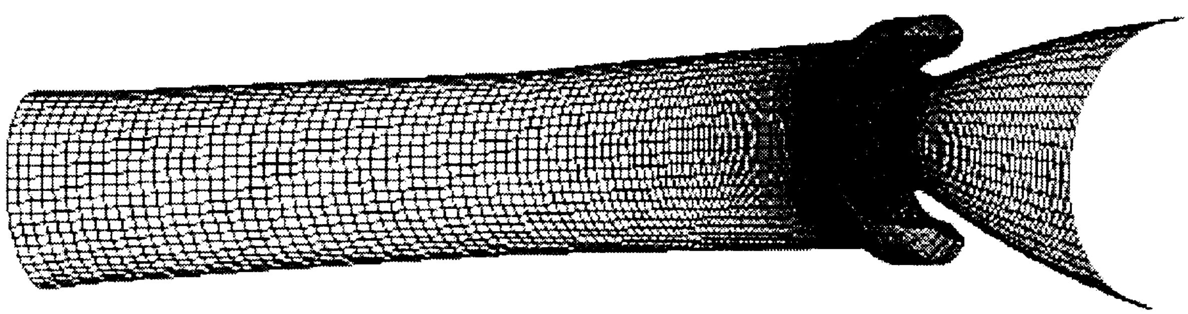
[redacted] N95-28733

1995122312

MARSHALL SPACE FLIGHT CENTER SURFACE MODELING AND GRID GENERATION APPLICATIONS

Robert W. Williams, Theodore G. Benjamin, and Joni W. Cornelison
NASA George C. Marshall Space Flight Center
Computational Fluid Dynamics Branch
Marshall Space Flight Center, Alabama

Gimballed Nozzles in Solid Rocket Motors

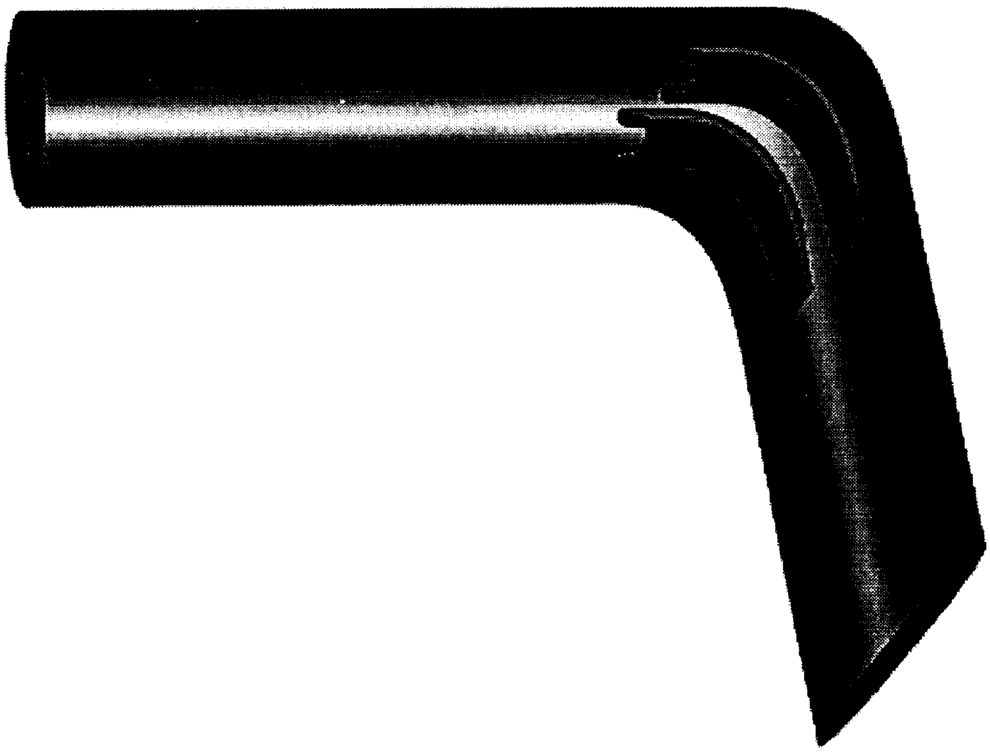


The Solid Rocket Motors (SRM) used by NASA to propel the Space Shuttle employ gimballing nozzles as a means for vehicular guidance during launch and ascent. Gimballing a nozzle renders the pressure field of the exhaust gases nonaxisymmetric. This has two effects: (1) it exerts a torque and side load on the nozzle; (2) the exhaust gases flow circumferentially in the aft-dome region, thermally loading the flexible boot, case-to-nozzle joint, and casing insulation. The use of CFD models to simulate such flows is imperative in order to assess SRM design.

The grids for these problems were constructed by obtaining information from drawings and tabulated coordinates. 2D axisymmetric grids were designed and generated using the EZ-Surf and GEN2D surface and grid generation codes. These 2D grids were solved using codes such as FDNS, GASP, and MINT. These axisymmetric grids were rotated around the center-line to form 3D non-gimballed grids. These were then gimballed around the pivot point and the gaps or overlaps resurfaced to obtain the final domains, which contained approximately 366,000 grid points. The 2D solutions were then rotated and manipulated as appropriate for geometry and used as initial guesses in the final solution. The analyses were used in answering questions about flight criteria.

For more information, contact Dr. Edward J. Reske.

