Institute for Computational Mechanics in Propulsion (ICOMP)

Seventh Annual Report – 1992

May 1993

NASA

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION
★★ NASA
Lewis Research Center
ICOMP
Case Western
Reserve University
Ohio Aerospace
Institute

Institute for Computational Mechanics in Propulsion (ICOMP)

Seventh Annual Report – 1992

Complied and edited by
Dr. Charles E. Feiler
ICOMP Executive Officer

Approved by
Dr. Louis A. Povinelli
ICOMP Director

May 1993

CONTENTS

INSTITUTE FOR COMPUTATIONAL MECHANICS

IN PROPULSION (ICOMP)

SEVENTH ANNUAL REPORT

1992

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in Cleveland, Ohio to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1992.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in September 1985, 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; to improve computer simulation of aerospace propulsion components; and to focus interdisciplinary computational research efforts. The specific areas of interest in computational research include: fluid mechanics for internal flows; structural mechanics and dynamics; and fluid-structural interactions. In 1990 a specific new focus was added with the formation within ICOMP of the Center for Modeling Turbulence and Transition (CMOTT).

The organization and operation of ICOMP as originally constituted were described in ICOMP Report No. 87-8 (NASA TM-100225), "The Institute for Computational Mechanics in Propulsion, (ICOMP), First Annual Report", Nov. 1987.

The ICOMP Sixth Annual Report for 1991 (ICOMP Report No. 92-1, NASA TM-105612) described the transfer of several administrative functions to the Ohio Aerospace Institute (OAI) along with a brief description of OAI itself. This association with OAI will be even stronger in 1993 with the relocation of the ICOMP research staff into the newly-opened OAI building. The daily operation of ICOMP continues to be provided by Program Director, Dr. Louis A. Povinelli, Executive Officer, Dr. Charles E. Feiler and OAI Director for Internal Programs, Dr. Theo G. Keith. The role of the Steering Committee, with membership as in 1991, has become more of an advisory nature reflecting the more mature stature of ICOMP.

The remainder of this report summarizes the activities at ICOMP during 1992. It lists the visiting researchers, their affiliations and time of visit followed by reports of RESEARCH IN PROGRESS, REPORTS AND ABSTRACTS published and SEMINARS presented. An overview summary of a special lecture series on turbulence and turbulence modeling, organized and conducted by Dr. Tsan-Hsing Shih of CMOTT, is also presented.

THE ICOMP STAFF OF VISITING RESEARCHERS

The composition of the ICOMP staff during 1992 is shown in figure 1. Fifty-eight researchers were in residence at Lewis for periods varying from a few days to a year. Figure 2 is a photograph of the ICOMP Steering Committee and the visiting researchers taken at a reception in July 1992. Figure 3 lists the universities or other institutions represented and the number of people from each. The figure lists thirty-eight organizations. Figure 4 shows the growth of ICOMP during its first seven years in terms of staff size, organizations represented and technical output as measured by the numbers of seminars, reports and workshops. The next sections will describe the technical activities of the visiting researchers starting with reports of RESEARCH IN PROGRESS, followed by REPORTS AND ABSTRACTS, and finally, SEMINARS.

RESEARCH IN PROGRESS

DARE AFOLABI, Purdue University

My work at ICOMP was carried out in February and March of 1992. During this period, research was conducted into the flutter of blades, bladed-disks, propfans and rotating machinery. These are vital components of various engines in propulsion systems.

The method of analysis utilized in my research derives from the qualitative theory of differential equations, and certain aspects of real algebraic geometry. I found that methods such as bifurcation theory and catastrophe theory, when correctly applied, provide fresh and illuminating insight into the flutter and stability of linear aeroelastic systems.

The following ICOMP Reports were written:

Afolabi, D.: Flutter Analysis Using Transversality Theory.
Afolabi, D; and Mehmed, O.: Flutter of Rotating Blades.

SURESH K. AGGARWAL, University of Illinois, Chicago

Research activity under the ICOMP program focused on the completion of computational work based on the KIVA-II code and the development of an all-speed spray code. The KIVA-II code was modified to simulate the Burke-Schumann type of flame supported by gaseous as well as liquid spray fuels. Detailed computations of these two flames with two chemistry models were completed. This effort is discussed in two papers (Authors: M. Mawid, D. Bulzan and S. K. Aggarwal) submitted for publication. The second part of the research focused on the development of an all-speed spray code. A major part of the time was spent learning about the structure of the code in consultation with Dr. J. S. Shuen and in the implementation of spray models into the code. This research activity is expected to be continued. I also presented a seminar, "Droplet Dispersion with Large Eddy Simulations."

