Numerical study of hot launch of missile inside a tube

PK Sinha and Debasis Chakraborty

Abstract
The impingement of hot rocket motor plume inside a canister is simulated numerically by solving three-dimensional Reynolds Averaged Navier–Stokes equations using commercial software. The computed methodology is first validated for cold flow jet impingement in a circular tube for different chamber pressure and the simulations captured all the finer aspects of blow-by flow conditions as reported in the literature. A very good comparison is obtained between experimental and numerical surface pressure distribution. The validated methodology is applied to simulate the hot launch of a missile from a canister. It is observed that for low annular gap between missile body and canister the motor plume interaction became intense and gave rise to a very significant base drag which may constrain the motion of the missile inside the canister.

Keywords
Jet impingement, computational fluid dynamic, tube launch

Date received: 21 April 2013; accepted: 4 December 2013

Introduction
For operational convenience, missiles are launched from canister. When the missile is fired inside the canister (hot launch option), the exhaust jet interacts with the canister surface. The hot jet impingement on the canister wall creates a complicated flow structure near the nozzle exit and cause severe mechanical and thermal load on the canister surface. In order to finalize the total system design, it is necessary to understand the complicated flow structure inside the launch tube.

The flow features in the canisterized missile base are very much dependent on the jet pressure ratio, nozzle divergence angle, and the canister tube geometry. The rocket exhaust flow of canisterized missile is mostly underexpanded. The underexpanded nozzle flow entrains air from the annular gap forming a shear layer at the plume boundary. The streamlines in the shear layer which do not possess sufficient energy to penetrate high pressure region downstream of the shock (created by plume impingement) will turn back toward the base. The exhaust that is turned upstream is known as blow-by flow. If the jet pressure ratio is lower or the (annular) gap between the nozzle exit and the canister tube is large, the system can act as an ejector and the reduction of base pressure is marginal. In the other case, where the jet pressure ratio is high or the annular gap between the nozzle exit and the canister tube is small, blow-by flow dominates and the missile is subjected to high base drag. Both these cases are schematically shown in Figure 1. The recirculation of hot exhaust in the region between the impingement shock and the missile base reduces the pressure value from the ambient and causes base drag which needs to be considered in the missile motion inside the canister. Because of space constraint, there exists a small annular gap between missile outer diameter and canister inner diameter. To support the missile, a co-axial obturator ring is sometimes placed at the base which further reduces the annular gap and restricts the entrainment from the ambient and further enhances the base drag. The flow field inside the tube needs to be analyzed carefully for canisterized system, both for the design of canister as well as for the missile motion.

Although the impingement of underexpanded supersonic free jet in a flat plate or inclined plate is
studied for many years both experimentally\textsuperscript{2–7} and numerically,\textsuperscript{8–11} the study of impinging jet in a confined environment is rather limited. Batson and Bertin\textsuperscript{12,13} obtained wall pressure distribution inside the launcher tube by conducting static rocket test with cold gas as well as with double base solid rocket propellant. The flow field of an underexpanded jet exhausting into a plenum from a square cross section launch tube was studied experimentally by Bertin et al.\textsuperscript{14} The tests demonstrated that strong shock wave interactions in the converging section enhance the recirculation of exhaust flow back into the launch tube and increase pressures surrounding the missile.

It is clear that the complex jet impingement process in confined environment needs further investigations. With the advent of powerful computers and robust numerical algorithms, computational fluid dynamics (CFD) is playing an important role to understand this complex flow physics and help to arrive at an efficient design of canister system. It is essential to validate the numerical tools to find out their range of applications and error band before using them in the design exercise. In this paper, a commercial code is used to explore the flow field of a hot launch canisterized missile after validating the computational tool against the static test results of motor firing in a long tube.

