[redacted]

N95-28732

19951223 11

8P3

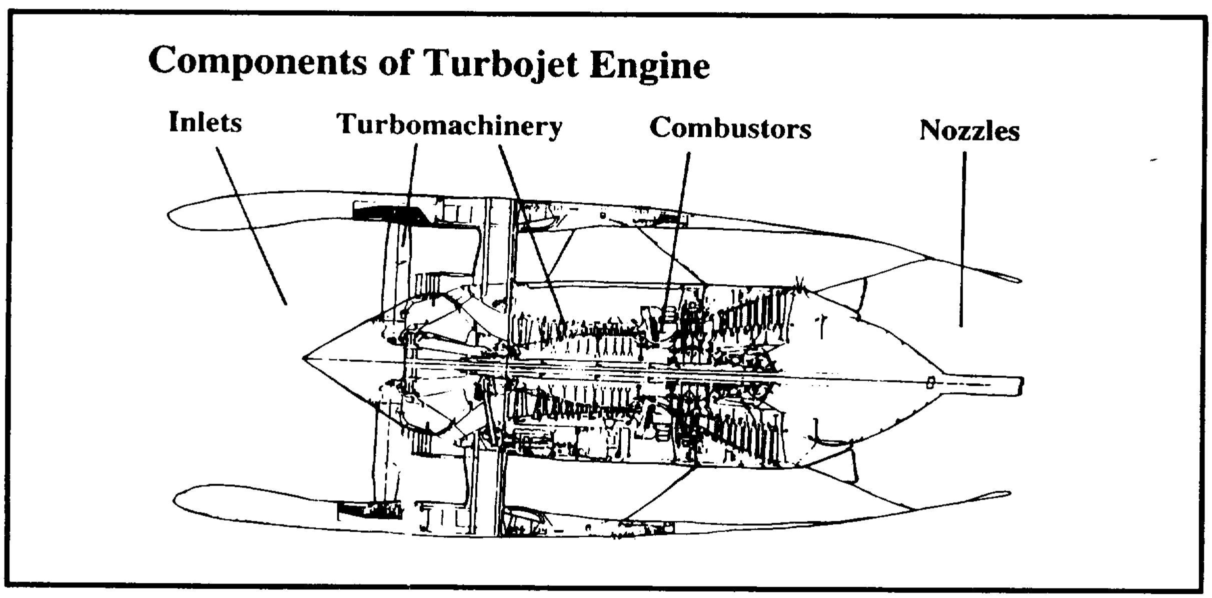
404944

SURFACE MODELING AND GRID GENERATION FOR AEROPROPULSION CFD

Yung K. Choo, [1] John W. Slater, [2] James Loellbach, [3] and Jinho Lee, [4]
NASA Lewis Research Center
Cleveland, Ohio 44135

SUMMARY

The efforts in geometry modeling and grid generation at the NASA Lewis Research Center, as applied to the computational fluid dynamic (CFD) analysis of aeropropulsion systems, are presented. The efforts are mainly characterized by a focus on the analysis of components of an aeropropulsion system, which involve turbulent viscous flow with heat transfer and chemistry. Thus, this discussion will follow that characterization and will sequence through the components of typical propulsion systems consisting of inlets, compressors, combustors, turbines, and nozzles. For each component, some applications of CFD analysis will be presented to show how CFD is used to compute the desired performance information, how geometry modeling and grid generation are performed, and what issues have developed related to geometry modeling and grid generation. The discussion will illustrate the following needs related to geometry modeling and grid generation as observed in aeropropulsion analysis: (1) accurate and efficient resolution of turbulent viscous and chemically-reacting flowfields, (2) easy-to-use interfaces with CAD data for automated grid generation about complex geometries, and (3) automated batch grid generation software for use with design and optimization software.



[1]Senior Scientist, CFD Branch, Internal Fluid Mechanics Division
[2]National Research Council Associate, CFD Branch, Internal Fluid Mechanics Division
[3]Research Associate, Institute for Computational Mechanics in Propulsion (ICOMP)
[4]Senior Engineer, NYMA, Inc.

[redacted]

Inlets

NASA Lewis inlet analysis from subsonic to hypersonic flight

Inlet flow characteristics to compute include:

Pressure Recovery
Distortions at compressor face
Cowl drag
Flow quantity and engine matching
Flow stability
Flow angularity

Surface Modeling / Grid Generation / CFD:

CAD software for surface modeling
NASA-IGES for data exchange
Interactive domain decomposition for multi-block grids
Space-marching, reduced-Navier-Stokes solvers on structured grids
Time-dependent, Navier-Stokes solvers on multi-block, structured grids
CFD codes are characterized by specialization to propulsion flowfields

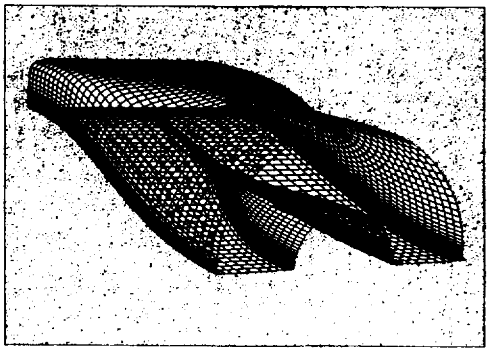
The purpose of the inlet is to provide the desired quantity and quality of airflow to the engine for stable production of thrust. The analysis of inlets at the NASA Lewis Research Center encompass flight speeds from subsonic to hypersonic.

