NASA/TM—1998-208397

ICOMP–98–01
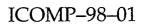
Institute for Computational Mechanics in Propulsion (ICOMP)

Twelfth Annual Report – 1997

Theo G. Keith, Jr., Karen Balog, and Louis A. Povinelli, Editors
Ohio Aerospace Institute, Cleveland, Ohio

National Aeronautics and
Space Administration

Lewis Research Center

June 1998

Available from

CONTENTS

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP)

TWELFTH ANNUAL REPORT

1997

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) was formed to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. ICOMP is operated by the Ohio Aerospace Institute (OAI) and funded via numerous cooperative agreements by the NASA Lewis Research Center in Cleveland, Ohio. This report describes the activities at ICOMP during 1997, the Institutes twelfth year of operation.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in September 1985. The overall purpose was to improve problem-solving capabilities in all aspects of computational mechanics relating to propulsion. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion.

The scope of the ICOMP program is to advance the understanding of aerospace propulsion physical phenomena and to improve computer simulation of aerospace propulsion systems and components. The specific areas of interest in computational research include: fluid mechanics for internal flows; CFD methods; turbulence modelling; and computational aeroacoustics.

This report summarizes the activities at ICOMP during 1997.

The following sections of this report provide lists of the resident and visiting researchers, their affiliations and educational backgrounds. Individual sections are provided which briefly describe reports of RESEARCH IN PROGRESS, the REPORTS AND ABSTRACTS published over the past year and the SEMINARS presented throughout the year. The agenda and overview of a very productive workshop held in 1997 is also given. The 2 day event entitled "Penn State Symposium" was held October 1-2, 1997.

THE ICOMP STAFF OF VISITING RESEARCHERS

The ICOMP research staff for 1997 is shown in Table I. A total of twenty-one researchers were in residence at ICOMP for periods varying from a few days to a year. The resident staff numbered nineteen while the visiting staff numbered two.

As usual, the resident researchers were very productive Table II provides a numerical summary of ICOMP during its twelve years of operation in terms of research staff size and technical output as measured by the numbers of seminars, reports and workshops.

Table I. - The ICOMP Research Staff-1997

A. Resident Staff.

Kumud Ajmani, Ph.D., Mechanical Engineering, Virginia Polytechnic Institute and State University, 1991. January, 1992–September, 1997.

Joongkee Chung, Ph.D., Mechanical Engineering, University of California, Berkeley, 1991. May, 1992–Present.

Datta Gaitonde, Ph.D., Mechanical and Aerospace Engineering, Rutgers University, 1989. September, 1995–December, 1997.

Duane R. Hixon, Ph.D., Aerospace Engineering, Georgia Institute of Technology, 1993. October, 1993–Present.

Bo-nan Jiang, Ph.D., Engineering Mechanics, University of Texas, Austin, 1986. October, 1987–Present.

Kai-Hsiung Kao, Ph.D., Aerospace Engineering Sciences, University of Colorado, 1989. November, 1992–September, 1997.

James M. Loellbach, Ph.D. expected 1995, Aeronautical and Astronautical Engineering, University of Illinois. May, 1992–Present.

Andrew T. Norris, Ph.D., Mechanical and Aerospace Engineering, Cornell University, 1993. June, 1993–Present.

Aamir Shabbir, Ph.D., Mechanical Engineering, State University of New York,Buffalo, 1987. June, 1991–Present.

Scott Sherer, M.S., Aeronautical and Astronautical Engineering, 1994. September, 1997–Present.

Shyue-Horng Shih, Ph.D., Aerospace Engineering, University of Cincinnati, 1993. June, 1993–Present.

Tsan-Hsing Shih, Ph.D., Aerospace Engineering, Cornell University, 1984. March, 1990–Present.

Erlendur Steinthorsson, Ph.D., Mechanical Engineering, Carnegie Mellon University, 1992. January, 1992–Present.

John Slater, Ph.D., Aerospace Engineering, Iowa State University, 1992. February, 1996–March, 1997.

Gerald Trummer, B.S., Michigan State University, 1982. September, 1995–Present.

Fu-Lin Tsung, Ph.D., Aerospace Engineering, Georgia Institute of Technology. March, 1993–Present.

Michael White, Ph.D. expected 1998, Mechanical Engineering, University of California, Davis. October, 1997–Present.

Shaye Yungster, Ph.D., Aeronautics and Astronautics, University of Washington, 1989. November, 1989–Present.

Ge-Cheng Zha, Ph.D., Mechanical Engineering, University of Montreal, 1994. July, 1997–Present.

B. Visiting Staff/Consultants.

Thomas Hagstrom, Ph.D., Applied Mathematics, California Institute of Technology, 1983. Associate Professor, Department of Mathematics and Statistics, University of New Mexico.

Eli Turkel, Ph.D., Applied Mathematics, New York University, 1970. Professor, Department of Mathematics, Tel Aviv University, Tel Aviv, Israel.

Table II. - ICOMP STATISTICS (1986 TO 1997)

	1986	1987	1988	1989	1990	1991	1992	1993	1994	1995	1996	1997
RESEARCHERS	23	43	50	46	47	49	58	64	50	30	33	21
SEMINARS	10	27	39	30	37	26	32	46	3	15	10	13
REPORTS	2	9	22	32	25	29	27	51	32	28	13	13
WORKSHOP/ LECT. SERIES	1	0	2	1	1	1	1	1	2	2	1	1
NO. OF PRESENTATIONS	7	0	21	14	15	21	15	33	40	23		

Research in Progress

RESEARCH IN PROGRESS

KUMUD AJMANI
Research Area: Development of Codes for Parallel Processing

The primary task for this year was performed in support of the "Trailblazer" project at NASA Lewis (under the auspices of the Lewis Hypersonics Program). The purpose of the task was to perform an extensive CFD study of the shock boundary-layer interaction between the engine-diverters and the primary body surfaces of the Trailblazer vehicle. Information gathered from this study would be used to determine the effectiveness of the diverters in preventing the boundary-layer coming off of the vehicle forebody from entering the main engines.

The first step in the CFD analysis involved the generation of individual, overlapping, three-dimensional grids around the surfaces of interest. The surface definitions provided by the Trailblazer team were processed through the ICEM-CFD program, in order to generate surface definitions which would be acceptable to the grid-generation code(s). Two grid-generation programs - GRIDGEN (for H-grids) and HYPGEN (for C-grids around conical surfaces) were used to generate a total of 6 volume-grids for the vehicle forebody, engine-pods and diverter surfaces.

The grid-generation stage was followed by the grid "assimilation" stage. The PEGSUS code was used to define the "holes" and "boundaries" for each grid. The information provided by the PEGSUS code was used as an input into the CHIMERA-based, grid-overset, upwind, finite-volume CFD code to perform the CFD simulations. The code uses the Baldwin-Lomax turbulence model in order to model turbulent regions of the flow.

Two sets of CFD calculations corresponding to Mach 2.5 and Mach 6.0 were performed. Three values of angle-of-attack (AOA) were used: -3, 0, and +3 degrees (for the purpose of a parametric study) for the Mach 2.5 case; the Mach 6.0 case was studied at 0 degrees AOA. A grid-refinement study was performed for the Mach 2.5, AOA = 3 degrees case, and the grid densities for the grid-converged solution were then maintained in all the other cases. This was done to ensure that all the CFD results obtained would be grid-independent.

Extensive post-processing of the results (particle-traces, contour plots) was performed for each test case. The results revealed the existence of a glancing side-wall boundary-layer interaction between the shock wave off of the engine-cones and the "diverted" boundary-layer between the pods. The CFD results indicated that the turning of the boundary-layer due to the pressure exerted by the engine-cone shock(s) might be sufficient enough that the "physical" diverters might be eliminated from the configuration, without any loss in vehicle performance. However, this phenomenon needs to be studied further, particularly since the elimination of the "physical" diverters would considerably simplify the overall design of the Trailblazer vehicle. Detailed CFD results of the computations performed in this task will be made available in an ICOMP report (to be published).

This work was done in collaboration and with the assistance of NASA colleagues Meng-S. Liou, Charles J. Trefny, Joe Roche, John Slater and Kai-H. Kao.

JOONGKEE CHUNG
Research Areas: Code Development for Unsteady Inlet Flows Using Parallel Processing, Iced Airfoils

A coordinated effort to incorporate new algorithms and boundary conditions developed in the past into the official version of NPARC (v3.0) was performed. Algorithm enhancement such as implicit subiteration was added for improved time accurate computations and multiblock time accurate runs were made possible as well as the introduction of dimensionality and performance monitoring for unsteady computations. Due to the importance of the compressor face and upstream flow disturbance time accurate boundary conditions for inlet research, an evaluation of existing boundary conditions and an effort to improve the Mach number based the boundary conditions was conducted and resulted in the addition of these boundary conditions and various time step management processes to the official version of NPARC. This modified version of NPARC is expected to continue to support controls-CFD inlet-related interdisciplinary research.

As a part of an effort for the development of an "Integrated Solution Process for Iced Airfoils" which combines a CFD code and an ice accretion code for accurate prediction of ice growth and performance degradation, a study on the effect of iced-geometry-smoothing was initiated. As a first step in this study, the degree of smoothing was defined by the number of control points for the given iced airfoil geometry. Then, reducing these number of control points in a systematical way provided various degrees of grid generation. This study will be continued by comparing CFD computed data such as pressure, lift, drag, flow separation, and wake flow patterns between the iced airfoils against any existing experimental data.

DATTA GAITONDE
Research Area: Electromagnetics and Fluid Dynamics

Previous efforts focused on developing tools for design of low observables were sustained. The final product was the maturation of a high-order accurate finite-volume based code to solve Maxwell's equations. One of the primary achievements was the development and implementation of efficient filtering techniques which enhance the robustness of high-order and optimized schemes without significant adverse impact on accuracy. This has eliminated the stability barrier which restrains the common use of high-order schemes for conservative wave propagation phenomena on curvilinear meshes[1,2,3].

A study was performed of crossing shock interactions under conditions of increasing interaction strength and asymmetry. In the first category, the observed computed topological bifurcations were correlated with the formation of various lines of coalescence and divergence evident in experimental and computed surface oil maps. The flow structure arising from asymmetric interactions was investigated with particular emphasis on i) vorticity dynamics, ii) shock-structure and iii) sidewall vortex loading[4].

Several efforts of the prior year were successfully published in archival journals[5-7]. The high-order algorithms developed for CEM have been implemented into the FDL3DI CFD code and are presently undergoing extensive testing. Preliminary results are highly encouraging.

References

[1.] Gaitonde, D., Shang, J.S. and Young, J.L., "Practical Aspects of High-order Accurate Finite-Volume Schemes for Electromagnetics", AIAA Paper 97-0363, 1997.

[2.] Gaitonde, D., Shang, J.S. "Optimized Compact-Difference-Based Finite-Volume Schemes for Linear Wave Phenomena", Accepted for publication, Journal of Computational Physics.

[3.] Gaitonde, D., "Optimized Schemes for Time-Domain Electromagnetics", Invited paper, 1997 URSI and IEEE/AP-S Meeting, Montreal, Canada, July 1997.

