JOURNAL OF SPACECRAFT AND ROCKETS
Vol. 50, No. 3, May–June 2013

Engineering Notes

Plume-Ducting System Design of Vertical Launcher Using Computational-Fluid-Dynamics Tools

S. Saha* and Debasis Chakraborty*
Defence Research and Development Laboratory,
Hyderabad 500 058, India

DOI: 10.2514/1.A32303

I. Introduction

VERTICALLY launched canisterized missiles are very convenient operationally. The design challenge of the vertical-launch system (VLS) is to contain the initial impact of the rocket jet plume and safely discharge the rocket exhaust gas quickly away from the launch installations during firings of the missiles. The schematic of a typical plume-ducting system of a vertical launcher is shown in Fig. 1. Compact VLS needs an innovative mechanical design that requires understanding of the flow structures of the exhaust plumes. Bertin and Korst [1] and Bertin et al. [2] have explained the complex flow features in the VLS. When the jet plume exhausts from the rocket motor it impinges on the bottom wall of the gas-gathering tank (GGT) and then exits through the uptake after circulating in the plenum chamber. Complex flow interactions may make the flow fluctuate, and these unsteady loads need to be considered for the structural design. The large adverse pressure gradient associated with a strong shock wave causes a significant fraction of the exhaust flow to move upstream into the annular gap between the rocket and launch-tube wall. Of concern to the designer of VLS is the possibility that, in the event of a restrained firing (hang fire condition), the reverse flow will impinge on the missile causing critical heat-transfer problems. The study of impinging jets in a confined environment in current literature is rather limited. Batson and Bertin [3] obtained wall-pressure distribution inside the launcher tube by conducting static rocket tests with cold gas as well as double base solid rocket propellant. Lee et al. [4,5] presented numerical solutions of a missile launcher for supersonic jet impingement. The effect of the plume-exit area of the plenum on the jet structure is studied, and the pressure and temperature rise in the plenum are compared with test data. It is clear that the complex jet-impingement process in a confined environment needs further investigation, and computational fluid dynamics (CFD) can play an important role to understand this complex flow physics and help to arrive at an efficient design of plume-ducting systems. But before using the computational methodology in the design exercise it is essential to validate the numerical tools to find out their range of applications and error band. In the present work, three-dimensional (3-D) viscous simulations are carried out to study the underexpanded plume impingement in a confined environment. 3-D Navier–Stokes equations are solved along with k-ε turbulence model using a commercial CFD solver [6]. The computational tool is first validated against experimental results of rocket-exhaust impingement in a confined environment and then used to design a compact plume-ducting system for a vertical launcher.

II. Computational Methodology

Unsteady 3-D Reynolds-averaged Navier–Stokes (RANS) equations with k-ε turbulence model are solved. A density-based solver with second-order spatially accurate Roe-flux-difference splitting scheme [7] and second-order implicit Euler scheme are used for spatial and temporal discretization, respectively. Rocket exhaust and air are considered as two different species with different thermodynamic properties, and their transport equations are solved. Wall function is used in the simulation, and the typical value of Y^+ is of the order of 30. For unsteady calculations, the time step (dt) is 10^{-5} s, the number of subiteration per time step is 20, and the residue fall of three decades is considered.

III. Validation Against Experimental Results of Tube-Launched Rockets

A highly under-expanded, supersonic, short-duration jet exhausting from a conical nozzle into a tube with an inside diameter slightly larger than the nozzle exit characterized the flowfield of tube-launched rockets. The experimental condition of a static test of a rocket motor with cold gas as well as double base solid rocket propellant inside a launcher tube carried out by Batson and Bertin [3] is taken as the test case for validation. The nozzle configurations for the hot and cold cases are slightly different. The throat diameter, exit diameter, total temperature, and ratio of specific heats for the cold-flow test cases are 28.6, 52.93 mm, 300 K, and 1.4 respectively, whereas the throat diameter, exit diameter, total temperature, and ratio of specific heats for the hot-flow case are 21.97, 52.45 mm, 3040 K and 1.23. The chamber total pressure varies between 9 to 16 MPa for the cold-gas case, whereas for the hot test the parameter varies between 37 to 44 MPa during the test duration. Axisymmetric simulations with 0.12 million nodes are carried out for the test cases. Mach number distributions in the symmetric plane for the cold and hot test cases are shown in Fig. 2. The plume from the nozzle exit is seen to impinge in the launch tube at a distance of 18.36 mm from the nozzle exit for the cold-flow case and 18.06 mm for the hot-flow cases. The nozzle-exit Mach number for both the cases are 2.8 and 3.0, respectively. Four and three distinct shock reflections are crisply captured in the simulation for the cold-flow and the hot-flow cases. The computed axial-pressure distribution at the launch-tube internal surface is compared with experimental values in Figs 3a and 3b, respectively. The pressure (pr ratio) is nondimensionalized by chamber pressure p_0, whereas the length (xratio) is nondimensionalized by exit radius r_e. A reasonably good match between experimental and computational values are obtained except near xratio $= 0$ where experimental values are higher (~15%) compared to the computational values for the cold case and (10%) for the hot case. Both the location of impingement and pressure peaks are well captured in the simulation.

