VENTING PERFORMANCE OF A TYPICAL PAYLOAD FAIRING USING TRANSIENT CFD APPROACH

Ankur Nagpal, Asish Raj S R, Sharjad AJ, Dipankar Das
Vikram Sarabhai Space Centre, Thiruvananthapuram.

ABSTRACT

Space launch vehicles like rockets are configured with payload fairing, which encapsulates the satellite and venting is necessitated by structural design constraints. A conventional approach to predict venting performance is to solve mass transfer equation for the compartment with coefficients of pressure and discharge as critical inputs along with geometric and trajectory details. In this work, venting performance of payload fairing is computed using transient flow simulations carried out using ANSYS Fluent, where external flow and venting flow through the compartment are solved in a coupled manner. Differential pressure on the compartment is compared between conventional approach and transient flow approach results.

Keywords: Payload fairing, differential pressure, vent whole, transient simulation, Fluent.

INTRODUCTION

Space launch vehicles have separate compartments assigned for stages, payload etc. Payload fairing is the uppermost region which protects the payload such as satellites from thermal and aerodynamic loads. The compartment gets filled with sea-level air before launch and as the vehicle gains altitude, there is reduction in free-stream pressure. The differential pressure thus created may lead to build up of structural loads exceeding the design criteria. Increase in structural strength for upper structures is limited by mass constraints, as it has first order payload loss. Hence adequate venting provision is required so as to maintain differential pressure within the design limits.

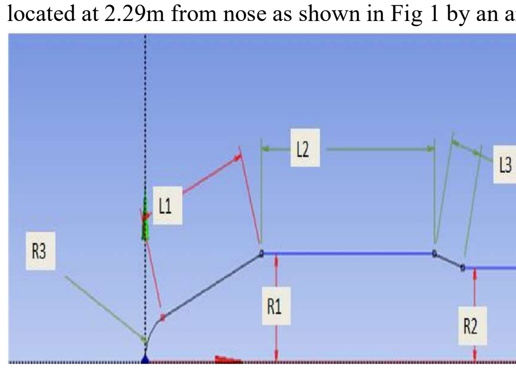
Venting performance calculation are conventionally carried out by solving mass transfer equation for the compartments[1, 2].This approach requires geometric information, trajectory details, coefficient of pressure (Cp) due to external flow and coefficient of discharge (Cd). Cp details are generally obtained from experiments or steady CFD calculations where influence of vent flow on external flow is neglected. Cd has strong dependence on external cross flow [3] and hence semi-empirical laws describing Cd behavior are used. In case of non-standard vent geometry, experiments are often carried out in presence of cross flow at various pressure and momentum ratio to characterize coefficient of discharge. Post flight analysis from various missions' show that pre-flight predictions with their bounds capture the flight trend in general [4].

An alternate approach, transient computational fluid dynamics simulations with connected flow domain on both external and internal regions, doesn't need inputs of Cp and Cd. External aerodynamic simulation is carried out in presence of Vent flow, where compartment pressure itself evolves as a solution of transient flow problem. Geometric and trajectory details are sufficient to carry out these simulations. Although This approach turns out to be computationally expensive, but increase in accuracy of venting performance prediction will justify the additional efforts.

In this paper, axisymmetric transient flow simulations are carried out for a launch vehicle payload fairing geometry [5] with typical trajectory conditions using ANSYS Fluent. Conventional mass transfer equations are also solved separately by using Cp inputs from steady simulation results. The venting performance thus obtained from two approaches is compared.

ANALYSIS DETAILS

The payload fairing geometry considered for this analysis is shown in Fig 1, and is adapted from [5] with 1m as the major diameter for heat shield. Vent hole is considered to be a circumferential slit of area 40.21 cm^2 and Width 1.28mm, located at 2.29m from nose as shown in Fig 1 by an arrow



R1	0.5m
R2	0.437m
R3	0.219m
L1	0.856m
L2	1.406m
L3	0.239m

FIGURE 1: GEOMETRIC DETAILS OF PAYLOAD FAIRING

Structured computational mesh is generated for external flow with sufficient refinement near the wall leading to total 55000 cells. First cell height is 5 microns and wall y+ <1 is ensured at the wall. The domain extends 30m on all sides and far-field boundary and wall boundary conditions are used. Steady simulations are carried out for Mach number range 0.5 to 1.5 and Cp at vent location is extracted for mass transfer calculations. Venting performance is computed using conventional approach considering typical trajectory conditions for the same Mach number range. Transient simulation is performed with M=0.5 solution as initial condition, and compartment pressure is patched to a given value initially. Time varying boundary conditions i.e. Mach number, Pressure and Temperature are specified at the far-field boundary using a user defined function (UDF).