KUMUD AJMANI (Postdoctoral), Virginia Tech

This work focuses on developing an implicit solver for parallel CFD applications on message-passing computer architectures, for obtaining steady-state solutions of the compressible Navier-Stokes equations. In this work, identical convergence rates for the global non-linear problem are obtained for the serial and parallel versions of the implicit solver. The GMRES algorithm belonging to the family of Krylov solvers is parallelized and used to solve large, sparse, non-symmetric, linear systems of equations. The linear system is preconditioned with LU-SGS (Lower Upper Symmetric Gauss Seidel) scheme and the preconditioning greatly contributes to convergence acceleration.

The specific machines used are an Intel iPSC/Hypercube (32 nodes) and an Intel Touchstone-Delta (512 nodes). The code is developed on the 32-node machine and benchmarked on the 512-node machine. The global domain is partitioned by a simple domain-decomposition strategy with each processor being assigned one domain. The resulting parallel code exhibits good scalability, superior parallel efficiency and excellent load-balancing. The convergence efficiency of the implicit parallel solver is identical to that of the serial solver.

Excellent performance numbers are obtained with the parallel code. Flow over a backward-facing step is used as a representative test case. The computational power (Mflops/sec) obtained varies as a function of the computational workload per processor. A maximum performance of 2300 Mflops/sec is obtained while computing a 513×513 grid on 512 nodes of the Delta machine. The parallel efficiency of the code remains above 80% as long as each processing node is assigned at least 1024 grid points.

Numerical modeling of communication patterns is also performed to gain an understanding of the underlying bottlenecks in the code. Run-time communication profiling reveals that the total communication times are independent of the nature of the domain partitioning. The total communication overhead constitutes roughly 5-7% of the total execution time, which shows that communication costs do not significantly affect the parallel performance.

Work is continuing towards examining better domain-decomposition ideas. Certain parallelizable preconditioners are also being investigated. Portable communication libraries are being included to generate

a code capable of performing on coarse and fine grained architectures. All these efforts will serve as important building blocks in the development of a 3D, general purpose solver for the Navier-Stokes equations.

ANDREA ARNONE, University of Florence, Italy

Recent progress in computational fluid dynamics along with evolution of computer performance is encouraging scientists to look at the details of the flow physics more and more. There are a variety of engineering applications where the unsteadiness of the problem can not be neglected (i.e. vortex shedding, natural unsteadiness, forced unsteadiness, turbomachinery stage analysis and rotor-stator interaction). In several branches of engineering, even if the flow is known to be unsteady, most of the analysis and designing tools used today are based on a steady or quasi-steady assumption. Therefore there is a strong interest in developing methodologies for efficient simulation of unsteady flow features.

Explicit schemes with accelerating techniques have proven to be very effective for steady problems. Unfortunately, the computational efficiency of those time-dependent schemes is achieved by sacrificing the accuracy in time. During my visit to ICOMP, a procedure was developed to show that the conventional steady-state acceleration techniques, specifically the multigrid techniques, can be reformulated and applied to unsteady Navier-Stokes problems as well, while, still achieving efficiency.

As one of the final goals of this research will be the study of unsteady phenomena in turbomachinery components such as rotor-stator interaction, we implemented the technique in the TRAF(2D/3D) codes. These two- and three-dimensional solvers were developed during a joint project between the University of Florence and NASA Lewis and were designed for turbomachinery blade row analysis.

To validate the procedure, three test cases were studied. First, vortex shedding over a row of circular cylinders in a laminar regime was examined, the interest being mostly in the flow periodicity and in the prediction of the Strouhal number. As a second application of natural unsteadiness, shock buffeting over a row of bicircular airfoils was studied. Finally, the last application concerned forced unsteadiness in turbomachines and simulates the effect of passing stator wakes on a rotor blade. A report on this study is in progress.

THOMAS F. BALSA, University of Arizona

We studied the development of a low-frequency instability mode in supersonic mixing layers. The Navier-Stokes equations are rationally simplified by asymptotic methods under the low-frequency and high Reynolds-number assumptions. This results in three coupled nonlinear equations for the vorticity, temperature and the amplitude of the disturbance, respectively. The largest nonlinear effects enter in a thin "boundary layer", called the critical layer, in which there is a very strong generation of vorticity by the baroclinic torque. Our work is based on the nonlinear critical layer concepts developed by M. E. Goldstein and his co-workers. Considerable interaction resulted with this group owing to the overlap of technical interests.