**Numerical methodology**

Present simulation uses a commercial CFD software CFX\textsuperscript{15} which is three-dimensional finite volume Reynolds Averaged Navier–Stokes (RANS) solver on structured or unstructured grid. The solver provides option to use different turbulence models, namely k–\(\varepsilon\), Renormalisation Group (RNG) k–\(\varepsilon\), k–\(\omega\), or Shear Stress transport (SST) turbulence model. The software has three major modules: (i) \textit{preprocessor}—imports grids from a grid generator and sets up the boundary condition, selects turbulence model, and initializes flow field; (ii) \textit{solver manager}—solves the flow field based on the grid and the boundary condition; and (iii) \textit{postprocessor}—helps in visualizing flow field data. The solver has different schemes for solving the advection term. For high speed flows, high resolution schemes are preferred for better accuracy and robustness.

**High resolution scheme**

In the solver, finite volume is created around the integration point by connecting the neighboring nodes. Solution fields and other properties are stored at the nodes. However, to evaluate many of the terms, the solution field or solution gradients must be approximated at the integration point. The advection term in the solver is modeled by expressing the finite element shape function \(\varphi\) at integration point in terms of nodal values of \(\varphi\) and can be cast in the form

\[
\varphi_{ip} = \varphi_{up} + \beta \nabla \varphi \cdot \Delta \Gamma
\]

where \(\varphi_{up}\) is the value at the upwind node and \(\varphi_{ip}\) is the value at the integration point. The high resolution scheme is a special nonlinear recipe for \(\beta\) at each node computed to be as close to 1 as possible without introducing new extrema. The advective flux is then evaluated using the values of \(\beta\) and \(\nabla \varphi\) at upwind point. The recipe for \(\beta\) is based on boundedness principle used by Barth and Jespersen\textsuperscript{16} which utilizes a multidimensional monotone reconstruction of cell averaged data and Roe’s flux function. This differs from the other upwind schemes for unstructured meshes which do not perform reconstructions of cell averaged data. This methodology involves first computing a \(\psi_{\text{min}}\) and \(\psi_{\text{max}}\) at each node using a stencil involving adjacent nodes (including the node itself). Next, for each integration point around the node, equation (1) is solved for \(\beta\) to ensure that it does not undershoot \(\psi_{\text{min}}\) and overshoot \(\psi_{\text{max}}\). The nodal values of \(\beta\) are taken to be minimum for all integration point values surrounding the nodes. The values of \(\beta\) are also not allowed to exceed 1. This algorithm is shown to be total variation diminishing in one dimension.

**Problem setup**

Chamber total pressure (\(P_0\)) and total temperature (\(T_0\)) condition are imposed at inflow plane and supersonic outflow condition was implemented at the outflow plane. No slip, adiabatic wall condition is imposed on all walls. Second-order spatially accurate high resolution scheme\textsuperscript{16} is employed to discretize the 3D RANS equation to capture the flow physics better.

**Figure 1.** Typical flow features in the tube at different chamber pressures: (a) no jet impingement on the canister wall, (b) jet impingement on the canister wall (reproduced from Bertin et al.).
Both standard k-ε turbulence model and RNG k-ε turbulence models with scalable wall function are studied to compare their predictive capabilities of confined supersonic jets. A fourth order decay of log-normalized maximum residue is considered as the convergence criteria. To find out the accuracy and the range of applications, the software has been validated for various complex aerospace problems including supersonic base flow, free stream and rocket exhaust interaction, and transverse sonic injection in supersonic stream and obtained very good match with experimental and flight measured values.

Validation study

Problem description and grid generation

The predictive capability of the CFD methodology is first assessed by numerically exploring the cold flow test condition of Bertin et al. In the experiment unheated compressed air at different chamber pressures was accelerated through different convergent–divergent nozzle–canister combination. In the present work, the C3-L1 geometry where the annular passage is blocked by obturator ring placed 1.27 cm upstream of the nozzle exit plane has been considered. The simulations are carried out for three different chamber pressures (P₀ = 2.71, 6.07, and 8.75 MPa). The schematic of the geometry and computational domain is provided in Figure 2. The length of the tube is 0.381 m and inner radius 0.01484 m. The nozzle has a length of 0.031 m and exit radius as 0.0124 m. The nozzle contour is generated as per the details given in Table 1. As the geometry is axisymmetric, a 5° sector of the configuration is selected for simulation. Both structured and unstructured grids are considered for the analysis. Two structured grids consisting of 0.2 and 0.4 million hexahedral points and one unstructured grid consisting of 0.24 million hybrid tetrahedral-prism element are generated. Commercial grid generator CFX-4 and ICEM-CFD is used to generate the structured and unstructured grids, respectively. The grids are made very fine through clustering near the wall and jet shear layer region to capture the important flow features accurately. Twelve prism layers were provided in the boundary layer and typical y⁺ and minimum grid spacing are of the order of 30 and 25 μm, respectively. Grid distribution in nozzle exit region with blown-up view near the wall boundary layer region is shown in Figure 3.