MASS TRANSFER ANALYSIS

The conventional approach is built upon a theoretical intensive approach using quasi-steady mass transport equations. This approach provides simple and elegant solution to the problem of venting with very less turnaround time. In this work, single compartment vent analysis program in Microsoft Excel. The differential pressure at each instant is found out by integrating mass transfer equation. The input values are Cp at vent location, Cd of vent hole, Volume of air and trajectory data as a function of time. The chamber is considered to be adiabatic and we consider the ideal gas equations.

Assuming the compartment as isothermal, isentropic flow through the vent hole to be quasi steady, we can write

$$\frac{d}{dt}(P_c) = \left(\frac{n}{V}\right)\left(\frac{P_c}{\rho_c}\right)\left(\frac{dm}{dt}\right)$$

Eq-1

Where, Pc is chamber pressure, ρ_c is the chamber air density, n is the polytropic index taken as 1 for isothermal process, V is the volume of the chamber. In reality, the process is a polytrophic one with index varying in the range 1.1 to 1.2. However, this study is limited to isothermal process only.

The mass flow rate is given by [1]

$$\frac{dm}{dt} = -C_d * A\sqrt{\left|\left(\frac{2\gamma}{\gamma-1}\right)P_c * \rho_c * \left(1-\left(\frac{P_l}{P_c}\right)^{\left(\frac{\gamma-1}{\gamma}\right)}\right) * \left(\left(\frac{P_l}{P_c}\right)^{\left(\frac{2}{\gamma}\right)}\right)\right|}$$

Eq-2

Where P_l is the local pressure, γ is the specific heat ratio, C_d is the discharge coefficient, and A is the area of the vent hole. Substituting equation (2) in equation (1) and rearranging we get

$$\frac{dP_c}{dt} = \frac{-Cd*A*n*P_c}{V}\sqrt{\left|\left(\frac{2\gamma}{\gamma-1}\right)R * T_c * \left(1-\left(\frac{P_l}{P_c}\right)^{\left(\frac{\gamma-1}{\gamma}\right)}\right) * \left(\left(\frac{P_l}{P_c}\right)^{\left(\frac{2}{\gamma}\right)}\right)\right|}$$

Eq-3

The chamber pressure P_c at each time instant can be numerically computed by using Euler integration.

$$P_c(n+1) = P_c(n) + \frac{dP_c}{dt}\Delta t$$

Eq-4

In order to establish a suitable time step, a representative input set is considered and computations are made for time steps of 0.01s, 0.1s and 1s and differential pressure is shown in Fig 2. Based on the time step convergence study, a time step of 0.1s for all further mass transfer calculations is assumed.

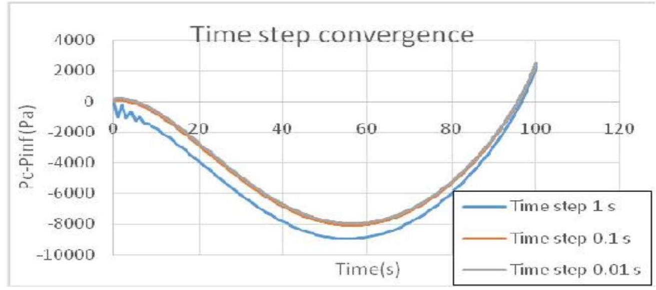


FIGURE 2: TIME STEP CONVERGENCE FOR DIFFERENTIAL PRESSURE FROM MASS TRANSFER COMPUTATION.

RESULTS AND DISCUSSIONS

Steady flow simulations are carried out at discrete Mach number conditions for the trajectory and Cp contour for A typical transonic condition i.e. M=0.9 is shown in fig 3 along with axial Cp distribution ploy.

