LMSC-HEC TR F225770

FINAL REPORT

ANALYSIS OF INTERNAL FLOWS RELATIVE TO THE SPACE SHUTTLE MAIN ENGINE

30 December 1987

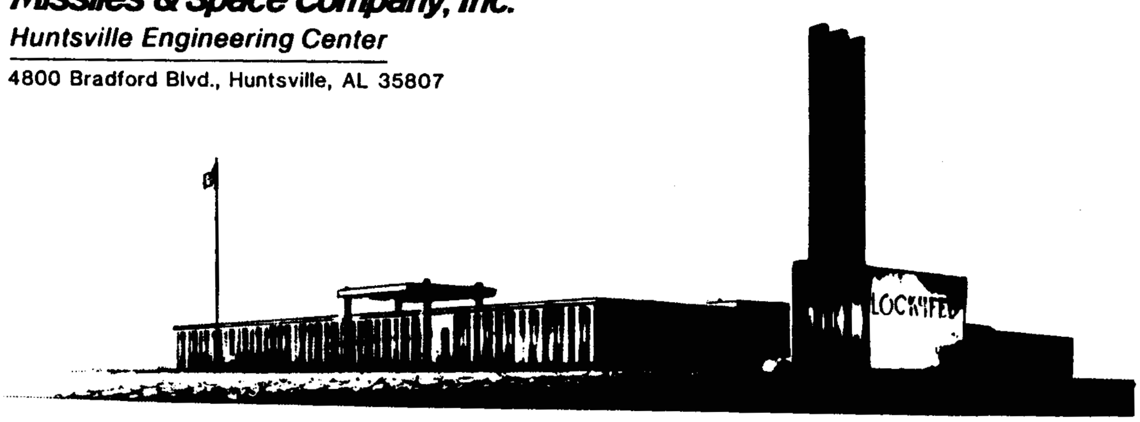
Contract NAS8-35984

Prepared for

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION
MARSHALL SPACE FLIGHT CENTER, AL 35812

By

Lockheed
Missiles & Space Company, Inc.
Huntsville Engineering Center
4800 Bradford Blvd., Huntsville, AL 35807



FOREWORD

This report was prepared by personnel of the Computational Mechanics Section of Lockheed's Huntsville Engineering Center. It constitutes final documentation of efforts performed under Contract NAS8-35984 for NASA-Marshall Space Flight Center.

The NASA-MSFC Contracting Officer's Representative for this research study was Mr. G.A. Wilhold, ED31.

CONTENTS

1. INTRODUCTION

One of the problem areas in the design of the Space Shuttle Main Engine (SSME) is in the second stage rotor in the high pressure fuel turbopump (HPFTP). Cracks have developed on the blade shank, apparently due to thermal stresses generated by strong thermal gradients. It is generally believed that the offending thermal gradients are caused by cold liquid hydrogen flowing directly over, and thus over-cooling, the aft side of the shank. A schematic of the current second stage rotor and platform seal region is shown in Fig. 1-1. The cold liquid hydrogen flows through the labyrinth seal into the cavity on the aft side of the second stage rotor and through the gap between the blade platform and platform seal, thus strongly cooling the aft side of the blade shanks. A proposed remedy for this problem is to divert the cold flow away from the blade shank region by means of a mechanical barrier or by providing a bypass for the cold flow through the labyrinth seal. A possible flow diverter and coolant bypass configuration is shown in Fig. 1-2.

The primary objective to the first task originally described by NASA-MSFC was to perform a three-dimensional CFD analysis for the flow environment in the region shown in Fig. 1-1 as well as that in Fig. 1-2, showing the base line flow diverter configuration, and to compare these results. Subsequently alternate diverter geometries were to be investigated to determine the optimum system design.

A second distinctly different task was to perform an unsteady, three-dimensional liquid oxygen (LOX) manifold analysis to describe the flow and phase distribution of oxygen as the main oxidizer value to the fuel preburner is opened. A CFD model would be provided for accurate description of the two-phase flow of oxygen in these passages so that the unsteady flow rates through the preburner injection elements can be simulated.

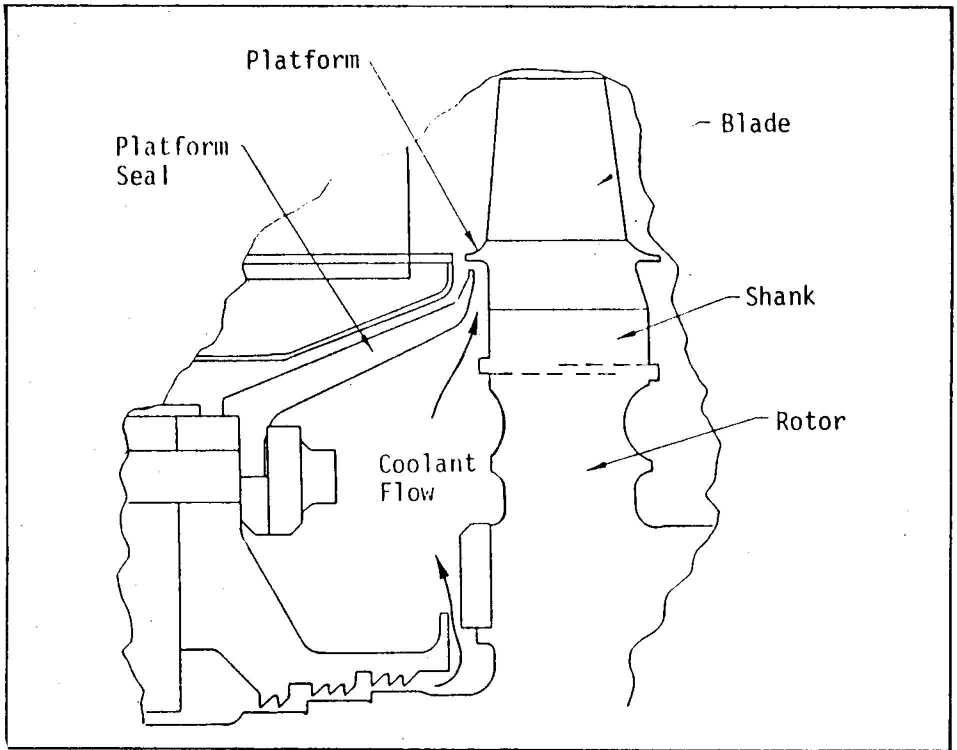


Fig. 1-1 Schematic of SSME Fuel Side Second Stage Turbine Rotor REgion and Coolant Flow

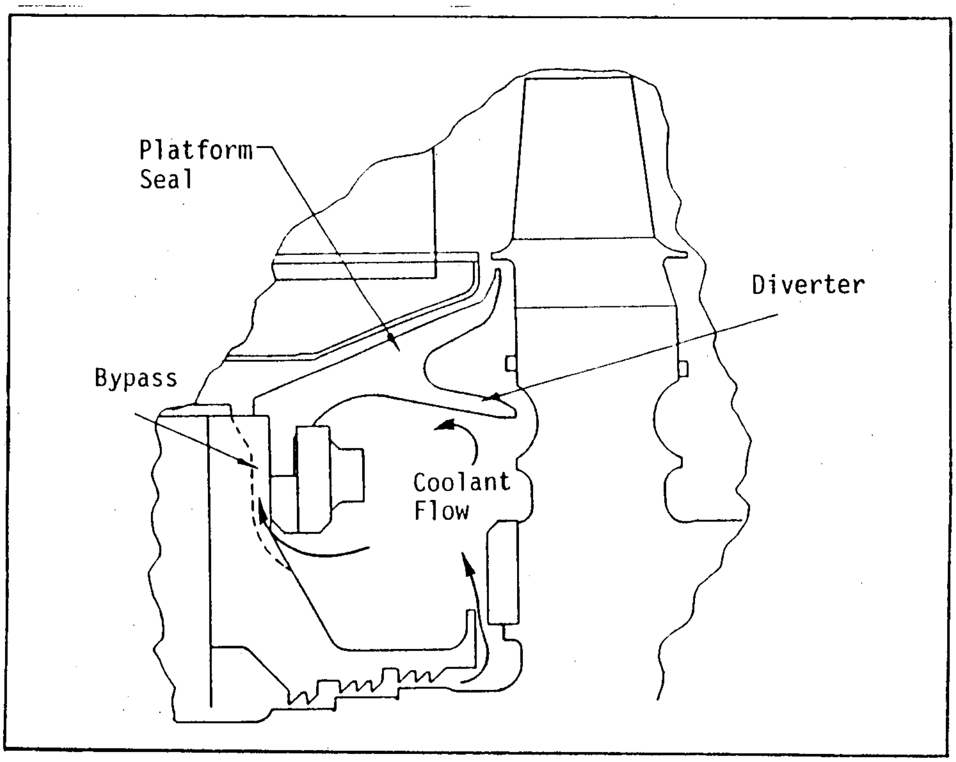


Fig. 1-2 Schematic of SSME Fuel Side Second Stage Turbine Rotor Region Showing Modified Aft Platform Seal with Flow Diverter and Coolant Bypass

Shortly after commencement of efforts on the flow diverter task described above, NASA-MSFC engineers abandoned the flow diverter concept. The contracting officer's representative for this contract redirected efforts as follows. In lieu of performing the flow diverter analysis it was directed that the Lockheed Computational Mechanics staff cooperate with the SSME computational staff at NASA-MSFC in developing and providing for NASA numerical capabilities for accurate analysis of much of the internal flows related to the SSME. This would involve consulting with the NASA CFD personnel as to which CFD techniques should be used, which appropriate CFD codes are available, and what modifications to the codes would be necessary for application to the SSME problem. Efforts and results for this redirected work are described in Section 2 of this report.

Efforts under the LOX manifold task were performed under a subcontract to the Software Engineers Consultants, Analysis (SECA) engineering firm of Huntsville, Ala. Description of work performed under this subcontract is described in Section 3 of this document.

2. CFD CAPABILITIES DEVELOPMENT

2.1 INTRODUCTION

Since the Mach number for most of the internal fluid flows to the SSME are below 0.2, it was decided, by the Contracting Officer's Representative, that an incompressible Navier-Stokes flow solver is the appropriate methodology to seek in a CFD code. An additional requirement is that the discrete solver in the code be implicit in nature. This is important for proper resolution of viscous boundary effects which could be responsible for undesirable separation and re-circulation regions. Such phenomena are more accurately detected when computational grid points are placed very close to the walls. This extremely small nodal spacing would adversely affect the usefulness of an explicit code because of the CFD limitation.

Several numerical methods have been developed (Refs. 1 through 6) to solve the incompressible Navier-Stokes equations in three-dimensional body-fitted coordinate systems. That which differs in these methodologies is the procedure for finding the pressure field so that the flow field is as close to divergence-free as possible. Satisfying this requirement is the main difficulty in solving incompressible flow problems numerically. The techniques of Refs. 1, 2, and 3 employ the pseudocompressibility approach. This is a time-iterative scheme which generates the pressure field so that the continuity equation is satisfied when a steady state solution is reached. Methods employed in Refs. 4, 5, and 6 utilize a successive pressure-velocity correction scheme for ensuring that the continuity equation is satisfied. Here a Poisson equation for pressure correction is used which is derived approximately from both the continuity and momentum equations. They are called SIMPLE methods (Semi-Implicit Method for Pressure-Linked Equations).

Much information was gathered from the scientific literature and individuals with experience in using codes based on both of these solution techniques. In general it was found that for similar FORTRAN coding structures the speeds, in CPU seconds per iteration per node, for both is comparable. A fundamental difference between the pseudocompressibility and the SIMPLE codes is that the former uses finite difference techniques whereas the latter uses finite or control volume techniques. This difference allows the pseudocompressiblity coding to be much less storage intensive. It was for this reason that this approach was selected as the one to pursue.