ALS LOX Sustainer Feed Line - Eighty-Degree Bend with Turning Vanes



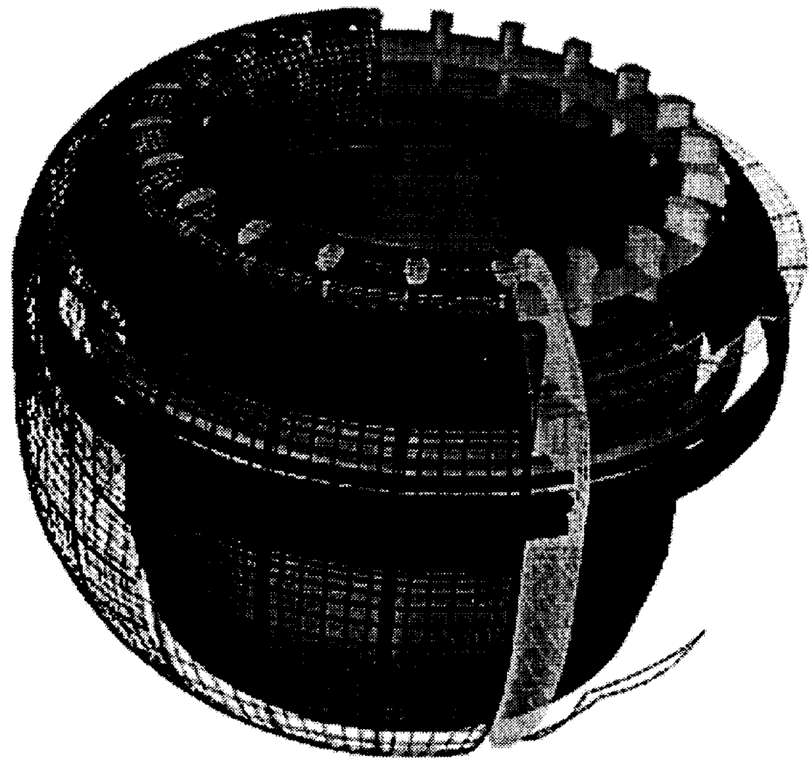
The purpose of this CFD study was to determine the vane loading and the amount of secondary flow downstream of an eighty-degree vaned pipe bend. The pipe in question was a liquid oxygen rocket engine feed line in the Advanced Launch System Propulsion module.

The geometry was generated on a Intergraph CAD system and transferred electronically by General Dynamics in NIGES format. The NIGES file was read into CAGI. Options were selected in CAGI to automatically discard many intermediate construction surfaces. The necessary surfaces were then extracted and written out in IGES format. The strong curvature of the guide vanes made it necessary to use a NURBS definition to properly model and maintain the blade shape. The IGES file was then read into GENIE++ where points were distributed on the surface and the volume constructed. Two grids were generated. Each grid had five zones totaling 115,000 and 360,000 grid points, respectively.

Since the version of INS3D that was used to solve the flow field did not support overset grids, an H-type grid was used to describe the geometry. This increased the complexity of adding the vanes. More recent releases of INS3D-UP have overset capabilities allowing greater grid generation flexibility.

For more information, contact Robert W. Williams.

Alternate Turbopump Development Turnaround Duct

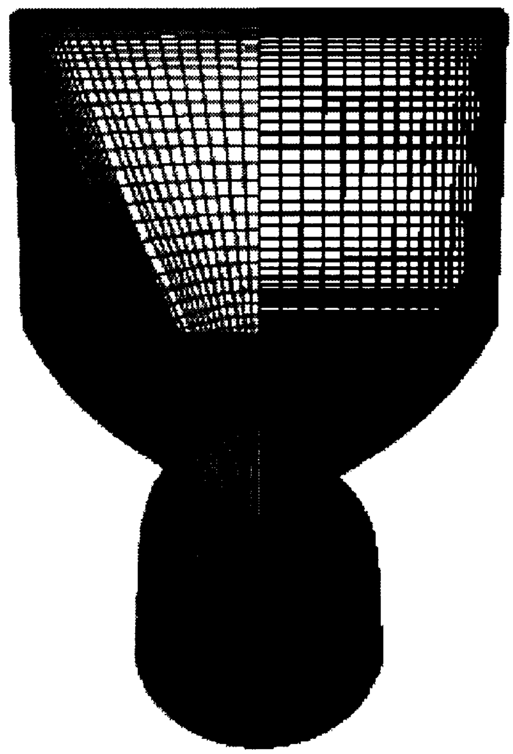


During testing of the ATD high pressure oxidizer turbopump at Stennis Space Flight Center cracks were found on the heat exchanger inner guide vane. A team was formed to assess the potential causes for the cracking and to propose changes to eliminate the problem. There was significant CFD analysis performed in support of this team. Over 50 2D or axisymmetric cases and six 3D cases were run. The actual turn-around duct geometry is 3D, but the 2D cases were run as axisymmetric to allow for rapid analysis of different configurations.

The geometries for the 2D grids were obtained partially from CAD systems and partially from faxed drawings. The grids were generated in GENIE++ and then smoothed using GEN2D. The 2D grids model a cross-section of the turn-around duct which contains a splitter vane, two heat exchanger vanes and heat exchanger coils. The 3D grid modeled the 22 struts on which the splitter vane is mounted. The data for the struts was obtained in the form of tabulated coordinates at midspan. Due to the nature of the 2D grids and to the time in which the grid was generated, the four intersecting boundaries along the strut were hand calculated. Later consideration indicates it may have been possible to use the surface-surface intersection option in GENIE++. A single passage between two struts was first generated and then rotated to obtain the full 3D grid. The 3D grid contains approximately one-half million grid points. REFLEQS was used to obtain the flow field for the 2D and 3D cases.

For more information, contact Joni W. Cornelison.