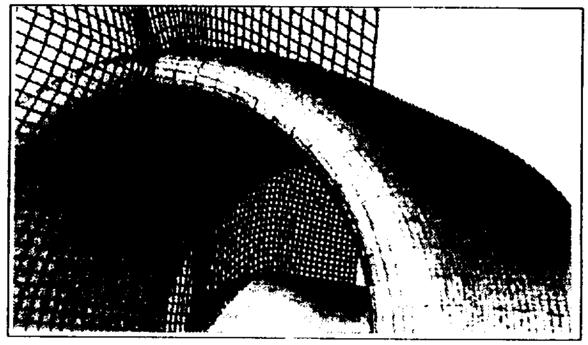
The CFD analyses are applied to determine such information as the mass flow rates, pressure recovery, distortions at the compressor face, inlet drag, flow angularity, and flow stability [1]. Most analyses involve steady-state flow; however, some unsteady flow analyses are conducted when examining inlet stability.

The primary CFD codes used at the NASA Lewis Research Center for inlet analysis include RNS3D [2] and NPARC [3]. The RNS3D code solves the reduced Navier-Stokes equations to efficiently model the flow using a space-marching method. The NPARC code solves the full, unsteady Navier-Stokes equations using time-marching methods. Both codes have undergone development to focus their abilities towards the analysis of propulsion flowfields, which are dominated by turbulent viscous effects. The codes include capabilities for bleed boundary conditions, turbulence models capable of predicting separation of flow from inlet walls, and compressor face boundary conditions. Both codes use structured grids primarily because of familiarity with the performance of the numerical methods and the desire to efficiently and accurately model turbulent viscous flows. The NPARC code uses multi-block, structured grids to model complex geometries. The blocks may overlap in a non-contiguous manner for difficult geometries.

Inlet Shape Design and Optimization

Bifurcated Transitioning S-Ducts **Subsonic Transport Inlet Optimization**





Use of CFD in design environment

Geometry defined by geometric parameters
Desire grid generation to be batch operation
Grid generation software as callable libraries

Two applications of CFD methods to inlet shape design and optimization studies are presented to illustrate how geometry is modeled parametrically and grids are generated as batch processes. This requires grid generation software to be accessible in a format callable by the design software.

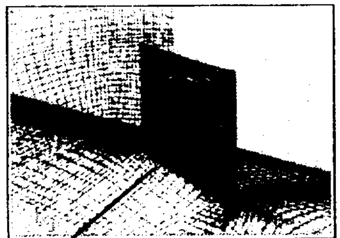
The first application is the analysis of bifurcated transitioning S-ducts for high-speed transport applications [4]. The analysis used the RNS3D and NPARC codes to compute pressure recovery and distortion at the compressor face. It is desirable to understand the relationship between the turbulent flow patterns and the shape of the cowl surface and the length of the inlet. The shape of the cowl was defined by parameters of an analytic equation. The structured grid was computed automatically with desired stretching for viscous resolution.

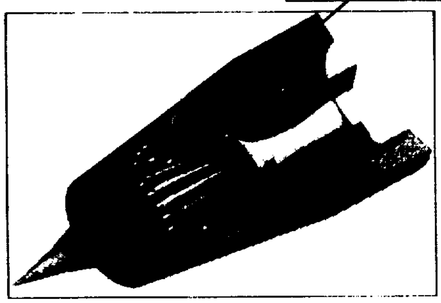
The second application is the shape optimization of a subsonic inlet [5]. The analysis modeled a 180 degree section of the inlet from the crown to the keel assuming symmetric geometry and flow conditions. At each section of the cowl, the profile was defined by a set of parameters of a superelliptic equation. The parameters varied as a function of the section angle. A three-dimensional grid was generated as a sequence of two-dimensional grids generated at each section angle of the inlet using an elliptic grid generation method from GRAPE2D. The NPARC code was used to compute the Mach number distributions on the surfaces of the inlet. The objective was to optimize the shape of the interior of the cowl such as to minimize the peak Mach number within the inlet. The optimization loop required the grid generation to be a batch process to allow regeneration of the grid at each optimization iteration.

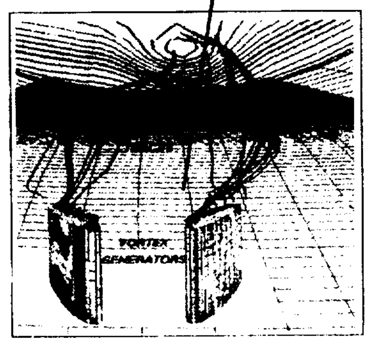
Geometric Complexity in Inlets

Geometric Features

Bleed slots, holes, passages
Vortex generators
Struts







Geometry Modeling and Grid Generation Methods

NURBS curves and surfaces, multi-block, structured grids
NASA-IGES interface

An inlet may consist of several complex geometric features, including bleed slots and holes, bleed passages, vortex generators, and struts. The struts are present due to structural and ducting requirements. The scale of the bleed slots and holes and the vortex generators are often small compared to the size of the duct, but their influence on the inlet flowfield may be quite significant. They are usually introduced to stabilize and to improve the flow. Because of their significance to the flow, an analysis usually includes their pressence either numerically or physically. RNS3D includes a numeric representation of a set of vortex generators as vortex sources in the flow. NPARC includes flow boundary conditions to simulate bleed surfaces.

