Investigation of Supersonic Retro Propulsion Flow Fields for Central Jets and Peripheral Jets on Re-Entry Bodies

Mrs. Thripthy Anil A S1 Mrs. Smitha2 Mr. Vishal Kaushik3
1PG Student 2,3Assistant Professor
1,2,3Department of Aerospace Engineering
1,2Malla Reddy College Of Engineering and Technology, Hyderabad, Telangana 3Priyadarshini College of Engineering, Nagpur, Maharashtra

Abstract— NASA's Entry, Descent, and Landing (EDL) space technology is introduced new techniques to achieve human exploration of Mars in the coming decades. One of those technologies, termed Supersonic Retropropulsion (SRP), involves initiation of propulsive deceleration at supersonic Mach numbers. The potential benefits afforded by SRP to improve payload mass and landing precision make the technology attractive for future EDL missions. NASA's EDL project spent two years advancing the technological maturity of SRP for Mars exploration. To model the flow field that exists for a vehicle employing SRP, multiple simulation method has been examined. Wind tunnel testing on scale model allows for visualization of the actual flow field structures using schlieren or shadowgraph images. Pressure ports on the body allow for data to be taken to characterize the effects of configuration and thrust level on the aerodynamics of the vehicle. In addition to characterizing the SRP environment, this datasets provide a validation database against with other modeling techniques can be compared. CFD approaches numerically solve the underlying flow equations to generate the solution of the expected flow structure and vehicle aerodynamics. This paper deals with the investigation on the effects of retro propulsion on the blunt body reentry vehicle in an opposing supersonic free stream. The focus is on aerodynamic properties for the application of EDL design and computational simulation development. This paper does not discuss non propulsive supersonic decelerators, detailed aero thermodynamic issues, slender body geometries or exhaust plumes in direction other than the free stream direction. This it also compare the results obtained in central peripheral nozzle locations as well as the computational simulation of supersonic retro propulsion flow fields and limitations of this work.

Key words: SRP, EDL AND PARAS-3D

I. INTRODUCTION

Supersonic retro propulsion, or the initiation of a retro propulsion phase at supersonic free stream conditions, is an enabling decelerator technology for high-mass planetary entries at Mars. The current knowledge on supersonic retro propulsion is largely derived from exploratory development efforts prior to the Viking missions in 1960s and the early 1970s, predominantly sub-scale wind tunnel testing. Preliminary computational results for a blunt body with two retro propulsion configurations are compared with experimental data for the location of prominent flow features and surface pressure distributions. This work is intended to provide an initial discussion of the challenges facing the computational simulation of supersonic retro propulsion flow fields. The rest of the paper is organized as follows. Vehicle geometry and grid generation methods are explained in section II. Computational results are presented in section III. Concluding remarks are given in section IV.

II. VEHICLE GEOMETRY AND GRID GENERATION

A. Vehicle Geometry

The configuration that we are considering here is spherically blunt aeroshell configuration. This configuration has a single nozzle aligned with the axis of symmetry. It is having a 14 degree slanted sidewall with total length is 2684.78mm. The forward face diameter is 3098.8mm and aft face diameter is 1994mm. The nozzle is a 15degree cone angle and exit, throat diameters are 387.5mm and 100.8mm. The full length of the supply line is not modeled in the computational geometry. Rather, a short cylindrical plenum is placed prior to the converging section of the nozzle. The boundary conditions are provided at the inflow boundary for each run condition. This configuration is used for the investigation of retro propulsion with various thrusts coefficients. Figures shows the CAD model of spherically blunted aero shell without nozzle and with nozzle.

Fig. 1: CAD model of spherically blunted aero shell without and with Central Single Nozzle with perspective view
B. Domain and Grids
Simulations have been carried at a free stream Mach number
2. Domain length in upstream to the model is 9.68D and
downstream to the model is 14.52D in the x axis with 200
grids. Domain length in the y and z axis had taken as
14.52D is in negative and 14.52D in positive and negative y
axis with 200 grids in y axis and zero in positive z axis with
100 grids.

<table>
<thead>
<tr>
<th>Domain</th>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>X/D</td>
<td>-9.68</td>
<td>14.52</td>
</tr>
<tr>
<td>Y/D</td>
<td>-14.52</td>
<td>14.52</td>
</tr>
<tr>
<td>Z/D</td>
<td>-0.00</td>
<td>14.52</td>
</tr>
</tbody>
</table>

Table 1: Domain and grids for spherically blunted aero shell
model (jet off & jet on)

C. Grids Refinements
CFD code PARAS 3D is used as a tool here. This code is in-
house developed code is extensively used for Launch
Vehicle Design. PARAS 3D code has Cartesian grid
formation; it uses Rectangular Adaptive Cartesian Mesh
(RAM) technique to capture body geometry, by splitting the
grid cells near the body and in region of large curvatures. At
each refinement step the mesh is refined on the basis of the
generated flow field. This means that the grid has been
adapted to capture the variations in the flow field more
accurately. The refinement takes place by the refinement/un-
refinement criterion specified in the PARAS 3D code. The
gas cells are different in each case due to refinement