[4.] Gaitonde, D., Shang, J.S., Garrison, T.J., Zheltovodov, A.A. and Maksimov, A.I. "Evolution of the Separated Flowfield in a 3-D Shock Wave/Turbulent Boundary Layer Interaction", AIAA Paper 97-1837, June/July, 1997

[5.] Gaitonde, D. Edwards, J.R. and Shang, J.S, "The Structure of a Supersonic 3-D Cylinder/Offset-Flare Turbulent Interaction", Journal of Spacecraft and Rockets, Vol. 34, No. 3, 1997, pp 294-302.

[6.] Gaitonde, D. and Shang, J.S., "Skin-Friction Predictions in a Crossing-Shock Turbulent Interaction", Journal of Propulsion and Power, Vol. 13, No. 3, 1997, pp 342-348

[7.] Gaitonde, D. and Shang, J.S., "Shock Pattern of a Triple-Shock Turbulent Interation", AIAA Journal, Vol. 36, No. 1, 113, 1998.

THOMAS HAGSTROM
Research Areas: Algorithms for Boundary Layer Value Problems, Domain Decomposition

Research work during FY97 involved three efforts:

High-Order Radiation Boundary Conditions

The issue of boundary conditions at artificial boundaries is crucial to the accurate simulation of long time wave dynamics, and hence of great importance to the development of computational aeroacoustics and electromagnetics. The work is focussed on the development of sequences of conditions which in some sense converge, with increasing order, to the "exact" or "transparent" condition. Issues to be addressed include both the practical implementation of such conditions, their mathematical analysis (in particular expected convergence rates), and their extension to more complicated systems.

We have considered two distinct but related sequences of approximate conditions, applied to the wave equation, Maxwell's equations, and the linearized Euler equations of gas dynamics. In joint work with J. Goodrich of NASA-Lewis, we are considering the classical Padé approximants to the symbol of the exact condition while with S.I. Hariharan of ICOMP we are considering conditions derived from progressive wave expansions. Recent progress includes:

a) An obstacle to the application of the Padé conditions to non-periodic problems has been the derivation of appropriate compatibility conditions at corners where artificial boundaries intersect. We have developed a promising new approach inspired by the work of Vacus on related problems for the wave equation. The idea is to differentiate the Euler equations and approximate boundary conditions sufficiently often and to combine the differentiated equations to produce the compatibility relations. A progam has been written to do this symbolically, and numerical experiments are underway.

b) The Perfectly Matched Layer (PML) technique of Hu has been implemented and the results compared with those obtained through the use of the Padé conditions. We found that although the PML gave good results for moderate times, the numerical solution became unbounded for longer times. Subsequent analysis showed that the PML system is only weakly well-posed.

c) A compact, recursive formulation of high-order boundary conditions based on progressive wave expansions has been found for the wave equation, the Maxwell system, the convective wave equation and the Euler equations linearized about a uniform flow. Numerical experiments with the wave equation and two-dimensional linearized Euler equations are currently underway. We have also found that, in three space dimensions, the new conditions are exact for solutions described by a finite numer of spherical harmonics.

d) A formulation of high-order boundary conditions for these equations on cylindrical boundaries has also been discovered. It involves a combination of the Padé and progressive wave conditions discussed above. It should turn out to be ideally suited for simulations of three-dimensional jet aeroacoustics.

High-Order One-Step Schemes for Hyperbolic Systems

It has long been recognized that high-order schemes can offer significant advantages in efficiency for the long-time solution of wave propagation problems. Recently J. Goodrich has proposed a variety of one-step schemes for hyperbolic systems. Some may be viewed as natural high-order multidimensional extensions of the Lax-Wendroff method while others extend the grid data to include derivative approximations. With J. Goodrich and R. Dyson work began on the extension and theory of the methods, focussing in particular on their use with Cartesian mesh discretizations of complex geometries and their stability for variable coefficient problems.

Numerical Computation of Flame Speed and Structure

The accurate simulation of combustion phenomena is of great importance in the development of propulsion systems. Dr. Hagstrom, jointly with K. Radhakrishnan of ICOMP, have a long term project to create a combustion simulator combining high-order numerical methods and the capability to include complex reaction and diffusion physics. A first, but important, stage in this project is the production of a code to compute one-dimensional phenomena. One version of the code computes flame speed and structure, with an emphasis on robustness, i.e., relative insensitivity to initial approximations to temperature and species profiles. The technique combines time-stepping with an automatic determination of a stabilizing inflow rate and convergence acceleration based on inexact Newton iterations. In the past year they have added a number of new features to the code and carried out more tests. In collaboration with R. Zhou, a student at the University of New Mexico, a preliminary version of a time-dependent code has been produced. Particular areas of progress include:

a) Inclusion of Newton iterations with inexact Jacobian to refine the estimates. With this enhancement, they have been able to achieve convergence of the flame speed to more than four digits for all cases so far considered.

b) Extensive testing with a variety of fuels (methane, propane, acetylene, methanol) over a variety of temperatures and pressures.

c) Implementation of a time-accurate code and its application to studies of methane ignition

DUANE R. HIXON
Research Area: Aeroacoustics

Work continued on the development and application of high-accuracy numerical schemes for the computational prediction of supersonic jet noise. The high-accuracy MacCormack-type schemes previously developed and applied[1,2] were further tested against benchmark problems to determine their linear and nonlinear performance[3]. A new class of compact MacCormack-type schemes were developed in collaboration with Dr. Eli Turkel, and showed improved performance[4].

The work on compact schemes led to a new and more efficient method of implementing high-order compact schemes using the MacCormack splitting method[5]. The sixth-order compact scheme derived in this work has been implemented in a 3-D multiblock generalized curvilinear coordinate Large Eddy Simulation Code which will replace the current code. Initial results show a large improvement in memory and CPU time compared to the 2-4 scheme.

The accuracy of generalized curvilinear coordinate transformations were investigated, and it was found that the formulation of the equations had a large impact on accuracy[6]. Results of this work were used when coding the new Large Eddy Simulation Code.

Work will continue in two directions: application and improvement of the curvilinear code. The first 3-D LES nozzle + jet application will be run in 1998, and improvements in the accuracy, speed, and robustness of the compact scheme will be investigated.

References

[1.] Hixon, R. "On Increasing the Accuracy of MacCormack Schemes for Aeroacoustics Applications", AIAA Paper 97-1586, March 1997.

[2.] Hixon, R., Shih, S.-H., and Mankbadi, R. R., "Effect of Coannular Flow on Linearized Euler Equation Predictions of Jet Noise", AIAA Paper 97-0284, Jan. 1997.

[3.] Hixon, R., "Evaluation of a High-Accuracy MacCormack-type Scheme Using Benchmark Problems", accepted for publication in the Journal of Computational Acoustics, 6/97.

[4.] Hixon, R. and Turkel, E., "High-Accuracy Compact MacCormack-Type Schemes for Computational Aeroacoustics", AIAA Paper 98-0365, Jan. 1998.

[5.] Hixon, R., "A New Class of Compact Schemes", AIAA Paper 98-0367, Jan. 1998.

[6.] Hixon, R., Shih, S.-H., Dong, T., and Mankbadi, R. R., "Evaluation of Generalized Curvilinear Coordinate Transformations Applied to High-Accuracy Finite-Difference Scheme", AIAA Paper 98-0370, Jan. 1998.

BO-NAN JIANG
Research Area: Flow Applications of the Least-Squares Finite Element Method

The main thrust of the effort has been towards the development, analysis and implementation of the least-squares finite element method (LSFEM) for fluid dynamics and electromagnetics applications. In the past year, there were four major accomplishments:

1. Dr. Jiang finished a monograph[1]. This is the first book devoted to the Least-Squares Finite Element Method (LSFEM) which is a simple, efficient and robust technique for the numerical solution of partial differential equations. The book demonstrates that the LSFEM can solve a broad range of problems in fluid dynamics and electromagnetics with only one mathematical/computational formulation. The book shows that commonly adopted special treatments in computational fluid dynamics and computational electromagnetics, such as upwinding, numerical dissipation, staggered grid, non-equal-order elements, operator splitting and preconditioning, edge elements, vector potential, etc. are unnecessary.

2. Dr. Jiang studied various theoretical aspects of the LSFEM. It was found that the analysis of the least-squares method for most (not only limited to elliptic) partial differential equations in engineering and physics can be based on the bounded inverse theorem. This establishes a unified mathematical framework of the least-squares method[2].

3. The widely used finite difference and finite volume algorithms for the solution of time-dependent electromagnetic problems solve only two Maxwell equations (in curl form) and ignore the divergence equations. It has been shown in previous work that the omission of the divergence equations in the numerical procedure leads to non-physical solutions. Joint work with Dr. Jie Wu, demonstrated that the most popular FD-TD method may introduce substantial errors in the modeling of microwave integrated circuit components and that the node-based time domain LSFEM can be used successfully with divergence equations and produces no erroneous solutions[3,4].

4. Joint work was performed with Drs. S.-T. Yu, J.Wu and J.C.Duh, on numerical simulations of three-dimensional Marangoni-Benard convection using the LSFEM. This is the first numerical solution for small containers and the results compared favorably with existing experimental data[5].

References

[1.] Jiang, B.N. (1998): *The Least-Squares Finite Element Method, Theory and Applications in Computational Fluid Dynamics and Electromagnetics*, Springer Series in Computational Physics, Springer-Verlag, Heidelberg (in press).

[2.] Jiang, B.N. (1998): "On the Least-Squares Method", Comput.Meth.Appl.Mech. Engrg. (in press).

[3.] Jiang, B.N. (1997): "The True Origin of Spurious Solutions and Their Avoidance by the Least-Squares Finite Element Method", in *Computational Electromagnetics and Its Applications*, eds. Campbell, T.G., Nicolaides, R.A., Salas, M.D., ICASE/LaRC Interdisciplinary Series in Science and Engineering, Vol.5, Kluwer Academic, Dordrecht, Netherlands, pp. 155-184.

[4.] Jiang, B.N. and J. Wu (1997): *"Importance of Divergence Equations in Computational Electromagnetics"* SIAM 45th Annual Meeting, July 12-18, Stanford University.

[5.] Yu, S.T., Jiang, B.N., Wu, J., Duh, J.C. (1998): *"Three-Dimensional Simulations of Marangoni-B'enard Convection in Small Containers by the Least-Squares Finite Element Method"*, Comput. Meth. Appl. Mech. Engrg. (in press).

KAI-HSIUNG KAO
Research Area: 3D Compressible Finite Volume Navier-Stokes Flow Solver

Dr. Kao's research was directed towards the project for a Coupled Aero/Thermal/Structure Analysis (CATS) of the secondary flow system in an air-breathing engine. The overall research aim of this work was to develop and demonstrate the CFD capability for predicting flowfields inside the rotating drum within the H.P. compressor of the Pratt & Whitney 4000 engine. An accurate numerical algorithm (AUSM) combined with an efficient and flexible grid generation technique (Chimera or DRAGON) was used. A turbulent model developed by CMOTT has been implemented and validated with a benchmark problem.

The following describes the accomplishments in FY97:

1. Code validation for drum-disk flows
    — Detailed comparison with measured values of velocity components and pressure.

2. Implement DRAGON grid technique
    — Investigate its effectiveness in terms of solution accuracy and grid capability

3. Conjugate heat transfer computation of the drum/rotor and shaft system
    — Investigate strategies for efficient and accurate coupling of flow and heat conduction codes, including data communication

4. Implement and validate CMOTT κ-ε turbulence model
    — Implementation has been completed
    — Validation has worked out very well. Good agreement with experimental results for supersonic compression corner case has been obtained.