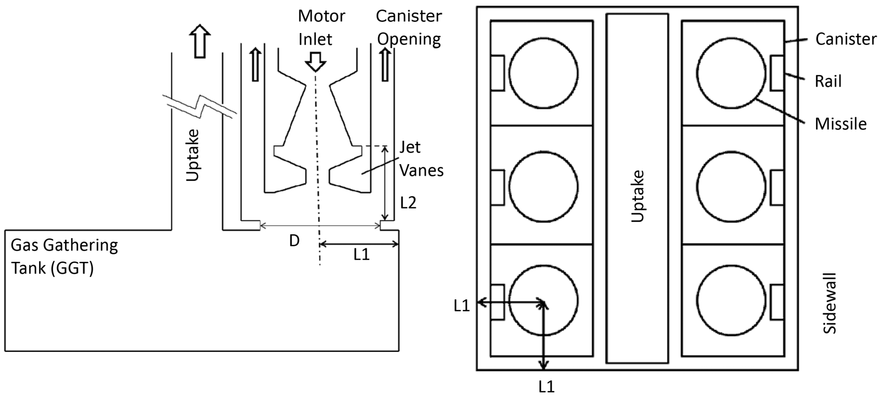
IV. Design of Compact Ducting for VLS

A vertical-launching system generally consists of six to eight missiles contained in canisters with a common GGT with a flat bottom and a common uptake at the center of the chamber for hot-gas disposal. Each canister is closed at the bottom (GGT end) to prevent entry of hot gas from the chamber into the canister, except the active canister (in which the missile is fired). Instead of a traditional

---

Received 23 December 2011; revision received 14 August 2012; accepted for publication 25 August 2012; published online 24 April 2013.

*Directorate of Computational Dynamics; debasis_cfd@drdl.drdo.in.



a) b)

Fig. 1 Schematic of vertical hot launcher a) elevation and b) plan view.

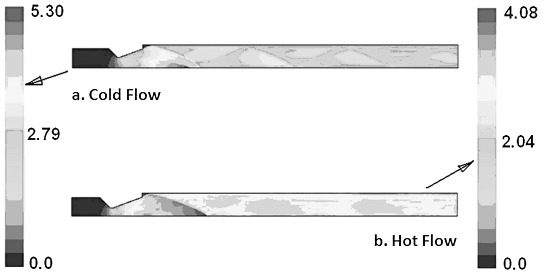
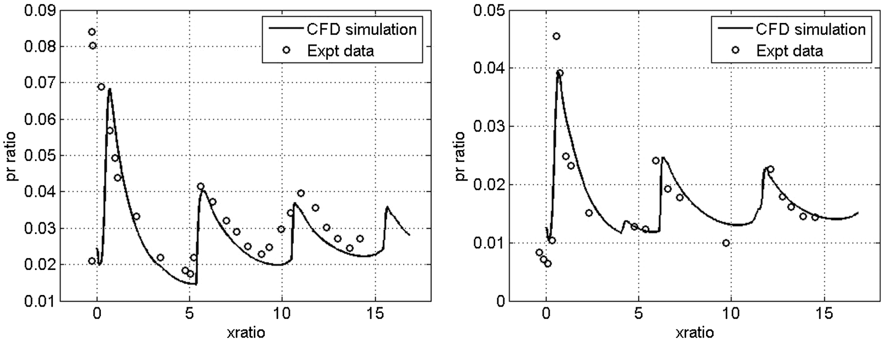


Fig. 2 Mach-number distribution in the symmetry plane.

one-duct system an alternate two-duct system is designed and analyzed in the present work for efficient disposal of hot rocket exhaust from VLS. The volume of the plenum chamber of the compact VLS is half of the volume of conventional design.