2.2 CODE DESCRIPTION

Much success (Refs. 7 and 8) had been enjoyed by the INS3D (Ref. 3) code. Thus it was the one chosen to obtain, learn, modify, and apply to SSME internal flowfield investigations. A basic unedited flow solver version of the INS3D source code was obtained from NASA-Ames. This version provides for inclusion of a turbulence model subroutine but does not contain one. At the time the code was acquired no user's manual was available but one has subsequently been documented (Ref. 9). This NASA document contains a detailed description of the code and its methodology. We present here a very brief description.

- INS3D is a three-dimensional fluid dynamics code for computing steady state incompressible flow fields in arbitrary fixed geometries. The finite difference solution procedure employs an implicit, approximate factorization scheme (Refs. 10 and 11) to solve the three-dimensional Navier-Stokes equations in primitive variables form.
- The formulation uses a three-dimensional coordinate transformation from Cartesian to general curvilinear coordinates. The pressure solution is based on the pseudocompressibility method. The Reynolds stresses are uncoupled from the implicit scheme and are lagged by one time step to facilitate implementing any form of eddy viscosity turbulence model which the user chooses. Explicit boundary conditions admissible are no slip, inflow, outflow, and periodic.

INS3D has been verified by computing fundamental fluid dynamics problems such as the flow through a channel (Ref. 3), flow over a backward-facing step (Ref. 12), and flow over a circular cylinder (Refs. 3 and 12). Three-dimensional cases include flow over an ogive cylinder (Ref. 13), flow through a rectangular duct (Ref. 3), wind-tunnel-inlet flow (Ref. 14), cylinder-wall juncture flow, and flow through multiple posts mounted between two plates (Refs. 15 and 16).

This is a very robust Navier-Stokes solver but is by no means designed to be a general code to be used by the uninitiated for application to an arbitrary problem. Grid generation is user supplied as are flow field initialization and application to symmetry boundaries. For multiple zoning major modifications to the code are required, and as mentioned previously, the user must supply an eddy viscosity subroutine for turbulence modeling.

2.3 CODE MODIFICATION

Several modifications were implemented into the basic INS3D source code obtained from NASA-Ames, of these only two are of significance to be reported here.

A subroutine for determining the value of the eddy viscosity at each node was added. Because of the convoluted geometries generic to the internal SSME structure a simple algebraic model was chosen. In much of the flow passages strong curvature effects are present, and as a result of the rapid turning of the fluid, the balance between strong pressure gradients and centrifugal forces dominate the evolution of the flow. It follows that although turbulent mixing is present it plays a secondary role in the physics of what is happening. It thus appears that the accuracy of the numerical technique used for solving the fluid dynamic equations is of paramount importance and will yield reliable predictions even with a relatively simple turbulence model.

The particular model chosen (Ref. 17) is a Prandtl-Van Driest mixing length type in which the eddy viscosity is proportional to the local vorticity. The mixing length is a function of the distance from a wall, and for the types of channel flows of relevance here, of the distance to the position of minimum vorticity. This kind of turbulence model is not difficult to code and has demonstrated surprisingly good agreement with experimental measurements in geometries like those found in the SSME (Ref. 18).

A second modification effected to the basic INS3D source code was an explicit multi-zone or multi-block procedure. The methodology of the code, as is true for most codes, is structured such that computations are preformed in a transformed curvilinear coordinate space, where the range of all three co-ordinates is from zero to one. Thus the calculations are actually done on a three-dimensional cube or block. Multi-blocking is a method of linking computations performed on two or more such blocks while preserving the physics which needs to be communicated between blocks or zones.

The procedure adopted is an explicit method (Ref. 7) which works well for subsonic flows and lends itself to straight forward coding. The actual coded scheme assumes that at least two planes of nodes in connected zones be in common. In each plane the nodal distribution in one of the curvilinear directions for both zones must be coincident. If positions of nodes in these two planes for each zone are not coincident in the remaining curvilinear directions, linear interpolation between nearest neighbors is used to obtain flow variables from one zone to the other.

Giving the forgoing geometric requirement the solution methodology is simple. For a two-zone computation the following would be used. Subsequent to time step N in Zone 1 flow variables are either transferred or interpolated to the inlet coincident plane of Zone 2. These values are held fixed during the time step sweeping in Zone 2. After this is done, flow variables are obtained on the second common plane of Zone 1 from a similar equating or interpolating procedure. This process is repeated until a converged solution is reached in both zones. Typically, the two-block procedure requires from 1.5 to 2.0 times the number of numerical iterations as would be required if both zones were solved as a single block.

2.4 TEST CASE RESULTS

Initial computations performed with the INS3D code were accomplished using the CRAY-XMP at the NASA-Ames research facility. Subsequently, NASA-MSFC obtained its own CRAY-XMP and we were required to transfer the code to MSFC and devote the necessary time required to get INS3D operational on this system. Accomplishing this task demanded considerable effort because of differences in the front end machine and communications network.

A two-dimensional geometry was generated having the dimensions of the FMOF version of the SSME/HGM fuel side turnaround duct. The computational grid, shown in Fig. 2-1, contained 3861 nodes and employed a grid stretched to both walls. This grid was used for almost all of the test cases, both laminar and turbulent, single and multi-block.

Results for laminar flow at a Reynolds number of 1,000 are presented in Fig. 2-2. Because of the two-dimensional nature of this flow the fluid is seen to accelerate near the inner wall around the turn and a large separated region is seen to have developed on the downstream side of the turn. These results can be compared to those for a turbulent flow computation at a Reynolds number of 1,000,000 using the turbulence model described in Section 2.2. These results are displayed in Fig. 2-3.

Multi-blocking test cases were first performed on a grid with the same nodal distribution as shown in Fig. 2-1 but used as a straight two-dimensional duct or channel. The two overlapped planes between the two zones was placed at mid-channel. Results of this computation are compared in Fig. 2-4. Both single and multi-block computations were carried to the same level of convergence. There is essentially no discernable difference in the results. Additional multi-blocking experiments were carried out using the turnaround duct geometry where the two overlapped planes from each block or zone were placed in the most precarious position just at the end of the 180-deg turn. These results are displayed for comparison in Figs. 2-5 through 2-9.

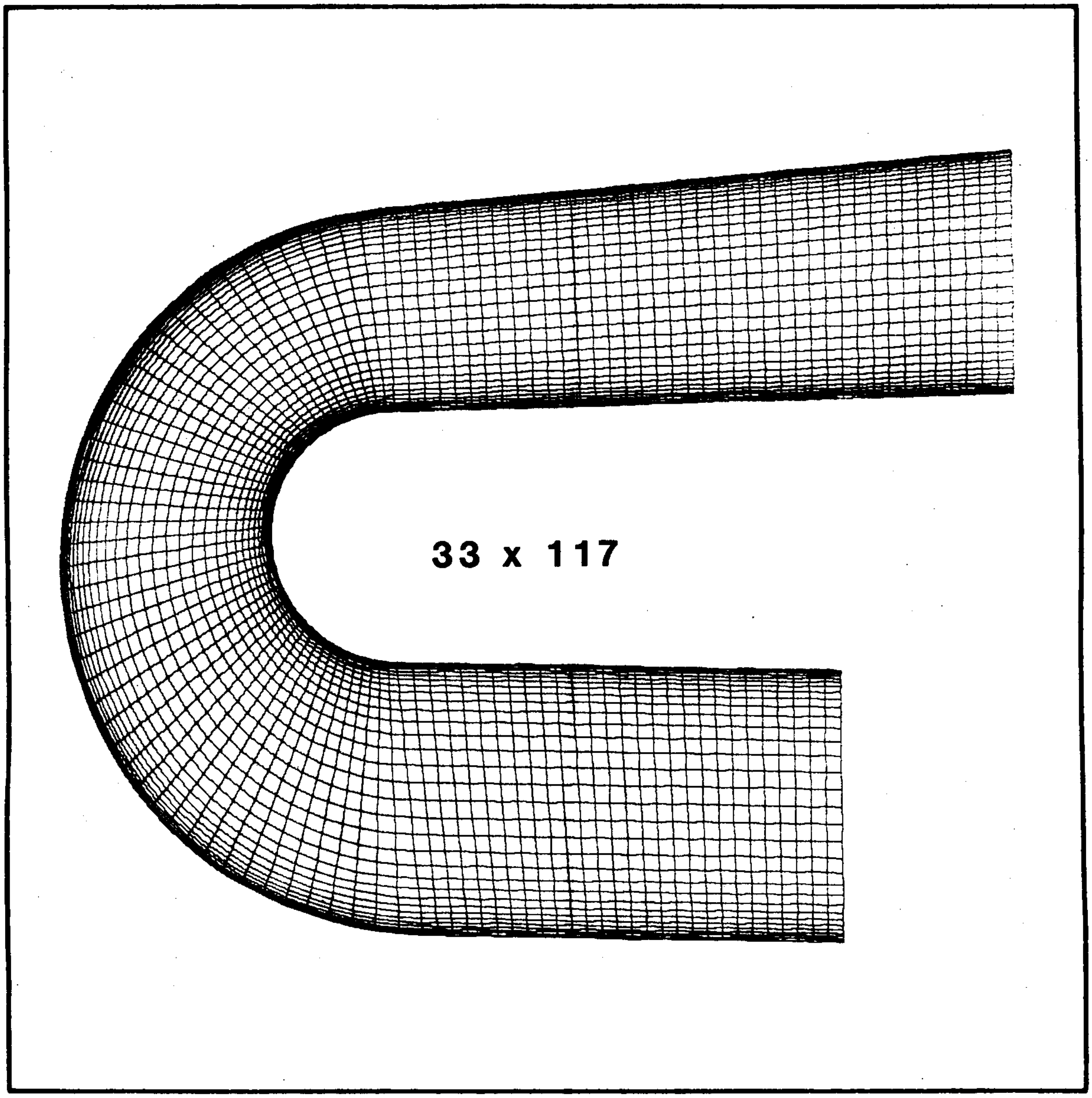


Fig. 2-1 Two-Dimensional Computation Grid for Testing Modifications Made to INS3D Code

