NUMERICAL ANALYSIS OF FLOW NON-UNIFORMITY IN THE HOT GAS MANIFOLD OF THE SPACE SHUTTLE MAIN ENGINE

J. Thoenes, S.J. Robertson, and A.W. Ratliff
Lockheed-Missiles & Space Company, Inc.
Huntsville, Alabama 35807

and

P.G. Anderson
Continuum, Inc.
Huntsville, Alabama 35808

Abstract

A numerical analysis was made of three-dimensional viscous flow in a conceptual hot gas manifold (HGM) for the Space Shuttle Main Engine High Pressure Fuel Turbopump (SSME HPFTP). A finite difference scheme was used to solve the Navier-Stokes equations including a mixing length turbulence model. The exact geometry of the SSME HGM was modeled using boundary fitted curvilinear coordinates and the General Interpolants Method (GIM) code. Slight compressibility of the subsonic flow was modeled using a linearized equation of state with artificial compressibility. Computations were performed on the CYBER 205 supercomputer. Experimental data were employed to provide boundary conditions for the computation. A time relaxation method was used to obtain a steady state solution.

Results are compared with experimental data and found to be in reasonably good agreement in terms of overall flow patterns, pressure distribution in the manifold, and total pressure loss along the flow path. In particular, flow separation regions in the transfer ducts as observed from the analysis are in good agreement with experimental data.

The results of this effort demonstrate the feasibility and potential usefulness of computational methods in assisting the design of SSME components whose function involves the flow of fluids within complex geometrical shapes.

Introduction

Much of the U.S. activity in space over the coming years will be dependent upon the Space Shuttle and its derivative versions as a principal space transportation system. This dependence requires improved designs or techniques to extend the life, upgrade performance, reduce weight, lower operational costs, and generally improve the functional capability of the main propulsion system. The engines for this main propulsion system are advanced high pressure engines operating on oxygen and hydrogen. A need therefore exists to investigate, develop, and define basic concepts in support of the main propulsion system improvements. A basic problem which requires investigation and improvement is that of flow nonuniformities found to occur in the turbopump exhaust manifolds, their transfer ducts as well as in the main injector of the current engines.

Development and verification tests of the SSME have shown that the three transfer ducts connecting the high pressure fuel turbopump (HPFTP) to the main injector have an uneven flow distribution with large areas of separated flow. The outer transfer ducts each carry approximately twice the amount of gas flowing through the center duct. The attendant higher dynamic pressures in the duct exits impose severe gasdynamic forces on the main injector liquid oxygen (LOX) posts. In the past, this has led to the installation of support shields, thereby trading structural integrity for a reduction in gasdynamic efficiency. In subsonic flow, pressure nonuniformities will also propagate upstream, thereby causing oscillatory loads on the turbine blades of the HPFTP.

One way to conduct a design improvement effort is to replace deficient hardware with redesigned components and determine the results by testing. Unfortunately, this process is very expensive, and practically prohibitive if more than one alternative improvement is to be investigated. An alternate way to attack the problem is to use a combined analytical/experimental approach. The advent of large core, high speed scientific computers in conjunction with the development of powerful numerical techniques has made the computational analysis of complex three-dimensional flow fields feasible. Realizing this opportunity, a joint analytical and experimental study was undertaken by Lockheed and Rocketdyne under the sponsorship of NASA. Based on a preliminary study of improvement alternatives, a conceptual two-duct HPFTP hot gas manifold was selected to be built and tested by Rocketdyne[1]. Test data obtained were then provided to Lockheed for comparison with computational results obtained from three-dimensional, viscous flow calculations. The methodology of the analysis and the results are briefly described in this paper.

Numerical Analysis

Methodology

This study employed the General Interpolants Method (GIM)[2] for constructing numerical analogs of the partial differential equations governing fluid flow. The domain of interest is first described by appropriate subdivisions into an assemblage of interconnected finite elements. A boundary fitted computational grid is generated using an algebraic interpolation scheme based on three-dimensional general curvilinear coordinates. Shape functions based on a set of generalized interpolants are then used to describe the behavior of all variables over each element. The discretized equations are multiplied by a set of weight functions and integrated over the element volume. A choice of weight functions which are orthogonal to the shape functions leads to explicit nodal analogs. As a result of this spatial discretization, the partial differential equations are reduced to ordinary differential equations with time as the independent variable. The two-step MacCormick algorithm was used to integrate these ordinary differential equations to a steady state solution.

