Aerodynamics and flow characterisation of multistage rockets

Srinivas G1, M V S Prakash*2

1Assistant Professor, Dept of Aero & Auto Engineering, Manipal Institute of Technology, Manipal University, Manipal, Udupi, Karnataka, India-576104.
*2UG student, Dept of Mechatronics Engineering, Manipal Institute of Technology, Manipal University, Manipal, Udupi, Karnataka, India-576104.

Abstract. The main objective of this paper is to conduct a systematic flow analysis on single, double and multistage rockets using ANSYS software. Today non-air breathing propulsion is increasing dramatically for the enhancement of space exploration. The rocket propulsion is playing vital role in carrying the payload to the destination. Day to day rocket aerodynamic performance and flow characterization analysis has becoming challenging task to the researchers. Taking this task as motivation a systematic literature is conducted to achieve better aerodynamic and flow characterization on various rocket models. The analyses on rocket models are very little especially in numerical side and experimental area. Each rocket stage analysis conducted for different Mach numbers and having different flow varying angle of attacks for finding the critical efficiency performance parameters like pressure, density and velocity. After successful completion of the analysis the research reveals that flow around the rocket body for Mach number 4 and 5 best suitable for designed payload. Another major objective of this paper is to bring best aerodynamics flow characterizations in both aero and mechanical features. This paper also brings feature prospectus of rocket stage technology in the field of aerodynamic design.

1. Introduction

The design of rocket model shape is very difficult job to the aerodynamic researcher because of its complexity in calculation of structural mass, inert mass and payload mass. However the designer also need to concentrate on the efficiency of the rocket in terms of thrust produced per unit time. Rocket model can make a simple way like cardboard tube completely bursting with black powder but to create an efficient and correct rocket implicates more number of complicated issues [1]. Some of the complicated difficulties like maintaining the desired combustion heat, maintaining the appropriate pressure of the fuel supplying into the combustion chamber, controlling the thrust levels producing from the rocket engine and most importantly the controlling the direction of motion. The rocket direction of motion again directly proportional to the aerodynamic shape of the rocket. This paper more concentrate on the desired shape to get better aerodynamic results in both single stage and double stage rocket model.
2. Literature survey on multi stage rockets

2.1 Forces on a rocket body

Aerodynamic study of the rocket body is the part of ballistics filed, whereas the spacecraft are studied in subdomains of the astrophysics. According to the rocket theory four forces lift, drag, gravity and thrust commonly act on the flying rocket [7]. Figure 1 shows four forces acting on flying rocket model. Any rocket travelling in the air must have thin in size and tall for providing high ballistic coefficient value as it makes very minimum drag losses. When high adequate speed in the exact direction and altitude are reached a stable escape velocity or orbit is obtained by addition centrifugal pseudo force and inertia can be significant due to path of the rocket around the centre of a celestial body [2-3].

![Figure 1](Reference: wikipedia.org)

**Figure 1.** Forces acting on a flying rocket body [Reference: wikipedia.org]

![Figure 2](Reference: wikipedia.org)

**Figure 2.** Computational fluid dynamic (CFD) rocket model solving flow chart
Rocket body when its flying at higher altitudes obviously it follows parabolic trajectory termed a gravity turn and this trajectory is frequently used at least during the initial part of a launch [4]. Rocket body can thus maintain even zero or low angle of attack which minimizes the stress due to the transverse forces on the launch vehicles, permitting a very frailer and later lighter, launch vehicle. Opposite to the direction of rocket body motion is drag force. This force decreases by increasing the acceleration of the rocket body and structural loads. Drag equation used to calculate the deceleration forces for fast moving rockets. The drag equation is showing in equation (1) where the term \( A \) is the reference area of the rocket body, \( v \) is the flow velocity, \( \rho \) is the mass density and \( F_D \) is the drag force.

\[
F_D = \frac{1}{2} \rho U^2 C_D A
\]  