The distinguishing feature of our work from previously published work is the appearance of all the modes (in the sense of a Fourier series) in the cross-stream velocity component. Recall that this velocity component is responsible for displacing a material volume into the high- and low-speed sides of the external streams. This results in the roll-up of vorticity.

The activity this summer focused on the development of a spectral computer code to capture this roll-up and the amplitude of the disturbance. The numerical results show this roll-up very nicely and additional calculations indicate that the "vortex sheet" underlying this low-frequency flow tends to develop sharp fronts reminiscent of nonlinear kink modes. Additional numerical results are required to confirm our last observation.

LISA BEARD (Graduate Student), University of Michigan

During my visit to ICOMP, I spent the bulk of my time working with The Center for Modeling of Turbulence and Transition (CMOTT). The project I helped with had a basic goal of studying the performance of low Reynolds-number, two equation (κ-ε) turbulence models in separated flow. First the validity of a 2D incompressible base solver to test the models in was established. The code, DTNS2D, created by J. Gorski at the David Taylor Research Center is based upon the method of pseudo-compressibility so that the incompressible fluid equations become hyperbolic and can be solved with time-marching methods. The validity of the code was established by investigating two laminar flow cases: flat plate flow results compared with the theory of Blasius and backward facing step flow compared with experimental measurements. Both

cases were in good agreement. The benchmark flow used to test the capabilities of various two equation turbulence models is a high Reynolds-number flow over a backward facing step. To-date most of the attention for this kind of flow has been placed on wall functions used in the near wall region which are not strictly valid in the separated region generated after the step. This established the motivation of the project to implement modified two equation models containing low Reynolds-number terms for near wall conditions, which are known to better predict near wall behavior, and test their performance compared to the wall function models in separated flow. Three low Reynolds-number models were implemented: Jones-Launder, Chien, and Shih-Lumley. However, even though these models better predict the very near wall behavior of the turbulence, they greatly underpredict the reattachment length of the shear layer. Preliminary results show that the wall function models obtain better estimates of reattachment length.

GILES J. BRERETON, University of Michigan

During my visit to ICOMP, I worked on a review article on progress in the understanding and prediction of unsteady turbulent flows. I also began to explore some turbulence modeling ideas based on results of rapid distortion theory, in collaboration with Dr. R. Mankbadi. The latter work has resulted in an improved model and a paper, to be presented at the International Conference on Near-Wall Turbulent Flows, Tempe, March 1993. The former article is still awaiting completion but will be submitted to a journal shortly. During my visit, I had numerous fruitful discussions with Dr. Mankbadi. I also had the opportunity to interact with Roy Tew (Stirling Engine Research), Dr. Moumir Ibrahim, Dr. Khairul Zaman, and Dr. Tsan-Hsing Shih amongst others.

TAWIT CHITSOMBOON (POSTDOCTORAL), OLD DOMINION UNIVERSITY

The mixer-ejector concept has been proposed to reduce the noise level of the High Speed Civil Transport (HSCT) aircraft, by reducing its exhaust jet velocity through mixing the hot jet exhaust with the slower entrained ambient air. To this end, the 3D flowfield in a 2D mixer-ejector of Pratt and Whitney has been analyzed using the MAWLUS CFD code (Multigrid Acceleration With L-U Scheme). The analysis has been rendered difficult due to the complexity in flow features and configuration of the domain. The analysis artificially created two distinguishable species of air, that of the hot core flow and that of the cold entrained flow, in order to unambiguously determine the extent of mixing the two streams. The conventional approach has been to use total temperature as a mixing indicator.

About 450,000 highly skewed, highly stretched grid points have been used in the calculation. Laminar results have been obtained with qualitative agreement with experimental results. It was reasoned that the flowfield is turbulence driven and thus requires a turbulence solution methodology. The k-ε turbulence model was then turned on. During the initial phase of the turbulence analysis, the solution showed a trend toward better agreement with experimental results. Unfortunately, the code failed shortly afterward. Many attempts have been made to make the turbulence model work properly. This includes: alternative treatments of source term stiffness; modeling the turbulence stress tensor as an incompressible flow; various treatments of smoothing terms; various treatments of wall functions; treating the viscous terms as partially implicit, etc. None of the remedies tried work. It now appears that simply reducing the time step of integration can stabilize the calculation without all the mentioned treatments. The code is now being run to see if this is, in fact, the case.

