

文章编号: 1000-8055(2009)07-1654-07

Numerical simulation of multi-phase combustion flow in solid rocket motors with metalized propellant

SHAFQAT Wahab, XIE Kan, LIU Yu

(School of Astronautics, Beijing University of Aeronautics and Astronautics,
Beijing 100191, China)

Abstract: Multiphase flow field simulation has been performed on solid rocket motor and effect of multiphases on the performance prediction of the solid rocket motor (SRM) is investigated. During the combustion of aluminized propellant, the aluminum particles in the propellant melt and form liquid aluminum at the burning propellant surface. So the flow within the rocket motor is multi phase or two phase because it contains droplets and smoke particles of Al_2O_3 . Flow simulations have been performed on a large scale motor, to observe the effect of the flowfield on the chamber and nozzle as well. Uniform particles diameters and Rosin-Rammler diameter distribution method that is based on the assumption that an exponential relationship exists between the droplet diameter, d and mass fraction of droplets with diameter greater than d have been used for the simulation of different distribution of Al_2O_3 droplets present in SRM. Particles sizes in the range of 1–100 μm are used, as being the most common droplets. In this approach the complete range of particle sizes is divided into a set of discrete size ranges, each to be defined by single stream that is part of the group. Roe scheme flux differencing splitting based on approximate Riemann problem has been used to simulate the effects of the multiphase flowfield. This is second order upwind scheme in which flux differencing splitting method is employed. To cater for the turbulence effect, Spalart-Allmaras model has been used. The results obtained show the great sensitivity of this diameters distribution and particles concentrations to the SRM flow dynamics, primarily at the motor chamber and nozzle exit. The results are shown with various sizes of the particles concentrations and geometrical configurations including models for SRM and nozzle. The analysis also provides effect of multiphase on performance prediction of solid rocket motor.

Key words: solid rocket motor nozzle; multiphase flow;
Rosin-Rammler diameter distribution method;
Roe FDS (flux differencing splitting); performance prediction

CLC number: V434 **Document code:** A

Nomenclature

p chambers pressure
 R grain outer radius
 L grain length, mm
 m_p mass of propellant
 I_{sp} specific impulse
 t_b burning time

A_c/A_t area ratio
 c_p specific heat at constant pressure
 T_c combustion temperature
 γ specific heat ratio
 ρ density
 n spread parameters
 $\bar{d}$ mean diameter of particle

Received: 2008-09-22; revision received: 2009-04-20

E-mail: shafqatwahabqq@yahoo.com

μ	molecular viscosity of the fluid
ρ_p	density of particle
d_p	particle diameter
Re_p	particle phase Reynolds number
V_{rp}	relative velocity between particles and gas velocity

Introduction

The aluminum powders are used in solid rocket motor propellant for the purpose of increasing the motor specific impulse and reduce combustion instabilities. As the aluminum burns with high temperature without any adverse effects on the detonation characteristics of the propellant, but it significantly affects the flow field of the chamber and nozzle. Several different metals have been considered and used in solid propellants with aluminum being extensively used in mass fractions of 12%–22%^[1]. High density and high heat of reaction are two factors which contribute to high impulse. It is known that the flow field of the solid rocket motor is very complicated due to the chemical reaction, particle evaporation, combustion, and break up and other complex characteristics like agglomeration and coalescence etc. So these effects cause nozzle erosion and significant exhaust products. Therefore aluminum behavior is one of the most important challenges faced by the solid propellant rocket industry^[2].

During the combustion of aluminized propellant, the aluminum particles in the propellant melt and form liquid aluminum at the burning propellant surface. Because of the physical properties of aluminum and its oxide, a large fraction of aluminum remains unreacted and in liquid state at the burning propellant surface. Several liquid droplets merge into large agglomerates^[1]. These agglomerates leave the propellant surface and continue to burn relatively slowly due to low volatility of aluminum. The vaporized aluminum reacts with the oxidizing species in the gas phase forming aluminum oxide. Whereas most of this aluminum oxide “smoke” diffuses outward into the flow field, a part is captured on the agglomerate surface and condenses to form an oxide

shell. The formation of aluminum oxide shell on the surface of the droplet further contributes to the “slowness” of aluminum combustion. So this results in reduced motor performance and possible that nozzle surfaces damage from possible impingement.