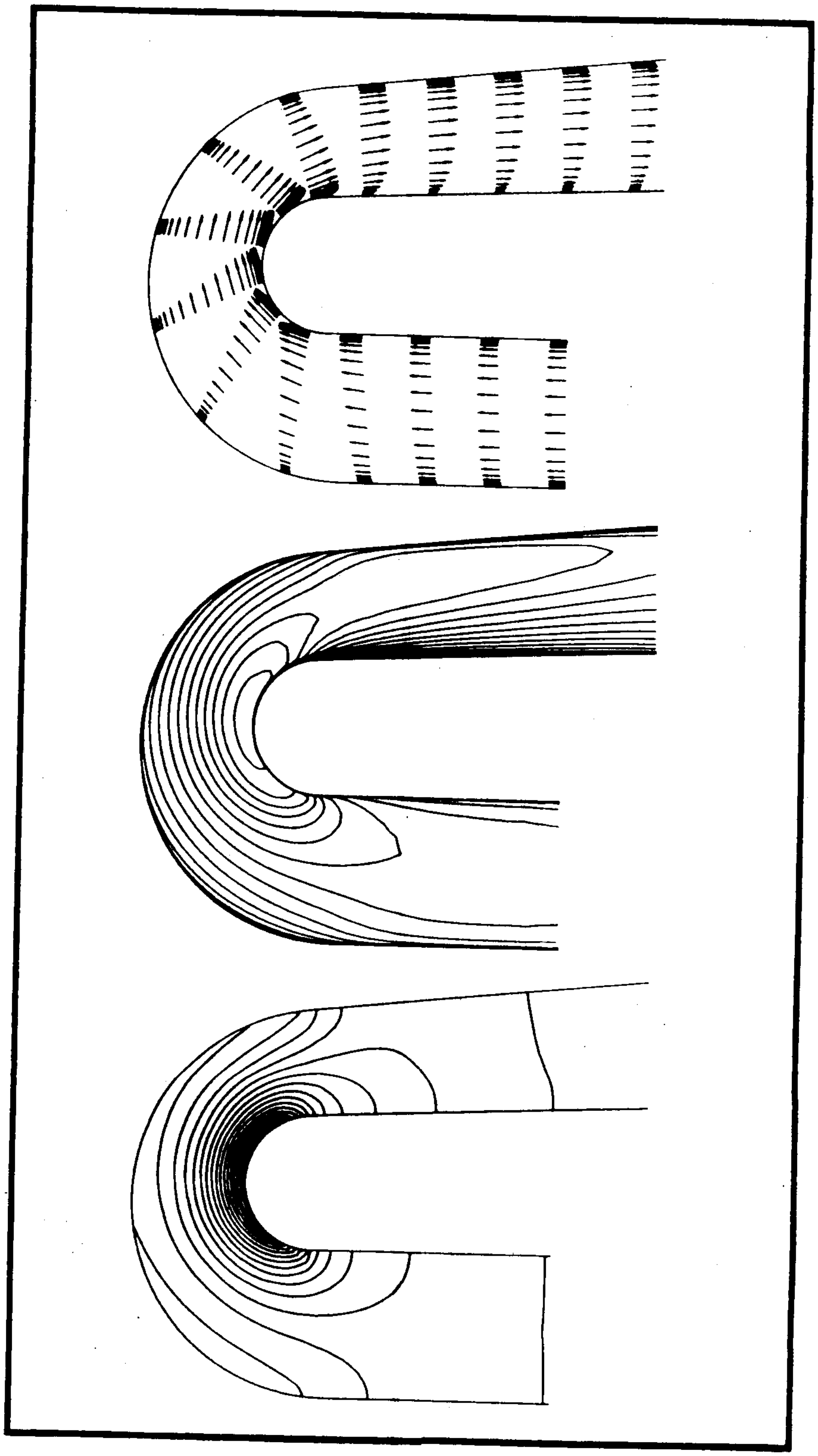


Fig. 2-3 Velocity Vectors (Top), Velocity Magnitude Contours, and Static Pressure Contours (Bottom) for Turbulent Flow at Re = 10^6 Using INS3D

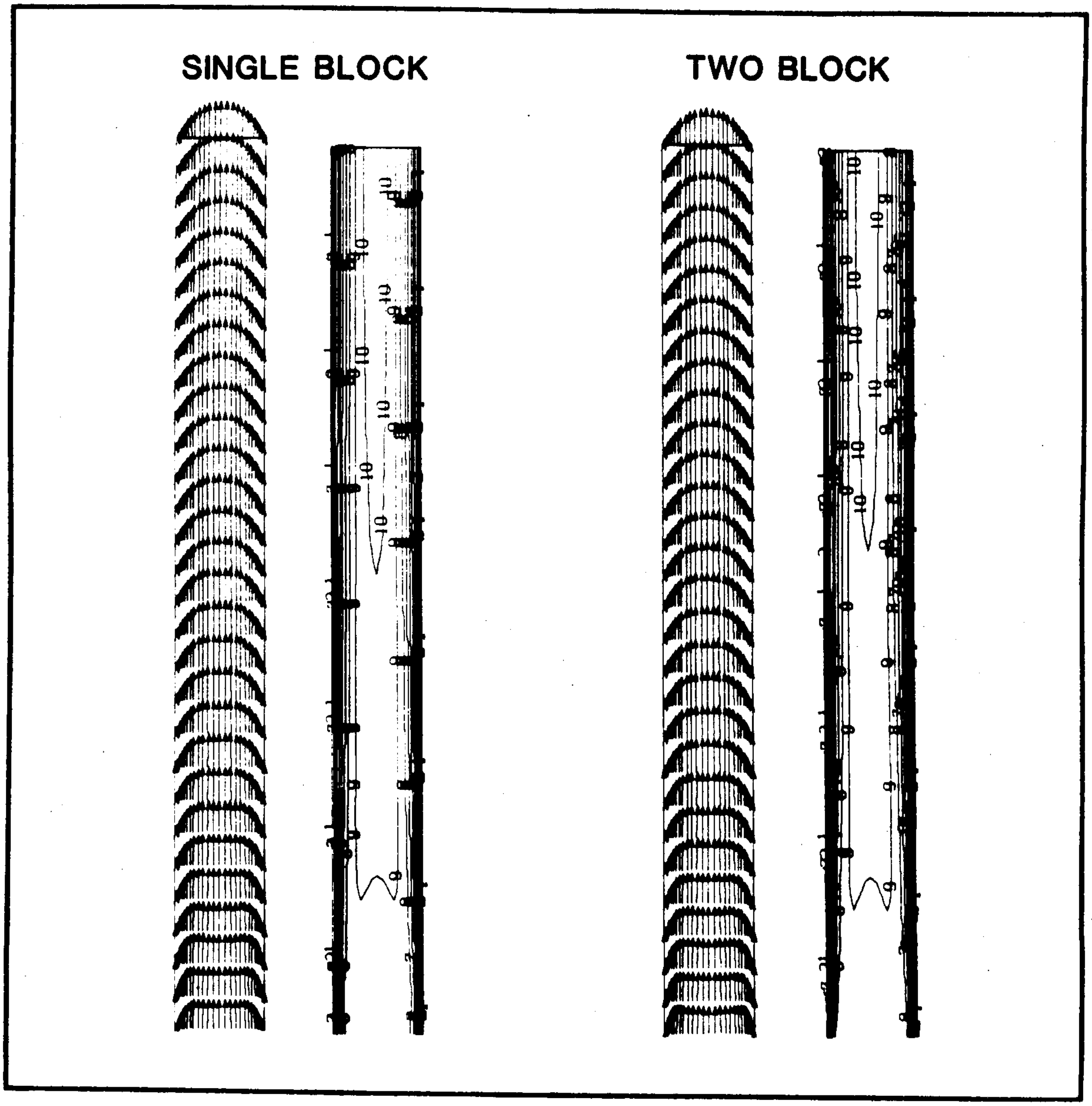


Fig. 2-4 Comparison of Velocity Vectors and Velocity Contours for Single and Multi-Block Cases in Straight Duct for Laminar Flow at Re = 500 with Blocking at Mid Channel

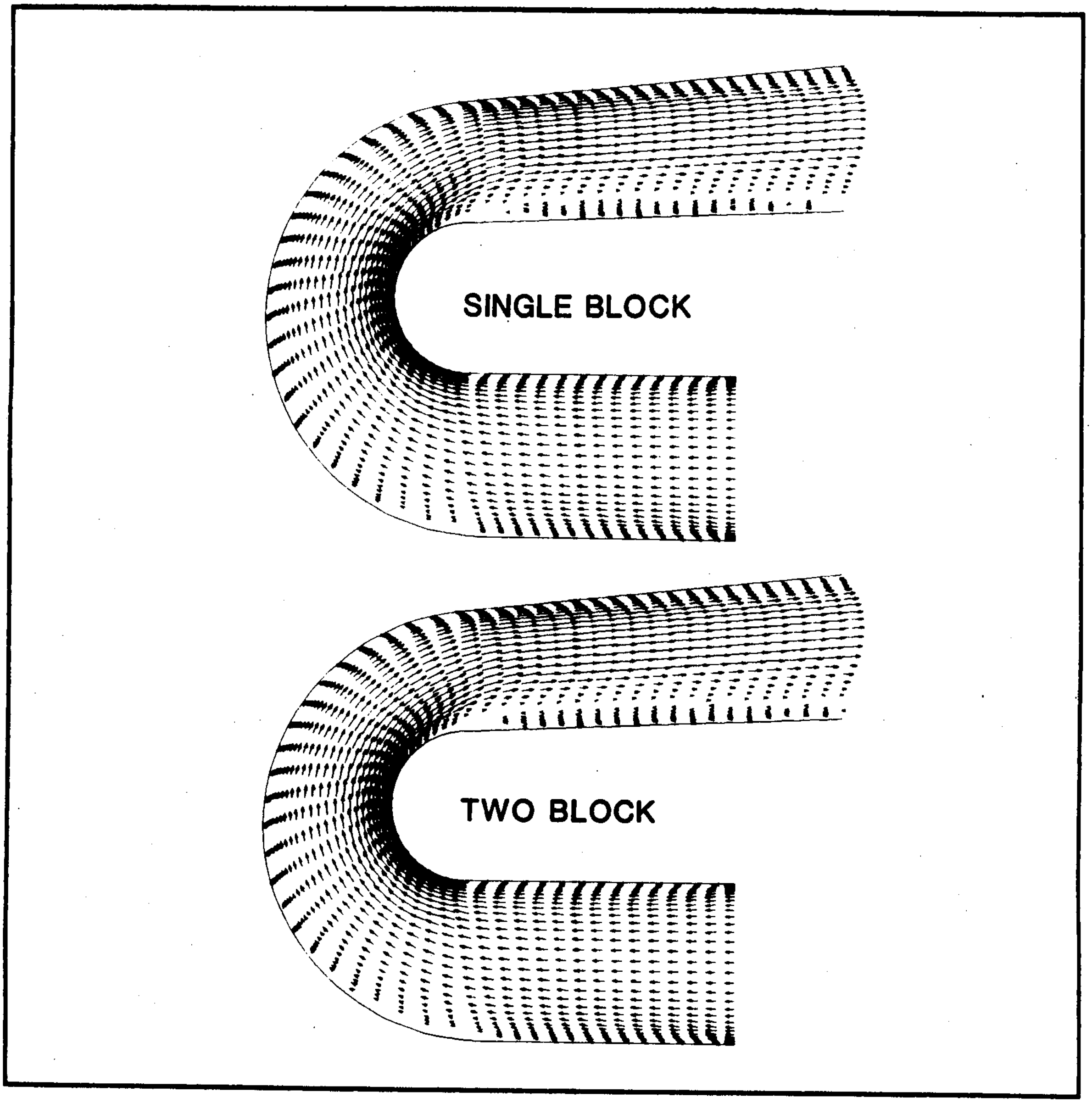


Fig. 2-5 Comparison of Velocity Vectors for Single and Multi-Block Cases in 2-D Turnaround Duct for Laminar Flow at Re = 500 with Blocking at Downstream Position of 180-deg Turn