Results and discussions

Typical flow features for chamber pressure of 8.75 MPa are represented through the Mach number contour in Figure 4. The jet exiting out of the nozzle is found to impinge on the tube at 7.3 mm away from the nozzle exit plane and gets reflected several times from the wall. The impingement shock, flow recirculation, Mach discs, etc. are crisply captured in the simulation. From computational results it is found that the impingement shock makes an angle of 24° with canister tube and the computed reflected shock makes an angle of 19.99° with canister tube in comparison to oblique shock angle of 20° from shock theory. Four prominent shock cells are evident from the contour plot.

Two turbulence models, namely k-ε turbulence model and RNG k-ε turbulence models were studied to assess their predictive capabilities. The computed surface pressures with the turbulence models are compared in Figure 5 for two different chamber pressures 2.71 and 6.07 MPa, respectively. The length (x/Rₙₑ) is

<table>
<thead>
<tr>
<th>Table 1. Nozzle profile along the length.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Convergent section x &lt; -1.308 cm</td>
</tr>
<tr>
<td>x (cm)</td>
</tr>
<tr>
<td>-1.562</td>
</tr>
<tr>
<td>-1.816</td>
</tr>
<tr>
<td>-2.070</td>
</tr>
<tr>
<td>-2.324</td>
</tr>
<tr>
<td>-2.578</td>
</tr>
<tr>
<td>-2.832</td>
</tr>
<tr>
<td>-3.086</td>
</tr>
</tbody>
</table>

Figure 2. Computational domain and nozzle geometry.
normalized by nozzle exit radius ($R_{Ne}$) and the pressure (pratio) is normalized with chamber total pressure. It is clear that the pressure distribution from 1.27 cm upstream nozzle exit plane ($x/R_{Ne} = 0$) till canister exit (maximum $x/R_{Ne} = 8$) obtained from both the turbulence model compare well with the experimental data. However, RNG $k-\varepsilon$ turbulence model is found to predict the pressure field marginally better in shock impingement region and especially the low pressure region compared to $k-\varepsilon$ turbulence model. These conclusions are consistent with the results of supersonic base flow simulation$^{19}$ carried out with $k-\varepsilon$ and RNG $k-\varepsilon$ turbulence model.

Effect of grid in the flow field is studied next by employing RNG $k-\varepsilon$ turbulence model for the simulations. Figure 6 compares the axial distribution of the launch tube surface pressure for chamber pressure 6.07 MPa with two structured grids involving 0.2 and 0.4 million points and unstructured grid with 0.24 million points. It is seen in the figure that almost identical results have been obtained from all three grids. The pressure kink near the second peak is absent with finer grid; thus, demonstrating the grid independence of the results. While a better result may have been obtained with the finer, structured grid, further simulations considered RNG $k-\varepsilon$ turbulence model, structured and coarse grid.

The velocity vector plots in the base region are plotted for all the three chamber pressures ($P_0 = 2.71$, 6.07, and 8.75 MPa) and shown in Figure 7(a) to (c), respectively. For all the cases, the chamber pressure is high enough for the plume to impinge on the launch tube and give rise to blow-by flow situation as described earlier. The size of the recirculation bubble is also seen to increase with chamber pressure. The computed pressure distribution along the tube length is plotted for all the three cases in Figure 8 to highlight the variation on predicted base pressure. It can be seen that the pressure distribution is nearly the same for all the cases except in the low pressure region, where the base pressure is little higher for highest chamber pressure case compared to other two cases. It is clear from the comparison of experimental and numerical data and detailed analysis of the flow field that the present methodology could predict both qualitative and quantitative features of blow-by flow situation encountered in the canistered launch missile.