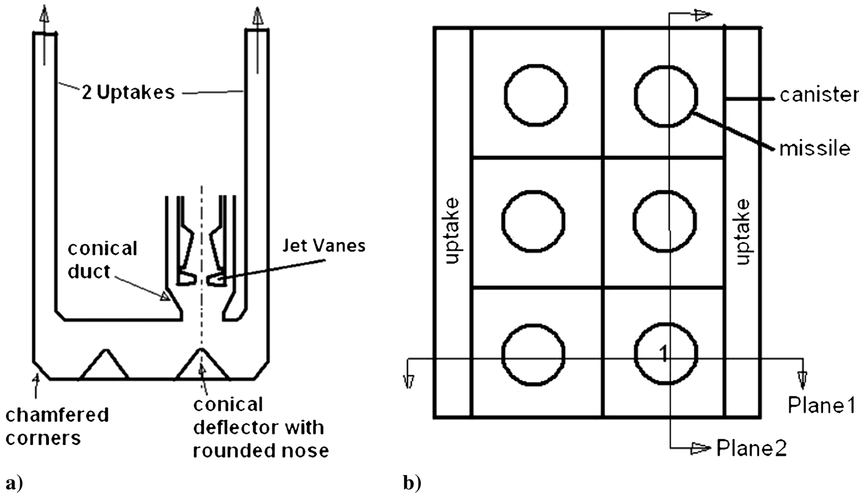
A. Geometrical Features of the Plume-Ducting System and Computational Grid

The schematic of the new plume-ducting system is presented in Fig. 4. In the design, conical plume deflectors with rounded noses were placed under each canister on the GGT bottom surface to deflect the plume jet with lower total-pressure losses. Two-side uptakes are provided at two GGT corners to give more angular space for



a) b)

Fig. 3 Pressure-distribution comparison in the launch tube a) cold flow and b) hot flow.



a) b)

Fig. 4 Schematic of alternate ducting system of VLS a) elevation and b) plan view.

the deflected jets to freely expand before exhausting to ambient. Chamfered corners are also provided for smooth turning of flow at corners and avoiding flow separation in the uptakes. A conical convergent duct from the canister to the GGT is provided to act as a diffuser for the expanding plume jet.

The computational domain consists of GGT, uptake, and active canister. In the active canister, the flowpath consists of the nozzle flowpath with four undeflected jet vanes in the nozzle exit, the internal annular passage created by the missile outer surface, and the canister inner surface. Figure 4b depicts a vertical plane (plane 1) that passes through the missile axis and is perpendicular to the two uptakes and another vertical plane (plane 2) that also passes through the missile axis but runs parallel to the uptakes. A multiblock structured grid that consists of 6.2 million hexahedral cells is generated in the computational domain using commercial grid-generator ICEM-CFD. Simulations are carried out with two different grids, namely 3.5 million and 6.2 million grids, and the surface of the bottom plate of the GGT is compared in Fig. 5. A close match between the two results demonstrates the grid independence of the solution.

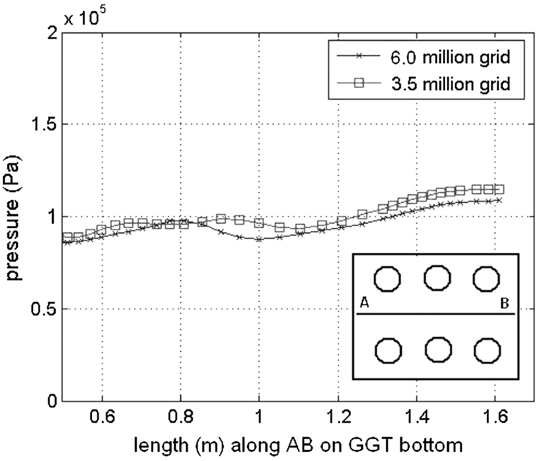


Fig. 5 Comparison of GGT-bottom-plate surface pressure with two different grids.