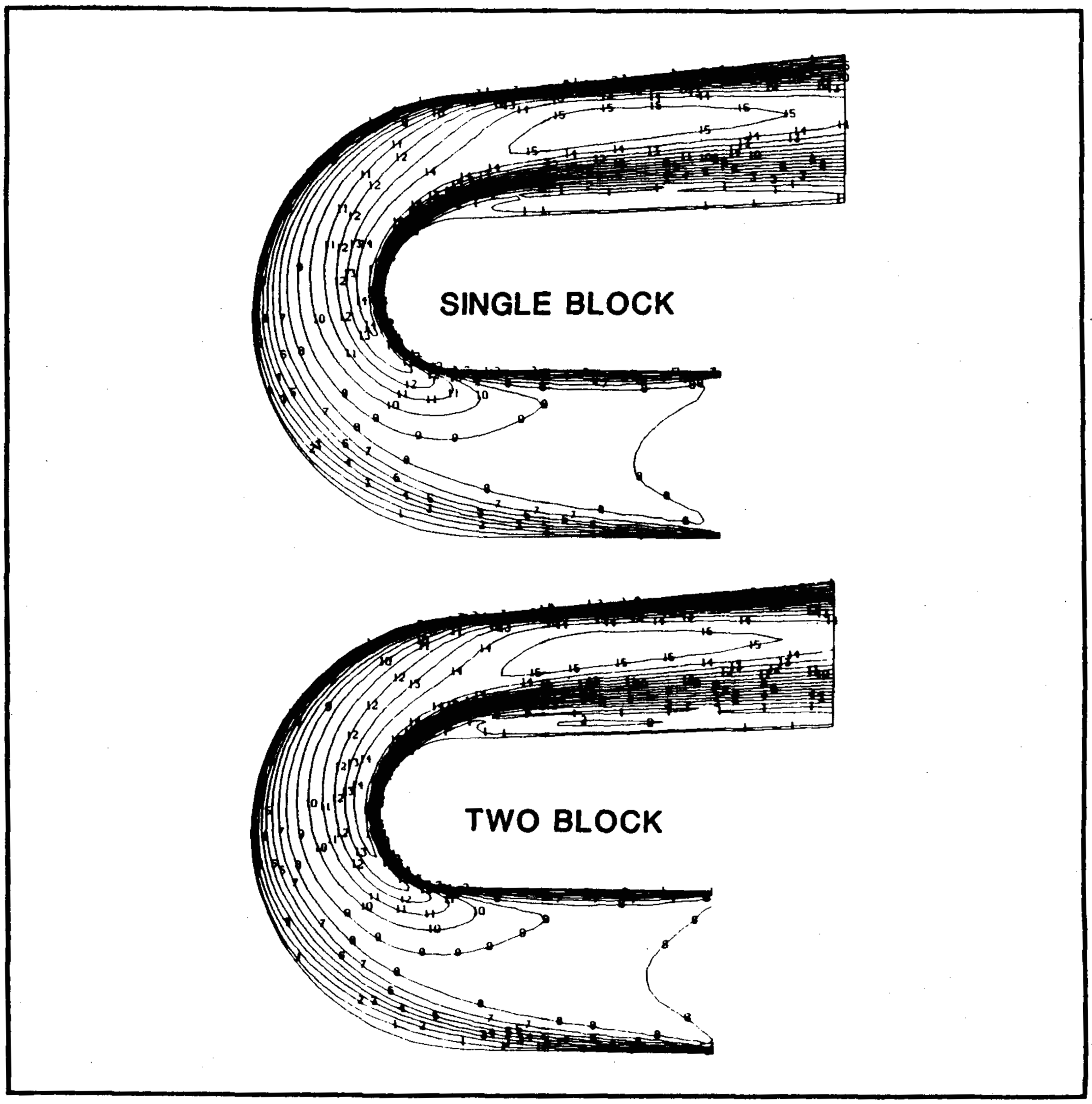


Fig. 2-6 Comparison of Velocity Contours for Single and Multi-Block Cases in 2-D Turnaround Duct for Laminar Flow at Re = 500 with Blocking at Downstream Position of 180-deg Turn

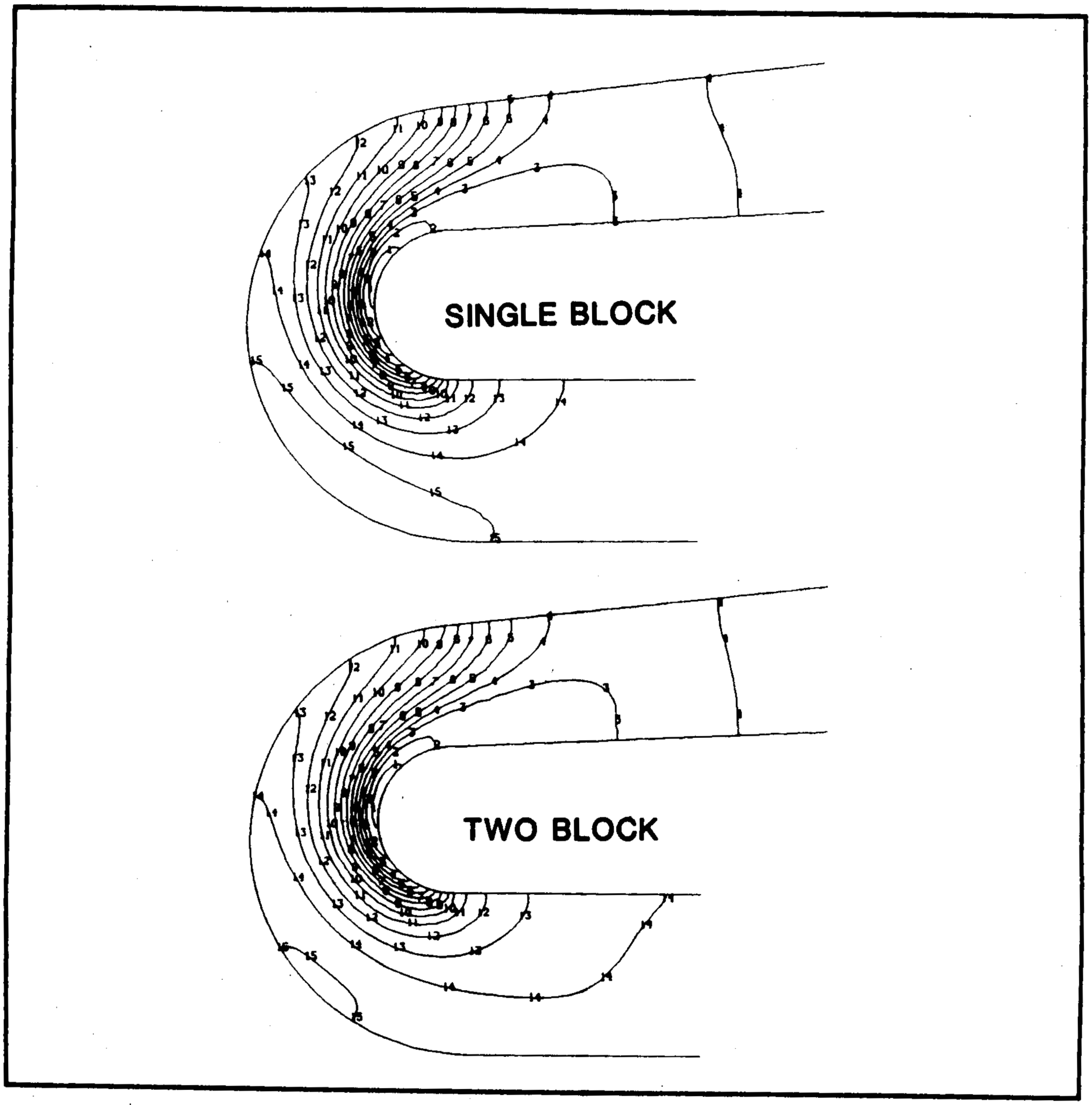


Fig. 2-7 Comparison of Pressure Contours for Single and Multi-Block Cases in 2-D Turnaround Duct for Laminar Flow at Re = 500 with Blocking at Downstream Position of 180-deg Turn

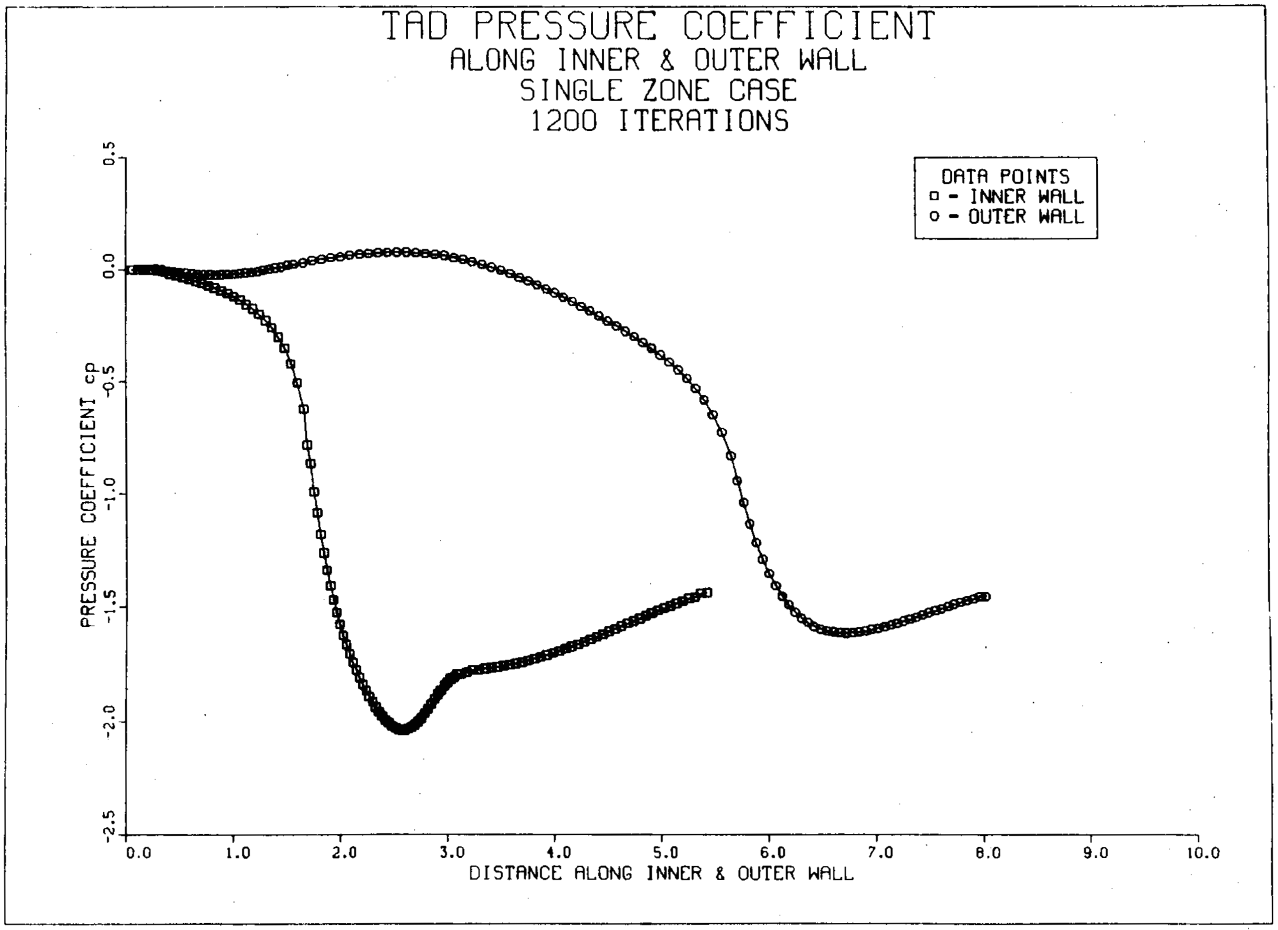


Fig. 2-8 **Single Block Results for the Linear Distribution of Pressure Along Inner and Outer Wall of 2-D Turnaround Duct for Case of Re = 500**

