Computers & Fluids 75 (2013) 22–34

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Analysis and design optimization of solid rocket motors in viscous flows

Mine Yumuşak *

ROKETSAN Missiles Industries, Ankara 06780, Turkey

ARTICLE INFO

Article history:
Received 28 August 2012
Received in revised form 1 January 2013
Accepted 5 January 2013
Available online 31 January 2013

Keywords:
Design optimization
Viscous flow
Turbulence
Solid rocket motor

ABSTRACT

The objective of this study is to develop a design tool that can be used in viscous flows. The flow analysis is based on the axisymmetric Navier–Stokes and k–ε turbulence equations. These coupled equations are solved using an explicit finite difference method. The accuracy of the analysis code is validated for viscous flows in solid rocket motor combustion chamber and nozzle. The gradient-based numerical optimization model is used to maximize the thrust of solid rocket motor under a constraint of propellant weight. The sensitivity analysis that measures the response of the flow with respect to a geometry perturbation is calculated by finite differencing. The optimization of design study employs a commercial optimization package. The performance of design optimization method is tested in solid rocket motor combustion chamber and nozzle design.

1. Introduction

Computational Fluid Dynamics (CFD) is extensively used in solid rocket motor (SRM) designs. In traditional design methods, engineers modify initial prototypes with their experience and analyze the modified configurations with CFD solvers. In general, these modifications are based on the trial and error approach and several configurations have to be analyzed before the final decision is reached. Traditional methods have some disadvantages. It is difficult to determine whether the final design is the best one, and computational time is very long. In order to reduce the computational time, sometimes simpler models are used for flow analyses. However, these models may reduce the reliability of the design results. In recent years, new advancements have been made to increase the reliability and efficiency of design methods. Automatic design methods developed by combining the CFD and optimization codes have several advantages. The experience and information needed for design can be reduced, and innovative designs can be made in the conditions that have never been tried before. The design cost can also be decreased by reducing the engineers' workload and design time. Another advantage of the automatic methods is that the design can be obtained as a solution to the optimization problem. In this way, if there is no solution with the specified design conditions, the design can be evaluated as a best available solution.

The performance of a rocket can be improved with aerodynamics and aero-thermodynamics design optimization. Minimizing the pressure and friction losses of internal and external flows may increase the range or payload capacity of rockets. Vehicle requirement such as performance characteristics, envelope constraints and mission profile play important role in combustion chamber design. Grain design and burning surface evaluation are very important in the design of combustion chambers. The essence is to evolve the burning surface and establish a relation between web burnt and the burning surface. The nozzle is another important component of rockets because its efficiency significantly affects rocket's performance. By minimizing the losses in the nozzle, the thrust of a rocket can be increased by keeping the grain geometry fixed.

The subject of solid rocket motor (SRM) design and optimization is quite vast. Many approaches from the gradient based class of methods, non-gradient based methods, heuristics and parametric methods have been used for design and optimization processes of SRM system parameters and sub-components. Kamran and Guazhu [1] present an approach using hyper-heuristic based on meta-heuristics method for combustion chamber, nozzle and grain designs. In this study, design objective is to minimize the gross mass of motor that would cater for a minimum inert mass along with maximum specific impulse. The grain design and burning surface evolution of a complex 3D solid rocket propellant grain are modeled by CAD software using parametric modeling. Anderson and Burkhalter [2] used genetic algorithms to design solid rocket motors as a component within an overall missile system. In this study, multiple goals, such as maximized range, minimized g-loading, minimized takeoff weight and maximized fuel volume are used to test the ability of genetic algorithms to work efficiently within a multidisciplinary framework. The 0-dimensional ballistics performance prediction code is coupled with the design tool to

* Address: Internal Ballistics Modeling Department. Tel.: +90 312 860 55 00; fax: +90 312 863 42 08.
E-mail address: myumusak@roketsan.com.tr

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.01.011

determine the most efficient configuration of the motor. Kumar et al. [3] carried out a parametric study to examine the geometrical influence on turbulent separated flows in SRMs. Five different physical models with different port geometries are examined. Results of interests such as reattachment length, size of the recirculation zone and the axial velocity variations are reported to illustrate the influence of transition region on the flow characteristics of turbulent mixed convection downstream of a solid rocket motor with divergent port. Such detailed results are needed for an integrated design and optimization of the high-performance solid rockets port geometry and its allied igniters with confidence. He concluded that the narrow port and long flow development ahead of the steep divergence are shown to favor flow separation, which might lead to high peak pressure, pressure-rise rate and thrust oscillations during the starting transient period of operation of motors with divergent ports.

The studies related to the design of rocket nozzles are not new. Optimization techniques used to design nozzle contours have been utilized since 1950. Rao [4] developed a method which optimizes a rocket nozzle contour for a given length or expansion ratio so as to achieve maximum thrust. Rao's method is based on the assumption of inviscid isentropic flow. This method has been used in many studies for different classes of rocket nozzle design. In a study by Farley and Campbell [5], three Rao optimized bell contour nozzles were compared to a 15° conical nozzle. The thrust produced by the optimized nozzles was greater than that obtained with the conical nozzle. In fact, bell contour nozzles have been used routinely for many years in large liquid rocket engines [6]. Conical nozzles are typically used only when fabrication and design costs outweigh performance. In recent years, more accurate design methods have been developed by using Navier–Stokes equations [7,8].