JAMES LOELLBACH
Research Area: 3D Structured Grid Generation Codes for Turbomachinery

Mr. Loellbach's research tasks during the past year were mainly in the area of computational grid generation in support of CFD analyses of turbomachinery components. This work was performed in cooperation with Dr. Chunill Hah of NASA Lewis Research Center and Fu-Lin Tsung of ICOMP. In addition to the grid generation work, he obtained a numerical simulation for the flow through a centrifugal gas compressor using an unstructured Navier-Stokes solver.

The work with Dr. Hah involves many different turbomachinery component analyses. These analyses were performed for NASA projects or for industrial applications. The work includes both centrifugal and axial machines, single and multiple blade rows, and steady and unsteady analyses. Over the past five years, a set of structured grid generation codes were deveoped by Mr. Loellbach that allow grids to be obtained fairly quickly for the large majority of configurations we encounter. These codes do not comprise a generalized grid generation package; they are noninteractive codes specifically designed for turbomachinery blade row geometries. But because of this limited scope, the codes are small, fast, and portable, and they can be run in the batch mode on small workstations.

During the past year, these programs were used to generate computational grids for a wide variety of configurations. In particular, Mr. Loellbach modified the codes or wrote supplementary codes to improve our grid generation capabilities for multiple blade row configurations. This involves generating separate grids for each blade row, and then making them match and overlap by a few grid points at their common interface so that fluid properties are communicated across the interface. Unsteady rotor/stator analyses were performed for an axial turbine, a centrifugal compressor, and a centrifugal pump. Steady-state single-blade-row analyses were made for a study of blade sweep in transonic compressors.

Mr. Loellbach also cooperated with Fu-Lin Tsung on the application of an unstructured Navier-Stokes solver for turbomachinery flow simulations. In particular, the unstructured solver was used to analyze the flow through a centrifugal compressor impeller.

ANDREW T. NORRIS
Research Area: Aerothermochemistry

Motivation and Objectives

The objective of this work is the development, implementation and validation of thermochemical models for use in combustion codes. The principal goal in this work was to achieve significant reductions in the CPU time required for reacting flow calculations, and to accomplish this without sacrificing accuracy. In addition, work was started on attempting to find a new method of calculating chemical reactions, based on thermodynamic considerations, rather than kinetic schemes. This work is being performed with John Marek of the Combustion Branch at NASA Lewis Research Center.

Work Performed

(1) Manifold Reaction Schemes.

During the year, further development of the Intrinsic Low-Dimensional Manifold code was undertaken. This code takes a full reaction mechanism, and automatically simplifies it into a reduced mechanism.

Work performed consisted of numerical enhancement of the ILDM code, to provide a more friendly user interface. The proposed inclusion of the ILDM code in the LSENS code was postponed until the LSENS code has finished a major upgrade.

In addition the ILDM code was installed in the National Combustion Code, in both the scalar PDF and the conventional assumed PDF combustion modules. A separate interface was written for the assumed PDF calculations, and included in the ILDM package. Preliminary results for the ILDM scheme show a 75% reduction in CPU time compared to the 12 species chemistry package.

Validation tests showed the accuracy of the ILDM scheme, parameterized by two scalars, to be of the same order of accuracy as a reduced mechanism containing 12 species and 10 rate equations for PSR calculations. This work was reported at the 1997 Joint Propulsion Conference.

(2) Neural Network Reaction Storage.

In cooperation with ABtech corporation, an investigation was performed into the feasability of using neural network techniques to store the reaction tables. Using this method we hope to eliminate the two big problems of table storage: Memory limitations. The memory problems are based on the need to obtain a good resolution for the stored function, which requires many table entries. For example a table may require on order of 100 entries to resolve a particular component. If the table is just a function of two parameters, this gives a 10,000 element table. But if a table is to be the function of five parameters, this corresponds to a 10,000,000,000 element table, or for one stored component, 40Gb of memory.

By using Neural network approaches, we hope to replace the whole table structure by a system of polynomial functions. This will eliminate the restriction on the number of parameters to describe the reaction, as the size of the polynomials is negligable. The speed of evaluating the polynomials will be more than the table look-up, but should be still significantly less than the conventional mechanisms.

Initial work has shown that a good agreement can be obtained between the neural network results and the table. Currently the error estimates have the maximum error at about 5% compared to the table predictions.

(3) NOx Post-Processor

The NOx Post processor, based on the ILDM reduced reaction scheme, was also worked on. Its capabilities were upgraded to include Jet A as a fuel and is currently being used by Pratt and Whitney for pollutant prediction in their design codes. This method was reported at the 1997 Joint Propulsion Conference.

(4) Thermodynamic Approach to Reduced Chemical Kinetics.

During the year, some advances in the thermodynamic approach to simplifying chemical kinetics was made.

One can consider the composition of a mixture to be a point in N-dimensional scalar space, with N being the number of species. Thus if you change the composition due to reaction, this appears as the motion of the point to another location on the scalar space. Now at each point in the scalar space, there is a chemical potential, also known as the Gibbs free energy. There is also one point in the scalar space where this is minimum; the equilibrium point. The application of chemical kinetic mechanisms results in the composition to move toward this point.

The thermodynamic approach is based on the premise that if we know the species in the reaction, we can determine the Gibbs free energy field for the entire scalar space. As all chemical reactions consist of a motion from a high chemical potential to the minimum, the reaction rate kinetics may be replaced by a simple caternary-type argument. That is, why not just slide "downhill" to the minimum and use the resulting path as the kinetic scheme?

The main advantage in doing this is that the need for a complex "full" mechanism can be avoided. These "full" mechanism consist of hundreds of rate expressions, each with at least three constants that have to be determined. The creation of these mechanims is very labor intensive, and the availablity is limited.

Work to date has been mainly in the area of setting up the nessesary numerical tools to analyse the resulting multi-dimensional spaces, and preliminary work on finding the potential and potential gradient.

Continuing Work

During the next year, the following projects are proposed:

(1)   Validation of the ILDM scheme in the NCC code.

(2)   Validation of the scalar PDF module in the NCC code.

(3)   Testing and development of Neural Network approach to storing reduced mechanisms.

(4)   Further work on the thermodynamic approach to simplification of chemical kinetic expressions.

In addition, details of the ILDM code implementation into the NCC code and some preliminary findings of the thermodynamic approach to reaction kinetics will be reported at the Joint Propulsion Conference in July 1998.

AAMIR SHABBIR
Research Area: Turbulence Modelling

Due to the three-dimensional nature of turbo-machinery flows the boundary layers on the hub and casing, as well as those on the blade surfaces, have to be resolved in addition to the flow details in the blade passage. For this reason, at present, the computation for multiple blade row machines, is done with the use of wall functions, rather than integration of equations of to solid surfaces.

A study was conducted to examine the feasibility of integrating the equations to solid surfaces. The Reynolds averaged three-dimensional Navier-Stokes equations that were solved using a finite volume formulation. The turbulent eddy viscosity in the mean flow equations was obtained from Chien's low Reynolds number turbulence model. The test case chosen was the NASA's transonic Rotor 37, which has been tested extensively by the turbo-machinery community to assess CFD tools. The details of the experiment can be found in Suder (199⁻) and those of the CFD method in Shabbir et al. (1996).

The grid used in the study employed about 0.9 million grid points. It was found that even then the y^+ values for the grid cells along the blade surfaces were an order of magnitude larger than what is required by the turbulence model for resolving the turbulence behavior ,i e., y^+ of unity). Therefore, it was concluded that at present the integration of equations of motion to solid surfaces is still not practical

Some effort was also spent in enhancing the existing turbulence model features in the APNASA code, which is used for CFD simulations of multiple blade row turbo-machines. A simulation of the NASA Lewis Large Scale Axial Compressor (nine blade rows) was also conducted using APNASA and results are being compared with the experimental data taken by Wellborn and Okiishi (199⁻)

References

Suder, K., (1996), Experimental Investigation of the Flow Field in a Transonic, Axial Flow Compressor With Respect to the Development of Blockage and Loss. NASA TM 107310.

Shabbir, A., J. Zhu, and M. L. Celestina (1996), Assessment of Three Turbulence Models In a Compressor Rotor. ASME Paper No. 96-GT-198.

Wellborn, S.W. , and T.H. Okiishi 1996, Effects of Shrouded Stator Cavity Flows on Multistage Axial Compressor Aerodynamic Performance, NASA CR 198536.

SCOTT SHERER
Research Area: Simulations of Electromagnetic Phenomenon

Dr. Sherer conducts his research at Wright-Patterson AFB. His primary responsibility is to perform research and computational simulations of electromagnetic phenomenon. This work focuses on three main areas of interest to the Air Force: 1) Fundamental modelling of the physical phenomena associated with electromagnetic scattering, 2) Solution algorithm development and validation, and 3) High-performance computing. In support of these areas, Dr. Sherer has thus far developed high-quality, structured, multi-block volumetric grids for various benchmark geometries as well as for more complex missile and aircraft geometries. He has also performed electromagnetic scattering simulations on geometries of interest to the Air Force using in-house Computational Electromagnetic (CEM) codes in order to accurately predict electromagnetic signatures. Currently he is performing in-depth validation studies of a higher-order, parallel-processing CEM computer code developed by an individual in the CFD Branch (AFRL/VAAC) and scheduled to be released to DoD users under the Common High Power Performance Computing Software Support Initiative (CHSSI) project. This work should be presented at next year's AIAA Aerospace Sciences Meeting. Dr. Sherer is also supporting the DoD Challenge project within the CEM group of AFRL/VAAC by performing grid generation tasks on the aircraft whose radar cross-section is being predicted.

SHYUE-HORNG SHIH
Research Area: Direct Prediction of Supersonic Jet Noise

Accurate prediction of jet noise generation and propagation is very important in developing advanced aircraft engines that will pass current and future noise regulations.

In supersonic jets, two major sources of noise are present: large-scale instabilities and small-scale turbulent eddies. The Large Eddy Simulation (LES) is currently the most promising tool for the prediction of supersonic jet noise, since it simultaneously calculates both the mean flow development and the associated noise generation and propagation. The existing LES code was applied to the test case of a Mach 1.4 heated round jet in a 400 ft/s uniform free stream. The full three-dimensional calculation was carried out to compute the unsteady flow characteristics of the jet plume. The jet was excited with disturbances at the jet plume inflow. Due to the lack of experimental data on frequency spectra at the nozzle exit, a single frequency and multiple frequencies excitations were imposed at the jet plume inflow. The jet shear layer mixing is enhanced compared to the axisymmetric disturbance case. The unsteady near field solutions is being collected to be applied to the Kirchhoff method to obtain far field noise.

The non-linear effects in jets could play important roles in noise generation mechanism. To reduce the computational costs for the LES method, an alternative approach has also been explored. Based on the LES concept, the fluid motion is split into three kinds of motion: a time-averaged motion, a large scale wavelike structure, and a fine scale random turbulence. Non-linear disturbance equations, which are derived from the Navier-Stokes equations, are solved with the prescribed mean flow. The large scale structures are computed directly while the effects of fine scale turbulence are modeled. The initial test of this approach on axisymmetric case is promising. Further three dimensional calculation is underway.

