# Numerical study of hot launch of missile inside a tube

Proc IMechE Part G: J Aerospace Engineering 2014, Vol. 228(14) 2604–2611 © IMechE 2014 Reprints and permissions: sagepub.co.uk/journalsPermissions.nav DOI: 10.1177/0954410014522609 uk.sagepub.com/jaero



**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.<sup>1</sup> 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

#### **Corresponding author:**

Downloaded from pig.sagepub.com at DEFENCE RESEARCH DEV LAB on February 11, 2015

Debasis Chakraborty, Directorate of Computational Dynamics, Defence Research and Development Laboratory, Kanchanbagh P.O., Hyderabad 500058, India. Email: debasis\_cfd@drdl.drdo.in

Directorate of Computational Dynamics, Defence Research and Development Laboratory, Hyderabad, India

studied for many years both experimentally<sup>2-7</sup> and numerically,<sup>8-11</sup> the study of impinging jet in a confined environment is rather limited. Batson and Bertin<sup>12,13</sup> 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.<sup>14</sup> 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 10<sup>15</sup> 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) preprocessor-imports grids from a grid generator and sets up the boundary condition, selects turbulence model, and initializes flow field; (ii) solver manager-solves the flow field based on the grid and the boundary condition; and (iii) 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

$$\varphi_{\rm ip} = \varphi_{\rm up} + \beta \nabla \varphi. \Delta \vec{r} \tag{1}$$

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<sup>16</sup> 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  $\varphi_{\min}$  and  $\varphi_{\max}$  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  $\varphi_{\min}$  and overshoot  $\varphi_{\max}$ . 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<sup>16</sup> 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.<sup>1</sup>).

Both standard k– $\varepsilon$  turbulence model<sup>17</sup> and RNG k– $\varepsilon$  turbulence models<sup>18</sup> 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,<sup>19</sup> free stream and rocket exhaust interaction,<sup>20</sup> and transverse sonic injection in supersonic stream<sup>21</sup> 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.<sup>1</sup> In the experiment unheated compressed air at different chamber pressures was accelerated through different convergentdivergent 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 ( $P_0 = 2.71$ , 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<sup>22</sup> and ICEM-CFD<sup>23</sup> 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  $\mu$ m, 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  $24^{\circ}$  with canister tube and the computed reflected shock makes an angle of  $19.99^{\circ}$  with canister tube in comparison to oblique shock angle of  $20^{\circ}$  from shock theory. Four prominent shock cells are evident from the contour plot.

Two turbulence models, namely  $k-\varepsilon$  turbulence model and RNG  $k-\varepsilon$  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/R<sub>Ne</sub>) is

Table 1. Nozzle profile along the length.

Convergent section x < -1.308 cm		Equations for divergent section (cm)	
x (cm)	r (cm)	For -1.308 ≤ X ≤ -1.031	
-1.562	0.856	$(x + 1.308)^2 + (r - 1.628)^2 = (0.813)^2$	
-1.816	0.937	For −1.031 < x ≤ 0.0	
-2.070	1.006	r = 0.759 + 0.364 (x + 1.308)	
-2.324	1.049		
-2.578	1.064		
-2.832	1.074		
-3.086	1.080		



Figure 2. Computational domain and nozzle geometry.



Figure 3. Typical grid distribution.



Figure 4. Mach no. distribution in symmetry plane.

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<sup>19</sup> 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 (obturator) is placed on missile outer diameter near the missile base which reduces the annular gap to  $\sim 0.00063 D_c$  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° 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.<sup>23</sup> Three-dimensional hexahedral grid has been



Figure 5. Comparison of experimental and numerical surface pressure with different turbulence models: (a)  $P_0 = 2.71$  MPa and (b)  $P_0 = 6.07$  MPa.



**Figure 6.** Surface pressure comparison with two different grids.

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



Figure 7. Velocity vector plot around base for (a)  $P_0 = 2.71$  MPa, (b) 6.07 MPa, and (c) 8.75 MPa.

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  $k-\varepsilon$  turbulence model and second-order spatially accurate high resolution scheme<sup>16</sup> for inviscid flux calculation are used in the simulations.



Figure 8. Comparison of pressure distribution along tube length for  $P_0 = 2.71$  MPa (case 1), 6.07 MPa (case 2), and 8.75 MPa (case 3).



**Figure 9.** Schematic representation of the missile-canister geometry.



**Figure 10.** Typical grid in the missile-canister computational domain.

### Results and discussions

Results with original gap. The first simulation is carried out by considering the 0.00063 D<sub>c</sub> 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 rocketexhaust.

Property	Value
Chamber temperature (T <sub>0</sub> )	3400 K
Chamber pressure (P <sub>0</sub> )	6500 kPa
Ratio of specific heat $(\gamma)$	1.11
Molecular weight	26.3 kg/kg mole
Specific heat at constant pressure $(C_p)$	303 I J/kg K
Thermal conductivity (k)	0.4242 W/m K
Dynamic viscosity (µ)	$9.4 imes10^{-6}$ kg/m s



**Figure 11.** (a) Mach number distribution in canister (downstream of the missile), (b) top, streamline pattern near support ring.