Geometric Treatment

The domain of interest is considered to be geometrically arbitrary in that any shape is represented as a bivariate blend of regular subdomains. Division into subdomains is made such that analytical functions can describe the shape of each edge (but it can be made by point specification and piecewise linear edges). Attention is then focused on each region. Regions are blended at the junction to provide a continuous full domain geometry.

The geometrical definition of the HGM was obtained from drawings supplied by Rocketdyne. The model includes the "fishbowl" proper and the transfer ducts, but not the turnaround duct. Struts at the transition from the turnaround duct to the fishbowl were neglected. This simplification might influence the flow properties at the fishbowl entrance but should have negligible effect on the overall flow in the bowl and the transfer ducts. Flow contours of the manifold were modeled accurately including the slight round off of the transfer duct inlet edges.

Representing the initial step in a parametric investigation of changes in the geometrical configuration, a second manifold using a flared transfer duct inlet section was modeled also. This configuration is characterized by a greatly increased radius of curvature at the transfer duct inlet, mainly in the lower inner duct quadrant.

Flow Analysis

The full set of differential equations solved by the GIM code consists of the equations for the conservation of mass, momentum (Navier-Stokes) and energy, supplemented by one equation of state. The differential equations are solved in the strong conservation law form for the conserved variables. The relaxation procedure is started from initial conditions based on a reasonable estimate of the steady state flow field. Primitive variables, including the pressure, are evaluated after the completion of each time step in a decoding procedure.

Transport properties are specified in terms of dynamic viscosity and Prandtl number for turbulent flow. Effects of turbulence were modeled first in simple form by applying an appropriate multiplier to the laminar viscosity, and second, by using a Prandtl-Van Driest algebraic turbulence model which evaluates the turbulent viscosity as a function of a local mixing length and the three-dimensional vorticity.

The initial guess for the velocity field is obtained by performing a multiple stream tube analysis to obtain approximate values for the magnitude of the velocity and the flow direction at each point in the flow field. These data, in conjunction with an assumed total pressure, can be used to initialize the primitive variables using isentropic relationships. Properties at the fishbowl inlet were based on measured data and held fixed during the relaxation procedure. The calculations were performed on a CYBER 205 supercomputer using slightly less than 8000 nodal points to discretize the computed flow region.

Results

A composite view of the computational (sparse) grid representing the fishbowl and the two transfer ducts is shown in Fig. 1. The geometric configuration is symmetric about a plane containing the pump axis. The inner flow surface, basically a circular cylinder, represents the turbine simulator, and the outer flow boundary is given by the spherical fishbowl wall. The exact position of the transfer ducts is more clearly seen as Fig. 2, representing an axial cut through the manifold and also showing the actual computational grid used in the calculations. The length and the exit face of the transfer ducts were determined by the location and orientation of pitot pressure probes installed in the experimental hardware.

Flowfield results obtained in the transfer ducts are shown in Figs. 3 and 4 in terms of a velocity vector map and a corresponding Mach number contour map, respectively. The region shown represents a longitudinal plane cut at an angle of approximately 45 deg to the fishbowl entrance plane (that is, a cut through the upper outer and lower inner quadrant of the transfer duct). Note the formation of a vortex, the location of which is in excellent agreement with experimental data which indicated a flow separation region in he same location. The separation region is more clearly delineated in the Mach number map of Fig. 4.

Measured and computed static and total pressures as functions of circumferential location in the fishbowl at an axial location roughly corresponding to the lower edge of the transfer duct inlet are compared in Figs. 5 and 6. The computed results for the two turbulence models mentioned previously are seen to be in very good agreement with the measured data. The rather close agreement between the two turbulence models can be attributed to the fact that in this analysis it is the configuration geometry that represents the dominant cause for turbulence in the flow.

