Fluent software is used. The axisymmetric nozzle is modelled and meshed using Gambit. The boundary conditions are specified and jet plume structure is analysed in Fluent. Grid Independent techniques are used to compare the results. The results are then compared to multi jets

Keywords—jet plume structure; Riemann Shock wave; sonic and supersonic nozzles;

1. INTRODUCTION

The hot exhaust gases coming from a highly under expanded nozzle expand rapidly, forming a plume with a very complex structure. Though very little reaction takes place within the jet, the aerodynamic structure of hot gases is complicated by internal shocks. As a result, various problems arise depending on the vehicle altitude and the proximity of the rocket nozzle to various adjacent surfaces. This may be either from the main vehicle or from surfaces perpendicular to the jet blast, such as being encountered during lunar landings and take-off. For flights of missiles within the atmosphere, jet plumes can produce flow separation over the missile afterbody, which in turn affects the stability characteristics.

In the viscous region along the periphery of the plume, considerable reaction does take place. Since the thickness of the mixing region is small compared to the diameter of the jet, at least near the nozzle, viscous effect is neglected in considering the aerodynamic structure of jet plumes

2. METHODOLOGY OF AXISYMMETRIC JETS

First, the jet is taken to be an ideal gas, and the internal structure and boundary are calculated. Second, the mixing problem with chemical reaction is considered, with one fluid being the external flow and the other being the jet fluid at those conditions occurring along the idealized boundary.

To simplify the problem, first calculations usually involve the assumption that the external fluid is at rest, so that a constant-pressure boundary condition results. The effects of altitude may still be included by varying the external pressure, but effects due to the velocity of the vehicle are neglected. Presumably, methods devised for the ambient atmosphere case may be extended to cover the case of an external with a relative velocity.

In this report, the aerodynamic structure is considered. An invisid fluid with constant specific heats is assumed to expand from a rocket nozzle into an ambient atmosphere. Computational solutions to the problem of calculating the jet boundary and internal shock structure are presented. Such solutions are important because they will allow accurate estimates to be made of the growth and development of the jet structure as altitude, fluid, and nozzle-exit conditions are varied.

3. COMPUTATIONAL PROCEDURE

The coordinates for the nozzle geometry is taken from the literature study [1] for various nozzle divergence angle. Using these coordinates, the sonic and supersonic nozzles are modeled with domain region to visualize the jet plume structure outside the nozzle. In this paper, modeling is done through GAMBIT.

The domain region is created with its length 10 times the nozzle diameter and domain height 5 times the nozzle diameter. Since we are considering axisymmetric nozzles and domain, they are drawn in 2D with a centre line, which is drawn as axisymmetric condition.

Then the appropriate boundary conditions are applied for the nozzle and domain. These regions are gone through meshing. Here we use structured mesh (quad mesh) for accurate results. This mesh file is saved as .msh file and exported to the analyzing software.

The analyzing software used is FLUENT which is used for post processing i.e., for seeing results. The mesh file is imported and the reference conditions (atmospheric pressure, temperature) are given. The inlet pressure and temperature also provided and the turbulence model for the flow is mentioned. Since unsteady flow is considered, time step should be small to give precise results and number of iterations should be provided appropriately and results are analyzed.

The procedure is repeated for various nozzle exit angles and the structure of jet plumes is studied.

4. MODEL DESIGN

The material used for nozzle is steel. The air at ideal gas is used for flow analysis. A cylindrical domain is created to capture the jet plumes structure with location and interference of shock wave (or expansion waves).
A. **Single Jet**

- The design geometry of nozzle is taken from the literature survey “Experimental and Theoretical Studies of Axisymmetric Free Jets” – NASA TR-R6. The word "data" is plural, not singular.
- The nozzle divergence angle for supersonic nozzle is taken as 5 degrees and for sonic nozzle it is taken as 0 degree.
- Both the throat area and exit area changes with change in divergence angle, which alters the exit Mach number and the structure of jet plumes.
- The coordinates are then plotted in gambit and 2-D nozzle is drawn as shown.

![Fig. 1: 2-D Nozzle](image)

B. **Multi Jets**

- Two nozzle with divergence angle 5° is taken and kept at centre to centre distance 0.0595 metres between the nozzle.
- The distance between the nozzle is taken with reference to c/d ratio = 1.75, from the paper "CFD Analysis of Twin Jet Supersonic Flow with Fluent Software".
- The coordinates are then plotted in gambit and 2-D nozzle is drawn as shown.

![Fig. 2: Multijet Nozzle](image)

5. **TECHNIQUES USED**

The model is meshed using Gambit with high precision. The mesh used is structured mesh (quad mesh) and the mesh is clustered inside the nozzle and at center part of the domain to capture the jet plume structure.

![Fig. 3: Face Mesh-Structured mesh(quadratic elements)](image)

6. **FLUENT ANALYSIS**

A. **Single Jet**

The plume structure is analysed and the inclination of jet boundary from the exit is found to be 45° for 5° nozzle divergent angle, which is shown in velocity contour.