Modular Thruster for a Tri-Propellant Engine

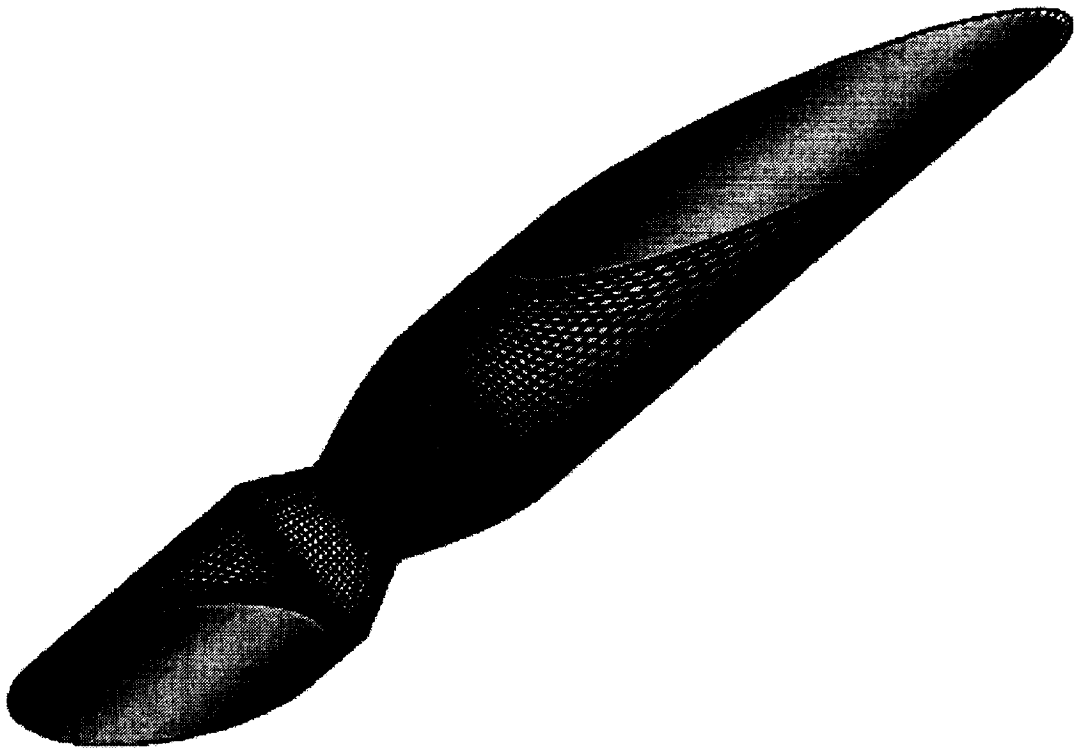


This geometry was received from Rocketdyne via an IGES file. The inlet region to the throat was a single NURBS surface; the exit from the throat was represented by two NURBS surfaces. All three surfaces were joined in NGP so that they could be split uniformly. Because there was no line of constant parametric value at the desired coordinates for splitting, the surface was split short of the desired coordinates and joined with a NURBS curve which lay on the symmetry plane. This was done in GENIE++ and these were combined on a NURBS surface. From that point in GENIE++, the flow field was discretized into an 87,451-point grid.

The flowfield was initialized using isentropic relationships and was solved in FDNS. This solution was checked for reasonableness and found acceptable. The purpose for this exercise was to develop a template for other similar geometries that are now beginning to be analyzed in tri-propellant engine design.

For more information, contact Theodore G. Benjamin.

Simplex Turbopump Nozzle



Turbine aerodynamic design support was provided for the Simplex turbopump, an in-house project at Marshall Space Flight Center. The turbine was designed as a supersonic/impulse turbine. Several 3D CFD calculations of the nozzle were performed using the FDNS code. Results from the analyses were provided to support design changes and rotordynamics, dynamics, stress and thermal analyses.

The single-zone nozzle grid was generated entirely in GENIE++ and contains approximately 190,000 grid points. It was necessary to model the nozzle with a full 3D grid since the inlet plane and the exit plane are rotated 90 degrees apart. The cross-section of the nozzle was generated as an O-grid. Later consideration indicated an H-grid may have eliminated problems in smoothing the exit surface and an H-grid would have been necessary in generating a combined nozzle and turbine blade grid. The elliptic inlet and exit surfaces were obtained by generating elliptic boundaries and rotating to the proper angle. For this grid this approach was found to work better than using the surface-surface intersection option.

For more information, contact Joni W. Cornelison.

Inducer Technology Model Volute Collector

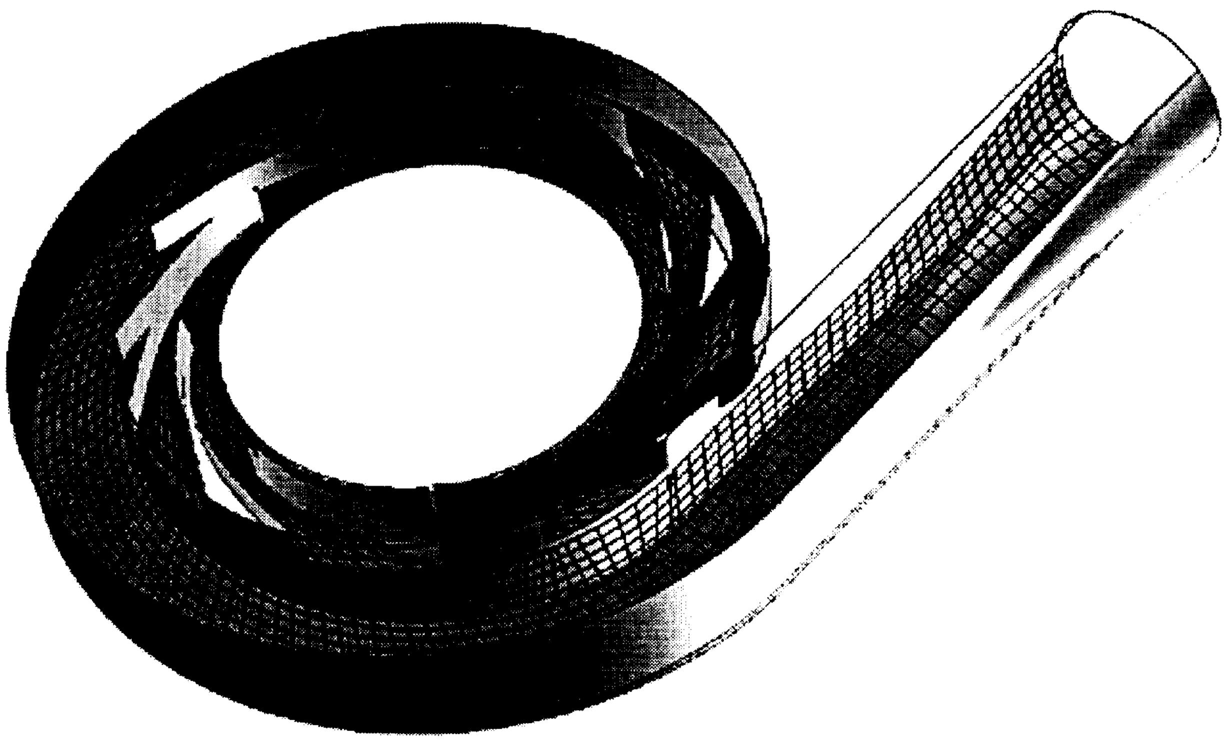
CFD analysis was performed on the volute in the ITM (Inducer Technology Model) to assess radial loads, total pressure losses, flow separations, and any unusual flow phenomena that might occur within the volute system.