**Simulation for hot launch of missile from canister**

The validated numerical methodology is applied for prediction of plume impingement during its motion inside the launch canister.

**Description of geometry and grid**

The schematic of the missile along with the canister is shown in Figure 9. Canister is a circular cylinder and the missile is placed concentrically with an annular gap of $0.038D_c$ ($D_c$ is canister diameter). A support ring (obturators) is placed on missile outer diameter near the missile base which reduces the annular gap to $0.00063D_c$ and severely restricts the atmospheric suction from the upper portion of the canister through the annular opening. The jet-vane based thrust vector control (TVC) system, used for maneuvering the missile during its initial phase of motion, is placed at the nozzle exit. One pair of opposite jet vanes is deflected by $3^\circ$ to account for jet vane misalignment. The length of the canister is about $11D_c$. The hot exhaust gas from motor comes out through the nozzle and interacts with the canister. The computational domain consists of nozzle, jet vane TVC system, and the canister geometry. One end of the canister is open to atmosphere. To model all the geometrical complexity (jet vane, support ring, etc.), a hybrid grid approach consisting of tetrahedral and hexahedral grid was employed using commercial software.$^{23}$ Three-dimensional hexahedral grid has been
generated in the canister region (consists of annular gap, missile base, and canister) with proper clustering in the regions of interest. In the complex nozzle–jet vane region, tetrahedral grid along with a prism layer on the nozzle wall to capture nozzle boundary layer has been generated. Twelve layers of prism grid are provided in 5 mm thickness to obtain a $y^+$ of 30. The grid points are properly merged at the interface of two grid regions. The interface region has been marked in Figure 10. An unstructured grid containing 2.8 million points was generated. The clustering of grid is maintained almost at the same level as in the validation case. Typical grid is shown in Figure 10.

**Simulation methodology**

In the present study, chamber pressure and annular gap are taken as constant during the missile travel within canister. Hence, the hot jet interaction with canister will remain the same for any missile position within canister and the simulations are carried out by placing the missile (nozzle exit plane) at the middle of the canister length. Rocket exhaust and air are considered as two different species and their transport equations are solved based on continuum hypothesis. The mixing of these two species (air and rocket exhaust) is considered and mixture viscosity and thermal conductivity are calculated according to Wilke’s formula and Saxena’s formula, respectively. The chamber conditions and thermo chemical properties of hot gas are provided in Table 2.

Total pressure and total temperature condition have been imposed at the inlet of the nozzle. Supersonic boundary condition is imposed at the outlet (exit of canister). No-slip boundary condition for velocity and adiabatic condition for temperature have been imposed on all the solid walls. Ambient
conditions are imposed for the inflow for the annular gap. Similar to the validation case, RNG k–ε turbulence model and second-order spatially accurate high resolution scheme\textsuperscript{16} for inviscid flux calculation are used in the simulations.

**Results and discussions**

**Results with original gap.** The first simulation is carried out by considering the 0.00063 \( D_{c} \) annular gap between support ring canister wall. Mach number distribution in the canister is presented in Figure 11. Flow recirculation region close to the support ring (blue region before plume impingement) and the shock cells is clearly seen in the figure. Blow-by flow phenomena, as discussed earlier, are also observed in the present case and some of the exhaust flow in the shear layer has turned backward as shown in the streamline plot in Figure 11. An asymmetry is seen in the recirculation region close to the base which is due to the presence of deflected jet vanes (\( 3^\circ \)) for moving the flow downward. The low pressure region near missile base (ahead of shock impingement) due to suction through the shear layer is clearly seen in pressure ratio contour plot in Figure 12. The canister wall pressure distribution is shown in Figure 13. The \( x \)-axis represents the axial length starting from support ring (normalized by canister diameter) and the \( y \)-axis represents the base pressure (normalized by chamber pressure). Two shock structures around the plume impingement region are due to the presence of the small backward facing step in the jet vane bracket. The subsequent shock reflections from the canister wall are also captured in the simulation. The low