Having demonstrated that available numerical analysis is capable of producing results which agree with experimental data, a principal goal of the effort undertaken here was to use the analysis to explore changes in the configuration which would result in more favorable flow behavior. As an example, a computational grid incorporating a greatly enlarged radius of curvature of the transfer duct inlet fairing, especially in the region just upstream of the duct flow separation region found in the experimental configuration, is shown in Fig. 7. For clarity, only the outer flow boundary is shown. Neither the duct location, orientation nor size of the transfer ducts was affected by this modification. Flowfield calculations for the modified configuration were then performed using initial and boundary conditions identical to those previously used for the nominal configuration geometry. The results are shown in Figs. 8 and 9, again in terms of a velocity vector and a Mach number contour map. The most significant finding is the diminution of the flow separation region (see Figs. 3 and 4) as a result of increasing the transfer duct inlet fairing radius of curvature.

Conclusions

Three-dimensional viscous flow in a conceptual twin-duct SSME hot gas manifold was numerically computed and the results were found to be in good agreement with measured data. The results of this study demonstrate the feasibilty and potential usefulness of computational fluid dynamics methods in the design and improvement of multi-dimensional flow configurations as complex as those of certain SSME components.

References

1. D.G. Pelaccio, F.F. Lepore, G.M. O'Connor, G.V.R. Rao, G.H. Ratekin, and S.T. Vogt, "Experimental Evaluation of an Advanced Space Shuttle Main Engine Hot-Gas Manifold Design Concept," AIAA/SAE/ASME 20th Joint Propulsion Conference, Paper AIAA-84-1463, June 1984.

2. L.W. Spradley, P.G. Anderson, and M.L. Pearson, "Computation of Three-Dimensional, Nozzle Exhaust Flow Fields with the GIM Code," NASA Contractor Report 3042, August 1978.

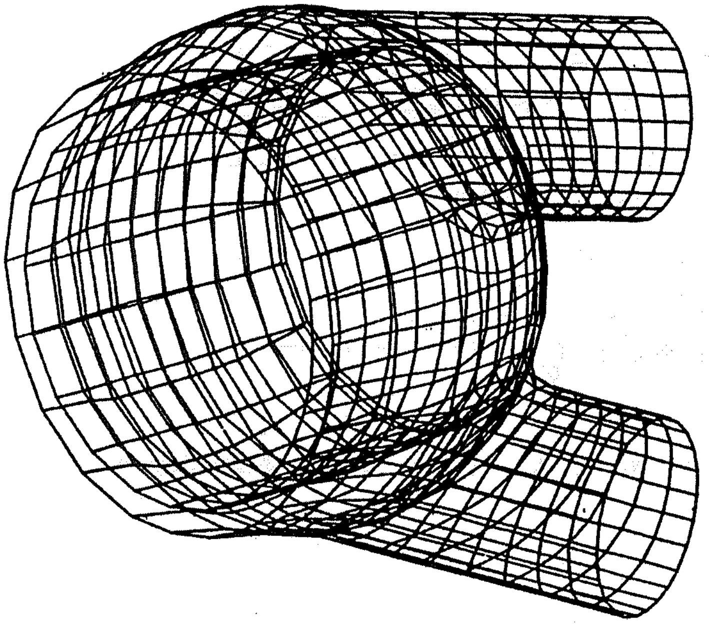

Fig. 1 SSME HGM Computational Grid, Sparse Representation, Viewed at 45 deg Angle to Bowl Axis