The volute and exit pipe geometries were obtained from drawings. A 2D cross-section of the volute was quickly generated using GENIE++ and then input to GEN2D for smoothing. The 2D axisymmetric grid was read back into GENIE++ and rotated about the centerline to form the 3D grid. The grid which models the exit pipe connected to the volute was generated totally in GENIE++. The inlet section of the pipe is rectangular and intersects with a conic section near the exit of the pipe. The intersection of these two surfaces was performed successfully by using NURBS surfaces and the surface-surface intersection option. Various modifications were made to the volute housing and were quickly generated using GENIE++. The two-zone grid contains approximately 45,000 grid points.

FDNS was used to perform the incompressible flow analysis on three cases. Each case showed secondary flows in the volute housing and exit pipe. Based on these results, a honeycomb ring will be inserted at the pipe exit to eliminate the secondary flow.

For more information, contact Joni W. Cornelison.

Simplex Diffuser and Volute

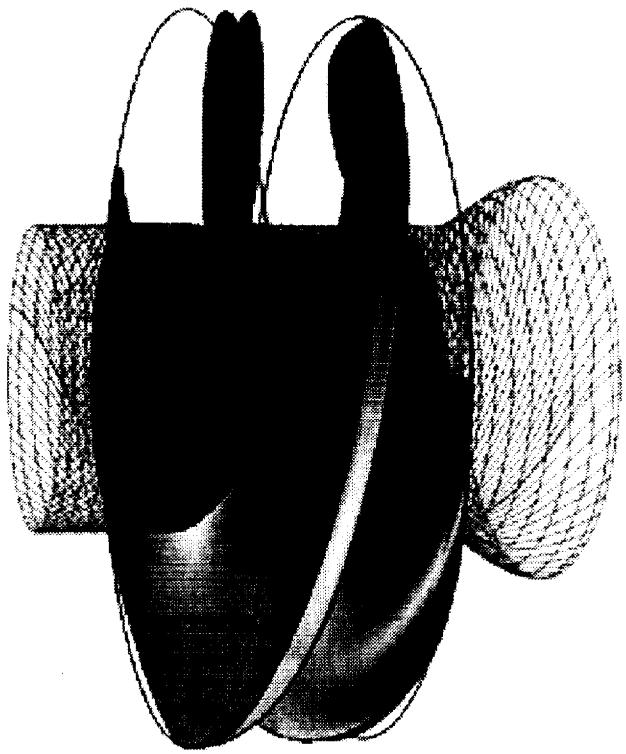


The Simplex Turbopump is an in-house project at Marshall Space Flight Center. The data for this geometry was extracted from drawings. The shape of the vanes is fully two-dimensional and it was modeled by fitting a curve through points which define the contour and stacking this contour axially. The volute is square in cross-section until the exit region, where the square cross section intersects with a conic section. This intersection was performed impressively well in GENIE++, which was utilized for the entire grid-generation process. The final grid contained 207,894 grid points in 10 zones.

This design was subjected to three-dimensional hydrodynamic analysis using the FDNS code. These analyses revealed a flaw in the initial design which caused separation on the pressure side of the vanes; this prevented the design from producing the desired diffusion. Alternative designs were proposed (and rapidly generated in GENIE++) and analyzed. As a result of the CFD analyses, one of these alternative vane proposals was incorporated into the final design before machining had begun.

For more information, contact Theodore G. Benjamin.

Simplex Turbopump - Inducer



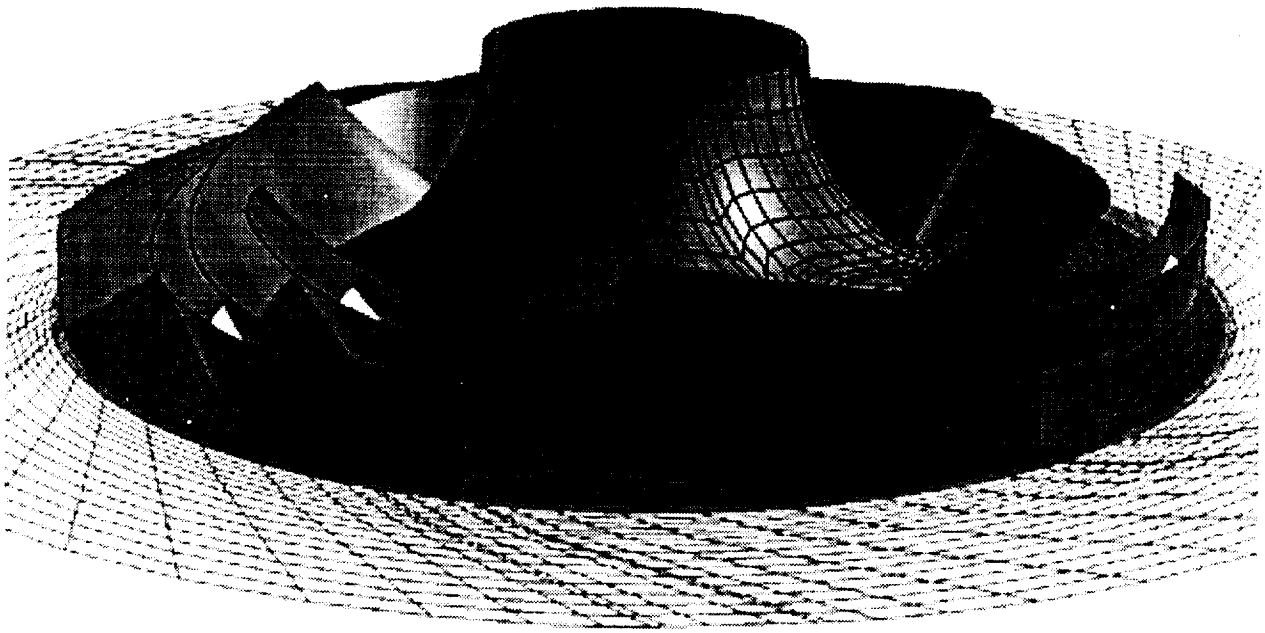
The Simplex Project is an in-house project at MSFC to design and build a turbopump to supply liquid oxygen to a hybrid rocket motor. The pump was required to deliver liquid oxygen at 570 gallons per minute and 1,500 pounds per square inch with a wheel speed of 25,000 revolutions per minute. The objective of this study was to use CFD during the design phase of the pump to predict performance and blade loading.