Another activity that the author has been involved in is cooperating with Dr. Shaye Yungster (ICOMP) in using MAWLUS to analyze the NASP 3D nozzle flowfield. The code seemed to perform well and the results compared very well with experiment.

A paper was co-authored (second author) with Dr. Yungster. The title of the paper is: "Numerical Analysis of a Single Expansion Ramp Nozzle with Hot Exhaust and External Burning". The paper was presented at the NASP Mid-Term Technology Review, Monterey, CA, April 21-24, 1992.

JOONGKEE CHUNG (Postdoctoral), University of California at Berkeley

A numerical study was performed to evaluate a 2D time accurate version of the PARC code which has a Runge-Kutta multistage solver sharing the same metrics, Jacobians, boundary conditions, Right Hand Side, and artificial dissipation as the steady implicit pentadiagonal solver.

For validation purposes, shock tube problems were chosen for comparison with 1D exact solutions. Pressure ratios of 10:1 and 500:1 were applied and numerical results were obtained for the axisymmetric case. Exact and numerical solutions at the center axis were compared for pressure, density, and U velocity. In these

computations, uniform time step size was used and coefficients of 3, 4 and 5 stage schemes proposed by Jameson et al (AIAA paper 84-0093) were modified to remove overshoots and wiggles near the shock. Additionally, several types of artificial dissipation sensors were studied to improve the accuracy of the solutions. Among them, a sensor based on entropy change produced the best results.

Further computations were performed to simulate the flow through a Variable Diameter Centerbody (VDC) supersonic inlet which has many phenomena such as shocks, boundary layer separations and mass flow through bleed holes. Both axisymmetric inviscid and laminar calculations were performed with or without the bleed. In case of inviscid flow, a stabilized normal shock (the started case) was captured in the diffuser section over a range of back pressure to free stream static pressure ratios. Once this ratio is raised over the critical value, the normal shock moves upstream and is expelled (unstart). To adequately deal with the formation of large separation regions for the viscous flow, bleed holes were placed upstream and downstream of the throat, which eliminated some separation and stabilized the terminating shock over a certain back to free stream pressure ratio range. To further improve the accuracy of the solutions, grid adaptations are planned and eventually turbulent cases will be run for realistic simulations.

DOMINIC DAVIS (Postdoctoral), University College, London

I am considering the possible mean-flow changes that may occur in the presence of viscosity in the problem of two low-amplitude interacting oblique waves in a shear layer. The growth rate of each wave is small compared to its phase speed and hence a critical layer exists (where the basic-flow speed and phase speed are equal). Inside this layer the waves interact nonlinearly and as well as affecting the boundary-layer growth, they also induce a relatively small vortex flow outside the critical layer. Previous results (Wu, Lee and Cowley 1992) have demonstrated that the waves may decay exponentially far downstream, whereas the vortex flow persists, in fact growing linearly with downstream distance, where it takes over as the dominant mean-flow perturbation. Eventually a new region comes into play, when the spreading critical layer has diffused entirely into the shear layer. Then the vortex flow is governed by viscous forces throughout the shear layer.

Secondly, I have been studying nonlinear vortex/wave interactions in 3D boundary layers. Specifically the interactions of two low-amplitude 3D Tollmien-Schlichting (TS) waves and their induced longitudinal vortex were considered based on the ideas of Hall and Smith (1989), but unlike their work, containing an order-one cross-flow. The nonlinear interaction near to the lower branch of the neutral curve is controlled by a partial-differential system for the vortex flow coupled with an ordinary-differential equation for each wave pressure. Three distinct possibilities emerge for the nonlinear behavior of the flow solution downstream: an algebraic finite-distance singularity; far-downstream exponential wave-decay; or far-downstream saturation. Which one occurs depends on the input amplitudes upstream, the wave angles and the size of the cross-flow.

The work involves both computational and analytical techniques and is performed in both high-Reynolds-number and incompressible regimes.

REFERENCES

Hall, P.; and Smith, F.T.: Nonlinear Tollmien-Schlichting/Vortex Interactions in Boundary Layers. Eur. J. Mech., 1989, B8, pg. 179-205.
Wu, X.; Lee, S.-S.; and Cowley, S.: On The Nonlinear Three-Dimensional Instability of Stokes Layers and Other Shear Layers to Pairs of Oblique Waves. 1992 (in preparation).