Table 1 Thermochemical properties of air and rocket exhaust

Properties	Exhaust gas	Air
Maximum total pressure, Mpa	9.0	0.1
Total temperature, K	3400	300
Specific heat at constant pressure, J/kg K	2138.6	1006
Molecular mass, kg/kg-mol	28	28.966
Thermal conductivity, W/mK	0.424187	0.0242
Molecular viscosity, Ns/m^2	9.34×10^{-5}	1.789×10^{-5}

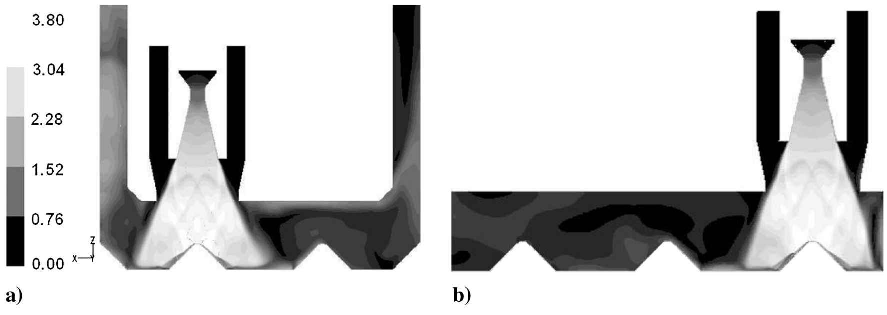
B. The Problem Setup and Boundary Conditions

Numerical simulations were performed in a hang-fire situation (the rocket motor is exhausting plume without being released from the canister), which is most critical for the design as it creates the most adverse flow situation within the GGT chamber. The jet vanes that are provided to control the missile in the initial launch phase are kept at zero deflection. Thermochemical properties of hot gas and air are summarized in Table 1. At the motor inlet, pressure and temperature conditions are prescribed. Because the flow through the canister opening is almost zero, flow is considered incompressible, and an air inlet at 1 bar pressure and 300 K temperature is used. Adiabatic wall conditions are imposed on all solid walls.

C. Results and Discussion

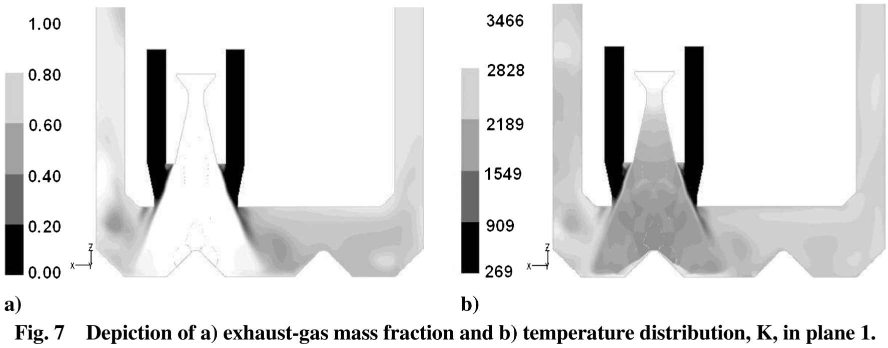
Mach-number distributions in two cross-sectional planes (plane 1 and plane 2) are shown in Figs. 6a and 6b. It can be seen in the figures that the jet retains its structure until it impinges on the conical deflector. The more bulging in the jet is due to the presence of the jet vane at the end of the rocket motor. The conventional single-uptake design showed a larger unsteadiness of jet flow, which causes a huge fluctuation of pressure load at the bottom GGT surface. The present design shows no distortion of the jet, and consequently, a lesser variation of pressure load is experienced. The computed temporal variation of nondimensional structural load (ratio of pressure integral on the bottom plate of GGT to motor thrust) is 2.8 times less than that of conventional design.

The exhaust-gas mass fraction and temperature distribution (K) of the alternate-ducting design in plane 1 is presented in Fig. 7. We could see that no hot gas from the GGT is recirculating back towards the missile. On the contrary some amount of cold air is entrained in the



a) b)

Fig. 6 Mach-number distribution a) plane 1 and b) plane 2.



a) b)

Fig. 7 Depiction of a) exhaust-gas mass fraction and b) temperature distribution, K, in plane 1.

