Numerical Investigation of under-expanded Missile Exhaust Plume Behavior

Shantanu Sharma, Manu Mohan, Sameer Karania, Satya Prakash
ARD&P Directorate, Aeronautical Development Agency, Bangalore, India
Email: satya@jetmail.ada.gov.in, karania@jetmail.ada.gov.in, shantanu@jetmail.ada.gov.in

1. Introduction:

For a number of decades numerous efforts have been put in understanding the exhaust plume behavior. Such a flow physics can be observed in powered missiles and rockets. While the plume structure has an effect on the stability and performance of missiles itself, it also affects the body, such as an aircraft from which the missile has been fired, coming in its wake. Ingestion of the exhaust plume by the air intakes is one such example in which the performance of the engine can get severely affected if the temperature of the ingested gases are higher than the acceptable value. All these and numerous other reasons call for an in-depth analysis of plume, which in itself has a complicated flow physics. Several aerodynamic complexities arises due to the fact that these exhaust jets are highly under-expanded which means the nozzle has not been able to expand the flow to match the ambient condition, and the pressure at the exit of the nozzle is higher than the ambient pressure. This gives rise to complex phenomenon like shock diamonds, Mach disk, barrel shocks. Apart from the complexities within the plume, the plume induced shock interacts with the boundary layer and this causes a shock induced separation which further adds to the complexity of the problem. Figure 1 shows schematically all the phenomenon occurring due to Plume interference [1] while Figures 2 and 4 shows a typical and detailed plume structure. As shown in Figure 2, there are three regions, namely a near field, a transition and a far-field region. The near field region can be further classified as an inviscid core and a viscous mixing layer, as shown in Figure 3. All the viscous phenomenon like mixing and reactions occur in the viscous mixing layer region. Figure 4 shows the above mentioned complexities in detail. It clearly depicts the Prandtl-Meyer expansion, which is typical for an under-expanded jet, as the flow needs to be accelerated and pressure has to be reduced to match the ambient condition. These waves get reflected from the plume boundary in the form of compression waves and result in the formation of Barrel shock, which, if strong, result in the formation of a normal shock at the center line, called as a Mach disc. This phenomenon is repeated downstream and result in the formation of shock diamonds which becomes weaker with enhanced mixing.

This study is an attempt to establish a numerical approach to model the flow physics of the plume with all its aforementioned complexities and to understand its behavior, which can act as a foundation for more complex studies like jet impingement on control surfaces of an aircraft, plume ingestion by aircraft air-intakes, etc., to name a few. Commercial CFD package, CFD++ was used to carry out the numerical computations [5]. The geometry and grids were prepared using Ansys ICEM CFD package [6].
2. Numerical Analysis:

2.1 Geometry modelling:

The geometry considered for present study is representative of a missile with ogive forebody and a straight aft-body without tail fins [1].

The configuration of the missile is a 13 calibre body with four calibre tangent ogive nose and a cylindrical after-body of diameter of $D = 63.5$ mm. The plume expands to supersonic speeds...
through a convergent-divergent nozzle having a design Mach number, $M_\infty$ of 2.7 and an exit diameter $D_e = 50.9$ mm. The geometry is shown schematically in Figure 5.

2.2 Grid generation:

The geometry missile is contained in a rectangular computational domain, shown in Figure 6, with the following extents:

- Length (along x-axis) = 570 times the Nozzle exit diameter $D_e$,
- Height (along y-axis) = 40 times the Nozzle exit diameter $D_e$

![Figure 6. Computational Domain](image)

A 2-D structured grid generation method was used to discretize the domain. Further O-grid splitting of the blocks insured a finer grids near the walls to capture the boundary layer effects. The grid was clustered in the regions where complex flow phenomenon like shocks, flow separation, boundary layer-Shock interaction etc. were anticipated to occur. The grid comprised of 2.22 million cell volumes. The node spacing was kept such that the $y^+$ was less than 1 for all the cases. Figure 7 shows the grid distribution near the nozzle and aft-body walls.

2.3 Simulation Methodology:

The plume simulation was attempted numerically as a 2-D axisymmetric problem. Flow solutions were computed using RANS approach with CFD++ software [5]. As shown in Figure 6, the computational domain comprises of Inlet, Outlet, Far-field, Symmetry and Nozzle-Inlet boundaries. All other boundaries were considered to be adiabatic walls with no-slip condition. The following boundary conditions are imposed:

a. **Inlet**: Pressure, Temperature and Normal velocity defined Inlet, where,

$$P = 101325 \text{ Pa}, \quad T = 288.15 \text{ K}, \quad \text{and } U = \text{Based on the free stream Mach number}$$

b. **Outlet**: Simple Back Pressure defined outlet, where,

$$P = 101325 \text{ Pa}$$

c. **Far-field**: Characteristic based Inlet/Outlet, where,

$$P = 101325 \text{ Pa}, \quad T = 288.15 \text{ K}$$

d. **Nozzle-Inlet**: Reservoir based Stagnation Pressure and Temperature, where,

$$P_0 \text{ and } T_0 \text{ values were based on the Pressure Ratio (PR).}$$
**e. Symmetry:** As the flow is supposed to be axis symmetric, only half of the geometry was considered on one side of symmetry in order to reduce the computational efforts.