ROBERT J. DEISSLER, Los Alamos National Laboratory

One project on which I have been spending considerable time is a study of Taylor-Couette flow with an imposed axial through-flow. This work has been done in collaboration with Dr. Wai-Ming To of Sverdrup Technology. In this system, a fluid exists between two concentric cylinders where the inner cylinder is rotating and where there is a through-flow parallel to the axes of the cylinders. For sufficiently large rotation rate the steady background flow is unstable and rolls (or Taylor vortices) form. Depending on the axial flow rate the instability can be either *absolute* (meaning that a localized perturbation grows at a given point in the laboratory frame) or, if the axial flow rate is sufficiently large, *convective* (meaning that the perturbation grows only in a moving frame of reference, i.e. a frame of reference moving in the axial direction, eventually damping at any given point in the laboratory frame). Under convectively unstable conditions noise near the entrance region will be amplified resulting in spatially growing waves and a *noise-sustained structure* [R. J. Deissler, J. Stat.

Phys. **40** (1985) 371; Physica D **25** (1987) 233; J. Stat. Phys. **54** (1989) 1459]. This is an ideal system to study the effects of noise and noise-sustained structure since a convective instability can exist for fairly low flow rates, meaning that the system is easily controlled and that there is no turbulence. This, of course, makes the solution of the Navier-Stokes equations much easier. Another advantage is that the Ginzburg-Landau equation can be derived for this system so that direct comparison between the Ginzburg-Landau system and Navier-Stokes system can be made.

We have done a linear stability analysis of this system showing the conditions under which the system is absolutely or convectively unstable. We have found that regions in parameter space exist for which spiral modes dominate. Previously we had developed a code using Chebychev collocation in the radial direction and Fourier (sine and cosine) transforms in the axial direction. This code reproduced much of the behavior seen in the experiment by Babcock, Ahlers and Cannel, Phy. Rev. Lett. **67** (1991) 3388, such as the selective and spatial amplification of noise near the inlet resulting in spatially growing waves and a noise-sustained structure of traveling Taylor vortices. We have also numerically solved for a more realistic developing background flow and have incorporated this into the code. However, we have discovered that some inaccuracy results from using Fourier transforms in such a system (it was less apparent with the previous background profile), since there is a large disparity in amplitude between the small amplitude noise near the entrance region and the saturated Taylor vortices further downstream. Therefore, we found it necessary to develop a code which has only local coupling in the axial direction. The code is now working and uses finite differencing in the axial direction, while still using Chebychev collocation in the radial direction. By introducing thermal noise into the code we have found that the noise-sustained structure in the experiment of Babcock et. al. is probably not thermal in origin, as previously suggested by Babcock, Cannel, and Ahlers, preprint (1992). However, the noise is still extremely small, being roughly an order of magnitude larger than thermal noise. A paper is in the process of being written.

Another project is the study of Marangoni convection (surface tension driven convection) in a fluid layer with two free surfaces. This project is in collaboration with Dr. A. Oron of the Technion in Israel and Dr. J. C. Duh of the NASA Lewis Space Experiments Division. We have analytically done a linear stability analysis for this system and have found that depending on the ratio of the Marangoni numbers of the top and bottom surfaces and in the limit of zero Biot number, the instability will be to either a finite wavenumber or to zero wavenumber. For the zero wavenumber case, a nonlinear evolution equation was derived. A paper is in the process of being written.

AYODEJI DEMUREN, Old Dominion University

Current research activities are in two basic areas. One is in the development of multigrid acceleration methods for the solution of compressible and incompressible Navier-Stokes equations for turbulent flows. The second is in the analysis of second-moment turbulence closure models for application to complex 3D flows.

The implementation of multigrid acceleration for incompressible flows has been reported in Demuren (1992). Progress has also been made in implementing multigrid acceleration in the Proteus computer code which was written in the IFMD for compressible flow applications. In the first phase of the work, convergence properties of the code were analyzed through von Neumann stability analysis methods and through actual computations of several test cases on three sets of grids with increasing levels of refinement. The results were presented in detail in an interim report (Demuren and Ibraheem 1992). It was shown that the convergence rate of the code is typical of single-grid methods based on approximate factorization. The error reduction rate is of order $[1 - 0(h^2)]$ so that it approaches unity as the grid is refined. The analyses also showed that it is a good candidate for multigrid acceleration, provided care is exercised in specifying artificial dissipation terms and in the choice of the CFL number. A fixed, V-cycle multigrid procedure similar to that used in the incompressible code was implemented in the 2D version of the Proteus code. The implementation tried as much as possible to adhere to the guiding principles of the Proteus code development. It was highly modular and transparent. Validation tests of the 2D code are now being completed and work is proceeding on the implementation for the 3D code.