<table>
<thead>
<tr>
<th>Type</th>
<th>Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gas Cells</td>
<td>17183780</td>
</tr>
<tr>
<td>Partial Cells</td>
<td>1083496</td>
</tr>
<tr>
<td>Body Cells</td>
<td>881466</td>
</tr>
<tr>
<td>Total Cells</td>
<td>19148742</td>
</tr>
</tbody>
</table>

Table 2: Grid cells for Spherically Blunted Aero shell (jet off)

<table>
<thead>
<tr>
<th>Type</th>
<th>Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gas Cells</td>
<td>5484897</td>
</tr>
<tr>
<td>Partial Cells</td>
<td>1083496</td>
</tr>
<tr>
<td>Body Cells</td>
<td>881466</td>
</tr>
<tr>
<td>Total Cells</td>
<td>7449859</td>
</tr>
</tbody>
</table>

Table 3: Grid cells for Spherically Blunted Aero shell (jet on
with Ct=3.00)

D. Boundary and Simulation conditions
The boundary conditions should give before starting the
simulation process. The model in PARAS will consider as a
solid wall by default in PARAS. Near upstream of model is,
upwind condition is prescribed at inflow, Upwind boundary
condition is for supersonic inflow condition and all free
stream parameter such as velocity, pressure and density have
to be specified. Outflow condition is prescribed as shift
condition; this condition will obtain exit flow variables
shifted from upstream neighbor cells. Flow variables are
allowed to change with the time-step. Far field such as outer
domain or side faces, shift condition is specified.

<table>
<thead>
<tr>
<th>Flow variables</th>
<th>Ct=0.00</th>
<th>Ct=3.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach Number</td>
<td>1.00E-005</td>
<td>1.00E-005</td>
</tr>
<tr>
<td>Pressure(pa)</td>
<td>820.05</td>
<td>4100.24</td>
</tr>
<tr>
<td>Temperature(K)</td>
<td>294</td>
<td>294</td>
</tr>
<tr>
<td>Density(kg/m3)</td>
<td>1.4</td>
<td>1.4</td>
</tr>
<tr>
<td>Specific heat ratio of gas</td>
<td>0.01</td>
<td>0.07</td>
</tr>
</tbody>
</table>

Table 5: Chamber conditions for Spherically Blunted Aero
shell

III. RESULTS
A. Spherically Blunted Aero shell with Jet off case
1) 0-degree Angle of Attack
The baseline case was Jet off at alpha=0 deg. The pressure
distribution is axisymmetric with the highest pressure that is
observed in the nose region and the pressure is decreasing
toward the shoulder. Shock stand-off distance (X=1.59m)
small and thus the position of the bow shock is so close to
the body. The Mach palette and Cp distribution is shown in
the Figure 3. The flow stagnates at the nose causing higher
Cp .Therefore due to expansion on the surface causes the Cp
falls down to shoulder. The Cp on Boat tail is nearly
constant. The supersonic wake formation aft of the body
causes to form reflected shock. The Cp value obtained is
1.6. In zero angle of attack case, the body is aligned parallel
to the free stream. Thus the force acting on the body is axial
force. So highest load experience on nose portion of the
body in axial direction .The coefficient of axial force
obtained here is .765
Investigation of Supersonic Retro Propulsion Flow Fields for Central Jets and Peripheral Jets on Re-Entry Bodies

(IJSRD/Vol. 4/Issue 10/2016/162)

All rights reserved by www.ijsrd.com

Fig. 3: Computational flowfield and Cp distribution for the zero nozzle configuration (AoA=0)

2) 5-degree Angle of Attack

The shock and wake region is unsymmetrical about the center line. The bow shock position is changed to leeward side with reference to nose. Cp is higher at the nose due to the deceleration of flow and it is decreasing towards the shoulder due to the expansion of flow up to shoulder. Thus higher Cp is experienced on windward side till shoulder. The gap between leeward and windward Cp distribution at nose is wider in this case as compared with five degree. The boat tail Cp on windward and leeward is constant. The supersonic wake formation on the aft body causes to form reflected shock wave. Cp achieved here is 1.61. The CA obtained is 0.76695.

Fig. 4: Computational flow field and Cp distribution for jet off case (AoA=5)

B. Spherically Blunted Aero shell with Jet on case (Ct=3.00)

1) 0-degree Angle of Attack

The Mach palette and Cp distribution is shown in figure 6. The flow features are similar to that of zero degree case. The flow field shows a distinct terminal shock called Mach disk. Here the plume expansion becomes larger and the terminal shock forms further away from the body. These plumes have a narrow expansion out of the nozzle, which allows for the correct jet boundary to form. This plume structure shows a smooth transition from nozzle exit to jet boundary up to the terminal shock. A very low value of Cp is observed here and it is negative. Because the fully developed jet plume with wide expanded jet boundary causes the flow is deflected away from the surface and hence a high subsonic flow field is created on the entire body. The flow field is similar in leeward and windward region. The highest Cp obtained here is -0.27 and it is in leeward side. Due to the unsteady nature of the flow causes a slight variation in pressure distribution.
is observed in windward and leeward side. The CA and CN for this case will be -0.0105 and 0.0068. And it is a small negative value and it is less than jet off case.