TSAN-HSING SHIH
Research Area: Turbulence Modelling

Objective

Aircraft engine combustors generally involve turbulent swirling flows in order to enhance fuel-air mixing and flame stabilization. It has long been recognized that eddy viscosity turbulence models are unable to appropriately model swirling flows. Therefore, it has been suggested that, for the modeling of these flows, a second order closure scheme should be considered because of its ability in the modeling of rotational and curvature effects. However, this scheme will require solution of many complicated second moment transport equations (six Reynolds stresses plus other scalar fluxes and variances), which is a difficult task for any CFD implementation. Also, this scheme will require a large amount of computer resource for a general combustor swirling flow.

This research is devoted to the development of a cubic Reynolds stress-strain model for turbulent swirling flows, and was inspired by the work of Launder's group at UMIST. Using this type of model, one only needs to solve two turbulence equations, one for the turbulent kinetic energy κ and the other for the dissipation rate ϵ.

14

Approach

The model developed in this research is based on a general Reynolds stress-strain relationship which is an explicit expression for the Reynolds stresses in terms of a tensorial polynomial of mean velocity gradients. It is derived from a generalized Cayley-Hamilton relation. This general formulation contains terms up to the sixth power of the mean velocity gradient and has eleven undetermined coefficients. Obviously, for any practical application, we need to truncate this polynomial. Shih, Zhu and Lumley (1995) suggested a quadratic formulation and determined the three relevant coefficients by using the realizability constraints of Reynolds stresses and a result from rapid distortion theory analysis. This quadratic model works quite successfully for many complex flows including flows with separation. However, our recent calculations of swirling flows show that the swirl velocity is not appropriately predicted, which verifies the finding of Launder's group at UMIST. Launder (1995) pointed out that "the weaknesses of the linear eddy viscosity model can not be rectified by introducing just quadratic terms to the stress-strain relation."

In this research, we retain the cubic terms from a general Reynolds stress-stain formulation and determine the coefficients by using a similar method used by Shih et al (1995) in deriving their quadratic model. The measured data from rotating pipe flows (Imao, Itoh and Harada, 1996) are used for proposing some numerical values of model constants. Modeled κ-ε equations are used together with the cubic model for mean flow calculations. The first test flow is a fully developed pipe flow rotating about its own axial axis with various rotation rates. The second test flow is a more complex flow with swirl and recirculation (Roback and Johnson, 1983). These two flows have detailed experimental data on mean velocity components which are used for comparisons with computational results from our turbulence models.

The detailed analyses and related references are reported in NASA TM113112.

Results

Our study shows that nonlinear cubic Reynolds stress models with modeled κ-ε equations seem to have the potential to simulate turbulent swirling flows encountered in aircraft engine combustors. The model proposed in this research appears simple and numerically robust for CFD applications in which the aircraft engine industry is particularly interested. However, further evaluations against other flows are needed in order to determine the flow range of the model's validity and to seek possible further improvements.

The proposed cubic Reynolds stress model can be combined with existing κ-ε model equations, yet the best combination needs further investigation and evaluation.

The proposed cubic model appears the simplest among other cubic or higher order models; however it requires about 15% more CPU time than does a linear κ-ε eddy viscosity model for a general 2D axisymmetric swirling flow. We expect that if a higher order model (e.g., fourth or fifth order) is used, then the CPU time for calculating Reynolds stresses will significantly increase and the model may become costlier for the calculation of a general 3D swirling flow.

Reference

[1.] Modeling of Turbulent Swirling Flows. NASA TM113112, ICOMP-97-08, 1997.

JOHN SLATER
Research Area: Research Efforts in Development of NPARC 2D/3D CFD Codes

Main Objective

The objective of the research was to develop a capability in the NPARC computational fluid dynamics (CFD) code to efficiently solve for unsteady airflows with moving geometry and grids. The application of interest was the unsteady flow in a high-speed aircraft inlet operating at the supercritical condition in which a terminal shock resides within the diffuser.

Background

Version 3.0 of the NPARC code solves the compressible Navier-Stokes equations on a multi-block, structured grid using finite-difference numerical techniques. It was developed primarily to solve for steady airflows. The analysis of a high-speed aircraft inlet operating at the supercritical condition requires understanding the sensitivity of the terminal shock to unsteady flow perturbations. In severe cases, the inlet may unstart, which involves the terminal shock being expelled from the inlet with a drastic decrease in aircraft performance. The inlet is restarted by varying the geometry of the inlet. For an axisymmetric inlet, the centerbody is collapsed slightly and translated forward. The analysis of this unstart/restart process using NPARC required improvements to the capability to solve unsteady flows and implementation of a moving grid capability. The result of this development effort was version 3.1 of the NPARC code.

Approach

Improving the capability to solve for unsteady flows involved implementing a Newton iterative method into the implicit method of NPARC. This resulted in a nominally second-order time accurate method for the implicit time integration. Other modifications were performed to allow the specification of time parameters and control of the time step size.

The modifications for the moving grid capability assumed a moderate level of grid motion associated with the motion of segments of the boundary grid relative to the rest of the grid of the block. This motion may be a rigid-body translation and/or rotation about a point or a deformation of the segment according to a coded relation. The remainder of the grid of the block deforms to accommodate the boundary motion. This requires that some regions of the grid be regenerated at each time step. Efficiency in the grid regeneration process is obtained by limiting the regeneration to only those regions in which there is grid motion. Thus, the grid becomes a computed function of time. For three-dimensional flow domains, the grid is assumed to be "quasi-2d" or axisymmetric in which the grid consists of planar grids with respect to the "l"-coordinate.

The flow equations and boundary conditions were expressed in an absolute frame of reference with the grid motion accounted for through the grid velocities. The velocity of each grid point was calculated from a time difference of the grids at two consecutive time levels.

Results

Several test cases involving moving grids were developed and computed to demonstrate the application and accuracy of the moving grid capability. A few simple test cases involving supersonic flow over a stationary wedge, a flying wedge, and a rotating flap on a flat plate showed good comparison with steady-state oblique shock theory. A test case involving the collapse of an axisymmetric bump in an annular duct showed good comparison with unsteady experiment data. The unstart/restart operation of the NASA variable diameter centerbody (VDC) inlet, which involved the translation and collapse of the centerbody, was demonstrated to show qualitative agreement with experimentally observed behavior.

ERLENDUR STEINTHORSSON
Research Area: Code Development for Flows in Complex Geometries such as Turbine Blade Coolant Passages

During the last year several tasks were performed. The tasks included, grid generation, flow solver development, enhancement of a block-merging utility (see description below) and report writing. The main task performed in last year's work was the enhancement of the capabilities of the TRAF3D.MB code, a flow solver used by several researchers at NASA Lewis Research Center for prediction of heat transfer in turbomachinery. The TRAF3D.MB code is a multi-block flow solver that utilizes an explicit Runge-Kutta time stepping scheme, coupled with some convergence acceleration techniques, to march solutions to steady state. In the multi-block scheme used in the code, each block of the multi-block grid is processed independently in every iteration, communicating the solution between blocks only at the end of an iteration. In an effort to test the effects of frequency of block-to-block communication on convergence rate, the code was modified to allow run-time control over communication between blocks, thereby allowing the user to specify communication in every stage of the Runge-Kutta time step, if so desired. At the same time, the code was modified to allow full run-time control over the Runge-Kutta scheme including the frequency of evaluation of artificial and viscous dissipation fluxes. Finally, all changes in the code were made with future parallelization of the code in mind.

The second main task over the past year was to develop useful multi-block grid topologies for turbine blades with tip clearance. For high-fidelity simulations of turbomachinery flows, we rely on multi-block grids generated by a commercial grid generator called GridPro. With GridPro, the grid generation task involves inventing grid-topologies that produce high quality grid systems. In general, this is not a great challenge. For turbine blades with tip clearance, however, this was a challenge as the grid needed to wrap around the blade and tip, be pitch-wise periodic, have smoothly varying grid spacing and be nearly orthogonal everywhere. This task was successfully completed. The topologies that were invented have been used in several simulations by Ali A. Ameri and have been found to serve well. The advantage of the grid generation approach used here is that once topologies have been invented that work for a specific class of geometries, the grid generation task is relatively straight forward and grids can be generated with minimum human effort. The multiblock grids themselves, due to the local structure and high quality of the grids results in less expensive and more accurate flow simulations than when lesser quality grids are used.

One result of using multiblock grid systems is that the grids typically contain a large number of small blocks. For performance reasons, it is desirable to merge the small blocks to form larger ones. This reduces memory overhead, improves vectorization on super computers and makes the grid easier to work with. In cooperation with Dr. David L. Rigby, we have developed a package for merging grid blocks automatically. During the year we worked on overcoming shortcomings in the package. At the present time, we have identified an algorithm that eliminates most of the shortcomings that were found. This new algorithm has not yet been implemented as the priority of the project was lowered.

The final task in 1997 was to write a detailed report on the TRAF3D.MB code. Due to the unanticipated time spent on grid generation and the block merging, the report was not completed as planned. Five chapters out of the expected 10 chapters have been completed. The report will be completed this year. The report will include chapters and sections to be written by Dr. Rigby and Dr. Ameri.

The first task during this year's work was to parallelize the TRAF3D.MB code on CRAY C90 computers (the eagle, von Neumann and Newton computers at NAS) and port the code to SGI computers (Turing at NAS). This task has been completed on time (about 150 hours of effort). In addition, the code can run in parallel on SGI computers as long as single precision calculations are sufficient. Execution in parallel using double precision on SGI computers awaits improvements in the compiler on the SGI computer or a somewhat larger effort to parallelize the code by employing message passing.

GERALD TRUMMER
Area: System Administration

Upgraded 2D plotting and 3D visualization software on the Aeromechanics Division's network of workstations, including XYPLOT, TECPLOT, FAST, FIELDVIEW. Integrated new workstations into the local network including visualization software installation, operating system upgrades and modifications. Supplied computer system support to the signature technology office as required. Created movies visualizing fluid flow and electromagnetic data, including time-accurate simulation of spiral vortex breakdown.

FU-LIN TSUNG
Research Area: Development of 3D Structured/Unstructured Hybrid Navier-Stokes Solver for Turbomachinery

The goal for the present research is to develop, validate, and apply a 3-D, Navier-Stokes, unstructured-grid solver for turbomachinery applications. In 1997, the main focus was on the validation phase. Cases tested include a shock tube for temporal accuracy, a backward-facing step, a turbine stator, and a turbine stage for spatial accuracy. Comparison between the data with experiment and other CFD codes shows the present unstructured solver is as capable as most structured solvers in predicting complex flow fields. However, to achieve a same level of resolution, generally the unstructured solver requires more cells. One contributing factor is that at this time, it is difficult to generate highly stretched, pencil shaped (three long edges sharing a vertex), tetrahedral cells. The presently used grid generator with advancing layer is able to generate highly stretched cells in the viscous layer. However, these cells are generally pancake shaped (three long edges sharing a face). In the backward-facing step case, the region near the step where the cells are extremely small on a 2D plane, many tetrahedral cells are needed in the transverse direction since pencil shaped tetrahedral cells are difficult to generate. Another factor is that the first point off the viscous surface (dn_1) is a fixed number through out the domain. This increases the number of cells since the most restricted dn1 must be used everywhere. Once these two limitations are removed, the number of unstructured cells can be reduced or the same number of cells can be better optimized for their distribution in the computational domain and thus enhance the unstructured grid's flexibility.