**f. Adiabatic/No slip condition walls:** The walls of the missile body were taken to be adiabatic no-slip boundaries, illustrated in Figure 7.

The turbulence effects are simulated using cubic k-ε model. The levels of turbulence in the far-field have been taken corresponding to free flight conditions. At the nozzle-inlet, turbulence has been specified in terms of length scale, equal to the nozzle inlet radius, and chamber velocity.

### 3. Results and Discussion:

An experimental investigation by Burt [2] provides pressure distribution over the aft-body of the missile, which has been used for data comparison and validation of the numerical approach in present study. Figure 8 shows the schematic of direction of jet and the aft-body. The data comparison is made on aft-body length from X/D=0 to -6.0 where X is measured along the length of the missile. The positive X-direction is taken to be along the jet expansion.

The plume was simulated for five Mach numbers and different pressure ratios. Table 1 defines the flow conditions:

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Pressure ratio (Chamber stagnation pressure/freestream pressure)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.9</td>
<td>88.0, 103.7</td>
</tr>
<tr>
<td>1.2</td>
<td>93.7, 155.3</td>
</tr>
<tr>
<td>1.5</td>
<td>93.7</td>
</tr>
<tr>
<td>2.0</td>
<td>93.7</td>
</tr>
<tr>
<td>3.0</td>
<td>93.7</td>
</tr>
</tbody>
</table>

Table 1: Flow conditions

### 3.1 Validation of Numerical approach:

---

Figure 7. Grid distribution  
Figure 8. Station wise schematic of the model
Figure 9 (a). Pressure plots: $M = 0.9$

Figure 9 (b). Pressure plots: $M = 1.2$

Figure 9(a) compares the computed pressures with experimental measurements at Mach 0.9. At higher free-stream Mach number of 1.2, the results are compared in Figure 9(b). The steep rise in the pressure magnitudes near the tail of the missile, marked by $X/D = 0$, can be attributed to the plume induced shock on the aft-body. As argued in [1], the dip in the pressure values, in experimental data for $M = 1.2$ case, can be attributed to the plume and strut interaction and wind-tunnel blocking effect.

3.2 Effect of Mach number and Pressure Ratios:

Figures 10(a) & (b) show the contours for Mach No. $= 0.9$ and different Pressure Ratios (PR). Figure 11 (a)-(d), shows us the contour lines for $PR = 93.7$ and different free stream Mach numbers.
The above contours show the effect of Mach number and Pressure ratio on the plume interference and Plume structure. A notable observation is increase in Pressure ratio (PR) changes the interior structure of the plume, significantly. This can be seen in Figure 10 (a)-(b), with the increase in plume pressure ratio, there is stronger expansion of plume and for higher ratio, the barrel shock reflects as a strong normal shock resulting in the formation of Mach disc at the center of the axis. Figures 11 (a)-(d) show us the effect of increase in free stream Mach number, and as can be seen in the figures, with the increase in free stream Mach number, the expansion of plume reduces, this can be attributed to the increased dynamic energy of the flow, which restricts the free expansion of plume. It can be seen in the pictures the leading compression waves getting pushed in the downstream direction, for higher Mach numbers, and so the plume induced shock also gets pushed in the downstream direction, resulting in a very steep rise in pressure values near the tail of the aft-body, as can be seen in the graphs shown in Figure 9.
4. Conclusion:

1. The predictions matched well with the experimental results and thus validate the numerical approach. This establishes the fact that the numerical approach can be used to study the behavior of plume from realistic missiles and aircraft.

2. The effect of Mach number and plume pressure ratio on the plume interference and its structure was analyzed. This revealed the fact that free-stream Mach number affects the plume interference predominantly, while the plume pressure ratio affects mostly the structure of the plume internally.

5. Scope for future work:

1. Further studies can be taken up with different turbulence models in order to further reduce the disagreement between experimental and Numerical results.

2. The study can be extended to more complicated 3-D analysis such as plume ingestion of air-intakes, impingement of jet on control surfaces of aircrafts etc.

3. High fidelity turbulence models like DES and LES can be taken up to study unsteady features like shock diamonds etc. in detail.

References:


