Prediction of solid rocket motor performance characteristics using computational fluid dynamics and validation with experimental data

Venkata Naga Mohan Manchiraju*a, Vijay Kumar Dwivedi*b
*aScientist, Advanced Systems Laboratory, Dr.APJ Abdul Kalam Missile Complex, DRDO, Ministry of Defence, Government of India, Hyderabad-500058,
*bDepartment of Mechanical Engg., IET, GLA University, Mathura, U.P., India
Email: a mvnmohan33@gmail.com, b vijay.dwivedi@gla.ac.in

Abstract. Solid Propulsion is widely used in Missile and Space applications. Accurate prediction of Specific impulse ($I_{sp}$) is important for design of Solid rocket motor. These predictions can be done using empirical relations. Obtaining these empirical relations require large amount of data. The renowned aerospace organizations use their own code for prediction. These codes are not open source and also developing a code requires big team effort with challenges of verification and validations. The present work is aimed at developing numerical model for prediction of Specific Impulse. The model is validated against the experimental data of static test of solid rocket motor taken from literature.

Key words: Computational Fluid Dynamics, Specific Impulse, Solid Propulsion

1. NOMENCLATURE

1.1 Symbols

- $F =$ Thrust in N
- $\dot{m} =$ mass flow rate in kg/s
- $A =$ Area in m$^2$
- $V =$ Velocity in m/s
- $P =$ pressure in Pa
- $\rho =$ density in kg/m$^3$
- $I_{sp} =$ Vacuum Specific Impulse
- $g_0 =$ Acceleration due to gravity in m/s$^2$

1.2 Subscripts

- 1 nozzle inlet or chamber
- 2 nozzle exit
- 3 atmospheric or ambient

2. INTRODUCTION

Propulsion is the act of changing the motion of a body. The propulsive force of a solid rocket motor is obtained by ejecting propellant at high velocity. Accurate prediction of propulsion parameters is important for design of Solid rocket motor. CFD is a useful tool for the prediction of propulsion
parameters. Many researchers have made use of CFD for their research work to study, design, analyze or predict solid propulsion elements and parameters.

CFD has a very important role in rocket propulsion. Chongankimet et al. [1] explained the role of CFD in development of rocket propulsion. Nozzle is a component that increases the performance of air breathing and non-air breathing engines. G Srinivaset al. [2] analyzed the performance of convergent nozzles. It has a good comparison with experimental results. To achieve dependable CFD predictions, it is important that the numerical model be based on first principle. The models based on correlation may not be fully reliable. Jiri blazeket al. [3] described a flow solver useful to simulate rocket motors. The reliability of code is of utmost importance for analysis of flow in nozzles. Bogdon-AlexandruBelegaet al. [4] used fluent 6.3 for analysis of convergent divergent nozzle with GAMBIT 2.4 software for numerical modeling and generation of mesh. Nathan Spottsset al. [5] used Metacompcfd++ software to study compressible flow through convergent conical nozzles. The results have good comparison with experimental data.

Laura et al. [6] used Hopsan a multi domain software to analyse the functioning of sounding rocket. The results obtained are comparable with real performance. The simulations were further useful for improvement of performance and increase in altitude a rocket can reach. Supersonic exhaust diffusers were also analyzed using CFD. M Srinivasa Rao et al. [7] performed numerical simulations for various rocket chamber pressures. The results have good comparison with experimental data. SukantaRogaet al. [8] analyzed the scramjet combustor using CFD. He identified important parameters for optimization of injection system. K. Schomberget al. [9] use CFD for design of high-area-ratio nozzle contours using circular arcs. The design offers an improvement in thrust coefficient and reduction in average length. S. Sahaet al. [10] analysed combustion instability using CFD in solid rocket motors. The analysis was validated against motor test data available in literature.

Specific Impulse is one of the important characteristic of rocket propulsion. It is a measure of fuel efficiency of the rocket, that is, the thrust imparted to the rocket per kilogram of the propellant expelled. If two different rocket motors have twodifferent values of specific impulse, then the motor with higher value of specific impulse is treated more efficient. This is because the motor will produce more thrust for the same amount of propellant. It gives us an easy way to size a motor during preliminary analysis. The rocket weight will define the required value of thrust. Dividing the thrust required by the specific impulse will tell us how much weight flow of propellants our motor must produce. This information determines the physical size of the motor. Accurate prediction of Specific impulse (I_sp) is important for design of Solid rocket motor.

3. OBJECTIVE

- To perform computational study on the rocket motor nozzle using AnsysFluent 14.0 for understanding the flow losses and compare results with published data
- To understand the flow losses involved in solid rocket propulsion and predict the vacuum specific impulse.