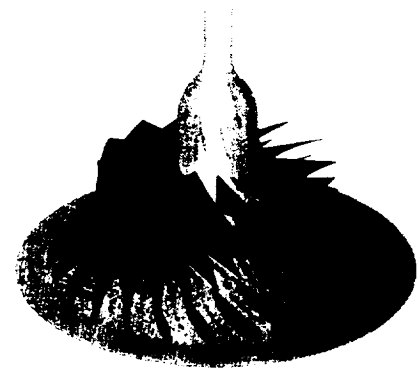
Some studies have begun to physically model bleed features and vortex generators as a way to bypass numerical modeling of these features. The figures above show some examples of grids generated about inlet geometries that include vortex generators. The high-speed inlet shown above made use of the capabilities of the ICEM DDN software to create a geometry model of the inlet from blueprint drawings [6]. Such a geometry model may come from a CAD effort. The ICEM CFD software was then used to create a multi-block, structured grid about the vortex generator. Some compromise of the vortex generator tip geometry was needed to simplify the multi-block format.

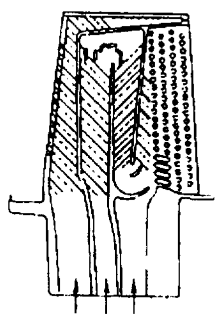
The F/A-18 inlet analysis shown above used the I3G/VIRGO and GRIDGEN software to create the geometry and multi-block, structured grid from axial station coordinates [7]. The turbulent viscous flow analysis examined the flow in the inlet for high angle-of-attack flow conditions. The analysis includes the geometric modeling of a pair of vortex generator within the duct.

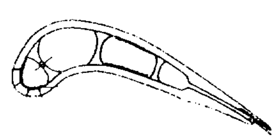
Turbomachinery Geometry Issues

Primary geometric features, such as endwall and blade shapes, are accurately represented.

Secondary features, such as cooling holes and seals, are often ignored to reduce cost.





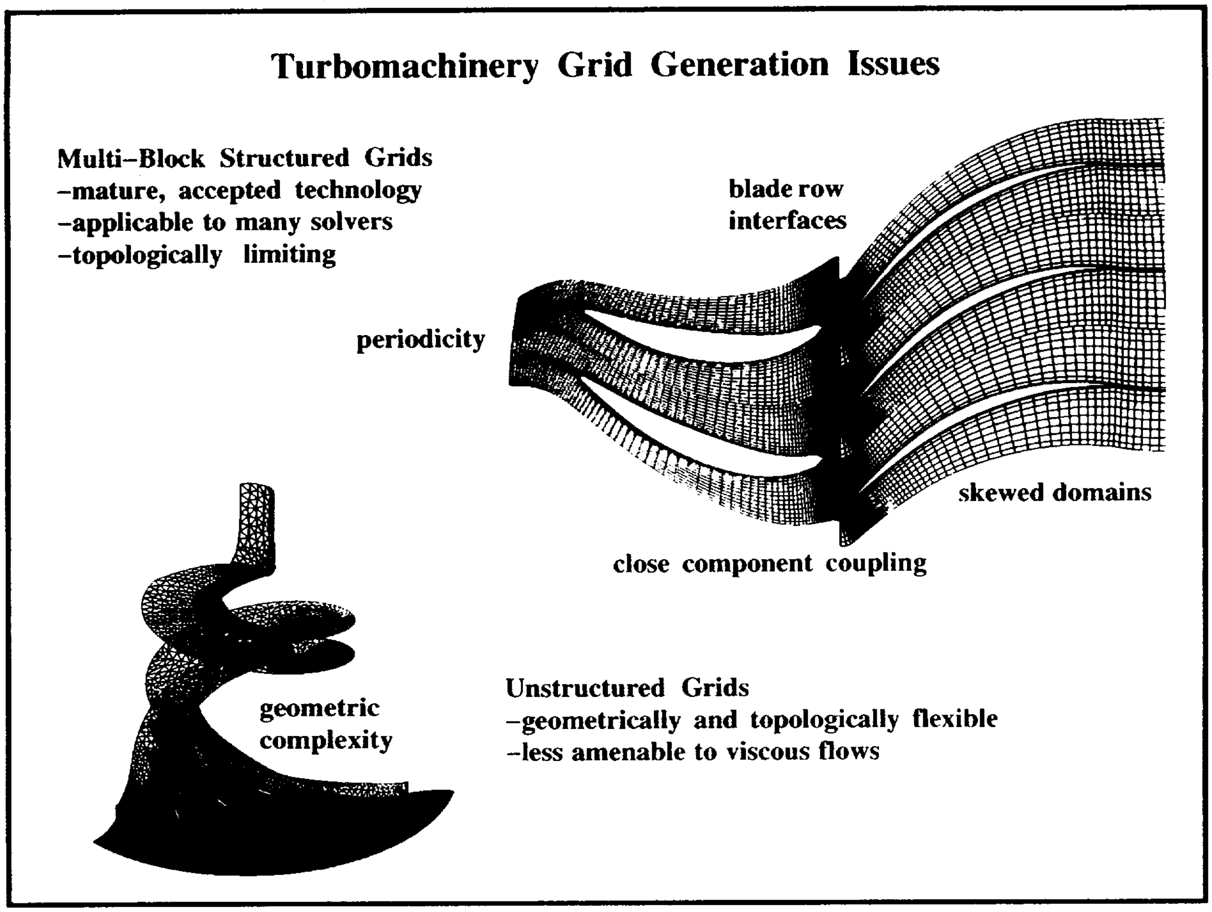


The cost of accurately representing small geometric details is very high, from both grid generation and flow solution standpoints.