ELI TURKEL
Research Area: Preconditioning Applied to Turbomachinery Flow Simulation

One of the major difficulties with solving steady state viscous problems is the large amounts of computer time required to achieve a steady state solution. This is especially problematical for low Mach number nonviscous flows due to the stiffness of the Euler equations at low Mach numbers. In addition numerous researchers have found that the accuracy of the numerical approximation deteriorates as the Mach number decreases to zero.

To resolve this difficulty we introduce a preconditioning matrix to equalize the eigenvalues of the modified system. It can also be shown that this also improves the accuracy of the numerical solution as the Mach number goes to zero. We are implementing this approach for turbomachinery problems with both rotor and stator components. In particular, we are concentrating on the LSAC and LSRR configurations.

MICHAEL WHITE
Research Area: High-order, Compact, Differencing, Unsteady flows

Working with high-order compact finite-differences for use in Electromagnetics and unsteady Fluids. Specifically, looking at radiation boundary conditions and the use of filtering for stability and accuracy for high-order compact finite difference schemes. For the filtering, explicit Shapiro type filters as used by Kennedy and Carpenter [App. Num. Math., 14, p.397-433 (1994)] are considered as to whether to filter the variables themselves or the incremental changes in the variables. This work complements that of Dr. Visbal and Dr. Gaitonde in this area.

SHAYE YUNGSTER
Research Area: High Speed Combustion and Detonation Waves

The research work for the 1997 focused on three different areas:

(1)   CFD Analysis of Rocket-Based Combined Cycle (RBCC) propulsion system

The goal of developing efficient single-stage to orbit reusable vehicles has brought renewed interest in Rocket-Based Combined-Cycle (RBCC) engines. For space applications, a typical RBCC engine will operate in four modes; 1) ejector-ramjet, 2) ramjet, 3) scramjet, and 4) all-rocket. While a considerable amount of research has been aimed at ramjet and scramjet systems, only a few studies have been directed to the ejector-ramjet and all-rocket propulsion modes. The objective of this work is to conduct computational studies of these two RBCC modes.

All-Rocket Propulsion Mode

The performance of a RBCC engine operating in the all-rocket mode (used for final ascent to orbit) was examined using the Navier-Stokes solver NPARC v3.0. This code was chosen because it has gone through extensive validation, has several turbulence models and is efficient in terms of CPU time required to obtain converged solutions. This code solves the Reynolds-averaged Navier-Stokes equations for a perfect gas using the implicit Beam and Warming algorithm. The equations are discretized using central differences with Jameson's artificial dissipation added for stability and to prevent oscillations around shock waves. The solution is efficiently obtained using Pulliam's pentadiagonal transformation. Two turbulence models were chosen for this study: 1) Chien's two-equation κ-ε model, and 2) Spalart-Allmaras one-equation model.

The CFD simulations were performed within the context of statistical Design of Experiments (DoE). This approach was necessary due to the large number of parameters (6) examined. A full parametric study of these six variables (each at three levels for example) would necessitate an enormous computational effort. With the DoE approach only 36 separate CFD solutions were required.

From the 36 cases run (on axisymmetric configurations), a response surface model was created using the computed Isp values integrated from the CFD results. Grid independence and the influence of turbulence modeling were analyzed, and careful attention was given to develop a convergence criterion, and to evaluate the effects of artificial dissipation on the solutions.

The results of the statistical regression analysis revealed the effects that each of the different parameters, and their interactions, had on Isp performance. Numerical flow visualization was used extensively to further understand the physical phenomena associated with the all-rocket propulsion mode. The results of this work, which was performed in collaboration with C. J. Steffen, T. D. Smith and D. J. Keller, were presented in AIAA paper 98-0954, AIAA paper 98-1612, and at the PERC 9th Annual Symposium, OAI, Oct. 1-2, 1997.

Ejector-Ramjet Propulsion Mode

In the ejector-ramjet operation mode, the fuel-rich rocket exhaust is mixed and burned with air captured by the inlet. This mode of operation typically covers the Mach number range from take-off to approximately Mach 3. The length required for complete mixing is currently unknown, and is of great interest due to the weight of the actively cooled mixing duct. The sensitivity of performance to the degree of mixing achieved within a given duct is also of great interest. The objectives of this work are:

1. to determine the length required for complete mixing of the air and rocket streams;

2. to determine the sensitivity of performance to the degree of mixing achieved within a given duct length;

3. to determine the validity of assumptions used in cycle analysis codes; and

4. to provide pre-test predictions and design guidance for the planned experiments.

An axisymmetric configuration that closely models the conditions in the RBCC reference vehicle currently being studied at NASA Lewis Research Center (known as "Trailblazer") is being analyzed with a CFD code developed at ICOMP. This code solves the time-dependent Navier-Stokes equations with finite-rate chemistry and real gas effects. It include a generalized chemistry capability, various options for turbulence models (some under development), and steady-state or time accurate marching algorithms. Calculations are being performed at various flight conditions and for different geometric configurations.

(2)   Development of Global RBCC Engine Cycle Code

In addition to experimental and CFD efforts, research work at the NASA Lewis Research Center is also focusing on developing a global RBCC engine cycle code for performance analysis of the various propulsion modes. As part of this effort, a computer program for analyzing the combustor component of the engine was developed using a one-dimensional approach based on the solution of the continuity, momentum, and energy conservation equations applied to a control volume around the combustor. The analysis assumes that the flow is in chemical equilibrium at the exit of the combustor. (The equilibrium computations are based on the CSDTTP chemical equilibrium program). Combustion efficiency, and viscous and heat transfer effects can be included.
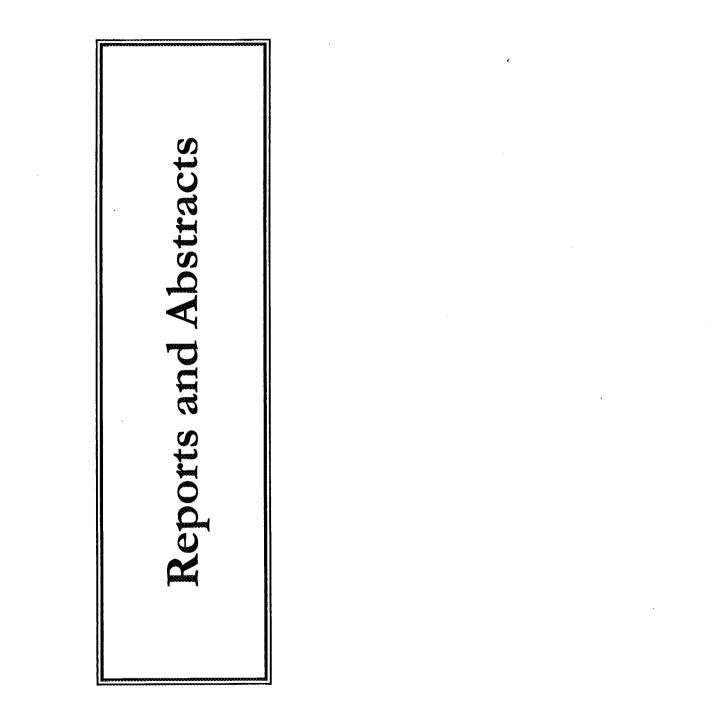
(3)   Detonation Wave Modeling

The development of detonation wave-based propulsion system concepts such as the ram accel- erator, oblique detonation wave engine and pulse detonation engine has renewed interest in hyper- sonic combustion. The goal of this work, which is carried out in collaboration with K. Radhakrishnan, is to both continue establishing validity of the CFD code we have developed, by application to standing oblique detonation waves for which experimental data were reported only recently, and improve understanding of detonation wave physics. In this study, we investigated the flow of a hydrogen-air mixture over a wedge-shaped projectile in an expansion tube. The computed solution was compared with experimental OH PLIF data. Computational studies such as this one can complement experimental efforts by providing detailed information about combustion initiation, flow structure and flow establishment time, a critical parameter in pulse facilities, in which the available test time may be too short to establish fully the reacting flowfield. Such calculations are also appropriate benchmark cases for evaluating numerical models for flow and combustion chemistry. Preliminary results of this work were presented at the Propulsion Engineering Research Center 9th Annual Symposium, Ohio Aerospace Institute, Oct. 1-2, 1997

GE-CHENG ZHA
Research Area: 3D Navier-Stokes Time Accurate Solutions Using Multipartitioning Parallel Computation Methodology

A parallel CFD code solving 3D time accurate Navier-Stokes equations with multipartioning parallel Methodology is being developed in collaboration with Ohio State University within the Air Vehicle Directorate, at Wright Patterson Air Force Base. The advantage of the multipartioning parallel method is that the domain decomposition will not introduce domain boundaries for the implicit operators. A ring structure data communication is employed so that the implicit time accurate method can be implemented for multi-processors with the same accuracy as for the single processor. No sub-iteration is needed at the domain boundaries.

The code has been validated for some typical unsteady flows, which include Coutte Flow, flow passing a cylinder. The code now is being employed for a large scale time accurate wall jet transient flow computation. The preliminary results are promising. The mesh has been refined to capture more details of the flow field. The mesh refinement computation is in progress and would be difficult to successfully implement without the parallel computation techniques used. A modified version of the code with more efficient inversion of the diagonalized block matrix is currently being tested.

Reports and Abstracts

1997 REPORTS AND ABSTRACTS

Keith, Theo G., Balog, Karen, and Povinelli, Louis A., Editors: "Institute for Computational Mechanics in Propulsion (ICOMP), Eleventh Annual Report - 1996", ICOMP Report 97-1, NASA TM 107476, May, 1997, 51 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by the Ohio Aerospace Institute (OAI) and funded under a cooperative agreement by the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities and accomplishments during 1996.

Yungster, Shaye (ICOMP); and Chen, Kuo-Huey (The University of Toledo): "Application of Chimera Grid Scheme to Combustor Flowfields at all Speeds", ICOMP Report 97-2, NASA CR 202321, January, 1997, 14 pages.

A CFD method for solving combustor flowfields at all speeds on complex configurations is presented. The approach is based on the ALLSPD-3D code which uses the compressible formulation of the flow equations including real gas effects, nonequilibrium chemistry and spray combustion. To facilitate the analysis of complex geometries, the chimera grid method is utilized. To the best of our knowledge, this is the first application of the chimera scheme to reacting flows. In order to evaluate the effectiveness of this numerical approach, several benchmark calculations of subsonic flows are presented. These include steady and unsteady flows, and bluff-body stabilized spray and premixed combustion flames. The results demonstrate the effectiveness of the combined ALLSPD-3D/chimera grid approach for analyzing subsonic combustor flowfields down to the incompressible limit.

Hixon, R. (ICOMP): "Evaluation of a High-Accuracy MacCormack-Type Scheme Using Benchmark Problems", ICOMP Report 97-3, NASA CR 202324, March 1997, 24 pages.

Due to their inherent dissipation and stability, the MacCormack sheme and its variants have been widely used in the computation of unsteady flow and acoustic problems. However, these schemes require many points per wavelength in order to propagate waves with a reasonable amount of accuracy. In this work, the linear wave propagation characteristics of MacCormack-type schemes are shown by solving several of the CAA Benchmark Problems.