![Fig. 4: Velocity Contours Of 5° Divergent Angle](image)

**TABLE I MODELS AND SETTINGS**

<table>
<thead>
<tr>
<th>Model</th>
<th>Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Space</td>
<td>Axisymmetric</td>
</tr>
<tr>
<td>Time</td>
<td>Unsteady, 2nd-Order Implicit</td>
</tr>
</tbody>
</table>

**TABLE II INLET SETTINGS**

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gauge Pressure (pascal)</td>
<td>3447000</td>
</tr>
<tr>
<td>Supersonic/Initial Gauge Pressure (pascal)</td>
<td>3435280</td>
</tr>
<tr>
<td>Total Temperature (k)</td>
<td>9780</td>
</tr>
<tr>
<td>Axial-Component of Flow Direction</td>
<td>1</td>
</tr>
<tr>
<td>Radial-Component of Flow Direction</td>
<td>0</td>
</tr>
</tbody>
</table>

**TABLE III PRESSURE FARFIELD SETTINGS**

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gauge Pressure (pascal)</td>
<td>101325</td>
</tr>
<tr>
<td>Mach Number</td>
<td>0</td>
</tr>
<tr>
<td>Temperature (k)</td>
<td>300</td>
</tr>
<tr>
<td>Axial-Component of Flow Direction</td>
<td>1</td>
</tr>
<tr>
<td>Radial-Component of Flow Direction</td>
<td>0</td>
</tr>
</tbody>
</table>

**TABLE IV OUTLET SETTINGS**

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gauge Pressure (pascal)</td>
<td>101325</td>
</tr>
<tr>
<td>Backflow Total Temperature (k)</td>
<td>300</td>
</tr>
<tr>
<td>Backflow Direction Specification Method</td>
<td>1</td>
</tr>
<tr>
<td>Axial-Component of Flow Direction</td>
<td>1</td>
</tr>
<tr>
<td>Radial-Component of Flow Direction</td>
<td>0</td>
</tr>
</tbody>
</table>

Wall boundary conditions are taken as no slip condition and non-viscous flow is considered.
The plot with Mach number in Y axis and axial distance from nozzle tail pipe in X axis with unit metre is shown below.

The jet plume structure for 0° nozzle divergent angle is shown in velocity contours.

Comparison is made for both nozzle divergent angles, red color indicates the plot of 0° nozzle divergent angle and white color indicates the plot of 5° nozzle divergent angle.

B. Multi Jets

Jet structure and its interaction vary with the distance between the jet nozzles, variation in nozzle divergence angle, jet pressure ratio, and change in ratio of specific heats.

Comparison is made for both nozzle divergent angles, red color indicates the plot of 0° nozzle divergent angle and white color indicates the plot of 5° nozzle divergent angle.

7. GRID INDEPENDENT STUDY

As the number of grid points increased, the truncation error is reduced, the error in the numerical solution would decrease and the agreement between numerical and exact solution would get better. Thus, a grid can be made finer and finer for better results. But however fine grids will give acceptable results, there is always a gap on number of node points due to processing capability limitation. So, different grids with varying number of node points are taken. When the numerical solutions obtained on different grids agree to within a level of tolerance specified, they are referred to as “grid converged” solution. The numerical solution becomes independent of grid as cell size is reduced. It is very important that the effect of grid resolution on the solution is investigated in every CFD problem.

As seen in figure 9, light blue and dark blue plots are in close agreement. Therefore optimized values of node points are 16004, and faces are 47373, where our CFD problem becomes independent of grid for further reduction of cell size.

8. CONCLUSION

• The Jet Plume structure varies with variation in nozzle divergence angle.
• Nozzle Divergence angle have small effect upon the primary wavelength of the jet, i.e., distance of normal shock wave (Riemann shock wave) from nozzle exit.
• Effects on inclination of jet boundary, jet mach number, existence, location and interference of shock (or expansion waves), diameter and distance of normal shock wave (Riemann wave) from nozzle exit in jet plumes are studied.
• When nozzle divergence angle increases, inclination of jet boundary increases.
• The results obtained are compared with the literature survey and further the procedure is continued for multi jets.
• Grid independent study is made and optimized value are found for our CFD problem.

ACKNOWLEDGMENT

There are few people whom I want to be very grateful and mention my sincere thanks individually. First of all, I would like to heartily thank our CEO & secretary Dr.P.Krishnakumar, Nehru Group of institution, who provide
us a platform to do my ME thesis work successfully. I would like to thank our respective Principal Dr. P. Maniarasan whose motivation and enthusiasm helped to complete this project work. It is a great pleasure to express my deep sense of unbound gratitude and sincere thanks to Dr. Sankar, Head of the department, department of aeronautical engineering, whose esteemed support and incessant encouragement enabled me to complete this thesis work successfully. I am extremely thankful to Mr. S. Senthilkumar, Assistant Professor, Aeronautical department who supported me in completing my project.

REFERENCES