Turbomachinery refers to a broad class of devices which transfer energy to or from a working fluid by the use of rotating components. Examples include nonreciprocating pumps and compressors, and turbines. Turbomachinery research at NASA Lewis is focused on applications to aircraft propulsion (radial and axial compressors and axial turbines in turbojet engines) and rocket propulsion (radial pumps and axial turbines in liquid fuel turbopumps). The objective of any turbomachinery component is to accomplish the desired transfer of energy to or from the working fluid with as little loss as possible in order to maximize the overall efficiency.

The typical level of geometric modeling used in numerical analyses of turbomachinery components consists of accurate representations of the channel boundaries (hub and tip endwalls) and the blade shapes. Small geometric features such as coolant holes and passages, tip clearance gaps, and seals between rotating and nonrotating components are often ignored in the interest of reducing the solution cost. The fluid-dynamic effects of certain small geometric details can sometimes be numerically modeled in the flow solver without accurate geometric representation, but usually to only a low order of accuracy. When the fluid dynamic effects of the small geometric details are of primary importance, they must be included in the geometric model. This is very costly, from both the grid generation and flow solution standpoints. Automated methods for topology determination and grid generation could be of use for such geometric complexity.

Geometry modeling techniques commonly used for turbomachinery configurations include CAD-based methods, NURBS representations, or simple discrete point representations of surfaces used in conjunction with different surface fitting and interpolation schemes. There currently is no real standard for geometry modeling of turbomachinery components.



A variety of grid types have been used in turbomachinery component analysis, ranging from single-block and multi-block structured grids to fully unstructured grids. Common structured grid topologies for turbine and compressor blades include C-, H-, and O-meshes. Multiple blocks are required for all but the simplest of configurations. Although the use of unstructured grids for turbomachinery analysis is increasing, the use of structured grids remains somewhat more common. This is due to the generally lower computational resource requirements of structured flow solvers and the availability of techniques for generating highly-stretched structured grids for viscous flows.

It is often only necessary to generate a grid for a single blade in a given row; the grids for the other blades in the row are copied and rotated from the original grid. Smooth transmission of flow properties across the boundaries of adjacent grids requires either that the grids be periodic or that the flow solver account for nonperiodic grids. Blade rows which rotate relative to each other are usually separated by a sliding grid interface, which therefore must be axially symmetric. These constraints, when applied to closely coupled, high-solidity blade rows, often result in computational domains with high inherent skewness and make the generation of nearly-orthogonal structured grids difficult. For viscous calculations, grid points should be clustered normal to solid walls and in the blade wake regions. This is difficult to achieve in unsteady, multiple blade row calculations in which the wakes of rotating components move with time. The use of unstructured grids can alleviate some of the problems mentioned above due to their greater geometric flexibility. Hybrid techniques combine both structured and unstructured grids in a single calculation in an attempt to exploit the advantages of each approach.

Turbomachinery Flow Solution Issues

Actual flows are unsteady and strongly affected by turbulence and viscosity.

Steady–state analyses of isolated components can usually be performed in body–fixed, rotating coordinate systems.

Mixing–plane interfaces can be used to couple components when unsteady interactions are not required.

Otherwise, time–accurate, unsteady, simultaneous solution techniques must be used for multiple components.

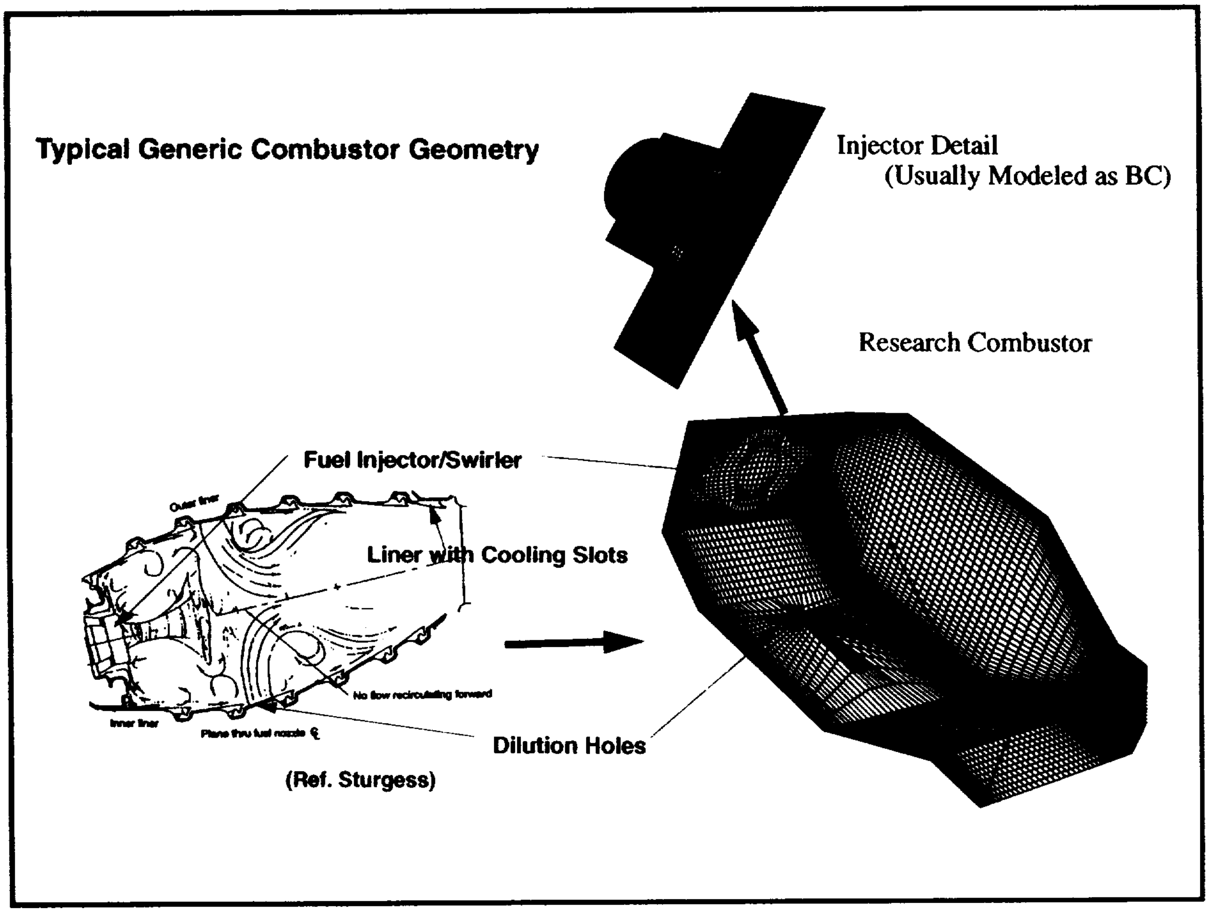
Several 3D Navier–Stokes solvers have been developed specifically for turbomachinery problems.