The ambient atmospheric pressure almost always different than exit pressure of the rocket jet. After rocket engine ignited on the ground the atmospheric pressure is maximum than pressure in the rocket jet. As the rocket body moves to higher altitude the jet pressure remains constant while the atmospheric pressure decreases, the rocket exit jet becomes slowly start growing, very concise becoming idyllically expanded although on its way to underexpansion. As the rocket body continues to increase the altitude, as the rocket is migrant into a vacuum, the rocket exit jet diameter increase to very high proportions. That is the reason as the rocket gains altitude rocket jet diameter increases. The jet diameter is again directly proportional to the area covered by the body miscellaneous component like nozzle, fins, engine location etc. This paper also discuss the entry and exit portion of the rocket body aerodynamic behaviour.

2.2 Rocket body staging

Rocket total mass includes structural, propellant, payload and also miscellaneous mass. Carrying this complete mass by the rocket to the higher altitude and reaching to the destination is very complicated to task to the researcher. There it helpful the concept called ‘staging’. The rocket leaves the excess weight like empty tanks, related engines during launch. Comparing with the speed of light, the extreme speeds that can be achieved with the staging is ideally limited only. To achieve better incremental velocity of the staging carried goes down geometrically with each additional stage required. This paper also discuss about the aerodynamic behavior of such additional stage to get incremental velocity using ANSYS software.

2.3 Safety and reliability of rocket

The safety and reliability of the rocket that depends on the design and materials used. Rocket reliability as per all physical components is based on the quality of engineering proportions in design and construction also. Due to enormous chemical energy release from exit of the rocket propellant consequences of coincidences can be austere. Generally most of the space missions have some issues either with the geometry or due to improper assembly of the sub domains of the rocker or propellant used in the rocket motor [9-10]. Therefore the body aerodynamic behavior also plays a vital role in the field of rocket science, in this paper focus is given to aerodynamic behavior for better safety and reliability of the rocket [5].

3. Role of computational fluid dynamics on rocket body

Experimental simulations on rocket models using the wind tunnel are more expensive than solving numerically using CFD. Experimental work involves higher cost for set up, complexity is more in construction and also difficult to set all the boundary conditions. Time for the analysis also more, need to undergo several prototypes to predict the accurate results through experimental work. Nowadays to overcome all these problems the numerical simulations are very much handy to researcher [6, 7].
To solve the aerodynamic behavior of the rocket model the well-known computational fluid dynamics (CFD) tool was used to predict the flow behavior properties. In this work computers are used to perform the aerodynamic as mentioned in the above theory calculations required to simulate the interaction of the air with rocket surface defined by boundary conditions. With the speed computational facility good results can be achieved. In this paper research yields CFD tool that improves the speed and accuracy of complications like turbulent or transonic flows. The basic theory and basic of all Turbulent flow are the Navier stokes equation see in equation(2) where the term ‘w’ indicates specific thermodynamic work, ‘v’ kinetic viscosity, ‘u’ velocity profile which helpful in many single phase fluid flows[8].

\[
\frac{\partial u}{\partial t} + (u \nabla) u - v \nabla^2 u = -\nabla w + g \tag{2}
\]

4. Methodology

In all the CFD methods same basic procedures are followed. During the pre-processing geometry defined based on the problem statement. After successful completion of geometry the body volume is divided into number of discrete elements or cells or mesh. In this paper for the better flow solutions structured grid was created. Consequently the physical modelling mentioned in the software for a given rocket model like enthalpy, species conservations and equation motion. The most important step in the CFD methodology is giving suitable boundary conditions for the rocket single stage and double stage involves specifying the fluid properties and behaviour at the boundaries of the problem statement. Once the setting of boundary conditions are done then simulations are started through solving the equations iteratively till solution get convergence under steady state or transient conditions. Lastly, the post processor is used for visualizing of the iterated solution and analysis of the rocket staging.

Figure 3: ANSYS workbench three stage rocket model.