Blade surfaces were output from an Intergraph CAD system and transfered electronically in IGES file format. The surfaces described in NURBS were read into GENIE++ where points were distributed on the blades. The hub and shroud surfaces were generated by projecting an arbitrary surface constructed at the root and tip respectively on a boundary of revolution. For grid resolution studies, changes were made to appropriate indices in the GENIE++ history file and then the grid generator was rerun using those inputs. All grids consisted of three zones and ranged in size from 25,000 to 680,000 grid points.

Flow solutions were obtained using FDNS in the relative reference frame. Results were post processed using FAST and an in-house code to evaluate and make plots of performance and blade loads.

For more information, contact Robert W. Williams.

Simplex Turbopump - Impeller



The Simplex Impeller was designed to operate in liquid oxygen at 25,000 revolutions per minute and deliver the fluid at 570 gallons per minute and 1,500 pounds per square inch. The purpose of this study was to aid in the design phase and determine performance and blade loading.

The geometry was transferred electronically from an Intergraph CAD system in IGES format. The NURBS surfaces were read into CAGI where streamlines were selected along periodic lines. The streamlines were set-up into an input file and read into TIGER. TIGER interactively generated the grid allowing control over Bezier curves connecting leading and trailing edges at the hub and tip. Several grids were generated in single and multiple zones. Grids ranged in size from 125,000 to 250,000 grid points. Later grids also modeled cavities between the impeller and diffuser. The cavities were added using GENIE++.

The flow field was solved using FDNS. Results were post-processed using FAST and an in-house code to calculate and plot performance and blade loading.

For more information, contact Robert W. Williams.

Advanced Liquid-Hydrogen Turbopump - Inducer / Impeller with Single Splitter

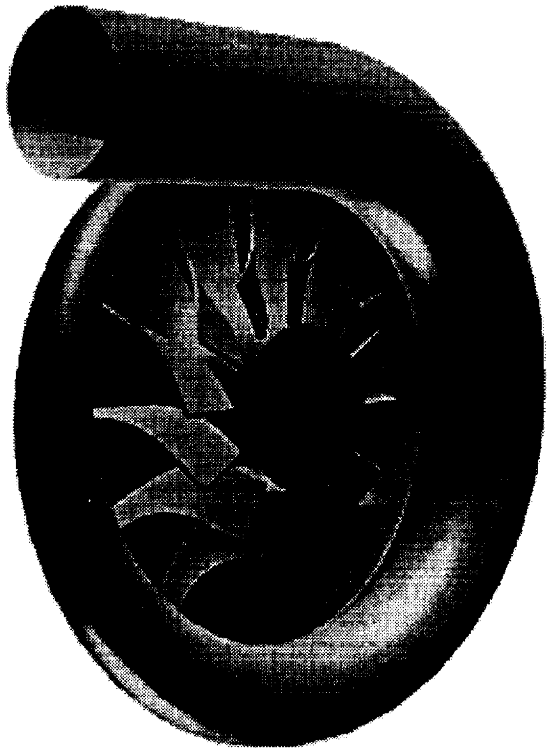
The development of the Advanced Long Life Turbopump is a cooperative agreement among the United States Air Force, NASA, and Pratt and Whitney. The pump was designed to deliver liquid hydrogen at 2,625 gallons per minute and has a rotational speed of 140,000 revolutions per minute. The pump is a combination inducer and impeller. In this analysis the pump was modeled with one set of splitter blades. It is shown above with three blades at the inlet and six blades at the exit. The actual geometry has three blades at the inlet and 24 blades at the exit.

Periodic conditions along each full blade were assumed simplifying the model to 120 degrees of the complete geometry. The geometry was received from P&W electronically in an ASCII text file. The geometry was represented by nine blade streamlines on each of the pressure and suction surfaces and the pump hub and shroud contours. The streamlines and contours were read into a translation program, the surfaces extracted, and written out into PLOT3D format. The surfaces were then read into GRIDGEN where points on the surface were redistributed and the volume was discritized. Two grids were generated. Each grid has four zones with the total number of grid points ranging from 16,000 to 116,000.

The problem was solved using FDNS in the relative reference frame at design flow conditions. Results were post-processed using FAST and an in-house code to calculate and plot performance and blade loading.

For more information, contact Robert W. Williams.