The choice of solver depends on what problem is being solved, and what information is desired from the solution.

Actual turbomachinery flows are unsteady due to the relative rotation of components. They are also strongly affected by viscosity and turbulence because of the influence of upstream components and the internal nature of the flow. Typical engineering values of interest include blade loadings, loss distributions, and heat transfer rates. Time-averaged values are useful for overall performance evaluation while unsteady values are used to determine peak loads and temperatures. Blade loadings can sometimes be accurately predicted using Euler methods. Accurate loss and heat transfer predictions, however, require Navier-Stokes techniques free from excessive numerical diffusion, and grid systems appropriate to the flow problem and solver.

The analysis of isolated turbomachinery components can often be performed in a steady manner by using body-fixed, rotating coordinate systems. Mixing-plane interfaces can be used to transmit tangentially-averaged solution data between components when unsteady interactions are not significant. When such interactions are significant, time-accurate, unsteady, simultaneous solutions for the components are required.

Several three-dimensional, Navier-Stokes solvers are used at NASA Lewis for turbomachinery analysis. These include the structured solvers HAH3D [8], RVC3D [9], TRAF3D [10], MSTAG [11], and BTOB3D [12] and the unstructured solver USM3D [13], among others.



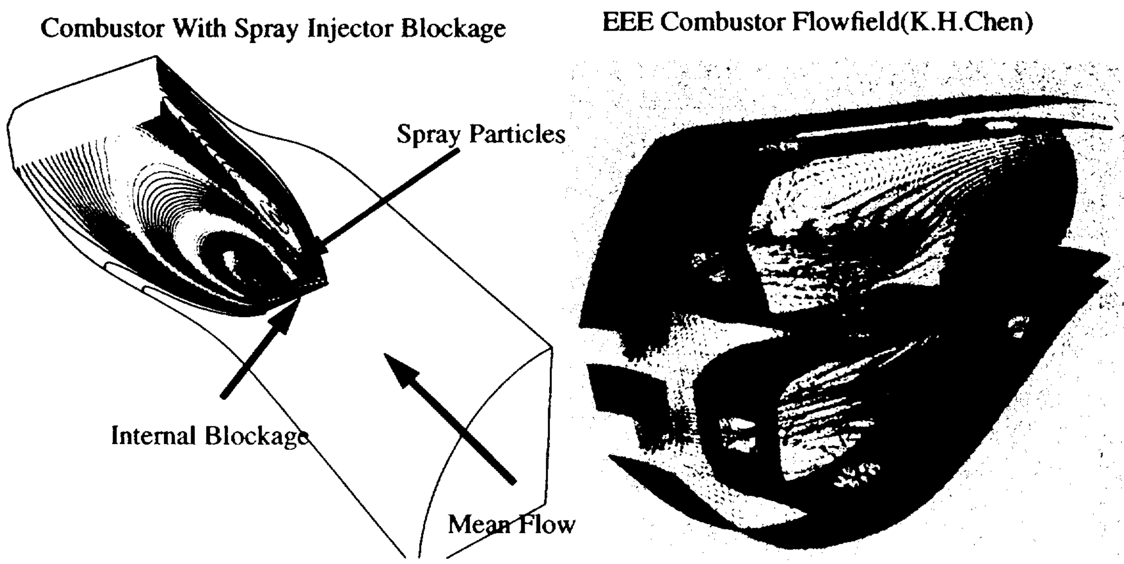
In any aeropropulsion system, the chemical potential of the fuel is converted into momentum gain in the combustor section. The next generation of commercial subsonic and supersonic aircraft require engines whose performance must be substantially better than current propulsion systems. This requires higher operating pressures and temperatures for the combustor. In addition, these engines must be capable of high performance with lower emissions levels to satisfy more stringent environmental regulations. Furthermore, these problems need to be resolved at a lower cost and in a shorter time frame.

Today's propulsion systems use very complex combustor and fuel injector geometries to obtain high combustion and mixing efficiencies for maximum gain of thrust from a given chemical potential. A typical gas turbine engine combustor geometry consists of a number of combustor "cans" arranged in annular form. The cannister illustrated in cross section above, contains a fuel injector with fuel atomizer, swirler, dilution holes, combustor liner with transpiration cooling holes, and slots with coolant injectors.