ORIGINAL PAGE IS
OF POOR QUALITY

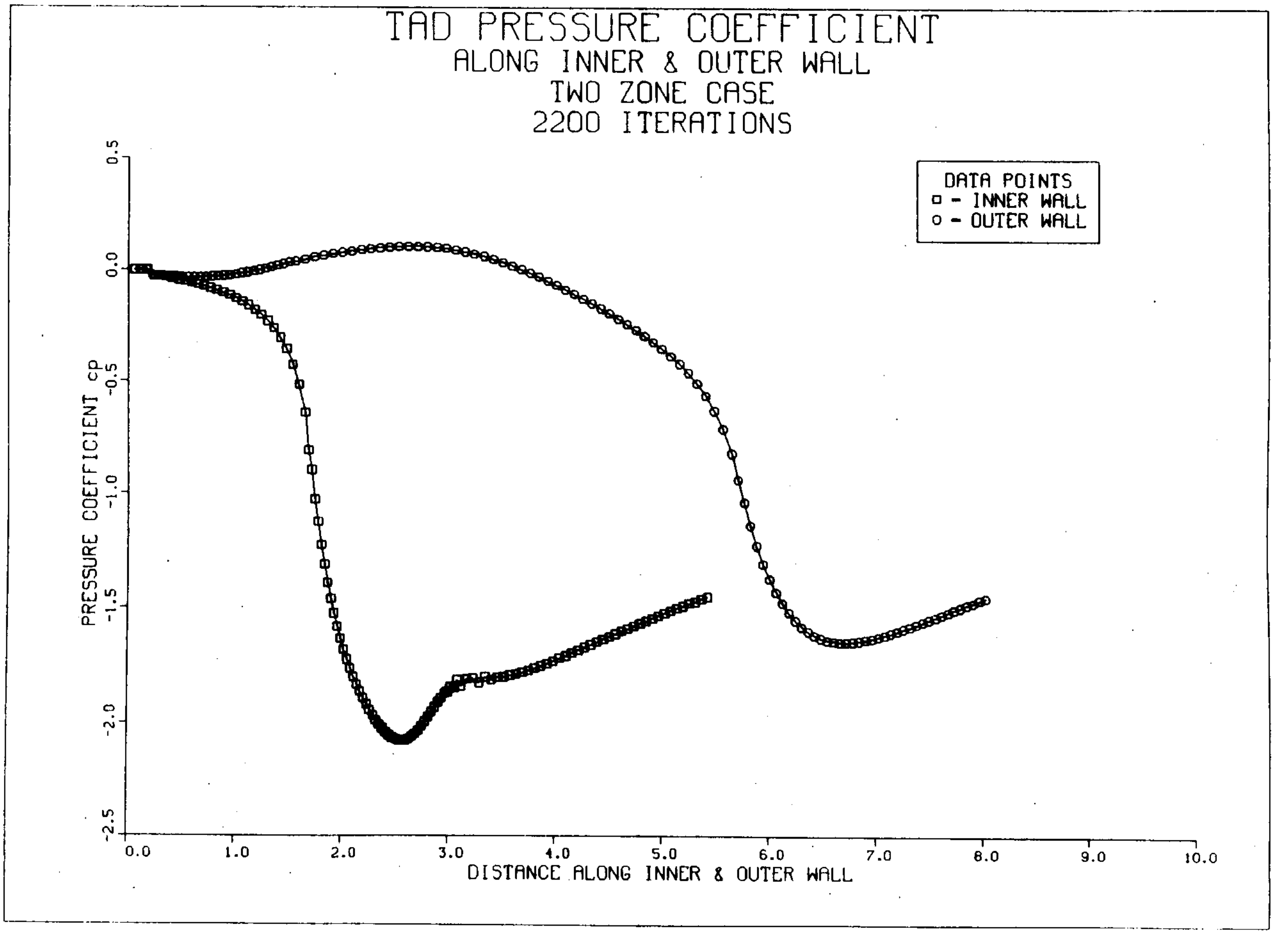


Fig. 2-9 Multi-Block Results for the Linear Distribution of Pressure Along Inner and Outer Wall of 2-D Turnaround Duct for Case of Re = 500

The static pressure distribution for any of these calculations is the most sensitive diagnostic for convergence as well as for comparison of solutions. Thus we present in Figs. 2-8 and 2-9 a plot of the pressure along the inner and outer wall for both single and two-block computations. The two curves for each lie directly on top of each other.

3. LOX MANIFOLD ANALYSIS

3.1 INTRODUCTION

The preburners of the Space Shuttle Main Engine still experience serious problems with respect to cracking of LOX posts and the frisbee in the inlet manifolds. Operation of the fuel preburner creates flows which cause turbine blade cracking in the high pressure fuel turbopump. The most likely cause of these problems is the extreme thermal environments caused by startup and shutdown of the engine. A detailed transient analysis of these flows is necessary in order to help alleviate these problems.

Operation of the fuel preburner is initiated by opening propellant control valves and igniting the flow with the augmented spark igniter (ASI). The configuration of the FPB is shown in Fig. 3-1. The flow passage from the fuel preburner oxidizer valve (FPOV) into the oxidizer manifold above the interpropellant plate is shown in this figure, and a cross-sectional view of this passage is presented in Fig. 3-2. The oxidizer flow to the ASI is supplied from a bleed line from the FPOV. The FPOV is shown in Fig. 3-3. The valves which control this flow are shown schematically in Fig. 3-4; notice that there is not a separate fuel control valve for each preburner. Flows from the fuel and oxidizer manifolds enter the FPB combustor through the injector faceplate, the layout of which is shown in Fig. 3-5. These figures were taken from Refs. 19 and 20.

The object of this study was to perform an unsteady, three-dimensional LOX manifold analysis to describe the flow and phase distribution of oxygen as the main oxidizer valve to the FPB is opened. A computational fluid dynamics model was developed to describe the filling and chill down of the LOX manifold.

ORIGINAL PAGE IS
OF POOR QUALITY

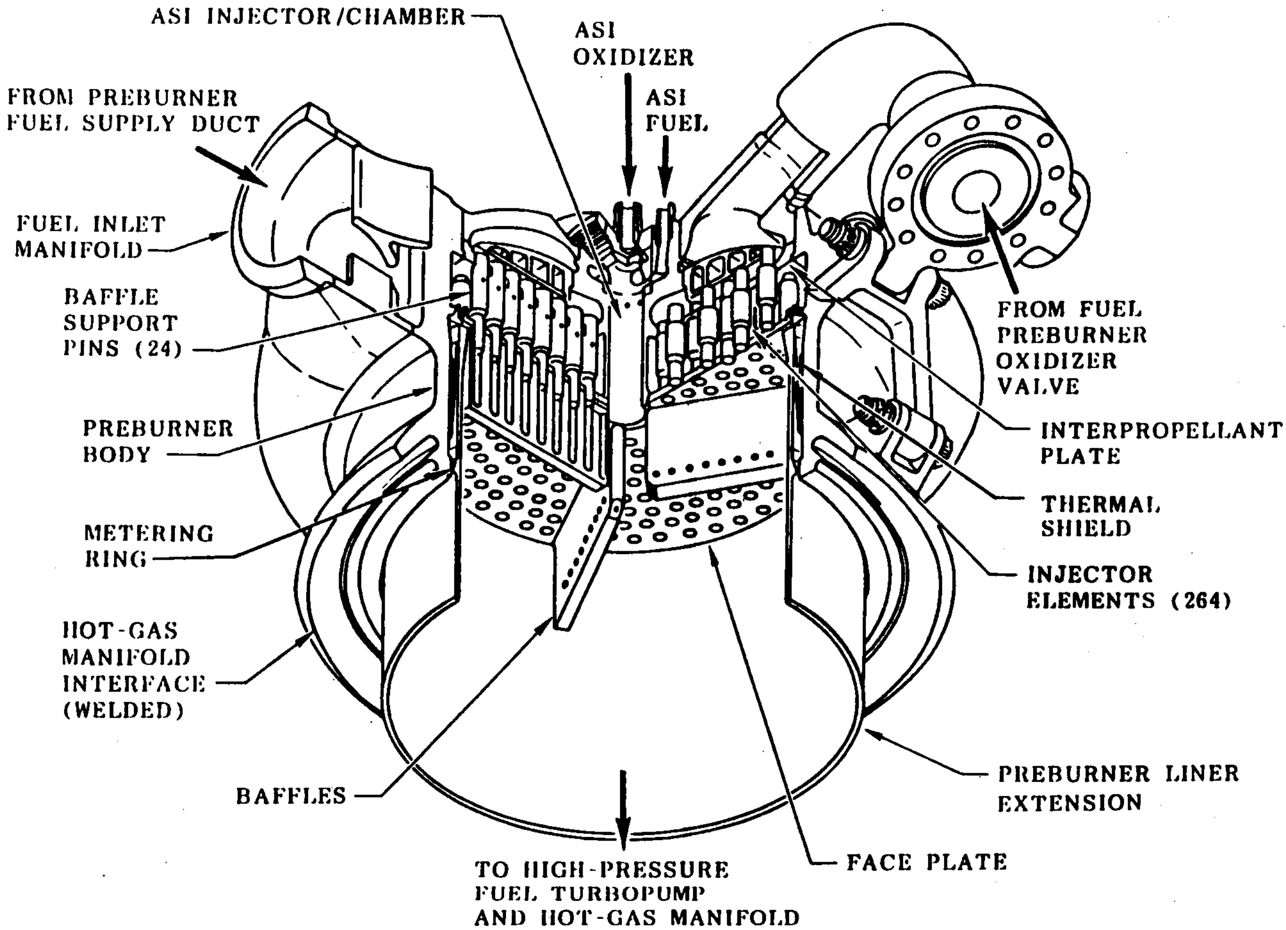


Fig. 3-1 SSME Fuel Side Preburner Configuration

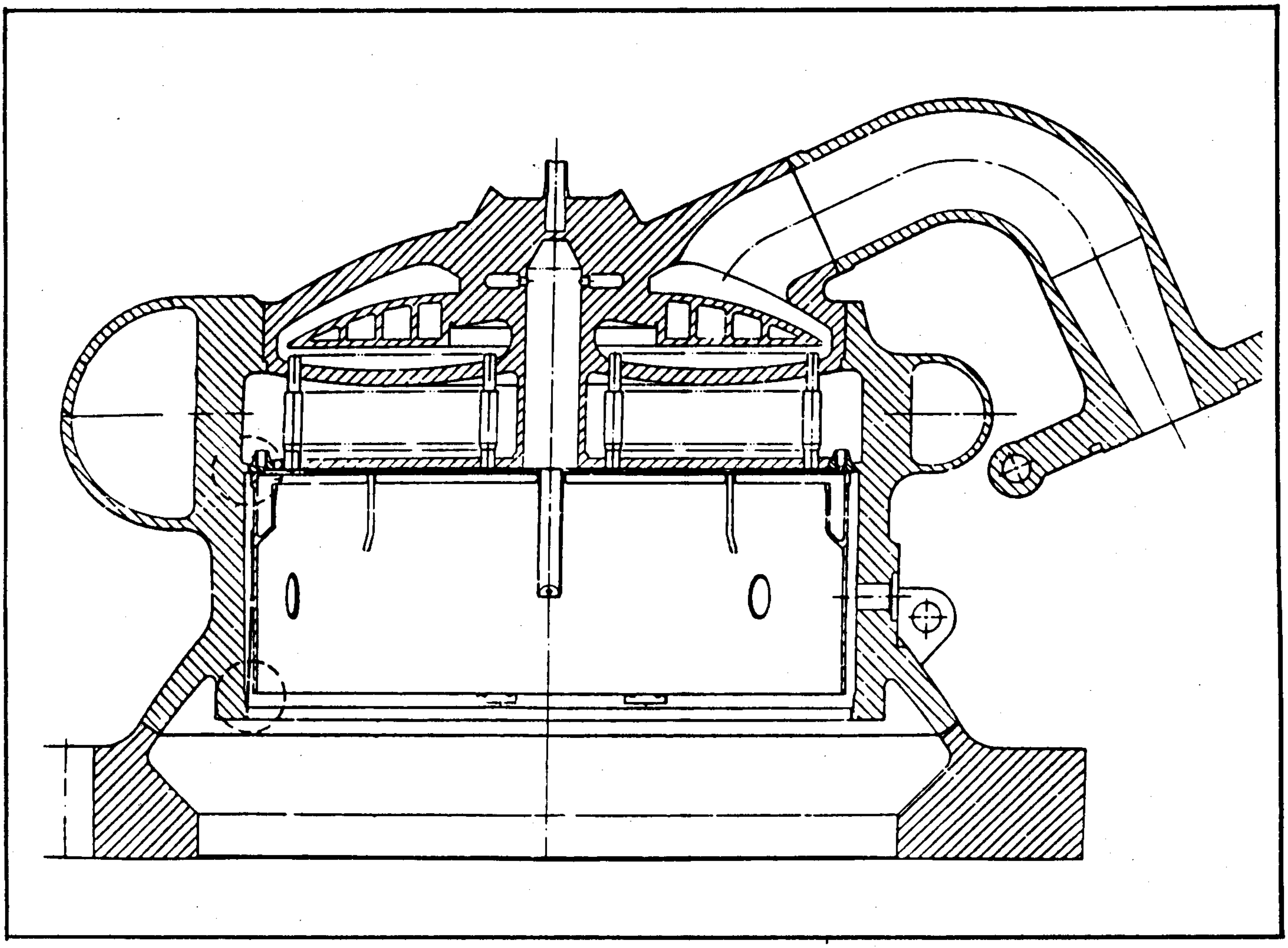


Fig. 3-2 Schematic for Cross-Section of Flow Passages of LOX Through Manifold and Interpropellant and Plate Above Fuel Side Preburner