**Fig. 6:** Computational flow field Cp distribution for jet on case with Ct=3.00 (AoA=0)

2) **5-degree Angle of Attack**

The Figure 7 shows the Mach palette and Cp distribution. By comparing this with zero angle of attack case we can understand that the flow field is deflected upwards due to 5 degree angle of attack. So jet plume shape also changed. The windward jet boundary is reduced and Mach disk is elongated into leeward side. The bow shock along the center line of the model is moved towards the leeward side and shock standoff distance also increased as compared with zero degree case. The flow field deflection in leeward side due to five degree angle of attack causes a highly subsonic flow region on leeward side. Thus higher Cp distribution is experienced in leeward due to fully attached flow.

**Fig. 7:** Computational flow field Cp distribution for jet on case with Ct=3.00(AoA=5)

Towards the boat tail region the more pressure distribution is experienced in windward due to the reattachment of the flow. Even though highest Cp is attained in leeward fore body and its value is -0.22. This value is slightly higher than the zero angle of attack case. Even at five degree angle of attack still the pressure distribution is negative on the surface. At five degree angle of attack the jet plume will take asymmetric shape and jet boundary elongated more into leeward. Hence the intermixed flow will deflect slightly towards the leeward side and modifying the pressure distribution. This gives the load on the body. The CA and CN value obtained here is -0.02225 and -0.00328.

**(c) 10-degree Angle of Attack**

**Fig. 8:** Computational flow field Cp distribution for jet on case with Ct=3.00(AoA=10)
The Mach palette and Cp distributions shown in figure 8. The shock stand off distance about the center line is \(x=7.5\) m and it is less as compared with five degree. Due to the deformed plume structure the windward jet boundary is reduced and Mach disk is elongated to leeward side. Bow shock along the center line of the model is changed position to leeward side. This changes modifies the pressure distribution over the surface. The flow stagnates somewhere between Bow shock and jet plume causing higher Cp and Cp obtained is 1.6 at nose. Therefore due to expansion of flow causes Cp falls down on the surface antCp on boat tail is nearly constant.

3) Comparison of Jet off and Jet on cases

The windward and leeward pressure distribution is symmetric in jet off case. By comparing Jet off and Jet on (\(Ct=3.00\)) pressure distribution at AoA=0 case we will get more Cp in jet off case which is at nose where deceleration is more due to absence of nozzle. Thus \(Cp(jet\ off)=1.6\) and \(Cp(jet\ on)=-0.11\) (leeward) and -0.18 (windward). At shoulder sudden expansion occurs shows steep fall and further expansion takes place at leeward side thus \(Cp(jet\ on)\) dominates here due to fully attached flow causes modified pressure distribution at 5 degree angle of attack and at boat tail occurring similar flow fields causes nearly constant Cp as shown in figure 10.
Fig. 11: Cp versus X directional distance in leeward and windward direction(AoA=10)

The pressure distribution is symmetric in windward and leeward in jet off case. By comparing Jet off(without nozzle) and Jet on(Ct=3.00) pressure distribution at AoA=10 case we will get more Cp in jet off case which is at nose where deceleration is more due to absence of nozzle. Thus Cp(jet off)=1.61 and Cp(jet on)= 1.28(leeward) and-0.22(windward). At shoulder sudden expansion occurs shows steep fall and further expansion takes place at leeward side but in Cp(jet on) nose, shoulder and boat tail occurring low Cp due to flow separation followed by expansion as shown in figure 11

IV. CONCLUSION

Supersonic retro propulsion is an enabling decelerator technology for high–mass planetary entries at Mars. SRP flow fields are the result of a complex interaction between typically highly under expanded jets and supersonic free stream. Then jet flows explaining various terms like mach disk, terminal shock etc. Accurate and consistent simulation of SRP flow fields is of significant interest to the current technology development community. A lack of modern experimental data at relevant conditions and incomplete existing data challenge efforts to validate existing computational tools approaches. This investigation presented a discussion of the relevant flow physics to provide insight into the effectiveness of inviscid computational analysis approaches in consistently and accurately capturing the relevant flow physics. Here the retro propulsion configurations with various angle of attack compared. The pressure data set are compared here. In Spherically blunted Aero shell, the jet off case is giving good pressure distribution at low angle of attack but at high angle of attack the jet on case with given Coefficient of thrust giving considerably good pressure distribution. This work represents here just an introductory step in developing steady, inviscid models for the design of SRP systems. Here we considered low thrust values only so it is not applicable to Mars Science Laboratory like experiments. However, based on comparison with various cases we found that this retro propulsion configuration flow solution capturing salient SRP flow features and detailed pressure coefficient comparison reasonably predict the forces for this SRP flows.

REFERENCES