Combustor analysis requires the prediction of (HC, CO, NOx) emission characteristics, exit pattern factors, liner wall temperature levels and gradients, ignition, smoke and soot production, flame stability, flame blowout and relight.

The development of a new combustor design is often very difficult and costly. Present design analysis methods consist largely of one- or quasi-two- dimensional empirical and semi-empirical analyses, previous experience, and a large number of expensive component testing with a long development cycle time [14] [15].

ALLSPD Low Speed Combustor Model Prediction Using Methanol Chemistry Model

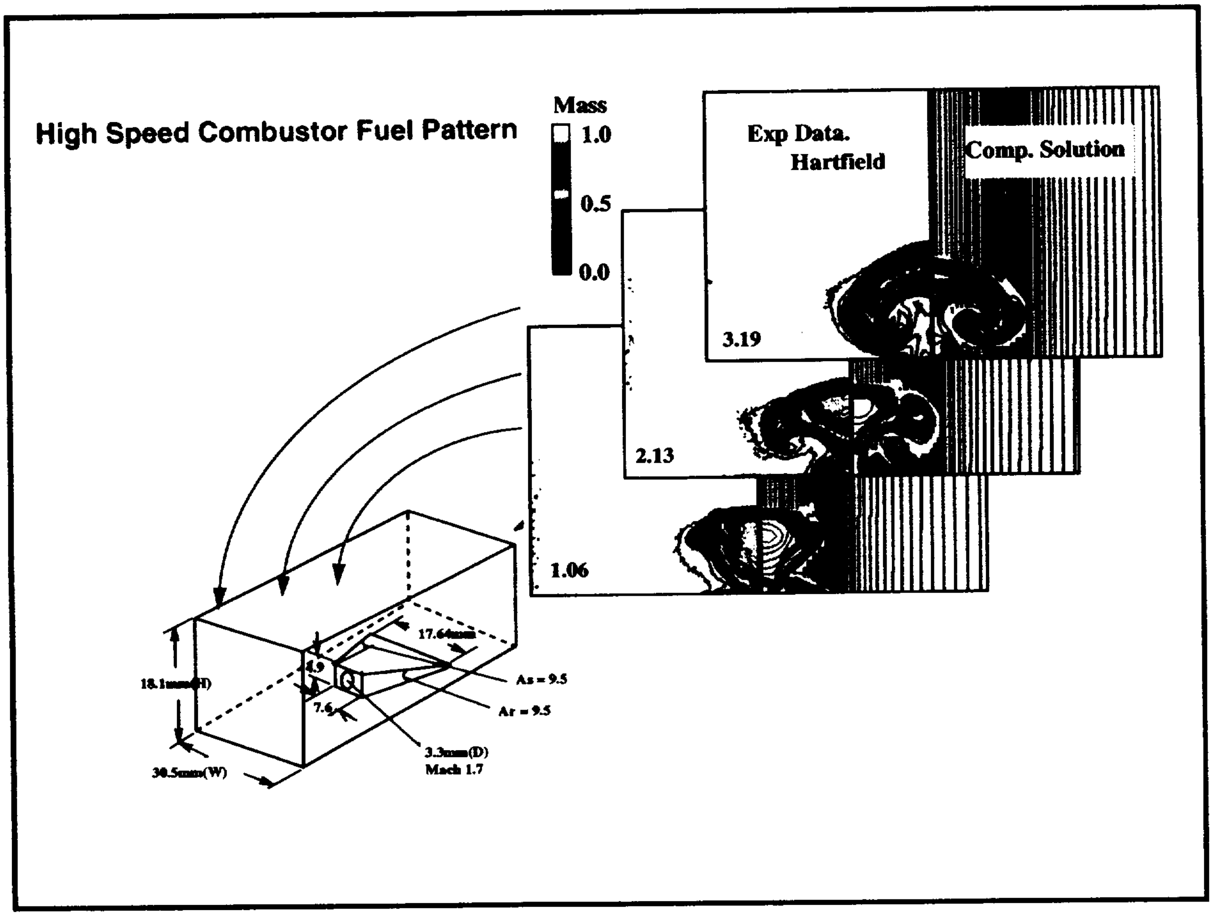


An improved analytical tools are needed to predict the performance of the combustor for complex combustor geometries. Design engineers would like to have analytical tools that contain the optimal balance of the best possible physics. These include accurate turbulence and chemistry models, fast and accurate numerical methods, the ability to handle complex geometries rapidly, and the capability to interface with other analytical tools.

In order to satisfy these challenges, fast and accurate analytical CFD tools are being developed at the NASA Lewis Research Center with refined physical models for combustor design. Two implicit LU solvers for reacting flows have been developed. The RPLUS code [16] is being developed for high-speed combustor flowfield analysis. The ALLSPD code [17] is being developed for low-speed combustor analysis. Both of these codes use the implicit treatment of the chemical source terms and have the capability to handle detailed finite-rate chemistry models, global chemistry models, Monte-Carlo PDF models [18], a low Reynolds number two-equation turbulence model, a spray chemistry model, and multiple-block structured grid systems.

The capability of the ALLSPD code is illustrated by the prediction of the temperature field inside of generic low-speed combustor shown above. A low-speed, low Reynolds number transition duct is used as a combustor, with combustion simulated using a methanol spray chemistry model. The figure also shows the flow pattern predicted by ALLSPD for the EEE combustor. Some of the more complex geometry, such as the combustor liner, is also modeled along with the spray chemistry.