ORIGINAL PAGE IS
OF POOR QUALITY

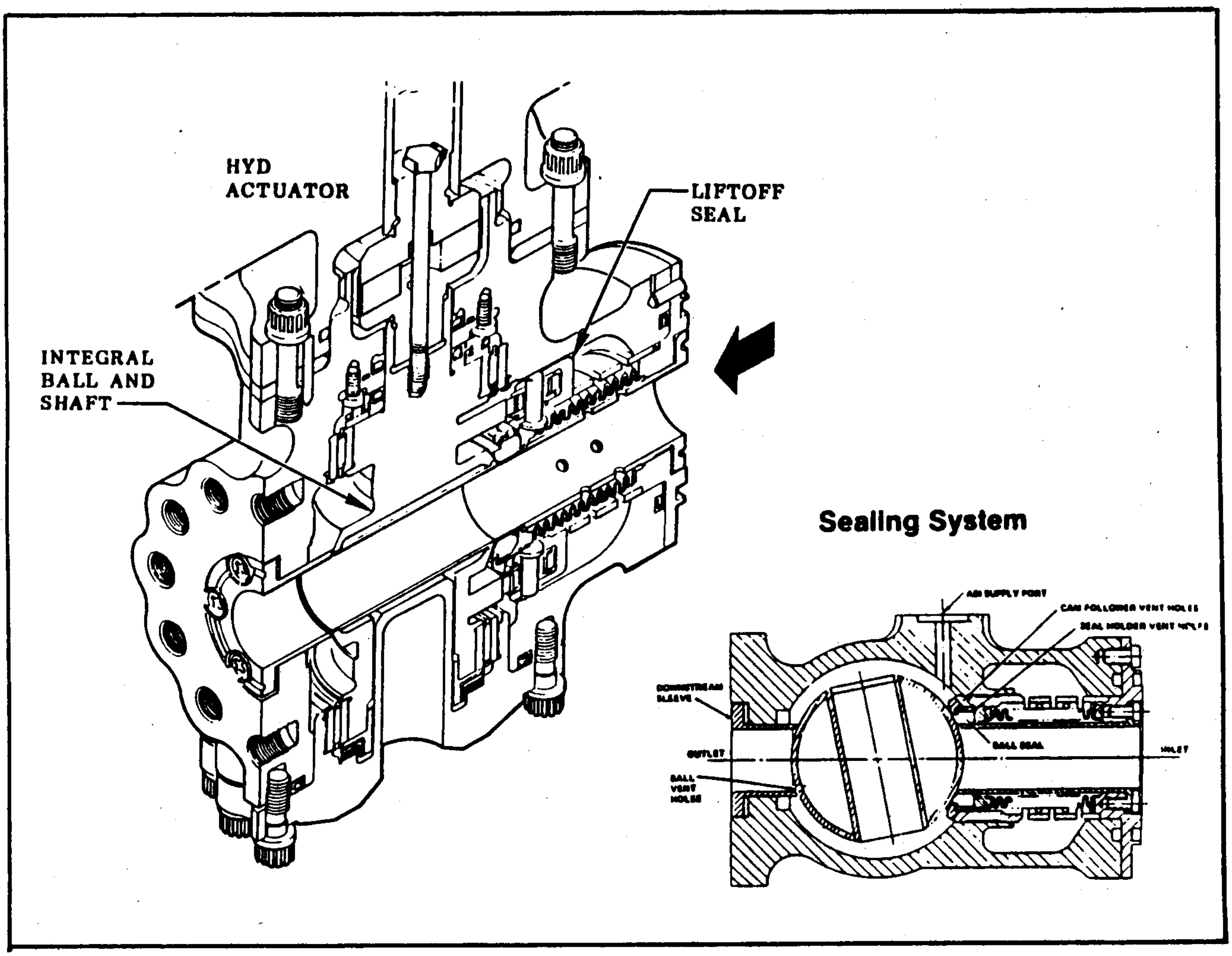


Fig. 3-3 SSME Fuel Preburner Oxidizer Valve Assembly and Sealing System

ORIGINAL PAGE IS
OF POOR QUALITY

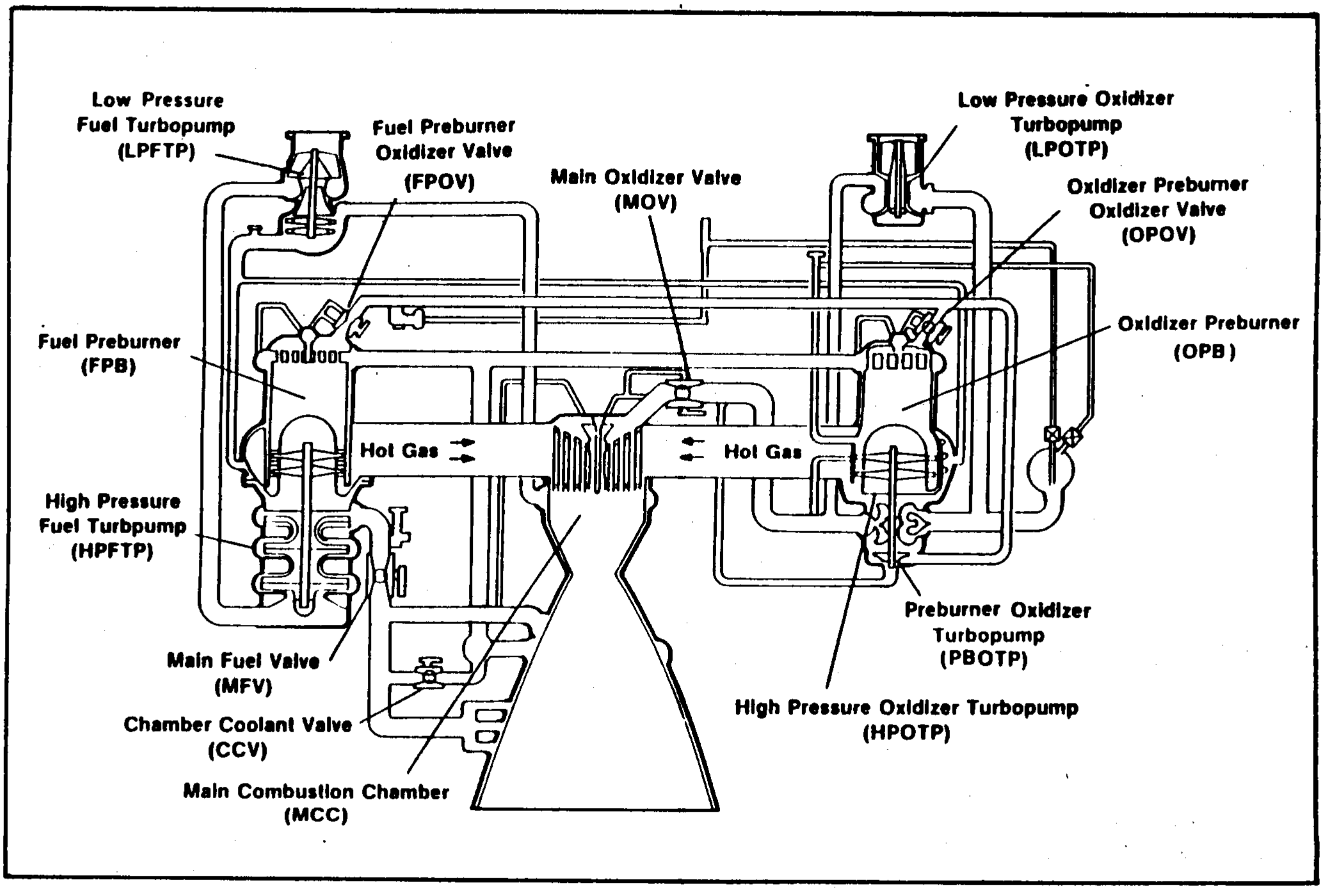


Fig. 3-4 SSME Schematic Showing Entire Gas FLow Path and Positions of Fuel and Oxidizer Control Valves

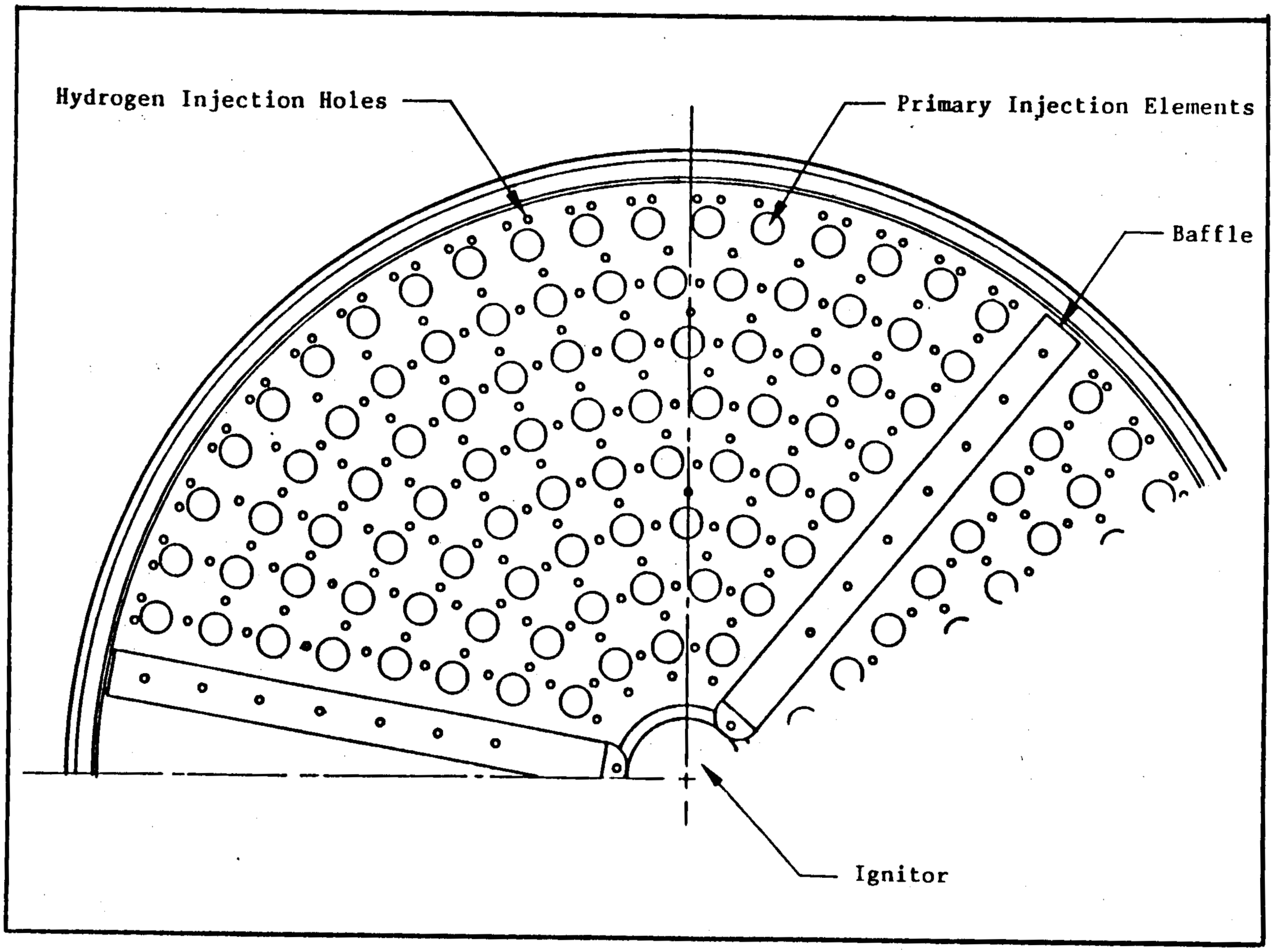


Fig. 3-5 Schematic of SSME Fuel Preburner Injection Faceplate Layout

Downstream boundary conditions were established to couple the manifold flow to the preburner flow so that the simultaneous operation of these two components could be simulated. Detailed modeling of the phase state of the oxygen is necessary so that the unsteady flow rates through the injectors can be simulated. This report described the results of this study.