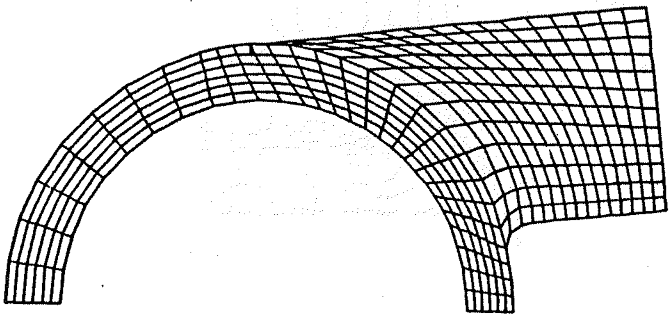

Fig. 2 SSME HGM Computational Grid, Viewed Along Bowl Axis

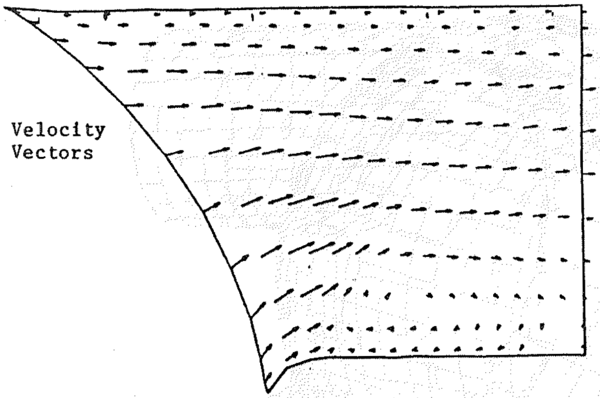

Fig. 3 SSME HGM Velocity Vector Map in Transfer Duct

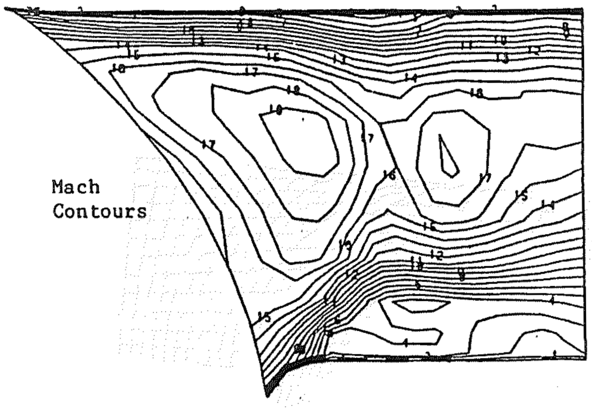

Fig. 4 SSME HGM Mach Contour Map in Transfer Duct

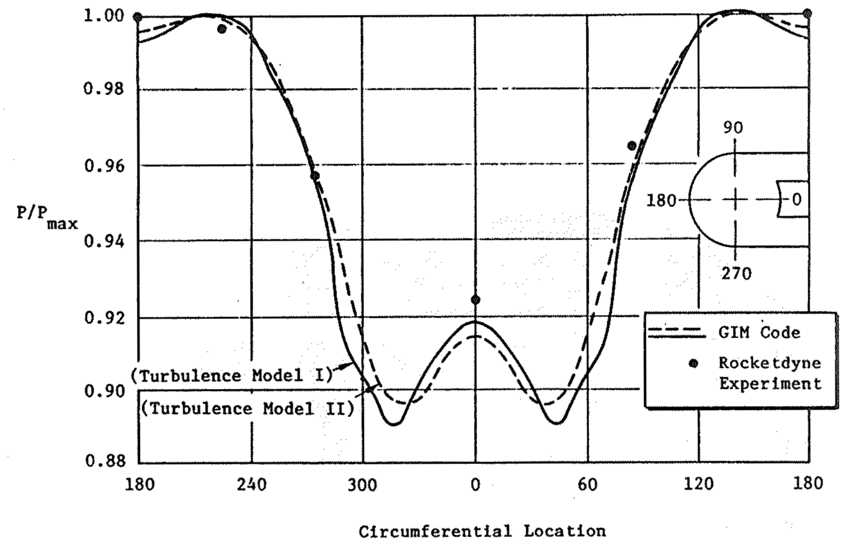

Fig. 5 Comparison of Computed Static Pressures to Measured Static Pressures in Bowl