A systematic study of second-moment turbulence closure models suitable for application to complex 3D flows has shown the models for the pressure-strain correlations proposed by Launder, Reece and Rodi (JFM, 1975) and Speziale, Sarkar and Gatski (JFM, 1991) to be most promising. The latter has the advantage of being able to predict the turbulence structure in a wall-bounded flow without recourse to the use of wall-reflection terms, treatment of which is ambiguous in many complex flows (Demuren and Sarkar 1993). Generalizations for curvilinear coordinates and incorporation in a multigrid code, which ensures efficient

convergence on any grid size, are presented in Demuren (1993). Extensions to compressible flow applications will follow.

REFERENCES

Demuren, A.O.: Multigrid Acceleration and Turbulence Models for Computations of 3D Turbulent Jets in Crossflow. Int. J. Heat and Mass Transfer (in press), 1992

Demuren, A.O.: On the Generation of Secondary Motion in Circular to Rectangular Transition Ducts. AIAA Paper-93-0681, 31st Aerospace Sciences Meeting, Reno, Nevada, 1993

Demuren, A.O.; and Ibraheem, S.O.: Convergence Acceleration of the Proteus Computer Code with Multigrid Methods. Interim Report submitted to NASA Lewis Research Center for the period ending July 31, 1992

Demuren, A.O.; and Sarkar, S.: Systematic Study of Reynolds Stress Closure Models in the Computations of Plane Channel Flows. ASME, J. Fluids Engineering (in press)

J. S. B. GAJJAR, University of Manchester, England

Boundary layer transition in 3D flows is fundamentally different from that in planar flows because of the occurrence of cross-flow instability, and to date not much work has been done to study the evolution and interactions of cross-flow vortices. The aims of this project are to extend some of the ideas of unsteady nonlinear critical layer theory to 3D flows and to address some of these issues. As a start the analysis of Gajjar (1992) was extended to include flows with multiple critical layers, and in particular the evolution of long wavelength, low frequency, cross-flow vortices in compressible boundary layers was investigated. The problem reduces to solving a set of coupled unsteady nonlinear critical layer equations for the upper and lower critical layers, with the evolution of the disturbance linked to the velocity jumps across both critical layers via an amplitude equation. Much of my effort here was devoted to developing the numerical code to solve this problem. The method based on Lagrangian coordinates developed by Goldstein & Wundrow (1991), and the Spectral collocation method of Gajjar (1992) are being used to solve the numerical problem. Currently the Lagrangian method is still being tested by comparing with the results for the one critical layer case of Goldstein & Wundrow, but it should be possible to obtain some new nonlinear results for the two critical layer case with the Chebychev method in the next few months.

MAX D. GUNZBURGER, Virginia Tech

The main thrust of our effort has been towards the development, analysis, and implementation of finite element least-squares algorithms for partial differential equations, with special emphasis on the Navier-Stokes equations. Dr. Bo-Nan Jiang of ICOMP and Professor C. L. Chang of Cleveland State University have been two leading pioneers in this field. Our work has been done in consultation with both of them, especially Dr. Jiang. Our own interest has been centered on understanding various beneficial properties of these types of algorithms. For linear problems it is well-understood that discretizations of this type lead to symmetric, positive linear systems. We have shown that for nonlinear problems, if Newton's method is used as a linearizer, this property is preserved in the neighborhood of the solution, regardless of the value of the Reynolds-number. The size of the neighborhood decreases with increasing Reynolds-number. However, least squares methods, coupled with a continuation method, offer the following advantage: you can solve the Navier-Stokes equations for any value of the Reynolds-number by solving a sequence of positive definite linear systems of equations. This of course, implies great advantages with regard to computer storage and CPU requirements.

Another potential advantage of least squares methods is that they allow for the use of equal order interpolation for all dependent variables. However, a potentially serious drawback has emerged. Contrary to the claims of some previous researchers, it seems that these methods are not optimally accurate if velocity boundary conditions are specified. We have carried out extensive computational studies of this issue, and have concluded that, although the method is not optimally accurate, it is nearly so. Thus, from a practical point of view, we have sufficient accuracy to make the method viable. (We should note that to our knowledge, all known practical methods that employ the vorticity as a dependent variable suffer from a similar loss of accuracy; in this regard, we are no worse off than with other methods.)

We are currently preparing a paper describing our experiences with least squares method for the Navier-Stokes equations. The paper will contain an analysis of errors, as well as the results of some calculations.

We have also been involved with flow optimization problems. Here our work deals with the design of algorithms for flow control. We have successfully implemented algorithms with heating and cooling controls and are now in the process of developing algorithms for the shape control of flows. This work is in the preliminary stage.