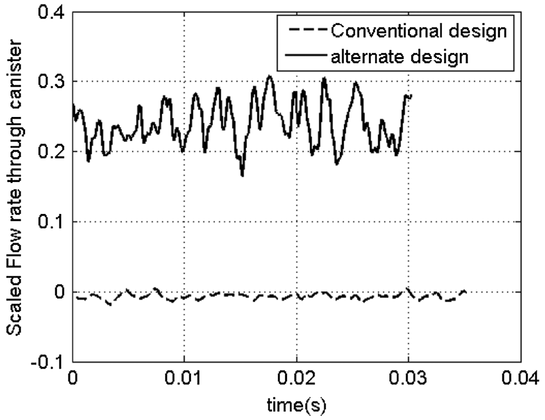


Fig. 8 Comparison of mass flow rate of gas flow through canister opening.

GGT through the opening between the canister and the missile. This feature of entrainment is very much desirable for the efficient thermal design of the VLS. Figure 8 compares the temporal variation of mass flow rate (scaled with motor flow rate) through the canister between the alternate and conventional design. A positive value means entrainment of cold ambient air into the GGT chamber through the canister, and a negative value means backflow of hot gas from the GGT to ambient through the canister. It is observed that 1 kg/s of rocket exhaust is recirculating back towards the missile base for the conventional design; the net mass flow rate of hot gas for the present design is almost negligible. In fact, 15 kg/s of cold air has entered into the GGT through the annular gap between the missile and canister.

V. Conclusions

Three-dimensional viscous simulations are carried out to study the underexpanded plume impingement in a confined environment. Unsteady Navier–Stokes equations are solved along with a k-ε turbulence model using commercial computational-fluid-dynamic tools. The computational tool is first validated against experimental results of a static test of a rocket motor with cold gas as well as double base solid rocket propellant inside a launcher tube with a diameter slightly larger than the nozzle exit. The simulation captured all the pertinent flow features and the computed surface pressure matches well with the experimental results both for cold- and hot-flow cases. The validated computational methodology is used to design a compact plume-ducting system for a vertical launcher. The alternate ducting system includes a number of novel features like two uptakes, conical deflectors, a convergent duct, etc., compared to the conventional plume-ducting system. The new design exhibits much well-behaved flowfield inside the plenum chamber with much less fluctuations, and the structural load due to pressure is 2.8 times lower compared to a conventional design. It is found that cold air is entrained into the gas-gathering tank through the annular space between the canister and missile surface thus reducing the thermal load in a vertical-launch system.

References

[1] Bertin, J. J., and Korst, H. H., "Launch-Tube Flow-Field Technology Based on Cold-Gas Simulations, Flight-Test Measurements and Theoretical Models," Fluid Dynamics Lab., Univ. of Texas Rept. 83-102, Austin, TX, April 1983.

[2] Bertin, J. J., Bertin, R. S., Yung, A., and Soohoo, G., "The Launch-Tube Flow-Field for a Vertical Launching System," AIAA Paper 88-0332, 11–14 Jan. 1988.

[3] Batson, J. L., and Bertin, J. J., "Rocket Exhaust Flow in Tube Launchers," *Journal of Spacecraft and Rockets*, Vol. 11, No. 11, Nov. 1974, pp. 739–740.
doi:10.2514/3.62170

[4] Lee, K. S., Hong, S. K., and Park, S. O., "Supersonic Jet Impingement Navier–Stokes Computations for Vertical Launching System Design Applications," *Journal of Spacecraft and Rockets*, Vol. 41, No. 5, Sept.–Oct. 2004, pp. 735–744.
doi:10.2514/1.13077

[5] Lee, K. S., Hong, S. K., and Ahn, C. S., "Numerical Investigations of Supersonic Jet Impingement on a Flat Wall in a Confined Plenum," *Proceedings of the 6th Computational Fluid Dynamics Conference*, National Taiwan Univ., Taiwan, Republic of China, 24–27 Oct. 2005; also Paper No. 2A-3.

[6] Ansys Fluent 6.3, Software Package, Ansys Inc., 2006.

[7] Roe, P. L., "Characteristic Based Schemes for the Euler Equations," *Annual Review of Fluid Mechanics*, Vol. 18, 1986, pp. 337–365.
doi:10.1146/annurev.fl.18.010186.002005

R. Cummings
Associate Editor