Figure 4: Representation of farfield around the rocket model in ANSYS workbench.
Through the public literature survey the standardized basic rocket model was selected. After completing the successful convergence on the profile the optimized rocket model was created. Figure 3 shows three stage rocket model. The same geometry was imported into the ANSYS workbench where the geometry was scale down accordingly. As like experimental test section set up, the similar geometry called ‘farfield’ created around the rocket model see in the figure 4. The structured grid was created around the rocket model 71 million cells for east stage see in figure 5. Near the body surface the density of the mesh elements are more to predict the accurate results like shock wave propagation and aerodynamic properties. After setting appropriate elements attributes, mesh controls selection the required mesh created. The CFD methodology broad perspective flow chart shown in figure 2.

![Figure 5: Mesh generation on rocket model in ANSYS.](image)

The most important dimension in the numerical simulation is setting the suitable boundary conditions to single stage, double stage and three stage rocket. The flow simulations was run for different Mach numbers like M=3, 4 and 5. Till the fuel consumed by the rocket each stage separation time and also the corresponding altitude also calculated. From the calculated altitude, the density and mass flow rate also given to the farfield using atmospheric data. By the time all three stages separated from payload full propellant mass consumed for additional incremental velocity to enter into orbital velocity. For all three different stages boundary conditions remains same only the Mach number entry velocity and exit pressure only changes every time. All these calculations theoretically very strong since the experimental data over multistage rocket very inevitable.

Comparison also made in this analysis especially at the nose tip notable changes also observed, very maximum at the inclined profile and that too end corner of the rocket model also. The assumption in this analysis is flow pattern similar for all models shapes as the length of the each stage consider to be same.

5. Results and Discussions

The simulations were performed for single, double and multistage from mach speed 4, 5 and 6. All cases shown from figure 5-32. However the numerical results were presented only mach speed 3 double stage rocket model met the design criteria because the boundary conditions are well matched with theoretical calculations see in figure 3-8. Comparatively the rocket body from double stage to multi stage mass and propellant varied independently by ± 10% for a nominal value resulting in a combination of 16 cases for each parameter. Apart from these results it may note that the nominal mach speed value 3 nose tip shockwaves are capable of handling losses due to the aerodynamic flow interactions to keep the body accurate incremental velocity, density, temperature, pressure within the design data base limit shown from figure 9-21.
The small values of rocket path angle at three different stage separation are 42 deg, 52 deg and 65 deg and altitude 78x10^3 ft, 82x10^3 ft and 86x10^3 ft respectively. The values were varied ±4000 ft and ±13 deg respectively. It is noted that these number differences can be controlled by the normal mach speed 3 and 4 satisfactorily and maintaining the rocket body incremental velocity, density, pressure and temperature with the design data base shown from figure 22-33.

**Figure 6**: Single stage rocket density contour at mach speed 3

**Figure 7**: Single stage rocket static pressure at mach speed 3

**Figure 8**: Single stage rocket velocity contour at mach speed 3

**Figure 9**: Single stage rocket density contour at mach speed 4
Figure 10: Single stage rocket static pressure at mach speed 4

Figure 11: Single stage rocket temperature contour at mach speed 4

Figure 12: Single stage rocket density contour at mach speed 5

Figure 13: Single stage rocket pressure contour at mach speed 5
Figure 14: Single stage rocket static temperature contour at mach speed 5

Figure 15: Double stage rocket density contour at mach speed 4

Figure 16: Double stage rocket density contour at mach speed 4
Figure 17: Double stage rocket pressure contour at mach speed 4

Figure 18: Double stage rocket static pressure contour at mach speed 4

Figure 19: Double stage rocket static pressure contour at mach speed 4

Figure 20: Double stage rocket total pressure contour at mach speed 4
Figure 21: Double stage rocket pressure contour at mach speed 4

Figure 22: Three stage rocket velocity contour at mach speed 3

Figure 23: Three stage rocket velocity contour at mach speed 3

Figure 24: Three stage rocket dynamic pressure contour at mach speed 3
**Figure 25:** Three stage rocket density contour at mach speed 3