Therefore the flow within the rocket is multiphase or two phase, because it contains droplets and smoke particles. The two phase flow representing bimodal distribution of aluminum oxide representing the smoke and the caps, which are on the order 1.5 and 100 μm in size respectively. Several combustion scientists and propulsion engineers have conducted many studies aimed at determining the size distribution of the Al_2O_3 droplets present in the motor, nozzle and exhaust plume, because of the large effect on the impulse efficiency. So this area still required a lot of attentions on the size distributions of the droplets entering the chamber and nozzle. Sabnis et al^[3] refer to Salita's^[4] quench bomb investigations, where the size distribution was found to be lognormal and bimodal. Typical values of the weighted mean diameters were 1.5 μm for the small particles; whereas the larger particles were 150 μm with the standard deviation of 0.2. However Whitesides et al^[5] analyzed a reusable solid rocket production propellant, and concluded that a single lognormal or polynomial distribution did not adequately fit the measured droplet diameter distribution. So such contradictory conclusion indicates that the droplet size distribution is sensitive to the propellant formulation.

Numerical simulation of the internal flow in the solid rocket motor and nozzle has been studied previously due to the importance of the flow field on the motor performance and reliability. The flow within the rocket is subsonic near the head end, but compressibility effects become important in the nozzle region^[6]. The flow transitions from a laminar state near the head end to a fully turbulent state farther downstream. Jayant S. Sabnis^[3] used Eulerian-Lagrangian two-phase approach suitable for the numerical simulation of the multiphase reacting internal flow in the solid

rocket motor with a metalized propellant. He used an Eulerian description to analyze the motion of the gas phase particulates while Lagrangian description is used for the analysis of the discrete phase that consists of the larger particulate in the motor chamber.

To provide design guides for maintaining high performance of the SRM (solid rocket motor), an accurate simulation of the gas-particle interaction is very important. Because of the complex flow field inside the SRM, limited experiment data is available for design purpose. The internal flow field analysis using a CFD (computational fluid dynamics) method can be utilized to obtain a better investigation for SRM, due to the recent progress in computing power. There has been some research conducted in the past for the solid rocket motor internal flow field analysis using the CFD method.

In this paper CFD simulation has been conducted for uniform particles size diameter to analyze its effect on the performance prediction of solid rocket motor. The Rosin-Rammler diameter distribution method and Eulerian-Lagrangian Approach^[7] has been used for the simulation of different distribution of Al_2O_3 droplets present in the motor. Because the distribution of the particles affects the performance of the motor, therefore prediction of the particles effect plays an important role for SRM design. The effect of injected droplets size distribution obtained with different models is investigated and shows the sensitivity of this distribution to the nozzle flow dynamics, primarily at the nozzle inlet and exit. The results are shown with various sizes of the particles, concentrations and geometrical configurations including models for conical and contours type nozzle, including the performance prediction.

1 Model description

1.1 Equation of motion of particles

The reacting multiphase flow in the solid rocket chamber can be effectively treated as consisting of a continuous phase composed of the

products of combustion from the binder, the ammonium per chlorate (AP), and reacted aluminum, and a discrete phase composed of particles containing aluminum and Al_2O_3 . Using Eulerian-Lagrangian approach trajectories of a discrete phase particles are simulated by solving an equation of motion for each dispersed phase particle^[7]. Trajectory calculations require the calculation of net force acting on the dispersed phase particles, it will be necessary to solve a set of coupled ordinary differential equations. This force balance equates the particle inertia with the forces acting on the particle, and can be written^[8]:

$$\left[\frac{dV_p}{dt} = \frac{3}{4} \frac{C_d \rho}{\rho_p d_p} \|V - V_p\| (V - V_p) + g \right] \tag{1}$$

The drag coefficient C_d can be obtained

$$C_d = \frac{24}{Re_p} (1 + 0.15 Re_p^{0.687}) \tag{2}$$

Reynolds number Re_p is defined as:

$$Re_p = \frac{\rho_g V_p d_p}{\mu_g} \tag{3}$$

Where V_p is the relative velocity between the particle velocity and the gas velocity.

