Studies on Internal Flow Choking in Dual-Thrust Motors

V. R. Sanal Kumar*
Vikram Sarabhai Space Centre, Trivandrum, Kerala 695 022, India
B. N. Raghunandan†
Indian Institute of Science, Bangalore 560 012, India
H. D. Kim† and A. Sameen†
Andong National University, Andong 760-749, Republic of Korea
T. Setoguchi†
Saga University, Saga 840-8502, Japan
and
S. Raghunathan∗
Queen’s University of Belfast, BT7 1NN Northern Ireland, UK
DOI: 10.2514/1.20748

Introduction

A detailed picture of the internal flow during the starting transient of high-performance solid rocket motors (SRMs) is of topical interest for several reasons in addition to the motor performance itself [1–12]. Despite the fact that many of the existing models could predict the internal flow features of certain classes of SRMs, none of these models could capture the unusual starting transient flow features such as pressure overshoot and pressure-rise rate often observed during the initial phase of operation of the dual-thrust motors (DTMs) [1]. Ikawa and Laspesa [8] reported that during the first launching of the space shuttle from the Eastern Test Range, the launch vehicle experienced the propagations of a strongly impulsive compression wave. This wave was induced by the SRM ignition and was emanating from the large SRM duct openings. The analysis further showed that the compression wave created by ignition of the main grain was the cause of the ignition overpressure on the launch pad [9]. Alestra et al. [10] reported that Ariende 5 launcher experienced overpressure load during the liftoff phase. The overpressure is composed of the ignition overpressure, which emanates from the launch pad, and the duct overpressure, which emanates from the launch ducts. Of late, Sanal Kumar et al. [1,2] reported that abnormal high-pressure overshoot in certain class of DTM during the startup transient is due to the formation of shock waves because of the fluid-throat effect, which has received considerable attention in the scientific community. This manuscript is the continuation of the previous connected note for establishing the intrinsic flow physics pertinent to internal flow choking in inert simulators of dual-thrust motors [1]. Note that the illustration of ignition pressure spike is deliberately set aside in this note for explaining the intrinsic flow physics pertinent to internal flow choking without complications arising from the propellant combustion.

This paper addresses the design challenges associated with development of high-performance dual-thrust motors because of their large size, high length to diameter ratio, and demanding thrust-time trace shape requirements. This note promises to produce an optimum and also a profitable flight motor design that meets all high-performance objectives while accommodating program development uncertainties.

Numerical Methodology

Numerical simulations have been carried out in inert simulators of DTM with the help of a two-dimensional standard $k$–$\omega$ turbulence model. This code solves standard $k$–$\omega$ turbulence equations with shear flow corrections using a coupled second order implicit unsteady formulation. In the numerical study, a fully implicit finite volume scheme of the compressible, Reynolds-averaged, Navier–Stokes equations is employed. Compared with other models, this model could well predict the turbulence transition in duct flows and has been validated through benchmark solutions [3–5,13]. Initial wall temperature, inlet total pressure, and temperature are specified. At the solid walls, a no-slip boundary condition is imposed. At the nozzle exit, a no-slip boundary condition is imposed. The nozzle pressure profile is imposed. Note that the motor exit geometry (nozzle) considered in this study is a short, straight duct followed by the convergent duct. Therefore, a radial axisymmetric pressure distribution at the exit was approximated analytically. Note that this is more realistic than the conventional exit boundary conditions. The Courant–Friedrichs–Lewy number is initially chosen as 3.0 in all of the computations. An algebraic grid-generation technique is employed to discretize the computational domain. The grids are clustered near the solid walls using suitable stretching functions. Ideal gas is selected as the working fluid.

Evaluation of Boundary Layer Blockage

It is assumed that the developing flow can be represented by boundary layer thickness together with a core in which the velocity is uniform. The approach applies equally to smooth or rough grain surfaces. In a two-dimensional flow model, boundary layer blockage is given by

$$\delta^* \frac{d}{2} = 1 - \frac{U}{U_{max}}$$  

(1)

where $\delta^*$ is displacement thickness, $U_{max}$ is the velocity on the axis, and $d$ is the diameter of the upstream port of the DTM. Although