Advanced Liquid-Hydrogen Turbopump Turbine and Volute



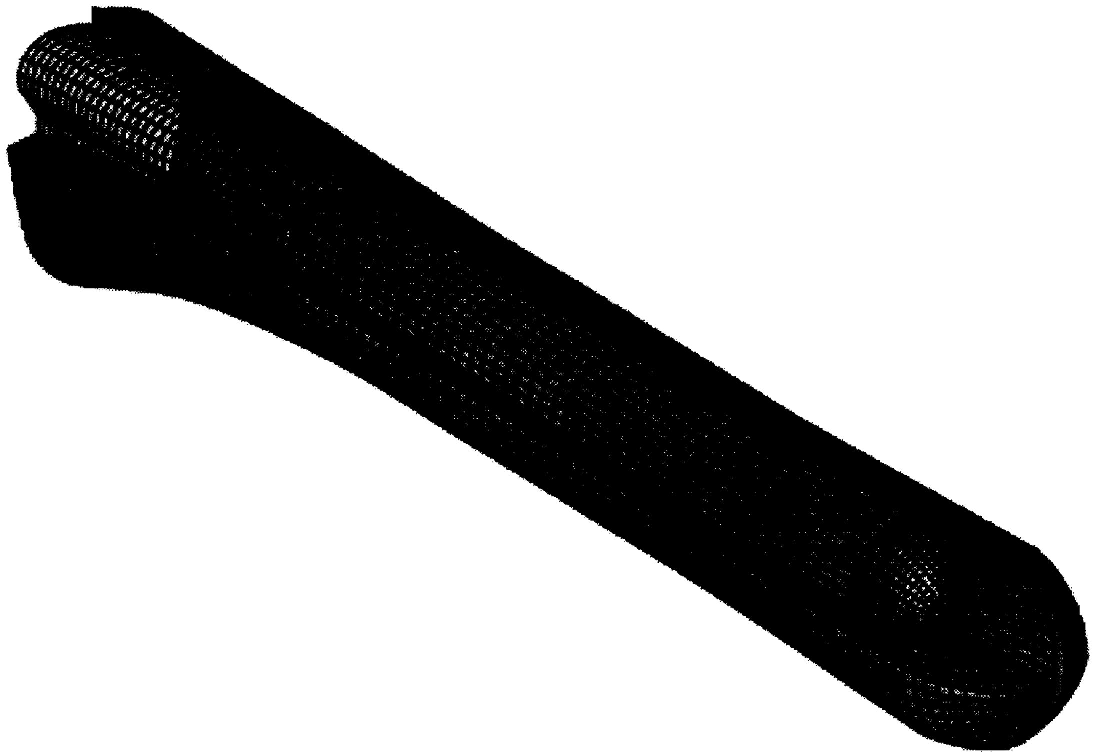
In support of the Air Force Phillips Lab ALT turbine aerodynamic design, NASA/MSFC is performing CFD parametrics to help determine optimum size and performance of this radial turbine. Unsteady CFD analysis is also being performed on the coupled volute and rotor. This turbine is in its design phase with the design work being performed by Pratt and Whitney.

The original surface grids for both sides of the rotor blade were obtained from an IGES file received from Pratt and Whitney. These surface grids along with the tabulated coordinates for the hub and shroud profiles were then input to TIGER. An initial volume grid was generated for a single blade passage and then input to GENIE++ to define the blade tip clearance region. The rotor grid currently contains three zones and consists of approximately 161,000 grid points. The volute grid is still in progress and when finished will be combined with a full 3D rotor grid.

Baseline balde analysis was performed using FDNS. The results indicate there are several problem ares that must be addressed.

For more information, contact Joni W. Cornelison.

Simulation of Bone Cement Flow During Hip Implant Insertion



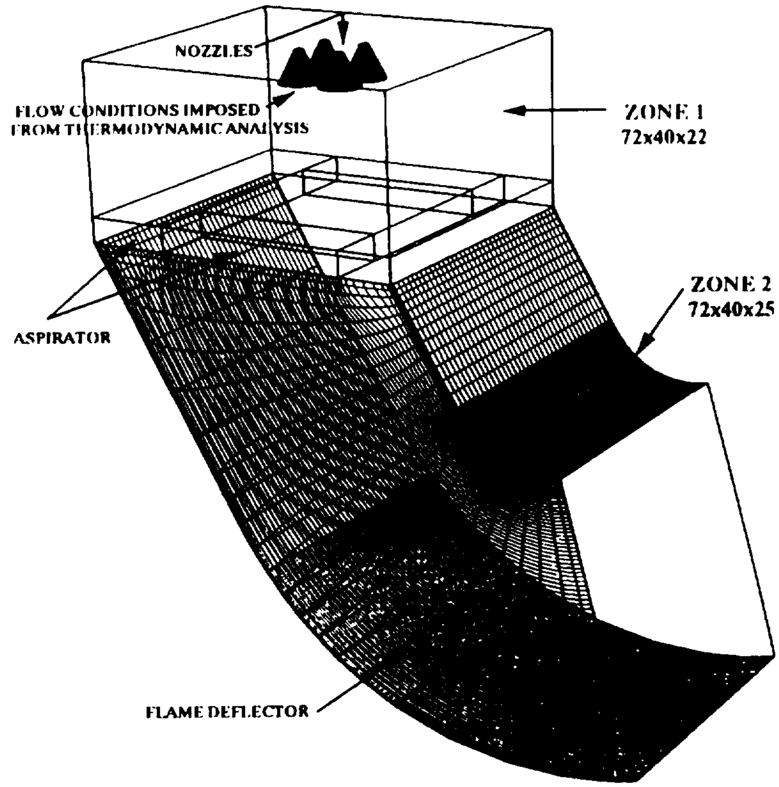
The purpose of this study was to simulate the flow of bone cement during hip implant insertion. For proper simulation of the induced cement flow, the surface of the stem and bone must move relative to each other at a velocity which corresponds to the insertion angle and rate. As a consequence, the grid must be continuously regenerated as the solution progresses. The mass flow of cement displaced out of the bone cavity is equal to the sum of the jacobian at each individual grid point.

The geometry definition was supplied by Howmedica Incorporated in IGES file format. The IGES file was read into CAGI where the hip-implant stem surface and the bone reamed-cavity surface were extracted and output in PLOT3D format. The two surfaces were then read into GENIE++ and transformed into NURBS surfaces for point distribution before building the discretized volume. The flow field for the 45,000 grid-point single-zone grid was then solved using FDNS modified to accommodate moving boundaries.

The analysis revealed the detrimental flow patterns which caused flaws in the solidified cement. The design was changed so that nonuniformity in the cement matrix was eliminated. A patent has been issued for this redesign.