1.2 Solid pocket motor geometry

The motor and nozzle geometry configuration adopted here is shown in the Fig. 1. It is a typical solid rocket motor with convergent and divergent nozzle. The cylindrical grain geometry with 2 000 mm length of motor was tried with contours nozzles. Moreover the propellant and nozzle parameters have also been dictated for all the cases.

Grain length,	$L = 2\,000\text{ mm}$
Mass of propellant,	$m_p = (2\,200 \pm 10)\text{ kg}$
Burning time,	$t_b = 30\text{ s}$
Grain outer radius,	$R = 800$
Mass flow rate,	$\dot{m} = 73\text{ kg/s}$
Area ratio,	$A_e/A_t = 11.7$

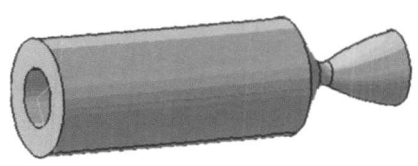


Fig. 1 Chamber, grain and nozzle geometry

1.3 SRM parameter values and material selection for multi phase

The following reference parameter value, with inert particle type material Al_2O_3 have been assumed for all the computations.

Chamber pressure, $p = 6.2 \text{ MPa}$
Combustion temperature, $T_c = 3200 \text{ K}$
Specific heat at constant pressure, $c_p = 1430 \text{ J/(kg} \cdot \text{K)}$
Ratio of specific heats, $\gamma = 1.19$
Density, $\rho = 3910 \text{ kg/m}^3$
Thermal conductivity, $\lambda = 0.1 \text{ W/m} \cdot \text{K}$

1.4 Roe scheme-FDS

Roe Flux differencing splitting(1980) based on approximate Riemann problem^[9] has been used through Fluent 6.3. This is second order upwind scheme in which flux differencing splitting method is employed. Upwind schemes are designed to numerically simulate more properly the direction of the propagation of information in a flow field along the characteristic curves. This scheme has been used with good success in computing the solution to non linear system is to solve an approximate Riemann problem rather than having to deal with the exact nonlinear iterative scheme.

The Rosin-Rammler distribution function is based on the assumption that an exponential relationship exists between the droplet diameter, d and mass fraction of droplets with diameter greater than d .

$$Y_d = e^{-\left(\frac{d}{\bar{d}}\right)^n} \quad (4)$$

here

$\bar{d}$ = Mean diameter of the particles

n = Spread parameter

Particles sizes in the range of 1– 100 μm are used, as being the most common droplets^[10-12]. In this approach the complete range of particle sizes is divided into a set of discrete size ranges, each to be defined by a single stream that is part of the group. In this case the particle size data obeys the distribution shown in Fig. 2

The Table 1 below summarizes the test matrix for the various droplets diameters distrib-

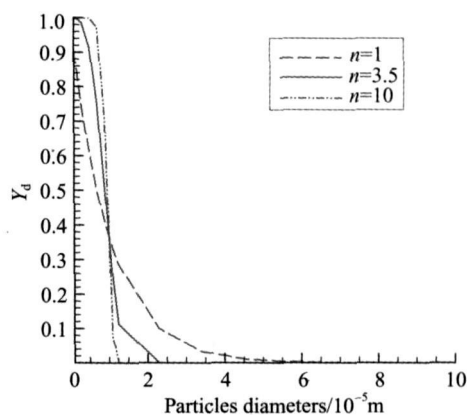


Fig. 2 Value of n ($n = 1, 3.5, 10$)

tions used. For the skewed logarithmic distributions different profiles have been chosen as diagrammatical represented in Fig. 2

Table 1 Particles diameters distributions and mass fractions for different spread parameters' n