An unstructured combustor solver with the majority of the physical model features contained in the ALLSPD code is also being developed.



The capability of the RPLUS code to predict the complex fuel injection patterns for a chemically reacting flowfield is shown above. The CFD prediction of the fuel penetration pattern generated by a swept ramp fuel injector is shown along with the experimental data. The fuel penetration and mixing is key to understanding and controlling combustion behavior.

For combustor analysis, additional chemical reaction / species transport equations are needed for the chemistry models, which can easily triple the number of equations which must be solved by the CFD codes. Three-dimensional combustor flowfield analysis with detailed chemistry models, until recently, has been limited to basic research efforts involving simplified geometries. The stiffness generated by the chemical time scales further restricts the convergence rate, which also restricts the size of the problem.

Research is being conducted to understand and to reduce the impact of uncertainties in the numerical modeling of turbulence, chemistry, and unresolved geometrical features. The combustor geometrical features such as fuel injectors and swirlers must either be numerically modeled as boundary conditions or resolved physically. Presently, the majority of these complexities are modeled as simple boundary conditions in order to reduce the computational cost.

The RPLUS and ALLSPD codes can be used with any structured grid generation software, such as CFD GEOM, GRIDGEN, NGP, GRIDPRO, and INGRID3D. The prevalent use of multi-block, structured grids has been due to the available wealth of mature numerical technology. As illustrated, the complex combustor geometry can be effectively treated using a structured multiple block grid topology with generalized block interface treatment.

Regardless of the topology, these codes will require: refined near-wall grid resolution with low cell aspect ratio, holes with various geometrical configurations, and smooth and nearly orthogonal grids. In addition, flexible internal grid structures for specification of blockages with irregular geometrical shapes fixed in space are also highly desirable.

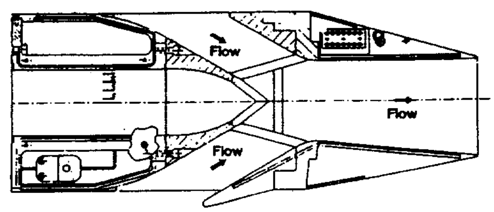
The analysis of next generation combustor designs requires that the CFD solvers have either the ability to accurately model or to numerically resolve the following key combustor design features: swirlers, fuel injectors, canister shapes with slots, dilution holes, cooling slots, and perforated walls.

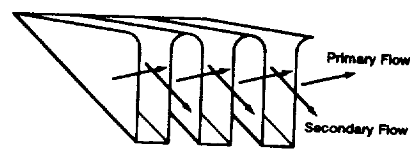
Nozzles

NASA Lewis nozzle analysis from subsonic to hypersonic flight

Nozzle flow characteristics to compute include:

Thrust
Flow mixing
Boattail drag
Noise





Mixer/Ejector Nozzle
Surface Modeling / Grid Generation / CFD

Time-dependent, Navier-Stokes solver on multi-block, structured grids
Interactive domain decomposition for multi-block, structured grids

The purpose of the nozzle is to efficiently convert the thermal energy of the combustion products into the kinetic energy of the exhaust jet and to straighten the gas flow. The analysis of nozzles at the NASA Lewis Research Center encompass flight speeds from subsonic to hypersonic.

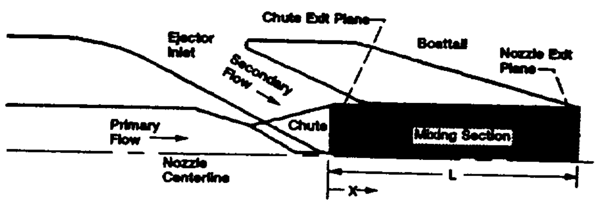
The CFD analyses are applied to determine such information as the thrust, boattail drag, and level of flow mixing of primary and secondary flows. The flows are dominantly turbulent viscous flowfields. The primary CFD code used for nozzle flow analysis is the NPARC code.

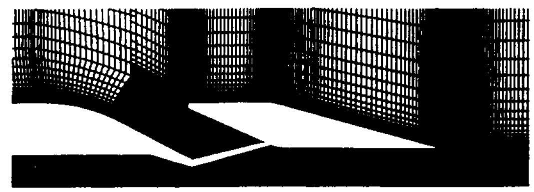
One example of the type of nozzle flow analysis being conducted at the NASA Lewis Research Center is the flow in a mixer/ejector nozzle during take-off and landing conditions [19]. Secondary flow is entrained into the primary jet flow through a set of lobed chutes introduced into the primary flow. The chute forms vortices which aid in the flow mixing. After take-off, the chutes retract back into the nozzle walls. The objective is to reduce the jet noise during take-off and landing. An application of such a nozzle would include high-speed civil transports.

Another significant problem under analysis is the computation of the boattail drag at transonic flow conditions. The objective is to generate a database of drag data for varying flow and geometric parameters.

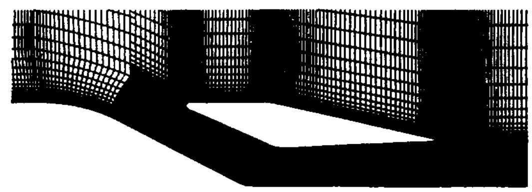
Geometry and Grids for Mixer/Ejector Nozzle

Mixer/Ejector Nozzle





(a) Primary centerline (A_s/A_{mix} = 1.0).



(b) Secondary centerline (A_s/A_{mix} = 1.2).