For more information, contact Francisco Canabal.

Pollutant Environment from RD-170 Propulsion System Testing

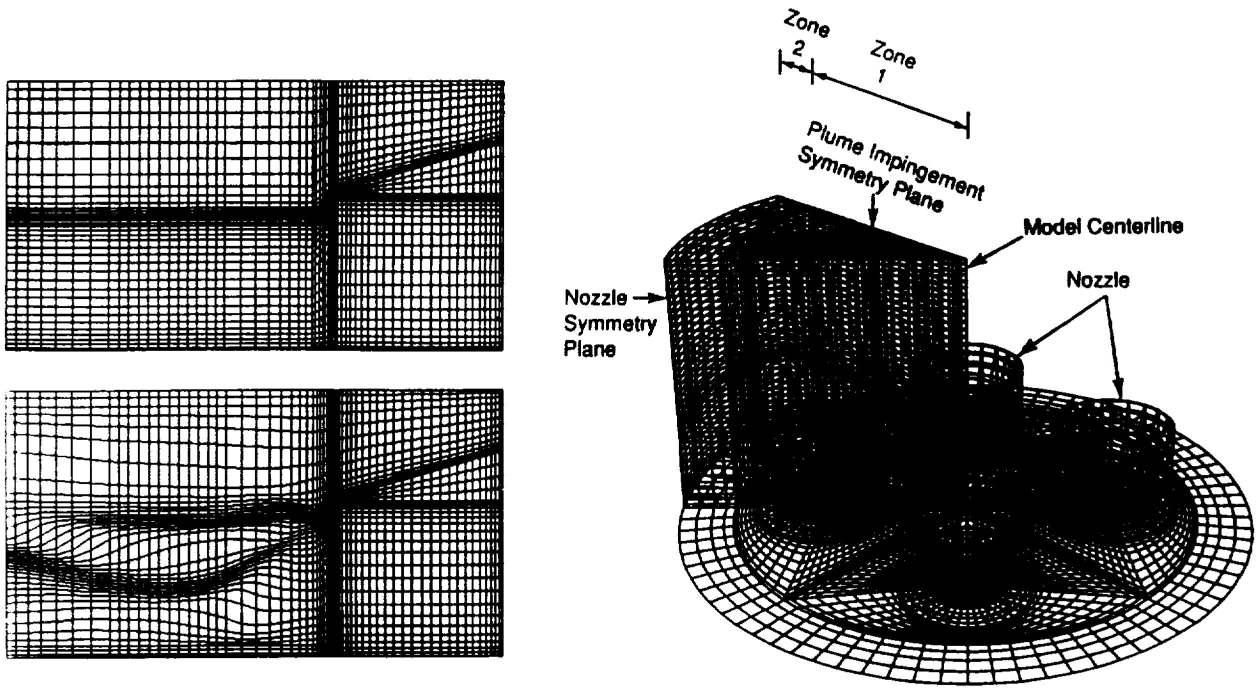


The objective of this study was to assess the exhaust-plume pollutant environment of the RD-170 engine during a hot-fire test on the F1 Test Stand at MSFC. A 3D simulation of the Russian-built kerosene engine was performed including: 3D air entrainment, 3D multiple-nozzle plume interaction and mixing with air, finite-rate after-burning reaction, plume impingement with flame bucket and plume quenching through deluge water, and 3D restricted plume expansion. Afterburning was modeled using 11-species, 18-reaction finite-rate chemistry. Water-quenching was computed assuming a homogeneous two-phase formulation.

Geometry was obtained from blueprints and test stand observation. Computational grid generation was performed using the EZSURF code. The edge curves of the nozzle exits, aspirator, flame deflector, and multi-zone block edges were generated first. Transfinite interpolation was then applied to create the initial surfaces. The flame deflector and nozzle exit surfaces were elliptically smoothed using Bezier curve and local redistribution techniques. The volume grid for the first block was created using two linear stackings -- one from the top of the block to the nozzle exit plane and then another from the nozzle exit plane to the bottom of the aspirator. The flame deflector block and subsequent atmosphere block volume grids were created using transfinite interpolation. The three-zone grid has a total of 300,000 points.

For more information, contact Dr. Ten-See Wang.

Adaptive Gridding of a Four-Engine Clustered Nozzle Configuration



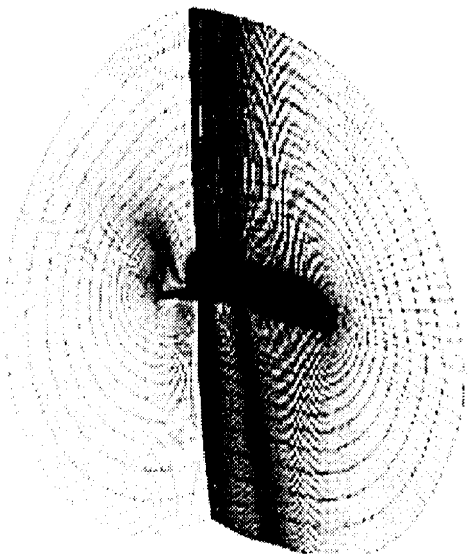
The objective of this study was to propose a computational methodology that could effectively anchor the base flowfield of a four-engine clustered nozzle configuration. For an efficient CFD calculation, a Prandtl-Meyer solution treatment was applied to the algebraic grid lines for resolution of initial plume expansion. As the solution evolved, the computational grid was adapted to the pertinent flow gradients.

Making use of the symmetrical nature of the geometry, a grid was generated for a small wedge of the complete flowfield. The "plume impingement symmetry plane" and the "nozzle symmetry plane" intersect along the "model centerline" at a 45-degree angle. Plume impingement and recompression would occur along the first symmetry plane and the centerline of the nozzle passes through the second symmetry plane. A two-zone baseline algebraic grid was generated with GENIE++. The first zone started from the base proceeding downstream, including the nozzle and plume region. The second zone (the outer shell) consisted of the ambient environment and a portion of the expanded plume. Grids ranged in size from 120,000 to 250,000 grid points. Initial plume-angle grid resolution was essential to the accurate prediction of base flow properties. Algebraic grids were generated for each pressure ratio resolving the initial plume angle by applying the isentropic Prandtl-Meyer plume-expansion theory.