3.2 TWO-PHASE OXYGEN FLOW

The initial transient flow of LOX into the oxidizer manifold involves the flow of a cryogenic liquid into motor hardware at ambient temperature, hence the initial oxidizer flowing into the fueled preburner is GOX. To mathematically model this two-phase flow process, the individual phase conservation equations written in the conventional manner for each point in space must be volume and time averaged. Whitaker (Ref. 21) and Slattery (Ref. 22) have presented a rigorous derivation of such equations for the simultaneous description of three phases. The equation set consists of conservation equations for each phase and surface balance equations to describe interfacial transfer. Ishii (Ref. 23) and others have presented suitably averaged equations for two phases as a "two-fluid model." If the flow is dilute with respect to one of the phases, a simplified "diffusion model" is obtained. Particulate flow in solid rocket motor plumes is modeled with a diffusion model. Harlow and Amsden (Ref. 24) have corrected the pressure terms in Ishii's derivation to arrive at the equation set given below. These equations implicitly include the interfacial area separating the two phases. Current modeling capabilities are not sufficient to predict such topological information. For the present, empirically determined flow regimes must be postulated to describe flow topology. The governing equations are stated as follows:

$$\{a_i\ d_i\}_t + \{a_i\ d_i\ \underline{V}_i\}_R = G_i$$

$$\{a_i\ d_i\ \underline{V}_i\}_t + \{a_i\ d_i\ \underline{V}_i\ V_i\}_R + a_i\ \{P\}_R = M_i - F_i$$

$$\{a_i\ d_i\ e_i\}_t + \{a_i\ d_i\ e_i\}_R + P(\{a_i\}_t + \{a_i\ \underline{V}_i\}_R) = L_i$$

where i = vapor, liquid; a is the volume fraction of the phase, d is density; $\underline{V}$ is velocity vector; e is internal energy; P is pressure. Braces with subscripts indicate appropriate partial differentiation with respect to the subscript, t for time or R for the position vector. G and L are interfacial exchange of mass and energy, respectively. Momentum exchange is separated into a part due to mass exchange, M, and a part due to friction and other forces, F. The interfacial exchange terms are functions of the principal variables and the interfacial area. The exact specification of the exchange functions is not critical to the solution method used to solve the equations, hence many representations of transport may be modeled. The only limitation is that these exchange rates vary slowly with respect to the inviscid terms on the left-hand side of the equations. The left-hand side of the equations control the complexity required of the numerical solution. Such methodology has been recently reviewed by Stewart and Wendroff (Ref. 25).

A solution of ten equations is required for a three-dimensional problem; one for each phase for energy and the three momentum equations, one for the volume fraction of one phase (the volume fractions must sum to unity), and one for pressure (implying a thermal equation of state for the mixture). Three time scales associated with specific classes of physical behavior must be considered in obtaining numerical solutions; these scales characterize interphase exchanges, sonic propagation, and fluid convection. Filling of the LOX manifold may be assumed to have interphase exchanges and pressure propagations which are in equilibrium with convective phenomena controlling the flow. Stewart and Wendroff's review identified two codes as being very useful for this type of problem: the TRAC code of Liles and Reed (Ref. 26) and an approximate implicit code developed by Spalding which is used in the PHOENICS code (Ref. 27). The method of Liles and Reed will be further evaluated for application to the LOX manifold filling analysis because it has been widely used in the nuclear industry. The TRAC code uses finite difference discretizations on staggered grids. Algebraic manipulations of the conservation equations and linearizations of the difference equations result in a pressure equation which contains all of the spatial coupling in the code. Once updated pressure values are obtained the remaining variables are simply calculated.

Current versions of this program are available from Los Alamos National Laboratory and are being requested through NASA. Further details of this code will be discussed in Section 4 of this report, but first other solution methods will be presented.

3.3 SOLUTION METHODS

If there were no velocity lag between the phases, it was shown by Stewart and Wendroff that the mixture sound speed is less than that of either of the pure phases. This implies that a computational fluid dynamic method which is efficient for either compressible or incompressible flow could be expected to solve the two-phase flow equations. Since SECA's ALFA code utilizes characteristic directions based on sonic velocity of the fluid to determine an integration algorithm, and since ALFA is currently being developed to treat both compressible and incompressible flow, the code should also solve the two-phase flow equations, even when there is velocity lag. The TRAC code has been requested by NASA but is not yet available to SECA; therefore, the ALFA code was evaluated for solving the LOX manifold transient problem.

ALFA is an explicit, unsteady Navier-Stokes solver developed from a finite-element point of departure, so that geometry and grid generation can be conveniently separated from the solution advancement portion of the program. Boundary condition and flow initialization definitions are also separated into distinct steps in the calculations. ALFA is structured such that initial flow property specification, integration/iteration analysis and outputting are treated in a modular fashion. The integration algorithm is biased to propagate pressure signals only in the characteristic directions. This is the program of choice to describe the combusting flow in the FPB. The integrator module is the only one which would require modification to describe the two-phase flow of oxygen.

The two-fluid model recommended in the previous section is seen to contain a single pressure at a given point which describes both fluids. This is a physically realistic assumption, but we shall show that is does impact the solution methodology. Other solution restrictions which are present even in single phase flows may be revealed by considering the following one-dimensional, simplified system of conservation laws.

$$\{U\}_t + \{F\}_x = 0$$

where U and F are n-component column vectors. The system can be quasi-linearized by considering F to be a function of U.

$$\{U\}_t + A\{U\}_x = 0$$

where A{U} is the Jacobian matrix, $\{F\}_U$. The system of equations is hyperbolic at this point, if there exists a similarity transformation such that

$$Q^{-1} A Q = M$$

where M is a diagonal matrix. The elements of M are the eigenvalues of A. If the eigenvalues are real, a computational algorithm can be devised which propagates pressure waves such that physically unrealistic signals are not generated.

The eigenvalues calculation just presented is utilized in SECA's ALFA code to provide a stable, efficient unsteady solution. If the two-fluid system of equations with one pressure possesses real eigenvalues, the ALFA code methodology would be expected to solve the model equations. Wendroff (Ref. 25) performed the eigenvalue determination for the one-dimensional, unsteady two-fluid conservation equations and found four real eigenvalues and two complex eigenvalues. This demonstrates that the current ALFA methodology is not appropriate for this application without modification.

Wendroff (Ref. 25) did not perform the eigenvalue determination to devise a computational algorithm, but to determine if the problem was ill-posed. Although the problem is ill-posed, two of the real eigenvalues do tend to give the system pressure wave propagation properties similar to those of gases.

Furthermore, interfacial phenomena involving friction and the assumed behavior of relative velocity and equality of pressure may be used to make the model become well-posed. Thus modifications could be made to utilize the ALFA code on this two-phase flow problem, but, since the TRAC methodology apparently works the problem in a satisfactory manner, further code development does not appear worthwile. It should be mentioned that artificial viscosity or dissipation which may be present in the numerical solution method may be sufficient to make the problem well-posed and avoid the difficulty just described. Regardless of whether or not suitable modifications can be made, the ALFA code will require further development for use in describing the LOX flashing in the fuel preburner manifold; therefore, TRAC should be used for near term studies.

In addition to a solution methodology for the left-hand side of the two-fluid equations, the interface transfer terms must be specified, and heat transfer within the solid walls of the LOX manifold calculated. All of these factors are already included in the TRAC code presented in the next section.

3.4 THE COBRA/TRAC CODE

The COBRA/TRAC code (Ref. 28) was developed by the Pacific Northwest Laboratory of the Battelle Memorial Institute to predict the thermal-hydraulic response of nuclear reactor coolant systems to break loss-of-coolant accidents and other anticipated transients. The code is used to solve the compressible three-dimensional, two-fluid, three-field equations for two-phase flow in a reactor vessel and the heat transfer within solid walls using COBRA-TF. The three fields are the vapor field, the continuous liquid field, and the liquid drop field. The primary reactor system is modeled with the TRAC portion of the analysis. The TRAC technology was developed at the Los Alamos National Laboratory (Ref. 26). The effort which was devoted to the development of this computational tool both at Battelle and Los Alamos was substantial; therefore, the conversion of this model from one describing water/steam to one describing LOX/GOX flow is well justified.

To utilize the COBRA/TRAC code, physical models of the mass exchange across the phase boundaries must be specified. The flow map to define interface topology included in the code is one developed by Dukler for vertical flow in pipes. A similar treatment for horizontal flows is not available because the effect of gravity on the liquid phase creates severe complications. These correlations will be assumed to be valid for the LOX manifold analysis. Their validity will be ascertained after some experience is developed in using the code. This is not a gross assumption because the cold GOX temperatures would cause the GOX and LOX densities to differ much less than water/steam densities. Nevertheless, all of the flow regimes in the code are expected to be required to treat the GOX/LOX flow. These regimes are shown in Figs. 3-6 and 3-7 taken from Ref. 28 for adiabatic and hot walls, respectively. The correlation equations contained in COBRA/TRAC will be assumed to be valid for oxygen when oxygen properties are used in place of those for water. The computed void fraction is used to shift from one flow regime to another.

Heat transfer within solid walls is treated with a nodal network model and convective and/or boiling heat transfer correlations; these will be utilized. The gap conductance and radiant exchange models will not be required. The quench front model will be useful in describing the LOX flashing.

The use of the COBRA/TRAC code is expected to greatly enhance the capability to simulate the transient behavior of the FPB startup and shutdown when coupled to the ALFA code for describing the propellant flow within the FPB combustor. The major remaining requirement is the correct specification of the physical properties of the propellant for these flow regimes.

3.5 PROPELLANT PROPERTIES

The physical properties for oxygen will be taken from the following sources. Liquid thermodynamic and transport properties from Ref. 29, gaseous thermodynamic properties from Ref. 30, and transport properties from Ref. 13. Liquid hydrogen properties are from Ref. 32. Transport properties are from