$n = 1$		$n = 3.5$	
Diameter/ μm	Mass fraction in range	Diameter/ μm	Mass fraction in range
1.0	0.9048	1.0	0.9997
2.0	0.8187	2.0	0.9964
3.0	0.7408	3.0	0.9853
5.0	0.6065	5.0	0.9154
7.0	0.4965	7.0	0.7505
9.0	0.4065	9.0	0.5007
11.0	0.3328	11.0	0.2475
12.5	0.2865	12.5	0.1126
23.0	0.1002	23.0	0
34.0	0.0333	34.0	0
45.0	0.0111	45.0	0
56.5	0.0035	56.5	0
67.0	0.0012	67.0	0
77.5	0.0004	77.5	0
88.5	0.0001	88.5	0
100	0	100	0

1.5 Boundary conditions

Following boundary conditions have been imposed on the configurations for discrete phase flow passing through the SRM chamber and nozzle model.

1) The droplets are injected at zero velocity and are assumed to be in thermal equilibrium with the surrounding gas.

- 2) A nonslip velocity is imposed for the head end, the nozzle walls, and walls at divergent part.
- 3) Supersonic out flow conditions are applied at the nozzle outlet.
- 4) For the axis, only a symmetry conditions has been applied.
- 5) Percentage of particles in the mass flow rate has been set.
- 6) Temperature 3200 K is set along the bottom and top injection surfaces.
- 7) Mass flow rate of 73 kg/s has been considered.
- 8) Designed pressure of 62×10^5 Pa has been set inside the chamber.

2 Results and flow field analysis

The computational domain and the grid for Solid rocket motor are shown in the Fig. 3. The computational grid used in these calculations consisted of 50 grid points distributed along the radial direction and 300 grid points distributed along the motor axis. The computational grid used in this study is generated from Gambit. Grid resolutions near the wall were appropriate to properly resolve the flow gradient at the walls. Numerical computational of the flow field and particle trajectories has been done using Fluent 6.3, which solve the multi phase flow simulations and its application on performance prediction of the solid rocket motor. The flow field is solved for a chosen burn time the particle are injected from the propellant surface and tracked through, until they reach the nozzle exit and impact on the wall. The results obtained here are discussed in detail.

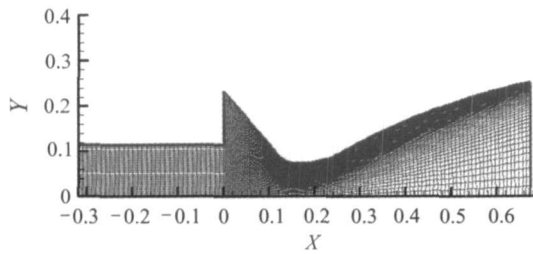


Fig. 3 Grid generation for SRM with contours nozzle

A computational fluid dynamics simulation has been performed in a representative rocket motor with contour exit profile shown in Fig. 1. Performance calculated on the basis of specific impulse for different models are tabulated in the Table 2 below. Specific concentration has been given to motor and nozzle flow field; however motor with the length of 2.00 m has been used. The propellant grain has 0.80 m outer diameters with propellant weight of 2200 kg.

In model A uniform particles distributions, the particles having one diameter size $d=10\mu\text{m}$ has been used. Fig. 4 shows the Mach number distributions along the motor centre line and exit plan of the nozzle flow with the above particles present. The specific impulse of 2037.20 is achieved.

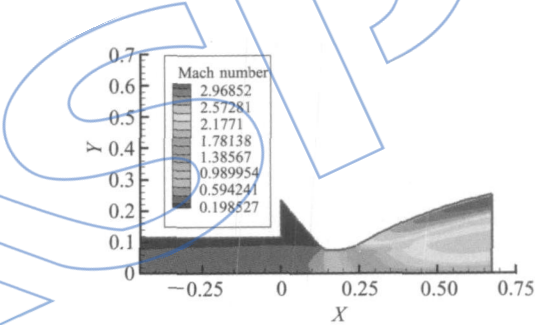


Fig. 4 Mach number contours for uniform particles group

To see the effect of turbulence Fig. 8 is shown smooth flow inside the chamber due to uniform particles (diameter $10\mu\text{m}$) distribution has not created any turbulence effect in the chamber as well as in the nozzle. Turbulence effect in case of multi group particles (diameter as per Table 1) is visible as shown in Fig. 9.