THOMAS HAGSTROM, University of New Mexico

In ongoing work with Professor S. I. Hariharan of the University of Akron and ICOMP, accurate radiation type boundary conditions are being developed and tested for unsteady compressible flow simulations in exterior domains. Our main goal has been to try to translate to the Euler equations accurate conditions for the wave equation. The latter are based on progressive wave expansions valid in the far field. We test our conditions (along with others) on the difficult problem of transonic flow past an impulsively started cylinder. This involves the propagation of a shock through the artificial boundary.

Past experience indicated that the direct application of high order conditions to, for example, the pressure leads to oscillations and instability when the shock reaches the boundary. Therefore, we have developed conditions involving the Riemann variables, also avoiding space derivatives. These are based on new progressive wave expansions we have developed for the linearized Euler equations, written in terms of the (radial) Riemann variables. This leads to a hierarchy of boundary conditions of increasing asymptotic accuracy. We have so far only implemented the lowest order (of accuracy) condition in this hierarchy. Encouragingly, we have found it to be stable, with accuracy comparable to conditions in current use. We are now in the process of deriving and implementing the second order condition.

In joint work with Dr. K. Radhakrishnan of NASA we are developing a high order numerical algorithm for simulating unsteady reacting flows governed by the zero Mach number asymptotic limit of the governing equations. Among the features we are building in are:

1. High order spatial and, eventually, temporal discretizations and adaptive meshing, as has been successfully used to study complex dynamic phenomena for simplified combustion models.

2. The ability to incorporate general reaction and diffusive transport mechanisms.

3. Solution of asymptotic equations which include a strong coupling between the reaction and the flow field, but which exclude the complicating feature of "fast" sound waves.

So far we have developed fourth and eighth order approximations of the convection and diffusion terms. The latter are elliptic, though nonsymmetric due to the boundary conditions. We have implemented a nonlinear preconditioned conjugate gradient squared algorithm to solve the implicit time-stepping equations, which at present is a second order splitting scheme. We are planning to implement a higher order coupled time integration algorithm based on BDF-type formulas. Experiments with the propagation of plane hydrogen-oxygen flames are underway. The fundamental goal of this research is to examine flame dynamics with more realistic models of the underlying physics.

In joint work with Dr. John Goodrich of NASA we are completing a comparative study of outflow boundary conditions for incompressible channel flows, using a streamfunction based solver in two space dimensions. Included are steady and unsteady flows over backward facing steps, in symmetric expansions, as well as vortical flows. We are finding that the asymptotic conditions we previously developed yield quite acceptable results, even for fairly complex flows at the artificial boundary.

S. I. HARIHARAN, The University of Akron

During this year a variety of interesting and useful problems related to computations of unsteady compressible flows was considered. The ongoing work with Professor Thomas Hagstrom has reached another level of maturity in terms of understanding and implementing far field boundary conditions for external flows governed by the Euler equations. Details of this work are described by Professor Hagstrom in his report. We are also in the process of extending this work to viscous flows.

The linearized Euler equations are becoming popular for simulations of unsteady effects in aerodynamics. However, there are unresolved issues related to computing solutions of these equations, such as construction of appropriate far field boundary conditions. Potential flow formulations that began several decades ago, also

based on the linearized Euler equations, have the advantage that the field equations are simple and the associated far field boundary conditions for these formulations are well understood. However, computational methods associated with these potential flow formulations do suffer with the inaccuracies associated with the prescription of Kutta condition along wake lines. These lines are part of the solution of the mean flow and should be known a *priori*. Alternate methods as well as formulations to address this problem are under investigation.

In the area of acoustics, in particular problems related to jet noise, there is a need for computational procedures for the accurate prediction of far field noise. A simplified boundary integral equation formulation has been derived for the prediction of the far field sound. This formulation also has a dual use of prescribing absorbing boundary conditions for the near field calculations. This formulation will be tested against existing procedures in the coming year.

M. E. HAYDER (Postdoctoral), Princeton University

The goal of my present research at ICOMP is to compute aerodynamic noise in a supersonic jet. In the past year, I collaborated with Professor Eli Turkel (Tel Aviv University/ICOMP) and Dr. Reda R. Mankbadi (NASA Lewis) in developing and validating the numerical models. Numerical models for plane and axisymmetric jets have been developed using the 2-4 scheme by Gottlieb and Turkel. Direct numerical simulation results using these codes compare very well with those of the linear theory. A study was made to improve nonreflecting boundary conditions. Also, a high order viscous correction to the 2-4 scheme was proposed. The axisymmetric code is being extended to a 3D round jet code. Pseudo spectral collocation is used in the azimuthal direction. At the present, work is concentrated in the following extensions of the models:
1. Implementation of a subgrid turbulence model and large eddy simulations.
2. Introduction of a nozzle and a study of noise originating from the exit of the nozzle.