Hixon, R., Shih, S.-H. (ICOMP); and Mankbadi, Reda R. (Cairo University): "Numerical Simulation of the Effect of Heating on Supersonic Jet Noise", ICOMP Report 97-4, NASA CR 202338, May, 1997, 21 pages.

The axisymmetric linearized Euler equations are used to simulate noise amplification and radiation from a supersonic jet. The effect of heating on the noise field of the jet is studied and compared to experimental results. Special attention was given to boundary treatment, and the resulting solution is stable and nearly free from boundary reflections.

Hixon, R., and Shih, S.-H. (ICOMP); and Mankbadi, Reda R. (Cairo University): "Effect of Coannular Flow on Linearized Euler Equation Predictions of Jet Noise", ICOMP Report 97-5, NASA CR 202339, May, 1997, 28 pages.

An improved version of a previously validated linearized Euler equation solver is used to compute the noise generated by coannular supersonic jets. Results for a single supersonic jet are compared to the results from both a normal velocity profile and an inverted velocity profile supersonic jet.

Norris, A.T. (ICOMP): "Application of Low Dimensional Manifolds in NOx Prediction", ICOMP Report 97-6, CMOTT 97-1, AIAA-97-3243, NASA CR 204137, August, 1997, 13 pages.

A new post-processing technique has been developed, based on the Intrinsic Low Dimensional Manifold (ILDM) method of Maas and Pope. The ILDM method is a dynamical systems approach to the simplification of large chemical kinetic mechanisms. By identifying low-dimensional attracting manifolds, the method allows complex full mechanisms to be parameterized by just a few variables: In effect, generating reduced chemical mechanisms by an automatic procedure. These resulting mechanisms however, still retain all the species used in the full mechanism. The NOx post-processor takes an ILDM reduced mechanism and attempts to map this mechanism to the results

of a CFD calculation. This mapping allows the NOx concentrations at each grid node to be obtained from the ILDM reduced mechanism, as well as other trace species of interest. Because a mapping procedure is used, this method is very fast, being able to process one million node calculations in just a few minutes.

Norris, A. T. (ICOMP): "Automated Simplification of Full Chemical Mechanisms", ICOMP Report 97-7, CMOTT 97-2, AIAA 97-3115, NASA CR 204138, August, 1997, 13 pages.

A code has been developed to automatically simplify full chemical mechanisms. The method employed is based on the Intrinsic Low Dimensional Manifold (ILDM) method of Maas and Pope. The ILDM method is a dynamical systems approach to the simplification of large chemical kinetic mechanisms. By identifying low-dimensional attracting manifolds, the method allows complex full mechanisms to be parameterized by just a few variables: In effect, generating reduced chemical mechanisms by an automatic procedure. These resulting mechanisms however, still retain all the species used in the full mechanism. Full and sketetal mechanisms for various fuels are simplified to a two dimensional manifold, and the resulting mechanisms are found to compare well with the full mechanisms, and show significant improvement over global one step mechanisms, such as those by Westbrook and Dryer. In addition, by using an ILDM reaction mechanism in a CFD code, a considerable improvement in turn-around time can be achieved.

Shih, Tsan-Hsing, Zhu, Jiang, Liou, William, (ICOMP); Chen, Kuo-Huey (The University of Toledo); Liu, Nan-Suey, (NASA Lewis); and Lumley, John L. (Cornell University): "Modeling of Turbulent Swirling Flows", ICOMP Report 97-8, CMOTT 97-3, NASA TM 113112, August, 1997, 54 pages.

Aircraft engine combustors generally involve turbulent swirling flows in order to enhance fuel-air mixing and flame stabilization. It has long been recognized that eddy viscosity turbulence models are unable to appropriately model swirling flows. Therefore, it has been suggested that, for the modeling of these flows, a second order closure scheme should be considered because of its ability in the modeling of rotational and curvature effects. However, this scheme will require solution of many complicated second moment transport equations (six Reynolds stresses plus other scalar fluxes and variances), which is a difficult task for any CFD implementations. Also, this scheme will require a large amount of computer resources for a general combustor swirling flow. This report is devoted to the development of a cubic Reynolds stress-strain model for turbulent swirling flows, and was inspired by the work of Launder's group at UMIST. Using this type of model, one only needs to solve two turbulence equations, one for the turbulent kinetic energy κ and the other for the dissipation rate ε. The cubic model developed in this report is based on a general Reynolds stress-strain relationship. Two flows have been chosen for model evaluation. One is a fully developed rotating pipe flow, and the other is a more complex flow with swirl and recirculation.

Zhu, J., and Shih, T.-H. (ICOMP): "An NPARC Turbulence Module With Wall Functions", ICOMP Report 97-9, CMOTT 97-4, AIAA 96-0382, NASA CR 204142, August, 1997, 11 pages.

The turbulence module recently developed for the NPARC code has been extended to include wall functions. The Van Driest transformation is used so that the wall functions can be applied to both incompressible and compressible flows. The module is equipped with three two-equation κ-ε turbulence models: Chien, Shih-Lumley and CMOTT models. Details of the wall functions as well as their numerical implementation are reported. It is shown that the inappropriate artificial viscosity in the near-wall region has a big influence on the solution of the wall function approach. A simple way to eliminate this influence is proposed, which gives satisfactory results during the code validation. The module can be easily linked to the NPARC code for practical applications.

Zhu, J., and Shih, T.-H. (ICOMP): "CMOTT Turbulence Module for NPARC", ICOMP Report 97-10, CMOTT 97-5, NASA CR 204143, August, 1997, 37 pages.

This is a user's manual of the CMOTT turbulence module, version 2.0, developed for the NPARC code. The module is written in a self-contained manner so that the user can use any turbulence model in the module without concern as to how it is implemented and solved. Three two-equation turbulence models have been built into the module: Chien, Shih-Lumley and CMOTT models, and all of them have both the low Reynolds number and wall

function options. Unlike Chien's model, both the Shih-Lumley and CMOTT models do not involve the dimensionless wall distance y^+ in the low Reynolds number approach, an advantage for separated flow calculations. The Van Driest transformation is used so that the wall function can be applied to both incompressible and compressible flows. The manual gives the details of the turbulence models used and their numerical implementation. It also gives two application examples, one for subsonic and the other for transonic flow, for demonstration. The module can be easily linked to the NPARC code for practical applications.

Tweedt, Daniel L., Chima, Rodrick V. (NASA Lewis); and Turkel, Eli (ICOMP): "Preconditioning for Numerical Simulation of Low Mach Number Three-Dimensional Viscous Turbomachinery Flows", ICOMP Report 97-11, NASA TM 113120, 31 pages.

A preconditioning scheme has been implemented into a three-dimensional viscous computational fluid dynamics code for turbomachine blade rows. The preconditioning allows the code, originally developed for simulating compressible flow fields, to be applied to nearly-incompressible, low Mach number flows. A brief description is given of the compressible Navier-Stokes equations for a rotating coordinate system, along with the preconditioning method employed. Details about the conservative formulation of artificial dissipation are provided, and different artificial dissipation schemes are discussed and compared. The preconditioned code was applied to a well-documented case involving the NASA large low-speed centrifugal compressor for which detailed experimental data are available for comparison. Performance and flow field data are compared for the near-design operating point of the compressor, with generally good agreement between computation and experiment. Further, significant differences between computational results for the different numerical implementations, revealing different levels of solution accuracy, are discussed.

Jayasimha, D.N. (Ohio State University); Hayder, M.E. (ICOMP); and Pillay, S.K. (NASA Lewis): "Parallelizing Navier-Stokes Computations on a Variety of Architectural Platforms", ICOMP Report 97-12, NASA TM 113158, 30 pages.

We study the computational, communication, and scalability characteristics of a Computational Fluid Dynamics application, which solves the time accurate flow field of a jet using the compressible Navier-Stokes equations, on a variety of parallel architectural platforms. The platforms chosen for this study are a cluster of workstations (the LACE experimental testbed at NASA Lewis), a shared memory multiprocessor (the Cray YMP), distributed memory multiprocessors with different topologies — the IBM SP and the Cray T3D. We investigate the impact of various networks, connecting the cluster of workstations, on the performance of the application and the overheads induced by popular message passing libraries used for parallelization. The work also highlights the importance of matching the memory bandwidth to the processor speed for good single processor performance. By studying the performance of an application on a variety of architectures, we are able to point out the strengths and weaknesses of each of the example computing platforms.

Kvernadze, George (University of New Mexico); Hagstrom, Thomas (ICOMP and University of New Mexico); and Shapiro, Henry (University of New Mexico): "Locating the Discontinuities of Bounded Function by the Partial Sums of its Fourier Series I: Periodical Case", ICOMP Report 97-13, NASA CR 206534, 35 pages.

A key step for some methods dealing with the reconstruction of a function with jump discontinuities is the accurate approximation of the jumps and their locations. Various methods have been suggested in the literature to obtain this valuable information. In the present paper, we develop an algorithm based on identities which determine the jumps of a 2π-periodic bounded not-too-highly oscillating function by the partial sums of its differentiated Fourier series the algorithm enables one to approximate the locations of discontinuities and the magnitudes of jumps of bounded function. We study the accuracy of approximation and establish asymptotic expansions for the approximations of a 2π-periodic piecewsie smooth function with one discontinuity. By an appropriate linear combination, obtained via derivatives of different order, we significantly improve the accuracy. Next, we use Richardson's extrapolation method to enhance the accuracy even more. For a function with multiple discontinuities we establish simple formulae which "eliminate" all discontinuities of the function but one. Then we treat the function as if it had one singularity following the method described above.

Seminars

1997 SEMINARS

Huynh, H. T. (NASA Lewis Research Center): "A Simple Upwind Scheme"

Upwind schemes are currently very popular in CFD, but they are generally perceived as difficult to forumulate and implement. In this talk, a very simple upwind scheme for the Euler equations is presented. The tools are: the midpoint rule, a linearization, a diagonalization, and the mean value theorem. The scheme then reduces to asking which way does the wind blow? Also, a straight forward geometric "entropy condition" is introduced which cures the sonic glitch problem.

Rigby, David (NYMA Inc): "Heat Transfer Prediction in Internal Coolant Flow Passages"

To achieve high performance and efficiency, gas turbine engines operate at high temperatures and pressure ratios. Currently, and for the foreseeable future, engine materials are unable to withstand these conditions without some form of cooling. Acceptable metal temperatures are maintained by passing relatively cool air from a compressor stage through complex internal passages in the turbine blades. Cooling must be done with a minimum amount of air since increases in performance due to increased operating temperature are partially offset due to added load on the compressor.

The Coolant Flow Management Team at NASA Lewis is studying many aspects of gas turbine cooling. The objective is to develop a methodology that can be used effectively to design and evaluate turbine cooling schemes. Results are presented demonstrating the use of very general structured multiblock grids for solution of the Navier-Stokes equations. The solver presently used, TRAF3D.MB, is efficient and has been used for many geometries. The code has been used in the past for heat transfer and performance prediction of the external gas path. Results presented in this work include flow in a channel with ribs and bleed holes, as well as flow through a channel with a 180 degree turn. Issues regarding grid generation and turbulence modeling will be discussed. For complex geometries, grid generation draws a significant amount of attention. The present results demonstrate that complex problems can in fact be handled using the multiblock technique. A newly developed block merging algorithm, the Method of Weakest Descent, will also be presented.

