

# Engineering Notes

ENGINEERING NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes should not exceed 2500 words (where a figure or table counts as 200 words). Following informal review by the Editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

## 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, BT71NN 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 Ariene 5 launcher

Presented as Paper 4147 at the 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Tucson, Arizona, 10–13 July 2005; received 26 October 2005; revision received 10 February 2006; accepted for publication 13 February 2006. Copyright © 2006 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved. Copies of this paper may be made for personal or internal use, on condition that the copier pay the \$10.00 per-copy fee to the Copyright Clearance Center, Inc., 222 Rosewood Drive, Danvers, MA 01923; include the code \$10.00 in correspondence with the CCC.

\*Scientist/Engineer, Propulsion Group, and Korea Science and Engineering Foundation Postdoctoral Research Scientist; rsanal@hotmail.com.

†Professor and Chairman, Department of Aerospace Engineering.

‡Professor and Head, School of Mechanical Engineering.

§Visiting researcher; currently Postdoctoral Fellow, Engineering Mechanics Unit, Jawaharlal Nehru Center for Advanced Scientific Research, Bangalore, India.

||Professor of Mechanical Engineering.

¶Professor and Head of Aeronautical Engineering, Bombardier Aerospace, Royal Academy Chair, Associate Fellow AIAA.

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 DTMs 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 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

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

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