The flow physics in a SRM is quite complex. In a SRM, the flow accelerates from the head-end, resulting in streamwise inhomogeneity, undergoing transition to turbulence in the midsection of the motor, becoming fully turbulent further downstream and finally reaching supersonic conditions at the exit of the nozzle. The velocity profile and turbulence characteristics near surfaces determine the convective heating loads and therefore affect both insulation thickness requirements and the potential erosive burning of the propellant. Early attempts to investigate the internal flow-field of a SRM were mainly concerned with the distribution of the mean velocity. In a SRM, near the head-end, the flow is laminar and its velocity profiles can be determined by the laminar similarity theory derived by Culick [9] and Taylor [10]. For uniform injection at a constant mass flow rate, and for laminar and incompressible flow, the laminar similarity velocity profile was given as:

$$\begin{aligned} \frac{u}{u_{inj}} &= \frac{x}{r_w}\frac{\pi}{2^{1-\alpha}}\cos\left[\frac{\pi}{2}\left(\frac{r}{r_w}\right)^{\alpha+1}\right] \\ \frac{v}{u_{inj}} &= -\left(\frac{r_w}{r}\right)^{\alpha}\sin\left[\frac{\pi}{2}\left(\frac{r}{r_w}\right)^{\alpha+1}\right] \end{aligned} \tag{1}$$

where the value of α is 0 for planar and 1 for axisymmetric flows. In the equation above, u_{inj} is the injection velocity on propellant surface, u is the axial velocity, v is the radial velocity, x is the axial direction, r is the radial direction and r_w is the propellant port radius.

The experiments performed by Dunlap et al. [11] and Yamada et al. [12] showed that the mean flow field is accurately represented by Culick's laminar velocity profile in the forward region of a cylindrical port rocket chamber. Later, Traineau et al. [13] conducted an experimental study on a nozzleless rocket motor at high injection rates and concluded that the laminar velocity profile persists in the mean flow, even if the flow is highly turbulent over the duct cross section.

Several numerical simulations have been performed to examine the flow-field within a rocket combustion chamber. Sabnis et al. [14] and Tseng [15] used conventional single-point turbulence closure schemes at low injection rates and obtained a good comparison of the mean flow field with the experimental data, but over predicted turbulence intensity levels within the chamber. Apte and Yang [16] conducted two dimensional simulations of flows through nozzleless rocket motors at very high injection rates to achieve a better comparison of turbulence intensity and axial velocity profiles compared with the second-order turbulence closures. Apte and Yang also performed a Large Eddy Simulation (LES) of internal flow development in a three dimensional rectangular rocket motor. Nicaoud et al. [17] performed Direct Numerical Simulation (DNS) at high injection rates in an attempt to reproduce flow conditions representative of a SRM in order to investigate the effect of a high blowing rate on the wall layer.

The objective of this study is to develop a design tool that can be used for rocket combustion chamber and nozzles. The reliability of design results depends on the accuracy of flow model used in design method. In order to capture the viscous flow physics, the axisymmetric Navier–Stokes and k–ε turbulence equations are solved simultaneously. The classical k–ε turbulence model is modified to predict the mean-flow velocity profile and turbulence intensities accurately near the mass injecting propellant surface and wall regions. A gradient-based finite difference numerical optimization method is used for design optimization. In gradient based design optimizations, the derivatives of objective function with respect to design variables are needed. In literature, these derivatives are called sensitivities. The accurate and efficient calculation of sensitivities is important for the performance of design method. There are two methods to obtain sensitivity derivatives. The first is the analytical approach, which calculates sensitivities by analytically differentiating the governing equations with respect to the design variables. The analytical method provides accurate and efficient sensitivity calculations. However using this method is not easy; developing a sensitivity code requires considerable amount of programming effort. The analytical method is more efficient if the analysis code uses an implicit numerical solution method. The other method is the finite difference approach, which requires the flow field solution for each perturbed design variable. The main advantage of the finite difference approach is that it is much easier to implement than the analytical approach. It does not require an additional programming effort to build a dedicated sensitivity code. Even though the finite difference approach requires flow solution evaluations for the perturbation of each design variable, the computational cost can be reduced if the geometry perturbations are small and the flow solution iterations for each perturbed geometry start from an already converged base solution. However, the finite difference sensitivity calculations have accuracy problems. The accuracy of the finite difference sensitivity derivatives depends on the choice of the magnitude of the design variable perturbations or the finite difference step size. However, the accuracy of the finite difference sensitivities can be improved by finding the optimum step size that can minimize the norm value of total error in sensitivities [18]. In the present study, sensitivities are calculated using the finite difference method and numerical design model uses a commercially available constrained optimization package DOT [19]. It searches for a feasible direction and step size to minimize or maximize a specified objective function under a set of constraints. The performance of design optimization model is demonstrated for solid rocket motor combustion chamber and nozzle designs.