With the combination of experimental investigations and numerical predictions, a greater understanding of the mechanisms can be achieved. As the flowfields become better understood, it is hoped that the development of simplified models can result.

Bui, Trong T. (NASA Lewis Research Center): "Large-Eddy Simulation of Turbulent Flows Using an Unstructured Finite Volume Algorithm"

Turbulence dominates practical internal flows in inlets, ducts, and nozzles and significantly affects the noise and performance of aircraft jet engines. CFD tools need to be developed in order to provide accurate predictions of these turbulent flows and allow the engineer the opportunity to explore the underlying flow physics in order to design better aeropropulsion flow components that can be used in the aerospace industry.

To study turbulent flow physics, direct numerical simulation (DNS) and large-eddy simulation (LES) techniques have been successfully used. For example, it was through DNS and LES of turbulent boundary layers and channel flows that the near wall turbulent structures responsible for the generation of turbulent kinetic energy were discovered and analyzed in detail. In both DNS and LES, the unsteady, fluctuating turbulent flow is directly calculated as a function of time. However, in LES, only the large scale fluctuations are calculated, and the effect of the small scales is modeled using subgrid scale (SGS) models. Since the the small scales are more isotropic, and they carry only a small fraction of the total turbulent energy, it may be possible to accurately simulate the overall turbulent flow field with relatively simple SGS models.

Current LES of turbulent flows have been restricted to flows with simple geometries and/or low Reynolds numbers because of limitations in the CFD algorithms used in these simulations and the limited computational capability provided by the existing serial computer architectures. For simulation of flows with complex geometries, unstructured finite volume methods have proven to be very useful. In addition, the large increase in computational power provided by the new massively parallel computer systems allows larger CFD simulations to be conducted.

To investigate the feasibility and accuracy of unstructured finite volume methods for turbulence simulations, LES of turbulent flows in ducts is being investigated using an unstructured, finite-volume CFD algorithm with least-square reconstruction. The simulation code was designed from the beginning to take advantage of the additional computational capability provided by recent massively parallel computer systems such as the Cray T3D and the IBM SP2. In this talk, results obtained from work currently in progress will be presented, and lessons learned from using unstructured methods on parallel computer systems for a turbulence simulation will be discussed.

Jacqmin, David (NASA Lewis Research Center): "Calculations of Two- and Multi-Phase Flows Using Smeared Interface Methods"

Smeared-interface methods are new, relatively easy to use methods for calculating two-phase flows with surface tension. With them, phase interfaces are approximated as a finite thickness width that is resolveable on a fixed numerical grid. Surface tension forces are distributed over the width of the interface. The movement of the interface is calculated using either front tracking or front capturing techniques. The thermodynamic/energetics of two phase flow will be discussed and it will be shown how to derive smeared-interface models variationally, from model energy functionals. The variational approach has the advantage of yielding energy conserving numerical methods and of showing how to consistently and easily calculate surface tension forcings. Convergence of the resulting models and of the numerical solution of these models will be discussed. Applications will be given to sloshing flows, capillary oscillations, and to droplet interactions.

Yao, Minwu (Ohio Aerospace Institute and Computational Microgravity Lab, LeRC): "Fluid Dynamics of Viscoelastic Liquid Bridges"

Filament stretching devices are one of the most promising techniques that have been developed for providing accurate measurements of transient elongational stress growth functions for viscous polymer solutions. Such filament stretching devices are currently being developed at NASA LeRC for a space experiment entitled: "The Extensional Rheology of Non-Newtonian Materials". The flight experiment is aimed at investigating the extensional rheological properties of non-Newtonian fluids under microgravity conditions. The measured rheological data will be used to improve constitutive models for complex fluid flows. The novel instrument, Microgravity Extensional Rheometer, has been listed as new NASA flight hardware that can be used by other fluid physics investigators in future studies of various non-Newtonian materials.

In the filament stretching apparatus, a cylindrical liquid column is generated between two circular plates and then elongated by separating the ends at an exponential rate. The resulting kinematics of the liquid column, approximate an (it is hoped) ideal uniaxial elongational flow. After appropriate corrections for surface tension, gravity and inertia, the extensional viscosity is determined from the time-varying axial force exerted on the rigid end-plates of the device. Despite the rapid proliferation of filament stretching rheometers, there are few theoretical or numerical studies of these devices which verify the assumed form of the kinematics and the resulting dynamic evolution of the fluid stresses.

Dynamic analysis of the transient free-boundary motion of a non-Newtonian material is computationally challenging. Analytical solutions are available only under certain restrictive assumptions, and numerical solutions have been limited to small total deformations. The present work studies extensional deformation of an axi-symmetric filament incorporating surface tension, viscoelasticity and inertia. The geometry and material properties used in the modeling are based on the current experiment design, and several differential-type constitutive models are used to simulate the viscoelastic behavior of the fluids. Numerical solutions are obtained using POLYFLOW, and Hencky strain values of 4 and larger can be achieved. The dramatic difference in the dynamic response between viscoelastic and Newtonian fluids will be shown. Other topics that will be covered include: the effects of initial geometry, gravity, pinning conditions at the end-plates, finite nonlinear elastic extensibility of the macromolecules and shear-thinning effects.

Dong, Thomas Z. (NRC Research Associate, NASA Lewis Research Center): "Issues in Computational Aeroacoustics (CAA)"

As a new branch of CFD, computational aeroacoustics (CAA) determines acoustic waves generated by aircraft via numerical methods. The characteristics and objectives of aeroacoustic problems are often quite different from those of commonly encountered CFD problems. These differences lead to many computational issues which are unique to CAA. Computational aeroacoustic problems are time-dependent by definition and the amplitudes of the acoustic waves are often many orders of magnitude smaller than those of the mean flow variables. The preservation of wave propagation properties of the

governing PDE (Euler or Navier-Stokes equations) such as dispersion and dissipation is of critical importance in designing numerical schemes for CAA problems. Because of the small amplitudes of the acoustic waves, non-physical reflections from the boundaries of the computational domain will also degrade the accuracy of the solutions. In this seminar, the above issues and a number of numerical methods of CAA will be discussed. The results of applications of these methods to aeroacoustic problems will also be presented.

Hixon, Duane (ICOMP): "High-Accuracy MacCormack-Type Schemes for Aeroacoustic Applications"

MacCormack schemes are widely used in the computation of linear and nonlinear unsteady fluid dynamic problems due to their ease of use. However, these methods require many points per wavelength to resolve a wave and small time steps to propagate waves accurately.

A new family of high-accuracy MacCormack-type schemes for aeroacoustic problems will be described. The performance of these schemes will be shown for benchmark problems of the two CAA Workshops. Applications of these schemes will be shown for the calculation of large-scale instability noise in supersonic jets using the linearized Euler equations.

Shih, Shyue-Horng (ICOMP): "Jet Noise Predictions"

Jet noise suppression has become a critical issue for the development of advanced aircraft engines. Understading the noise generation mechanism and its prediction are vital to the development of noise suppression techniques. The major difficulty in predicting jet noise is that the sound radiated by the jet is produced by the unsteady turbulent fluctuations in the jet exhaust. In theory, Direct Numerical Simulation (DNS) can be used to directly predict the noise radiated from the entire range of turbulent eddies. However the grid resolution required to simulate high-Reynolds number turbulent flows makes DNS impractical due to current computer limitations. As such, Large-Eddy Simulation (LES) of jet noise represents the most thorough technique currently available for jet noise prediction. In LES, the larger turbulent eddies are directly predicted, while the effects of the smaller eddies are modelled. Since the large eddies are believed to be more efficient in radiating sound in supersonic jets, this approach will predict the majority of the noise produced.

In this talk, the LES approach for supersonic jet noise computation will be presented. A zonal approach used to speed the calculation by splitting the computational domain into a nonlinear sound generation region and a linear acoustic propagation region will be discussed. The far-field radiated noise is then obtained by coupling the near-field LES solutions of the sound source to the far-field extension techniques in the linear acoustic propagation region.

Slater, John (NASA Lewis Research Center): "Study of CFD Methods Applied to Rapidly Deforming Boundaries"

This seminar will discuss a study that was performed using various computational fluid dynamics (CFD) methods to understand their effects on the analysis of the flow field induced by the rapid collapse of a flexible bump in an annular duct with initially stagnant conditions. This flow represents a good test case for CFD methods for deforming boundaries because the flow is due entirely to the boundary motion. The study examined methods for implementing explicit and implicit time integration, modeling the bump collapse, imposing moving surface boundary conditions, modeling the grid dynamics, computing the numerical flux, and imposing the geometric conservation law. Good agreement was obtained between the CFD results and the time-varying static pressure readings obtained from an experiment. Significant results showed the crucial importance of the bump collapse model and the wall boundary conditions. The geometric conservation law was of critical importance for the explicit method, but not for the implicit method.

The seminar will also discuss other issues of CFD for moving grids, the modification of NPARC for moving gnds, and some earlier work in applying moving grids for the analysis of the unstart/restart operation of a high-speed inlet.

Hui, W. H. (Hong Kong University of Science and Technology): "Accurate Computation of Discontinuous Flow — The Role of Coordinates"

Coordinate systems play a crucial role in accurate computation of discontinuous flow. For two-dimensional steady supersonic flow the system based on streamlines and their orthogonals is found to be optimal in that it is the most robust and it produces infinite shock and slipline resolution. Such an optimal coordinate system, however, is shown not to exist for general three-dimensional steady supersonic flow. Nevertheless, a system using stream surfaces as coordinate surfaces

has the property of crisply resolving slip surfaces. Extension to unsteady flow based on material surfaces or stream surfaces also retains this favorable property.

Stewart, Mark (NYMA, Inc.): "Program Verification for Scientific and Engineering Codes"

A procedure for analyzing, checking and documenting the meaning or semantics of a scientific and engineering code will be presented. This procedure uses well established compiler techniques to recognize physical and mathematical formulae as well as geometrical location within a grid stencil.

Prototype software will be demonstrated which implements this procedure. A theoretical analysis of this procedure is possible by comparing compiler parsing rules with the fundamental algebraic manipulations of equations.

The ability to locate some semantic errors and document semantic concepts in a scientific and engineering code should reduce the time, risk, and effort of developing and using these codes.

Fu-Lin Tsung (ICOMP): "An Unstructured-Mesh Solver for Turbomachinery Applications"

Inner workings of the TUSM (turbomachinery unstructured mesh) code, an unsteady three-dimensional viscous solver designed for turbomachinery applications, will be presented in this seminar.

TUSM solves the unsteady Navier-Stokes equations using a finite volume method on tetrahedral meshes. An implicit integration scheme is used to march the solution in time and an upwind scheme is used for inviscid flux computations. Turbulence closure is achieved via a one-equation model. Sample calculations including backward-facing step, turbine stator, compressor rotor, and stage computations will be presented.

Bing-Gang Tong (Graduate School and Department of Mechanics University of Science and Technology, China): " Stability and Bifurcation Studies of Blunt Body Wakes by Using LDGM"

The low-dimensional Galerkin method (Noack and Eckelmann, 1992) has been generalized to compute wake flows behind translatory-oscillating and rotatory-oscillating 2-D circular cylinders in a uniform stream. By using it in discretizing the Navier-Stokes equations, an open flow wake system with an infinite degree of freedom may be reduced to a low-dimensional dynamical system with a finite number of degrees of freedom for low Reynolds numbers. The instability and dynamical bifurcation of wake flows behind stationary and uniformly rotating cylinders is predicted in terms of low critical Reynolds numbers. In respect to flow control, some nonlinear behavior of wakes behind oscillating cylinders is simulated numerically. A Floquet stability analysis is made for periodic wake flows.