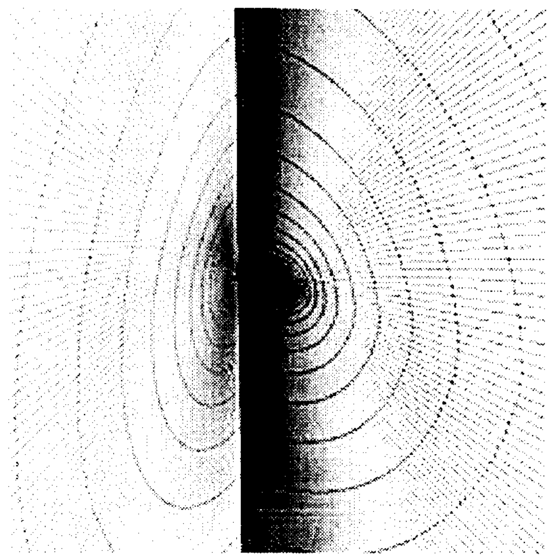
The SAGE code was used to refine the initial plume angle resolved algebraic computational grid. Since mach number contour was closely associated with the plume boundary layer and the pressure gradient follows the recompression shock, these two flowfield variables were used as pertinent grid-adaptation parameters. Adaptation was applied downstream of the nozzle lip to maintain the initial plume-expansion angle. Grid-line clustering follows both the plume boundary layer and the recompression shock.

For more information, contact Dr. Ten-See Wang.

WB001 Wing-Body Vehicles



The volume grid for supersonic flow simulations (159x129x43)



The volume grid for transonic flow simulations (159x129x65)

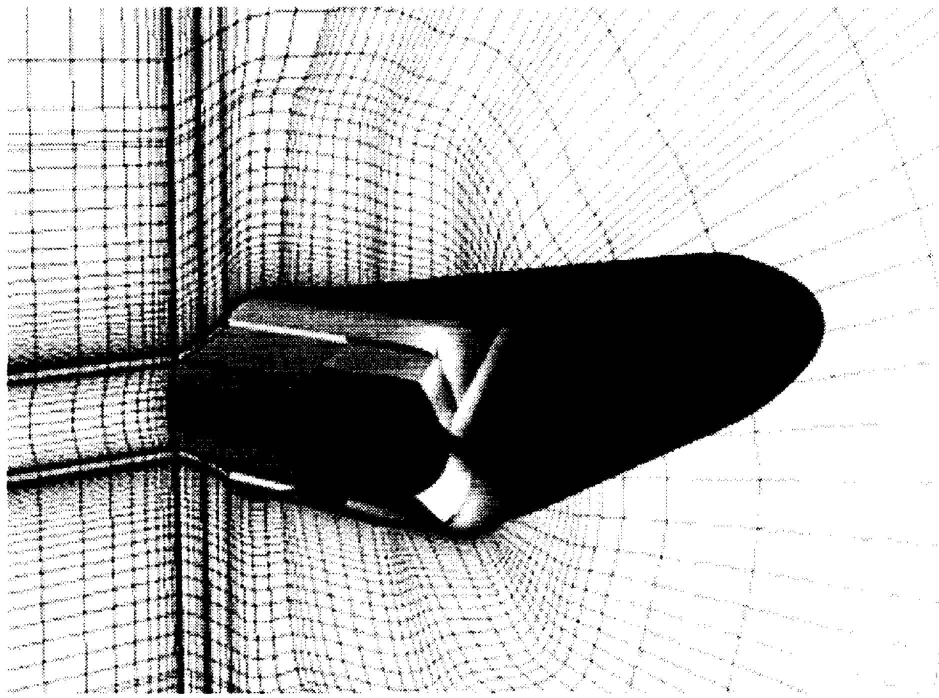
The original surface grid for the WB001was obtained from NASA-LaRC/MS66. This single-block grid contained 161x129 points. The volume grids were generated using HYPGEN. Since HYPGEN is a hyperbolic grid generator, there was no need to define the far field surface. Also, in order to obtain a smoothly stretched grid and sometimes to avoid negative volume grids, it was required to have a viscous spacing off the body. Subsequently, some layers near the surface could be removed for inviscid calculations.

The computational domain for each of the transonic and supersonic cases had to be properly generated to fully capture the physical phenomena. The figure on the left shows the volume grid generated for supersonic simulations. This grid was used for high supersonic simulation (M_∞=5.72); therefore, the far field considered was close to the body, about 1 body length. This helped reduce the number of grid points used for the computation, thus less CPU time was required for the execution. Likewise, for transonic simulations (M_∞=1.1) the far field had to be extended as far as possible to ensure free stream conditions. To be conservative, a far field of 10 body lengths was generated. The figure on the right shows the volume grid generated for transonic simulations.

It is noticed that the number of grid points in the k-direction was increased for the transonic simulations in order to retain a reasonable stretching ratio. Also, the number of grid points in the i-direction was changed so that mesh sequencing could be performed during flow computations.

For more information, contact Bruce T. Vu.

Lifting Body with Aerospike Engines



The original surface geometry was provided as 18 coarse surfaces defining the body, wings, aerospike engines and body flaps. These surfaces were manipulated using a NURBS surface tool. The GENIE++ general purpose grid generation system was used to generate the volume grids. All the important features, such as fins, body flaps, nozzles, and base regions, have been included in this configuration. These grids were designed to aid in calculating plume aerodynamic interactions at subsonic conditions. For supersonic simulations the grid lines at the base regions could be modified to account for the plume angles. These structured grids contain 6 zones, totaling more than one-half million grid points, to be used for inviscid calculations.

For more information, contact Bruce T. Vu.

STRUCTURED PATCHED GRID TECHNOLOGY