**Table 2. Chamber conditions and properties of rocket exhaust.**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chamber temperature (( T_{0} ))</td>
<td>3400 K</td>
</tr>
<tr>
<td>Chamber pressure (( P_{0} ))</td>
<td>6500 kPa</td>
</tr>
<tr>
<td>Ratio of specific heat (( \gamma ))</td>
<td>1.11</td>
</tr>
<tr>
<td>Molecular weight</td>
<td>26.3 kg/kg mole</td>
</tr>
<tr>
<td>Specific heat at constant pressure (( C_{p} ))</td>
<td>3031 J/kg K</td>
</tr>
<tr>
<td>Thermal conductivity (( k ))</td>
<td>0.4242 W/m K</td>
</tr>
<tr>
<td>Dynamic viscosity (( \mu ))</td>
<td>( 9.4 \times 10^{-6} ) kg/m s</td>
</tr>
</tbody>
</table>
pressure (pressure ratio $\sim 0.002$) near the missile base 
subtracted from ambient pressure is integrated on the 
base area (missile base and support ring area) for the 
calculation of base drag. The computed base drag 
coefficient based on nozzle exit parameters and missile 
diameter is about 0.13. This high base drag severely 
constrains the missile motion in the canister and may 
affect the mission adversely.

Effect of increased gap. Parametric studies are carried 
out to assess the impact of annular gap between sup-
port ring and canister on the base drag by performing 
simulation with larger annular gap ($\sim 0.0063 D_c$). The 
blown-up view of the Mach number distribution near 
the annular gap (in 0.9–1.1 scale) and the pressure 
distribution downstream of the missile base are pre-
sented in Figure 14. It is seen that the flow in the gap 
is choked ($M = 1$) and as a result the flow from upper 
portion of the canister got restricted and the base 
pressure (pratio $\sim 0.004$) has increased by about 
60% compared to smaller annular gap (pratio 
$\sim 0.0025$). However, the base pressure level with 
increased annular gap is still quite low compared to 
ambient and this has caused a base drag only 10% 
lower compared to original gap. It is necessary to 
consider the effect of additional base drag caused 
due to impingement of exhaust jet in the vehicle can-
ister while considering the vehicle’s motion within the 

Conclusions

The flow within tube/canister arising out of the inter-
action of hot jet is investigated numerically. Three-
dimensional RANS equations are solved along with 
RNG k-$\varepsilon$ turbulence model and scalable wall func-
tion. The solution methodology has been validated 
against the experimental data for cold jet impinge-
ment on canister wall and a good comparison of 
experimental and computational values is obtained. 
The simulation captured all the finer details of blow-
by flow phenomena observed in the plume impinge-
ment in a tube. The effect of computational grid and 
turbulence model on the flow field is investigated and 
it is observed that RNG k-$\varepsilon$ turbulence model per-
formed marginally better in predicting the flow field in 
the canister. The validated methodology is applied to 
estimate the base drag of a canister launched missile 
for the assessment of hot launching option. An order 
of magnitude increase in annular gap has increased 
the base pressure by 60% and brought down the 
base drag by 10% of its original values. It is observed 
that the base drag occurred due to jet impingement on 
the canister is significant and it can adversely affect 
the motion of the missile in the canister.

Funding

This research received no specific grant from any funding 
agency in the public, commercial, or not-for-profit sectors.

Conflict of interest

There is no conflict of interest of the present work with 
other literature.
Acknowledgement

The authors wish to acknowledge the constant support and encouragement of Shri A.K. Chakrabarti, Ex-Director, DRDL for pursuing the task. The authors also wish to thank Sri Soumyajit Saha, Scientist, DRDL for his help in the simulation work.

References

16. Barth TJ and Jespersen DC. The design and application of upwind scheme on unstructured meshes. AIAA paper 89-0366.

Appendix I

Notation

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_c$</td>
<td>canister diameter</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$P_0$</td>
<td>chamber total pressure</td>
</tr>
<tr>
<td>$R_{Ne}$</td>
<td>nozzle exit radius</td>
</tr>
<tr>
<td>$T_0$</td>
<td>chamber total temperature</td>
</tr>
<tr>
<td>$x$</td>
<td>axial distance</td>
</tr>
</tbody>
</table>