PENN STATE SYMPOSIUM

PENN STATE SYMPOSIUM
October 1-2, 1997

Objective and Scope

Each year since 1989, the Annual Symposium on Propulsion has brought together a select group of engineers and scientists to share their most recent research results, present the status of current programs, and to discuss the important trends in propulsion.

This year, the Symposium is comprised of two sets of sessions running parallel. The first set of sessions cover a broad range of programmatic and technical issues related to Airbreathing and Combined-Cycle Propulsion, Advanced Propulsion Concepts, and Rocket Propulsion. The second set of sessions provides recent results from research in Spray Combustion, Combustion Dynamics, and Heat Transfer and Fluid Mechanics. Each paper is given by an invited speaker.

A tour of Plum Brook Test Facility in Sandusky, Ohio will also be provided as part of the Symposium. Plum Brook is the home of the Hypersonic Tunnel Facility, capable of Mach 5-7 freejet testing in non-vitiated flows; the Spacecraft Propulsion Research Facility, built to test a full scale Centaur upper stage in high altitude/near vacuum conditions; and the Cryogenics Propellant Tank (K-site), a 25-ft diameter door that was created for testing of a variety of rocket vehicle engine tank systems. In addition to these premier propulsion facilities, Plum Brook is also the site of the world's largest vacuum chamber, the 100 ft dia x 125 ft high Space Power Facility.

Workshop Agenda

Wednesday, October 1, 1997

7:30 a.m.	Registration
8:00 a.m.	Welcome and Introductory Remarks *Don Palac, Manager, RBCC Projects, NASA Lewis Research Center* *Robert Santoro, Penn State University* *Don Campbell, Director of NASA Lewis Research Center*

8:10 a.m.   **Session I. Air Breathing and Combined Cycle Propulsion**
Chair: Chuck Trefny, NASA Lewis Research Center

Computational Fluid Dynamics Simulation of a Low-Speed Rocket-Based Combined Cycle
W. Watkins, P&W and L. Chiapetta, UTRC

The Flight Acceleration Simulation Test Facility (FAST)
W. Swartwout and G. Roffe, GASL

A Leading Hypersonic Combined-Cycle Propulsion Application: Single Stage to Orbit (SSTO), Vertical Takeoff and Landing (VTOL), Using Supercharged Ejector Scramjet (SESJ) Engines
J. R. Olds, GIT, R. W. Foster, Teknos and W. D. Escher, Kaiser Marquardt

Design of Rocket-Based Combined-Cycle Launch Vehicles Using Multi-Variable Optimization Techniques
T. Rice and D. M. Van Wie, JHU/APL

Advantages of Air Liquefaction in Combined Cycle Systems
A. A. duPont, duPont Aerospace

The Air Ain't for the Birds Alone
A. Siebenhaar, GenCorp Aerojet

Integrated Experimental/Analytical Studies of the Ejector Mode of a RBCC Engine
R. J. Santoro, C. L. Merkle, S. Pal and W. E. Anderson, Penn State University

Rocketdyne RBCC Concept Development
P. Ortwerth, A. Ketchum and G. Ratekin, Rocketdyne

RBCC Rocket Mode Parametric Performance Analysis Using CFD
T. D. Smith, C. J. Steffen, Jr., NASA Lewis Research Center, S. Yungster, ICOMP and D. J. Keller, RQS, Inc.

Session II. Spray Combustion
Chair: Marlow Moser, University of Alabama, Huntsville

Droplet Breakup in Highly Turbulent Flowfields
T. D. Prevish and D. A. Santavicca, Penn State University

High Frequency Modulated-flow Fuel Injectors
J. Dressler, Fluid Jet Associates

Supercritical Vaporization of LOX Droplets Using Molecular Dynamics
T. L. Kaltz, L. N. Long and M. M. Micci, Penn State University

Raman Spectroscopy Experiments for LO_2/GH_2 Rocket Combustion at Fuel- and Oxidizer-Rich Conditions
S. Yeralan, S.Pal and R. J. Santoro, Penn State University

Transcritical LOX Droplet Gasification
R. D. Woodward and D. G. Talley, Phillips Laboratory

Fiber Optic Probe for Primary Zone Fuel Distribution Measurements in Actual Turbine Combustors
J. G. Lee and D. A. Santavicca, Penn State University

Fundamental Ligament Shedding Frequencies in a Swirl Coaxial Injector
N. O. Rhys, M. Moser, UAH and R. Eskridge, NASA Marshall Space Flight Center

Parallelization of Lagrangian Spray Combustion by Domain Decomposition
K.-H. Chen, University of Toledo/NASA Lewis Research Center and D. Fricker, USARL/NASA Lewis Research Center

12:00 p.m.	Lunch at Plumbrook
2:00 p.m.	Tour of Plumbrook Test Facility

October 2, 1997

8:10 a.m.

Session III. Advanced Propulsion Concepts
Chair: Gerald Smith, Penn State University

Advanced Propulsion Concepts at the Jet Propulsion Laboratory
S. Leifer, R. H. Frisbee and J. R. Brophy, NASA Jet Propulsion Laboratory

Dense Plasma Focus for Space Propulsion
C. Choi, Purdue University

Optimization of a Gasdynamic Mirror Fusion Propulsion System
W. J. Emrich, Jr., NASA Marshall Space Flight Center

Antiproton-Catalyzed Microfission/Fusion Propulsion Systems for Exploration of the Outer Solar System
B. Dundore, J. Fulmer, G. Gaidos, J. Laiho, R. A. Lewis, G. A. Smith and S. Chakrabarti, Penn State University

Status: Breakthrough Propulsion Physics Program
M. G. Millis, NASA Lewis Research Center

Low-Power Microwave Arcjet Thruster Development
D. Nordling and M. M. Micci, Penn State University

AIMStar: Antimatter Initiated Microfusion for Pre-cursor Interstellar Missions
B. Dundore, G. Gaidos, R. A. Lewis and G. A. Smith, Penn State University

Session IV. Combustion Dynamics
Chair: Kevin Breisacher, NASA Lewis Research Center

Dynamic Response of a Premixed Swirl Injector Flame to Longitudinal Acoustic Disturbances
T. Wang and V. Yang, Penn State University

Experimental Investigations of High-Pressure Gas Turbine Combustion Instabilities
J. C. Broda, S. Seo, S. Pal and R. J. Santoro, Penn State University

An Optical Study of Unstable Combustion
K. K. Venkataraman, L. H. Preston, D. W. Simmons, B. J. Lee, J. G. Lee and D. A. Santavicca, Penn State University

An Experimental Study of Combustor Air Swirler Acoustic and Fluid Dynamic Sensitivities
J. M. Cohen, UTRC and J. R. Hibshman, VPI

Quick Stability Calculations for Longitudinal and Tangential Modes in Pre-mixed Gas Turbine Combustors
Z. Yang, AYT Corp., K. Breisacher, NASA Lewis Research Ctr., K. Radhakrishnan, NYMA and A. Oyediran, AYT Corp.

Active Control of Combustion Instability in a Lean Premixed Dump Combustor
J. G. Lee, C. M. Jones, and D. A. Santavicca, Penn State University

Automated Simplification of Full Chemical Mechanisms: Tabulation Implementation
A. T. Norris, ICOMP

Simulation of Unsteady Hypersonic Combustion Around Projectiles
S. Yungster, ICOMP and K. Radhakrishnan, NYMA, Inc.

12:00 p.m.   Lunch at the Ohio Aerospace Institute
Keynote Speaker: Martin Kress, Deputy Director, NASA Lewis Research Center

1:30 p.m.   **Session V. Rocket Propulsion**
Chair: Roger Woodward, Penn State University

Recent Advancements in TRW Space Propulsion Technology
R. Sackheim, TRW

High Energy Density Propellants: Propulsion Benefits for Launch Vehicles
B. Palaszewski, NASA Lewis Research Center

An Integrated Injector Design Methodology: Experimental and Analytical Developments
P. K. Tucker, NASA Marshall Space Flight Ctr., and S. Pal, M. J. Foust, M. Lehman and R. J. Santoro, Penn State Univ.

Low Cost Propulsion using High Density Storable and Clean Propellants
W. E. Anderson, D. Krause and T. Lewis, OSI and J. Rusek, NAWC China Lake

A Comprehensive Model of RDX/GAP Propellant Combustion
Y. C. Liau, S. T. Thynell and V. Yang, Penn State University

Ignition Characteristics of Aged Composite Propellants under CO_2 Laser Heating
A. Ulas and K. K. Kuo, Penn State University

Effect of Curing Agent on the Burning Rate of GAP Monopropellant
G. C. Harting, G. A. Risha, A. Peretz, and K. K. Kuo, Penn State University

Session VI. Heat Transfer and Fluid Mechanics
Chair: Ray Gaugler, NASA Lewis Research Center

Study of Blade Tip Clearance Effects for Turbomachinery Applications
J. Feng, V. Ahuja and C. L. Merkle, Penn State University

Analysis of Tip and Casing Treatment Effects on Heat Transfer and Efficiency in Gas Turbine Rotors
A. Ameri, AYT Corp.

An Aerothermal Investigation of Boundary Layer Fences for Use in Turbine Endwall Regions and Internal Coolant Passages
D. H. Rizzo and C. Camci, Penn State University

Boundary Layer Thickness and Heat-Release Effects on Mixing Layers
D. Schwer and C. L. Merkle, Penn State University

Hydraulics of Swirl Propellant Injectors
V. Bazarov, MSAI

Summary of GH_2/GO_2 Raman Spectroscopy Experiments for a Shear Coaxial Injector
M. Lehman, S. Pal and R. J. Santoro, Penn State University

Aerodynamic and Heat Transfer Research in a Transonic Turbine Blade Cascade
P. W. Giel, NYMA, Inc.

REPORT DOCUMENTATION PAGE

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE June 1998	3. REPORT TYPE AND DATES COVERED Technical Memorandum

4. TITLE AND SUBTITLE

Institute for Computational Mechanics in Propulsion (ICOMP)
Twelfth Annual Report – 1997

6. AUTHOR(S)

Theo G. Keith, Jr., Karen Balog, and Louis A. Povinelli, Editors

5. FUNDING NUMBERS

WU–523–36–13–00

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER

E–11224

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, DC 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA TM—1998-208397
ICOMP–98–01

11. SUPPLEMENTARY NOTES

ICOMP Program Director, Louis A. Povinelli, organization code 5000, (216) 433–5818.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Categories: 34 and 64   Distribution: Nonstandard

This publication is available from the NASA Center for AeroSpace Information, (301) 621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT (Maximum 200 words)

The Institute for Computational Mechanics in Propulsion (ICOMP) was formed to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. ICOMP is operated by the Ohio Aerospace Institute (OAI) and funded via numerous cooperative agreements by the NASA Lewis Research Center in Cleveland, Ohio. This report describes the activities at ICOMP during 1997, the Institute's twelfth year of operation.

14. SUBJECT TERMS

Numerical analysis; Computer science; Mathematics; Fluid mechanics

15. NUMBER OF PAGES
44

16. PRICE CODE
A03

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102