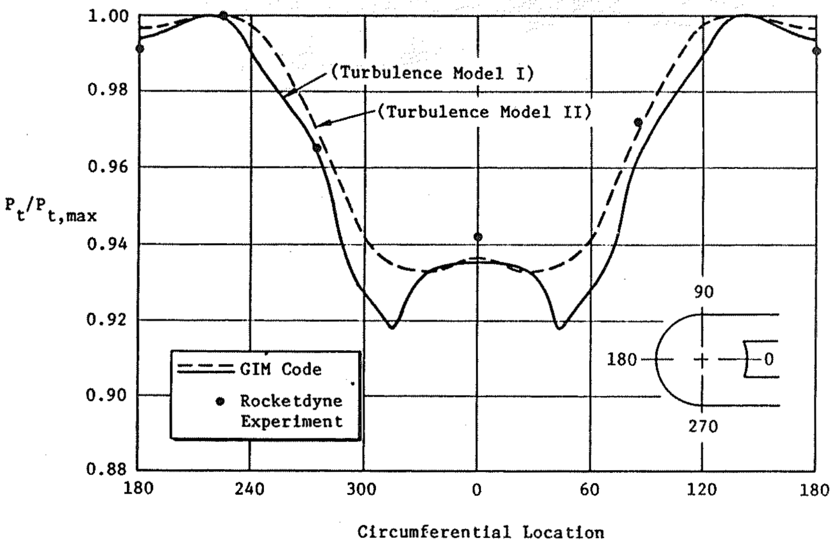

Fig. 6 Comparison of Total Pressures to Measured Data in Bowl

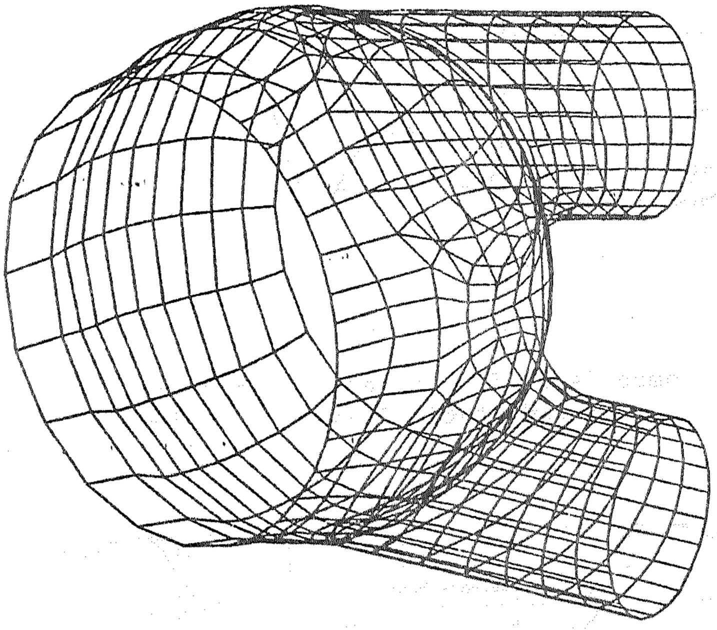

Fig. 7 SSME HGM Modified Grid

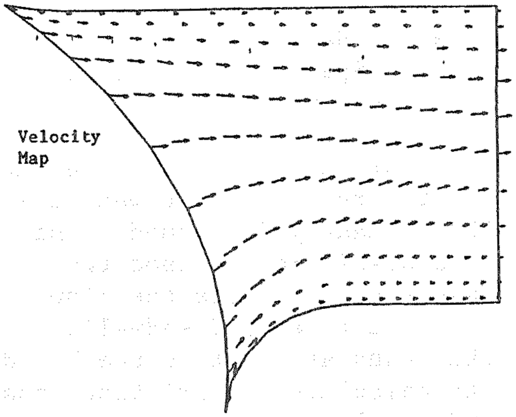

Fig. 8 Transfer Duct Velocity Vector Map in Modified Configuration

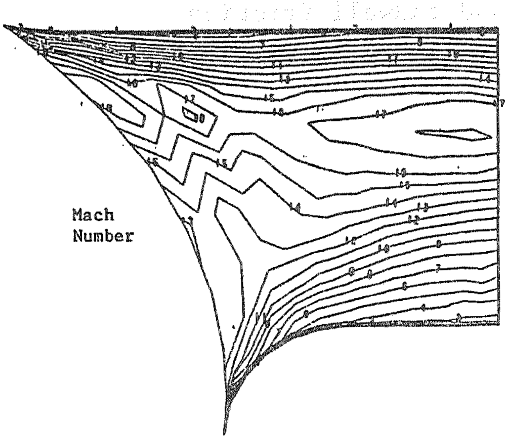

Fig. 9 Transfer Duct Mach Number Contour Map in Modified Configuration