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 support ring and canister on the base drag by performing simulation with larger annular gap ( $\sim 0.0063 \text{ D}_c$ ). 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 presented in Figure 14. It is seen that the flow in the gap is choked (M = 1) and as a result the flow from upper



Figure 12. Pressure (pratio) distribution in the canister for original annular gap in the symmetry plane.



**Figure 13.** Canister wall surface pressure (pratio) distribution.

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 canister while considering the vehicle's motion within the canister.

# Conclusions

The flow within tube/canister arising out of the interaction of hot jet is investigated numerically. Threedimensional RANS equations are solved along with RNG k-e turbulence model and scalable wall function. The solution methodology has been validated against the experimental data for cold jet impingement on canister wall and a good comparison of experimental and computational values is obtained. The simulation captured all the finer details of blowby flow phenomena observed in the plume impingement 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 performed 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.



**Figure 14.** Flow parameters near the increased annular gap: (a) pressure distribution  $(p_s/p_0)$  in the canister and (b) blown-up view of Mach number distribution near the annular gap.

### 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

- Bertin JJ, Idar ES and Booker DL. Secondary flows in a rocket launcher tube. J Spacecraft Rockets 1981; 18: 119–126.
- Knight CV. Experimental investigation of two-dimensional, supersonic flow impingement on a normal surface. AIAA J 1973; 11: 233–235.
- Kalghatgi GT and Hunt BL. Occurrence of stagnation bubble in supersonic jet impingement flows. *Aeronaut Quart* 1976; 27: 169–185.
- Lamont PJ and Hunt BL. The impingement of underexpanded axisymmetric jets on wedges. J Fluid Mech 1976; 76: 307–336.
- Alvi FS and Iyer KG. Mean and unsteady flowfield properties of supersonic impinging jets with lift plates. AIAA Paper No. 99-1829.
- Nakai Y, Fujimatsu N and Fujii K. Experimental study of underexpanded supersonic jet impingement on an inclined flat plate. *AIAA J* 2006; 44: 2691–2699.
- Ramanujachari V, Vijaykant S, Roy RD, et al. Heat transfer due to supersonic flow impingement on a vertical plate. *Int J Heat Mass Transfer* 2005; 48: 3707–3712.
- Wu J, Tang L, Luke EA, et al. Comprehensive numerical study of jet-flow impingement over flat plates. J Spacecraft Rockets 2002; 39: 357–366.
- Tsuboi N, Hayashi K, Fujiwara T, et al. Numerical simulation of a supersonic jet impingement on a ground. SAE Trans Sect 1 J Aerospace 1991; 99: 2168–2180.
- Rizk MH and Menon S. Numerical simulation of impinging jets. AIAA Paper No. 86-0279.
- Kim KH and Chang KS. Three-dimensional structure of a supersonic jet impinging on an inclined plate. J Spacecraft Rockets 1994; 31: 778–782.

- 12. Batson JL and Bertin JJ. Rocket exhaust flow in tube launchers. J Spacecraft Rockets 1974; 11: 739–740.
- Batson JL and Bertin JJ. Experimental study of flow field produced when an underexpanded rocket exhausts into cylindrical tube. AIAA Paper No. 73-1227.
- Bertin JJ, Bertin RS, Yung A, et al. The launch-tube flow-field for a vertical launching system. AlAA Paper No. 88-0332.
- 15. User Manual, CFX-10, Ansys Inc., 2004.
- Barth TJ and Jespersen DC. The design and application of upwind scheme on unstructured meshes. AIAA paper 89-0366.
- 17. Wilcox DC. Multiscale model for turbulent flows. *AIAA J* 1988; 26: 1311–1320.
- Yakhot V and Orszag SA. Renormalization group analysis of turbulence. J Sci Comput 1986; 1: 3–51.
- Dharavath M, Sinha PK and Chakraborty D. Simulation of supersonic base flow – effect of computational grid and turbulence model. ASME J Aerospace Eng 2010; 224: 311–319.
- 20. Saha S, Rathod S, Chandramurty MSR, et al. Numerical simulation of base flow of a long range flight vehicle. *Acta Astronaut* 2012; 74: 112–119.
- 21. Aswin G and Chakraborty D. Numerical simulation of transverse side jet interaction with supersonic free stream. *Aerospace Sci Technol J* 2010; 14: 295–301.
- 22. User Manual, CFX-4, AEA Technologies, 2001.
- 23. User Manual, ICEM-CFD 11, Ansys software Ltd.

# Appendix I

Notation

D <sub>c</sub>	canister diameter
Μ	Mach number
P <sub>0</sub>	chamber total pressure
R <sub>Ne</sub>	nozzle exit radius
$T_0$	chamber total temperature
Х	axial distance