In model B multi phase flow with Rosin Rammler diameter particles distribution and spread parameter $n=1$ has been used. Minimum diameter of 1×10^{-6} m and maximum of 0.0001 m with total flow rate of 23 kg/s has been selected.

Fig. 5 shows flow field for Mach number contours details, with specific impulse of 2030.95. In model C and model D two phase flow with same particles diameter distribution and spread parameter $n=3.5$ and 10 has been used. The specific im-

pulse of 2032 and 2033 has been achieved respectively.

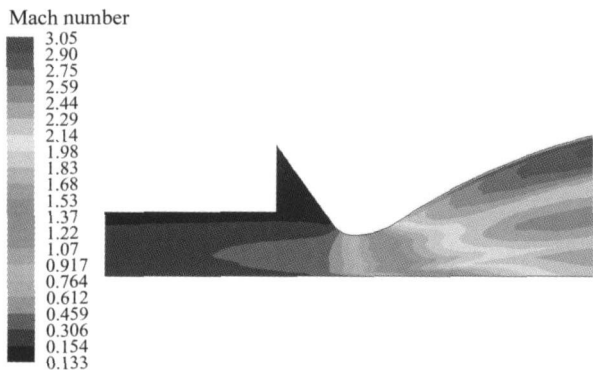


Fig. 5 Mach number contours for multi-group particles

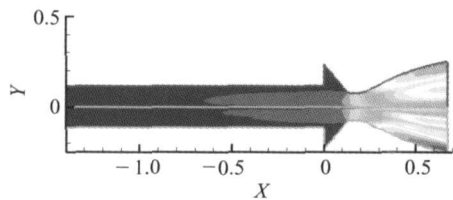


Fig. 6 Comparison between uniform particles and multi-phase flow with spread parameter 3.5

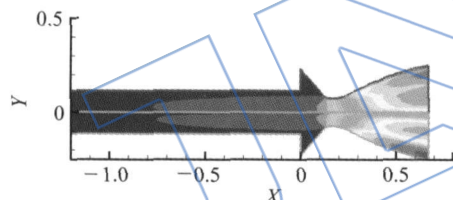


Fig. 7 Comparison of multi-particles flow field for $n=1$ and $n=3.5$

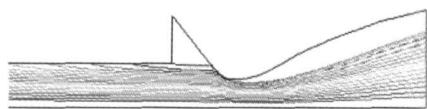


Fig. 8 Particles track for uniform diameter size-group particles



Fig. 9 Particles track for multi-group particles

The simulation presented above it is concluded that particles distributions has major role in performance prediction of the solid rocket motor. Therefore if we are more specific or close to the diameter distribution of particle with, it is

giving more exact results. As from the Table 2 below best performance is achieve through model-B with spread parameter $n=1$. From simulations it is also seen that particle with more diameters are moving close towards the centre or axis and lighter particles are going away from the axis Fig. 8 This paper also indicate that particle distributions has 2% – 3% performance difference, so multi group is more perfect method to be applied for SRM performance prediction.

The effects of uniform particles and multi-group particles on Mach numbers flow field are shown in Fig. 4 and Fig. 5. The Mach number is less than 0.2 at the nozzle inlet, reaches to 1.0 at the throat for both the cases. The maximum Mach number of 2.97 at the nozzle exit is reached for uniform particles and maximum of 3.02 is reached for multi-group particles with spread parameter one. The contours are affected strongly by the particles concentration. This will affect the motor performance.

The most important effect is in the supersonic part of the nozzle. As seen through comparison till sonic line flow field for uniform particles and multi-group particles is the same but in the divergent part of it flow field simulations are different as clearly visible in the Fig. 6 and Fig. 7. This variation in the flow field is actually creating the difference in the performance prediction also.