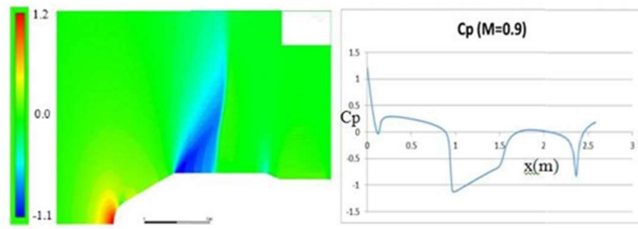


FIGURE 3: Cp CONTOUR ON THE PAYLOAD FAIRING AT MATCH NUMBER 0.9

Flow acceleration beyond stagnation point on the spherical nose portion, near constant pressure on cone, acceleration at cone-cylinder junction and terminal shock in cylindrical region are visible in Fig 3. It can be observed that value of Cp at the vent location is -0.15. In the transonic region, values of Cp show drastic variation at the vent location as shown in Fig 4. In the subsonic region values of Cp are near -0.25 owing to pressure recovery occurring in cylindrical region. Beyond Mach number 0.9, terminal shock passage occurs leading to sudden variation in Cp at vent location. At supersonic speed, as Mach number increases, Cp Approaches 0 at the vent location on cylindrical region. These Cp values are used for mass transfer calculations in single compartment venting performance analysis.

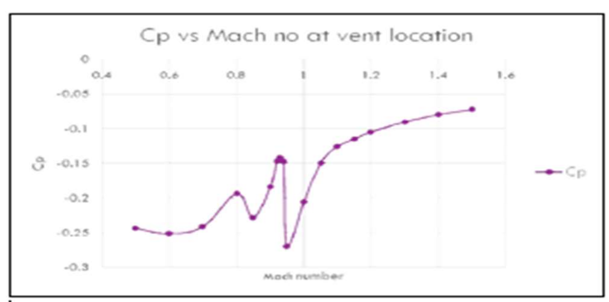


FIGURE 4: Cp VARIATION AT VENT LOCATION.

Transient flow simulation has been carried out with time-step of 0.01s and 0.05s with 50 inner loop iterations while employing dual time stepping transient method. At t=0, payload fairing is initialized with pressure 1000Pa more than the ambient pressure and simulation is carried out for 37.5 seconds. Mach number increases from 0.5 to 1.5, ambient pressure decreases from 85kPa to 20kPa and temperature drop from 290K to 220K during the time considered. The values of differential pressure i.e. (Pc-Pinf) thus obtained are shown in

Fig 5. Variation in differential pressure trend is seen during 15-20s, due to transonic freestream flow. The maximum differential pressure observed is -6kPa, which is generally in the acceptable design limits. Also shown if Fig 5 is the differential pressure computed from vent code where mass transfer computations are carried out with Cp values discussed earlier and a constant Cd = 0.2. The value of Cd can be derived from the transient flow simulation data by normalizing actual mass flow rate by ideal mass flow rate. It is observed that the two approaches show similar trend throughout the critical Mach number regime studied.

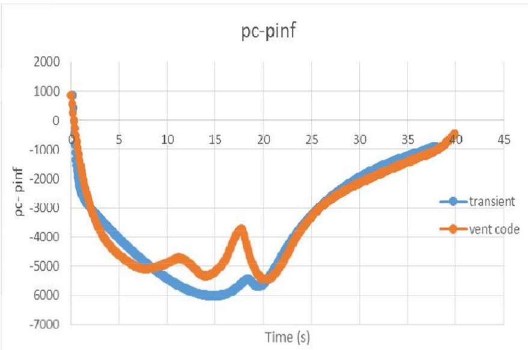


FIGURE 5: DIFFERENTIAL PRESSURE VS TIME FROM TRANSIENT FLOW SIMULATION AND MASS TRANSFER COMPUTATION.

CONCLUSIONS

Transient flow simulation approach has been used to compute single compartment vent performance and the same is compared with that computed using conventional mass transfer calculations. The two approaches show a good match for the value of Cd derived from transient flow computation. The merit of transient approach is in the fact that values of Cp and Cd are not required as inputs and also the interaction between external flow and vent flow is considered appropriately

REFERENCES

[1] M.S.Sastry, 1998. “Aerodynamic Design Criteria Monograph For Launch Vechicle”, VENT HOLE DESIGN, Thiruvanthaapuram: VSSC, ISRO.

[2] Francisco, J., 1975. "'NASA Space Vehicle Design Criteria," Compartment Venting, NASA , CALIFORNIA: NASA

[3] Mehta, R. C., 2002. Estimation and Comparison of Discharge Coefficient for Satellite Launch Vehicles Pay Load Fairing Vents. s.l., 20th AIAA Applied Aerodynamics Conference.

[4] Mehta, R. C., 2017. Analysis of Payload Compartment Venting Of Satellite Launch Vehicle, Advances in Aircraft And Spacecraft Science, 4(4), pp. 437-447.

[5] Rao, G. D., 2008. “Heat Flux Estimation of Bulbous Heat Shield at Hypersonic Mach Number” Vijayawada, 2nd International Conference on Recent Advance in Experimental Fluid Mechanics.