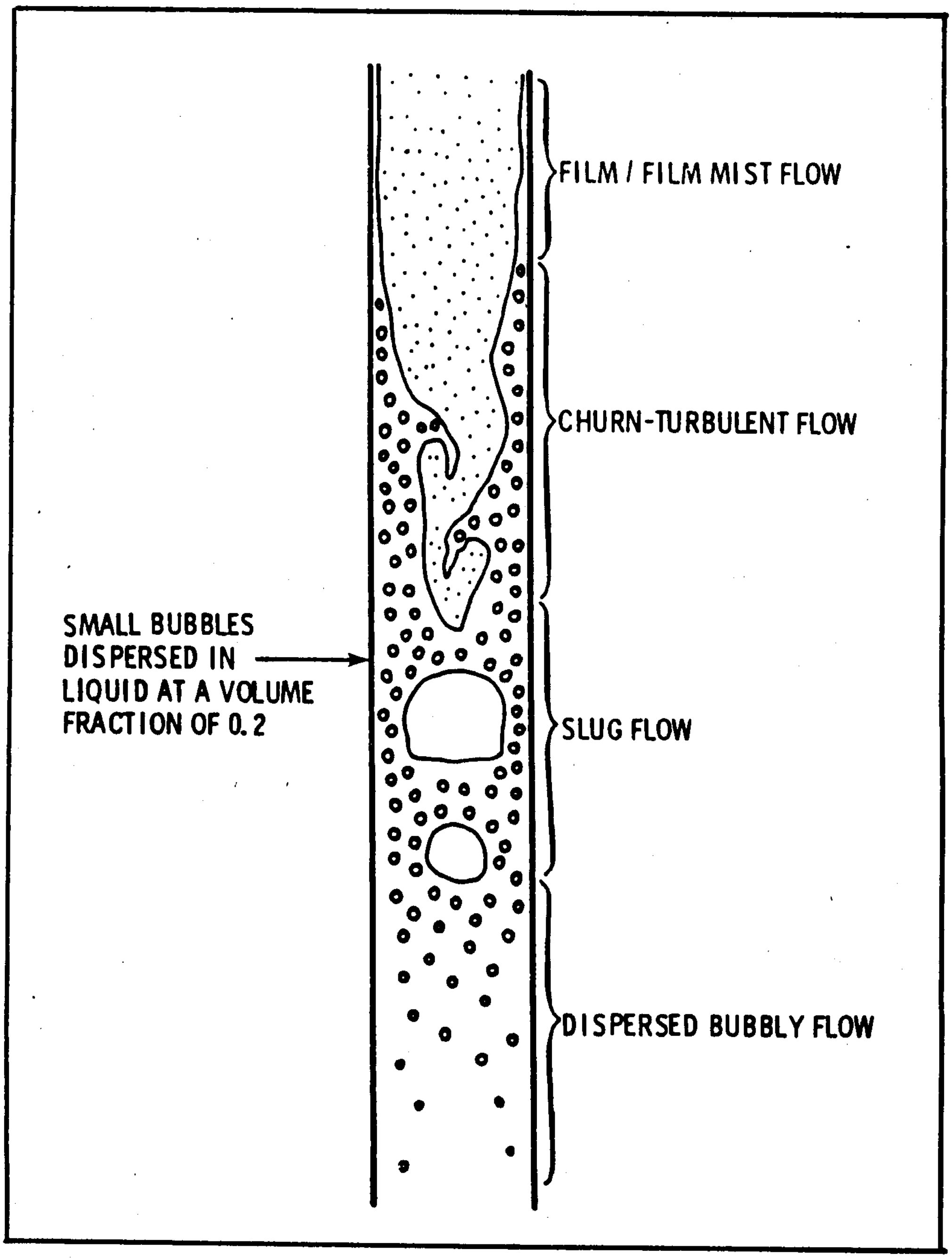


Fig. 3-6 Schematic Showing Normal Regimes in a Two-Phase Flow

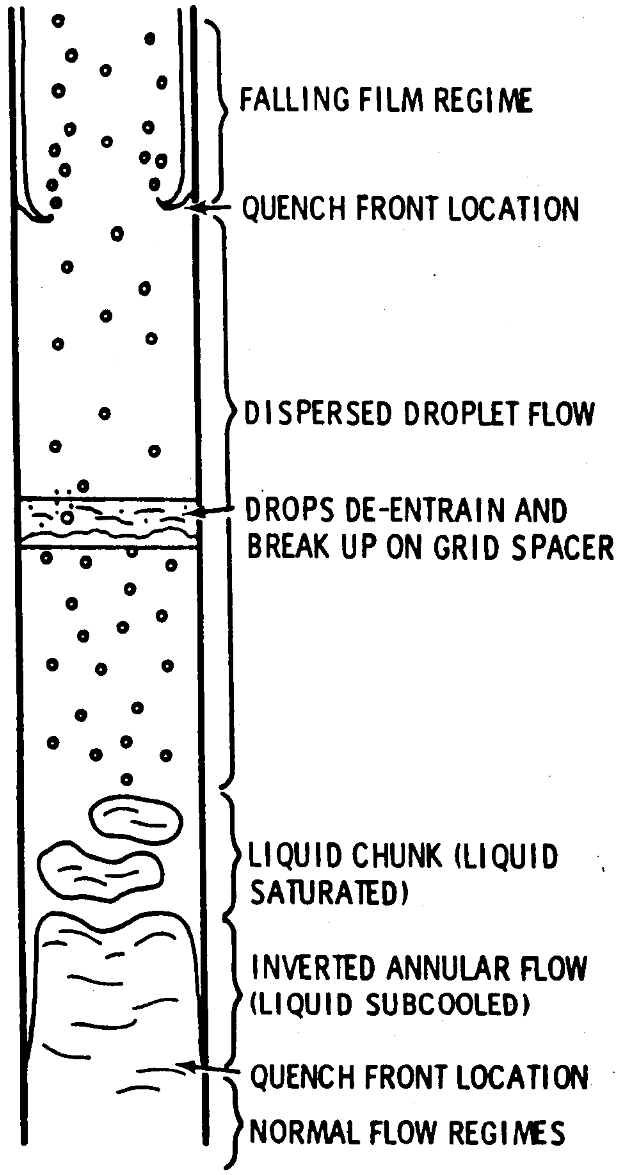


Fig. 3-7 Schematic Showing Two-Phase Flow Regimes for Case of Hot Wall FLow

the same source as for oxygen. The only discrepancy noted to date is that the volume expansivity for oxygen (Ref. 29) appears to have the wrong sign. Most substances expand when heated; LOX is not an exception. The magnitude of the values given in the NBS technical note agree with those given by Rowlinson (Ref. 33), but the signs do not. The NBS TN uses the signs given in the tables to calculate the speed of sound tables; hence the sound speeds are too high. These data are not used directly in the COBRA/TRAC code, but the apparent error should be anticipated.

The COBRA/TRAC code uses tables of property data which must be created from the cited references for LOX/GOX. This code also uses temperatures as flags for certain calculations; these flags must be changes to reflect oxygen properties.

4. SUMMARY AND CONCLUSIONS

Cooperative efforts between the Lockheed-Huntsville Computational Mechanics Group and the NASA-MSFC Computational Fluid Dynamics staff (ED32) has resulted in improved capabilities for numerically simulating incompressible flows generic to the Space Shuttle Main Engine. A well established and documented CFD code has been obtained, modified, and applied to laminar and turbulent flows of the type occurring in the SSME Hot Gas Manifold. The INS3D code has been installed on the NASA-MSFC CRAY-XMP computer system and is currently being used by NASA engineers.

Studies to perform a transient analysis of the FPB have been conducted. The COBRA/TRAC code is recommended for simulating the transient flow of oxygen into the LOX manifold. Property data for modifying the code to represent LOX/GOX flow has been collected. The ALFA code has been developed and recommended for representing the transient combustion in the preburner. These two codes will couple through transient boundary conditions to simulate the startup and/or shutdown of the fuel preburner. SECA is currently conducting a study, NAS8-37461, to implement this modeling effort.

5. REFERENCES

1. Chorin, A.J., "A Numerical Method for Solving Incompressible Viscous Flow Problems," J. Comp. Phys., Vol. 2, 1967, pp. 12-26.

2. Steger, J.L., and Kutler, P., "Implicit Finite-Difference Procedures for the Computation of Vortex Wakes," AIAA J., Vol. 15, No. 4, April 1977, pp. 581-590.

3. Kwak, D., J.L.C. Chang, S.P. Shanks, and S. Chakravarthy, "An Incompressible Navier-Stokes FLow Solver in Three-Dimensional Curvilinear Coordinate System Using Primite Variables," AIAA Paper 80-0253, Reno, January 1984.

4. Vanka, S.P., B.C-J. Chen, and W.T. Sha, "A Semi-Implicit Calculation Procedure for FLows Described in Boundary-Fitted Coordinates Systems," Num. Heat Trans., Vol. 3, 1980, pp. 1-19.

5. Shyy, W., S.S. Tong, and S.M. Correa, "Numerical Recirculating Flow Calculation Using a Body-Fitted Coordinate System," Num. Heat Trans., Vol. 8, 1985, pp. 99-113.

6. Maliska, C.R., and G.D. Raithby, "A Method for Computing Three-Dimensional Flows Using Non-Orthogonal Boundary-Fitted Coordinates," Int. J. for Num. Meth. Fluids, Vol. 4, 1984, pp. 519-537.

7. Chang, J.L.C., and D. Kwak, "On the Method of Pseudo Compressibility for Numerically Solving Incompressible Flows," AIAA Paper 84-0252, 1984.

8. Chang, J.L.C., D. Kwak, and S.C. Dao, "A Three-Dimensional Incompressible Flow Simulation Method and its Application to the Space Shuttle Main Engine, Part 1, Laminar Flow," AIAA Paper 85-0175, 1985.

9. Rogers, S.E., D. Kwak, and J.L.C. Chang, "INS3D - An Incompressible Navier-Stokes Code in Generalized Three-Dimensional Coordinates," NASA TM-100012, November 1987.

10. Beam, R.M., and R.F. Warming, "An Implicit Finite-Difference Algorithm for Hyperbolic Systems in Conservation-Law Form," J. Comp. Phys., Vol. 22, 1976, pp. 87-110.

11. Briley, W.R., and H. McDonald, "Solution of the Three-Dimensional Compressible Navier-Stokes Equations by an Implicit Technique. Proceedings of the Fourth International Conference on Numerical Methods in Fluid Dynamics, Lecture Notes in Physics," Vol. 35, Springer Verlag, New York, 1975, pp. 105-110.

12. Rogers, S.E., D. Kwak, and U.K. Kaul, "On the Accuracy of the Pseudo Compressibility Method in Solving the Incompressible Navier-Stokes Equations," Appl. Math. Mod., Vol. 11, 1987, pp. 35-44.

13. Zilliac, G., "A Computational/Experimental Study of the Flow Around a Body of Revolution at Angle of Attack," NASA TM-88329, 1986.

14. Champney, J.M., T.J. Coakley, and U.K. Kaul, "Incompressible Viscous Flow Simulations of the NFAC Wind Tunnel," ATM-FR-25-012, 1986.

15. Kaul, U.K., D. Kwak, and C. Wagner, "A Computational Study of Saddle Point Separation and Horseshoe Vortex System," AIAA Paper 85-0182, 1985.

16. Rogers, S.E., D. Kwak, and U.K. Kaul, "A Numerical Study of Three-Dimensional Incompressible Flow Around Multiple Posts," AIAA Paper 86-0353, 1986.

17. Chang, L.C., D. Kwak, S.C. Dao, and R. Rosen, "A Three-Dimensional Incompressible Flow Simulation Method and its Application to the Space Shuttle Main Engine, Part II - Turbulent Flow," AIAA Paper 85-1670, 1985.

18. Chang, J.L.C., "A Study of Turbulent Flow in an Axisymmetric Diffusion Duct," Fifth SSME CFD Working Group Meeting, Huntsville, Ala., 21 April 1987.

19. Anon., "SSME Orientation (Part A-Engine)," Contract NAS8-27980, Rocketdyne Division, Rockwell International, Canoga Park, Calif., October 1982.

20. David, D., "Space Shuttle Engine Study, Vol. II, Interim Report," Contract 8-36176, Aerojet Technical Systems Co., Sacramento, Calif., April 1985.

21. Whitaker, S., Adv. in Heat Transfer, Vol. 13, 1981, pp. 119-203.

22. Slattery, J.C., Momentum, Energy, and Mass Transfer in Continua, McGraw-Hill, New York 1972.

23. Ishii, M., Thermo-Fluid Dynamic Theory of Two-Phase Flow, Eyrolles, 1975.

24. Harlow, F.H., and A.A. Amsden, J. of Comp. Phys., Vol. 17, 1975, pp. 19-52.

25. Stewart, H.B., and B. Wendroff, J. of Comp. Phys., Vol. 56, 1984, pp. 363-409.

26. Liles, D.R., and W.H. Reed, J. of Comp. Phys., Vol. 26, 1978, pp. 390-407.

27. Spalding, D.B., "The Numerical Computation of Multi-Phase Flows," to be published, 1984.

28. Thurgood, M.H., et al., NUREG/CR-3046, PNL-4385, Vol. 1, Battelle Memorial Institute, Richland, Wash., 1983.

29. Anon., NBS Technical Note 384.

30. Stull, D.R., and H. Prophet, NSRDS, NBS 37, 1971.

31. Svehla, R.A., NASA TR R-132.

32. McCarty, R.D., and L.A. Weber, NBS TN 617, April 1972.

33. Rowlinson, J.S., Liquids and Liquid Mixtures, 2nd ed., Butterworth, London, 1969.