4. METHODOLOGY FOR CFD PREDICTIONS FOR ROCKET PROPULSION

The work flow for the CFD predictions for rocket propulsion is given below:

4.1 Selection of CFD Software

In the present paper, the computational study is done on the rocket motor nozzle using ANSYS Fluent 14.0 for understanding the flow losses involved in nozzle and predicting the vacuum specific impulse I_sp and thrust. The code solves the following Navier-Stokes equations through finite volume method.
4.2 **Input Parameters**

The rocket motor used for testing had an overall length of 720 mm and outer diameter of 198 mm. The throat diameter is 41 mm and nozzle exit diameter is 317.7 mm. The motor had a propellant weight of 23.345 kg and operational pressure of 7.34 MPa.

4.3 **Pre-Processing**

4.3.1 **Geometric Modelling.** Using the coordinates as in ref 5 a 2D axi-symmetric geometric modelling is done using AutoCAD software. The Model is given in ‘Figure 1’.

![Figure 1.2axi-symmetric Geometric Model](image)

4.3.2 **Meshing and Validation.** Four different mesh sizes were taken for the setting up the model. All the meshes were Cartesian meshes as this kind of mesh is best suited for the problem. The elements chosen were quads. The four different quad sizes selected are 1mm, 0.5 mm, 0.3 mm and 0.1 mm. The meshes 0.3 mm and 0.1 mm are close to the actual tested mass flow rate within 2 percent. But the number of elements of mesh 0.1 mm is around 91,000. The initial solution took about 3 days to solve in a desktop. Due to computational constrain mesh of 0.3 mm is used for further calculation and prediction. The solutions arrived at are mesh independent.

4.4 **CFD Setup**

4.4.1 **Boundary Conditions and Gas Properties.** The CFD setup containing Boundary conditions and Gas properties is tabulated in Table 1

| Table 1 CFD setup |
|-------------------|
| **General**       | Solver Type: Density Based |
|                   | 2D Space: Axis Symmetric Model |
|                   | Ideal Gas equation |
| **Models**        | Energy Equation: on |
|                   | Viscous Model: k-omega model |
| **Boundary Conditions** | Inlet Pressure: 7.34 MPa |
|                   | Inlet Temperature: 3410 °K |
|                   | Outlet pressure: 0.001 atm |
|                   | Nozzle wall: No slip and adiabatic heat flux. |

4.4.2 **Run for Chosen No of Iterations and Validation.** For running the iterations the residuals for each flow equation was set at 10^-10. Initially 10 iterations were run for initialization under hybrid initialization. Then multiples of 500 iterations were run till the residual condition was met. It was noticed that there was no further change after 3700 iterations. The residuals were also found within acceptable limits. The residual plots is shown below in ‘Figure 2’.
4.5 Post Processing

4.5.1 Contour Plots

4.5.1.1 Pressure Contour Plot. The pressure contour plot is shown in ‘Figure 3’ the boundary condition defined at chamber inlet is 7.34 Mpa. The CFD predicted the outlet pressure at nozzle exit as 3.39kPa. This is because of gas expansion in the nozzle. The expansion waves are noticed at the start of nozzle divergent contour.

4.5.1.2 Temperature Contour Plot. The boundary condition of gas temperature defined at chamber inlet is 3410˚K. The gas temperature dropped from 3410˚K to 938˚K at nozzle exit. This is because of gas expansion in the nozzle. The kinetic energy required to expand into empty space comes from heat energy that gives the gas temperature. So the temperature is dropped. The ‘Figure 4’ shows variation in temperature across length of nozzle.
4.5.1.3 Velocity Contour Plot. The gas velocity increasing form stagnant condition to 3070 m/s at the nozzle exit as solved by CFD. As the temperature is decreasing in nozzle, this energy is utilized by the gas to gain kinetic energy, thus increasing the velocity. The thrust developed by the nozzle increases with increase in exit velocity. The velocity contour plot is shown in ‘Figure 5’.

4.5.1.4 Mach No Contour Plot. The Mach No, a ratio of gas velocity to sound velocity, is increasing from stagnant condition to Mach 5 at the nozzle exit as solved by CFD. Mach No is increasing due to increase in gas velocity in the nozzle as well as reduction in sound velocity. The flow is sub sonic in the convergent section, sonic at the throat and Supersonic in the Divergent section of the nozzle. The thrust developed by the nozzle increases with increase in exit velocity. The velocity contour plot is shown in ‘Figure 6’.
4.5.1.5 Density Contour Plot. The gas density at the chamber was 6.67 kg/m$^3$. It has decreased to 0.012 kg/m$^3$ at the nozzle exit as solved by CFD. The mass flow rate is a function of Density, Area and Gas velocity. The mass flow rate is constant along the length of the nozzle. As the area of nozzle and gas velocity are increasing, the density of gas is decreasing. The density contour plot is shown in ‘Figure 7’.