**Figure 26:** Three stage rocket density contour at mach speed 4

**Figure 27:** Three stage rocket static temperature contour at mach speed 4

**Figure 28:** Three stage rocket total pressure contour at mach speed 4
Figure 29: Three stage rocket dynamic pressure contour at mach speed 4

Figure 30: Three stage rocket density contour at mach speed 5

Figure 31: Three stage rocket static pressure contour at mach speed 5

Figure 32: Three stage rocket total pressure contour at mach speed 5
6. Conclusions

The analysis and flow simulation over the multistage, double stage and single stage rocket body vehicle concepts, single stage at mach speed 4 and double stage at mach speed 6 were performed outstandingly to the real-time criteria application. A passive agreement between the single stage and multistage rocket body mach speed 4 and 6 staging does not result in reasonable stage separation because the vehicle propellant mass and relative aerodynamic losses went very much outside the design data limits. However these combinations well suitable for mach speed 3 due to lower shock propagation strength at the entry of the nose tip. Further mach speed 6 combination with three rocket configurations well feasible at aerodynamic behavior to have better aeromechanical features. In all the 16 cases it is also investigated the best flow properties like pressure, temperature, density and velocity at entry and exit face of the rocket body.

References

[1] Sivaraj G., K. M. Parammasivam, M. Gokulraj, 2013 Computational Analysis of Flow Behaviour over the Multistage Launch Vehicle with Strapons, Journal of Mechanical Engineering and Technology. Volume 1, Issue 1, pp. 54-65.
[2] N. Sreedhar, G. Ravindra Reddy 2013, Flow simulation over a Multi-Stage Launch Vehicle with Strapons using CFD techniques International Journal of Computer Trends and Technology, volume 4 Issue7, pp 2246-2249.
[3] 2015, Computational Fluid Dynamics (Cfd) In Launch Vehicle Applications Practice No. Pd-Ap-1311, http://llis.nasa.gov/lesson/758.
[4] Soon-Heum Koand Chongam Kim 1997 Separation Motion of Strap-On Boosters with BaseFlow and Turbulence Effects Journal of Guidance control and dynamics, Volume 20, No.4.
[5] S.K.Chakrabarty,K. Dhanalakshmi an J. S. Mathur Computation of flow past aerospace vehicles http://www.iisc.ernet.in/currsri/nov25/articles18.html
[6] Robert E. Bartels Pawel Chwalowski, Steven J. Massey, Jennifer Heeg, Carol D. Wieseman 2012 Computational Aeroelastic Analysis of the Ares Launch Vehicle During Ascent American Institute of Aeronautics and Astronautics, PP 1-20.
[7] Arash Naghib-Lahouti, Amir Nejat, Taravat Khadivi. 2012 Parametric Analysis of Aerodynamic Characteristics of Launch Vehicles with Strap-On Boosters ICAS 2002 CONGRESS, pp 264.1-264.12.
[8] Soon-Heum Ko, Chongam Kim, Kyu Hong Kim 2006 Separation Analysis of Strap-Ons in the Multi-Stage Launch Vehicle Using the Grid Computing Technique 44th AIAA Aerospace Science Meeting and Exhibit, Reno, Nevada.

[9] R. Venkatraman, S. Sankaran, G. Krishnaiah, P. Malleswara Rao, P. K. Vivekanand and T. Sundararajan 2015, Understanding Ignition over Pressure During Start Up of Typical Scaled Down Solid Rocket Motors, *Indian Journal of Science and Technology*, Vol 8(5), 481–488.

[10] Valeriy 2016 Developing a Test Stand to Verify Accuracy of Conducted Research at the Early Design Stages of Active Descent Systems of Spent Launcher Stages Valeriy Ivanovich Trushlyakov and Kseniya Alekseevna Rozhaeva, Indian Journal of Science and Technology, Vol 9(36), September 2016. DOI: 10.17485/ijst/2016/v9i36/102037.