Table 2 Comparisons for multi phase flow using contours nozzle

Models description	Performance prediction (specific impulse) / (m / s)
Model A (Two phase flow uniform particles distribution)	$I_{sp} = 2037.20$
Model B (Two phase flow, spread parameter $n=1$)	$I_{sp} = 2030.95$
Model C (Two phase flow, spread parameter $n=3.5$)	$I_{sp} = 2032.50$
Model D (Two phase flow, spread parameter $n=10$)	$I_{sp} = 2033.20$

3 Conclusion

Numerical flow field simulations have been performed for typical SRM with burning aluminum droplets generating aluminum oxide. For droplet size distributions, uniform particles diameters and Rosin-Rammler diameter distribution method has been used. Roe's FDS with Eulerian-Lagrangian approach, and Spalart-Allmaras turbulence model has been applied. Various sizes of particles with various concentrations have been used. The effects of these distributions on the flow field dynamics specifically at the SRM chamber and nozzle exit are observed. The analysis discussed also provides a sophisticated tool for solid rocket motor performance predictions.

It is seen that due to the drag force exerted on the particles and gas flow, the larger particles tend to collect more along the nozzle axis than the smaller one. This results in large size particles cloud density along the centre line which in turn slows down the gas flow field and performance is influenced to the greater extent. As larger the particles in flow field more particle-free zone near the nozzle wall and smaller the gas Mach number along the nozzle centre line. Therefore this study highlights the performance influence appears due to the diameters distribution and its particles concentrations in propellant formulation.

The results obtained can also provide the designer a basic guide line and potential improvements of SRM including the use of materials and ultimately the SRM grain design and nozzle geometry such that a better performance of the SRM can be achieved. There are still important points that have to be further investigated to

gain further insight into the multiphase flow dynamics and its impingements on the nozzle surface material that causes ablation phenomena.

References

- [1] Sabnis J S. Numerical simulation of distributed combustion in solid rocket motors with Metalized propellant[J]. Journal of Propulsion and Power, 2003, 19(1): 48-55.
- [2] Jackson T L, Najjar F, Uckmaster J B. New aluminum agglomeration model and their use in solid propellant rocket simulations[J]. Journal of Propulsion and Power, 2005, 21(5): 925-936.
- [3] Sabnis J S, deJong F J, Gibeling H J. A two-phase restricted equilibrium model for combustion of metalized solid propellant[R]. AIAA Paper 92-3509, July 1992.
- [4] Salita M. Quench bomb investigation of Al_2O_3 formation from solid rocket propellants: Part II - Analysis of data [C]//Proceeding of 25th JANNAF Combustion Meeting. Laruel M D: CPIA Publ. Chemical Propulsion Information Agency, Johns Hopkins University, 1988, 1: 185-197.
- [5] Whitesides R H, Dill R A, Purinton D C. Application of two phase CFD analysis to the evaluation of asbestos-free insulation in the RSRM [R]. AIAA Paper 97-2861, July 1997.
- [6] Najjar M F, Ferry J P, Haselbacher A, et al. Simulation of solid propellant rockets effect of aluminum droplet size distribution[J]. Journal of Propulsion and Power, 2006, 439(6): 1258-1270.
- [7] Rande V V. Computational flow modeling for chemical rocket engineering[M]. London, UK: Academic Press, 2002.
- [8] Cesco N, Lavergne G, Estivaleres J L. Simulation of the two phase flow in solid rocket motor[R]. AIAA-96-2640.
- [9] Tannehill J C, Anderson D A, Pletcher R H. Computational fluid mechanics and heat transfer[M]. 2nd ed. Bristol, PA: Taylor and Francis, 1997.
- [10] Price E W. Combustion of metalized propellants[C]//Kenneth K K, Summerfield M. Fundamentals of solid propellant combustion: Progress in Astronautics and Aeronautics. New York: AIAA, 1984, 90: 475-513.
- [11] Desnoyer D. Two phase nozzle and chamber flow in solid rocket motor[R]. AIAA-87-0364.
- [12] Ciucci A, Iaccarino G, Amato M. Numerical investigation of 3D two-phase turbulent flows in solid rocket motor [R]. AIAA-98-3966.