![Density Contour Plot]

**Figure 6.** Mach No Contour Plot

![Density Contour Plot]

**Figure 7.** Density Contour Plot
5.0 RESULTS AND DISCUSSION

5.1 Calculation of $I_{sp}$

The Mathematical calculations to determine the $I_{sp}$ and Thrust are given below:

$$\text{Thrust} = F = \dot{m}v_2 + (p_2 - p_3)A_2$$

Where

$F$ = Thrust in N
$\dot{m}$ = mass flow rate in kg/s = 5.96 kg/s
$A_2$ = Area at nozzle exit = 0.07926 m$^2$
$V_2$ = Axial Velocity at nozzle exit = 2859.5 m/s
$P_2$ = Static Pressure at nozzle exit = 9833.24 Pascal
$P_3$ = Ambient Static Pressure (Vacuum) = 101 Pascal

From this equation the thrust obtained from the CFD solution is 17814 N

The equation for $I_{sp}$ is:

$$I_{sp} = \frac{F}{\dot{m}g_0}$$

Where

$I_{sp}$ = Vacuum Specific Impulse
$F$ = Thrust in N = 17814 N
$\dot{m}$ = mass flow rate in kg/s = 5.96 kg/s
$g_0$ = Acceleration due to gravity = 9.81 m/s$^2$

The vacuum $I_{sp}$ calculated using the results of CFD is 305.1 s

5.2 Comparisons of propulsion characteristics CFD vis-à-vis Experimental data

| Parameter | From CFD code | Experimental data | Accuracy |
|-----------|---------------|-------------------|----------|
| $I_{sp}$  | 305.1 s       | 296.7 s           | 2.9%     |

The propulsion characteristics predicted through CFD is compared with the experimental data. The comparison table is given in Table 2. The $I_{sp}$ predicted using model developed in fluent is 305.1 s and matches very closely to experimental data. The results are within 3% accuracy. The pressure, velocity, temperature, Mach and density plots as discussed above follow the trend expected in an isentropic nozzle expansion. Thus it can be safely said the model developed is validated and the results can be used for design purpose.

6.0 CONCLUSIONS

CFD is a useful methodology in predicting the rocket motor / nozzle performance. The current project reveals that the accuracy of the prediction is better than 3%, which makes it reliable for the designers, scientists and engineer community. The CFD results can be used to accurately predict other parameters like temperature, pressure, velocity and density at any point in the nozzle. This data will help designers for selection of appropriate materials for nozzle fabrication. Flow features like expansion waves and oblique shocks occurring inside the nozzle can be captured accurately. This will help the designer to design the divergent contour free of any flow irregularities and flow separation. There is a future scope of improving the accuracy by using 3D simulations, which can be taken for further tasks up on availability of computational hardware.
7.0 REFERENCES

[1] Chongankim 2016 Roles of CFD simulations in developing rocket propulsion systems The 2016 structures congress (structures 16)
[2] G Srinivas et al 2017 IOP Conf. Series: Materials Science and Engineering 197, 01208
[3] J Blazek 2003 Flow Simulation in Solid Rocket Motors Using Advanced CFD, 39th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit AIAA 2003-5111
[4] Bogdan-alexandru belega and Trungducnguyen 2015 Analysis of flow in convergent-divergent rocket engine nozzle using computational fluid dynamics, International conference of scientific paper, AFASES 2015
[5] NSpotts and SGuzik and X Gao 2013 A CFD Analysis of Compressible Flow through Convergent-Conical Nozzles 49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference AIAA 2013-3734
[6] Laura, Navarrete-Martin and PetterKrus 2017 Sounding Rockets: analysis, simulation and optimization of a solid propellant motor using Hopsan6th CEAS air & space conference aerospace Europe 2017, CEAS 2017
[7] M. Srinivasa Rao, Afroz Javed and Debasis Chakraborty 2017 Numerical Characterization of Supersonic Exhaust Diffusers, Defence Science Journal 67(2) pp. 219-223
[8] Sukanta Roga 2019 J. Phys.: Conf. Ser. 1276 012041
[9] K. Schomberg and J. Olsen 2016 Design of High-Area-Ratio Nozzle Contours Using Circular Arcs Journal of propulsion and power. 32(1), pp.188-195
[10] S. Saha and D. Chakraborty 2016 Computational Fluid Dynamics Simulation of Combustion Instability in Solid Rocket Motor: Implementation of Pressure Coupled Response Function Defence Science Journal. 66(3) pp. 216-221
[11] Sutton G P 2001 Rocket Propulsion elements (New York: John Wiley & Sons)

ACKNOWLEDGEMENTS

While performing this work I came across many people, whose contributions helped my field of research in various ways. I express my gratitude to all of them. I thank Advisory Group for Aerospace Research and Development that published Technical report 230. The experimental data was taken from this technical report. I am thankful to Dr. MRM BABU, Director, Advanced systems Laboratory, Defence Research and Development Organization, Hyderabad for all the support extended for the current work. My specific thanks are due to Mr. Mrinmoy Biswas, Scientist, Advanced Systems Laboratory, and Hyderabad for extending his valuable support and resources in executing the current work. I thank GLA University for extending all the cooperation complete this work.