Geometry read from detailed drawings
Geometry simplified to include only flow passages
Flow domain is 1/2 wavelength of chute lobes
8 structured grid blocks, clustered for turbulent flow computation
Interactive domain decomposition and grid generation
Grids include about 1 million grid points

The mixer/ejector nozzle represents the typical geometric complexity of CFD analyses of nozzles. For the analysis discussed in reference [19], the geometric dimensions for the nozzle were read directly from detail drawings. Mechanical elements such as acuators, hinges and seals were not included in the geometry model. The flow domain consisted of 1/2 of the wavelength of a lobe of the chute with assumptions of flow symmetry. The I3G/VIRGO software package was used to define the surface geometry and grids. The INGRID3D software package was used to generate the volume grids. The grid contained 8 blocks with a total of about 1 million grid points. The grid used an H-grid topology. The grid was clustered near the walls for resolution of turbulent flow. One difficulty in generating the structured grid for the block containing the lobe of the chute was excessive grid skewing due to the S-shaped turn. This grid is typical of those used in the analysis of nozzle flows. The GRIDGEN software package is also used for grid generation.

Concluding Remarks

Some concluding remarks that can be stated from this overview of the geometry modeling and grid generation from the perspective of the CFD analysis of propulsion components include the following:

- The geometry modeling and grid generation activities at NASA Lewis are oriented to the application to components of aeropropulsion systems.
- The aeropropulsion flowfields are dominantly turbulent viscous flowfields with heat transfer and chemical reactions.
- The complexity of the geometry and flows and limitations of computational resources require a compromise between numerical and physical modeling.
- The majority of the analyses are performed using structured grids because of the maturity of the CFD methods as applied to the computation of viscous flows with and without chemical reactions.
- As computational resources increase, the capability to include greater geometric complexity improves. This requires an easy-to-use interface with CAD representations and more automation in grid generation.
- Automated grid generation must still allow proper control over the grid distribution to properly resolve the flowfield.
- Design and optimization processes use CFD and grid generation in a batch mode. Interactive-based software, perhaps, can include an option for either the use of a graphical user interface or the use of callable libraries.

REFERENCES

1. Anderson, B.H. and C.E. Towne, "Application of Computational Fluid Dynamics to Inlets," Practical Intake Aerodynamic Design, Ed. E.L. Goldsmith and J. Seddon, AIAA, Washington, D.C., 1993.
2. Towne, C.E., "RNS3D User's Guide: Version 0.8," January 1994.
3. NPARC Alliance Technical Team, "A User's Guide to NPARC: Version 2.0," November 1994.
4. Anderson, B.H. and K. Kapoor, "A Study on Bifurcated Transitioning S-Ducts for High Speed Inlet Application," AIAA-94-2812, June 1994.
5. Reddy, E.S. and D.R. Reddy, "Aerodynamic Shape Optimization of a Subsonic Inlet Using 3-D Euler Computation," Paper to be presented at the 31st AIAA/ASME/SAE/ASEE Joint Propulsion Conference, July 1995.

6. Slater, J.W., Joongkee Chung, Y.K. Choo, "Geometry Modeling, Grid Generation, and Flow Computations for Complex Geometries," AIAA-94-2717, June 1994.

7. Smith, C.F. and S.D. Podleski, "Installed F/A-18 Inlet Flow Calculations at 30 Degree Angle-of-Attack: A Comparative Study," AIAA-94-3213, June 1994.

8. Hah, C., "Calculations of Three-Dimensional Viscous Flows in Turbomachinery With a Low Reynolds Number Turbulence Model," AIAA Journal of Propulsion and Power, Vol. 3, No. 5, May 1987, pp. 415-422.

9. Chima, R.V, "Viscous Three-Dimensional Calculations of Transonic Fan Performance," NASA TM 103800, May 1991.

10. Arnone, A., Liou, M-S., and Povinelli, A., "Transonic Cascade Flow Calculations Using Non-Periodic C-type Grids," CFD Symposium on Aeropropulsion, NASA Conference Publication 3078, 1990.

11. Adamczyk, J.J., "Modeling Equation for Simulating Flows in Multistage Turbomachinery," ASME Paper 85-GT-220, 1985.

12. Dawes, W.N., "Development of a 3D Navier-Stokes Solver for Application to all Tyles of Turbomachinery," ASME Paper 88-GT-70, 1988.

13. Frink, N.T., "Recent Progress Toward a Three-Dimensional Unstructured Navier-Stokes Flow Solver," AIAA-94-0061, Jan. 1992.

14. Mongia, H.C., "Application of CFD in Combustor Design Technology," AGARD CP-536, Fuel and Combustion Technology for Advance Aircraft Technology, 1993.

15. Sturgess G.J., et. al., "Design and development of a research combustor for lean blowout studies," ASME Paper 90-GT-143, 1990.

16. Lee, J., "An Application of Two-Equation Model of Turbulence to Three-Dimensional Chemically Reacting Flows," AIAA-95-0734, January 1995.

17. Chen K.H. and Shuen J.S., "Three-Dimensional Coupled Implicit Methods for Spray Combustion Flows at All Speeds," AIAA-94-3047, June 1994.

18. Hsu A.T., Tsai, Y-L, and Raju, M.S., "A PDF Approach for Compressible Turbulent Reacting Flows," AIAA-93-0087, January 1993.

19. J.R. DeBonis, "Full Navier-Stokes Analysis of a Two-Dimensional Mixer/Ejector Nozzle for Noise Suppression," AIAA-92-3570, July 1992.