LIN-JUN HOU (Postdoctoral), Georgia Institute of Technology

The rate of convergence using different preconditioned matrices for the Least-Squares finite element method (LSFEM) has been studied. The diagonal-block preconditioned matrix is used in the Jacobi preconditioned conjugate gradient method (JPCG) instead of the Jacobi matrix. For better results, the trade-off is the calculation of each diagonal block matrix whose size is equal to the number of degrees of freedom for each node. However, the matrices for the reduced-order system in LSFEM are sometimes problem dependent and for matrices which are *strong diagonally dominated*, the Jacobi preconditioned matrix is a better choice.

Another project is a study of the 3D driven cavity flow at various Reynolds numbers. Due to the difficulties of existing finite element methods for large-scale computation, finite element solutions for this problem are scarce. Most numerical efforts are restrictive in mesh size and insufficient in resolution, due primarily to the limitations of numerical methods and computer resources. With the characteristics of LSFEM, the problem is solved by matrix-free iterative methods. Numerical results show that no steady solution exists for $Re < 400$, and for $Re \leq 400$ current results are in good agreement with those obtained by Iwatsu *et al* using a finite difference method with very fine mesh. The presence of Taylor-Görtler-like vortices is observed at $Re = 1000$. A further example, 3D backward facing step flow, was also examined. A simplified marker-and-cell (SMAC) finite-difference scheme is applied by Ikohagi and co-workers for the same 3D problem in curvilinear coordinates. However, no comparisons have been made with either the experimental data by Armaly *et al* or with other simulation results, and thus no primary conclusions can be really made. Current results show that at $Re = 277$, the reattachment length of the recirculating zone behind the step matches the experimental results and the vortex is stronger in the region of side walls and has negligible influence on the region near the mid-span. Study of velocity vectors along the span at different downstream locations indicates that even with $Re < 400$, 3D phenomena have occurred. Simulation of flow with higher Re numbers is in progress and 3D effects are expected to become more serious.

Present research also focuses on the development of an unstructured mesh generator for adaptive finite element analysis which is needed most in irregular heterogeneous regions.

J. MARK JANUS, Mississippi State University

The focal point of my two week visit at ICOMP involved the study of relative-motion subdomain-interface solution dynamics. The primary research tool used was a model problem which contained a relative-motion subdomain interface (i.e. an interior boundary condition). The "configuration" is one which Dennis Huff studies to demonstrate the need for accurate (nonreflecting) boundary conditions in turbomachinery flow simulations if one seeks acoustic information. It involves a cascade of flat plates undergoing very small amplitude oscillations creating acoustic disturbances throughout the domain. These disturbances propagate upstream and downstream and should pass "through" the inflow and outflow boundaries. Ideally, for the (sub)domains of my concern, these acoustic waves should pass "through" a relative-motion interior boundary, *undisturbed*.

During my visit, software developed for studying model problems of this type was debugged, made operational, and enhanced. The starting point was a code originally developed for the simulation of 3D rotating machinery. The code was modified for 2D cascades undergoing small amplitude oscillations during the final days of my stay last year and was not completely operational. Presently the code is capable of simulating an oscillating cascade with the downstream portion of the domain either fixed or in relative motion with the subdomain containing the flat plates. The cascade operates with zero interblade phase angle, creating acoustic disturbances (a cut-on propagating mode which travels streamwise, both upstream and down-stream). Although the results at the time of this report are preliminary with regard to interior boundary efficacy, test runs have been completed which confirm the distortion of acoustic waves due to grid motion. This is expected yet further study is warranted to determine the extent and impact of the distortion.

Several subdomain interface treatment methodologies were tested. These include an overlapped interpolation technique in which grid line continuity was not maintained from one subdomain to the other with an overlap of four cells (two from each subdomain). This necessitated the passage of data across the interface via a simple linear interpolation routine. A non-interpolated variant (referred to as clicking) was also tested. For the clicking routine grid continuity was maintained by uniform grid spacing at the boundary and appropriate time-step selection. This yielded a simple extraction-injection routine for data passage across the interface. The final approach tested was that of continuous grid skewing (distortion) to accommodate relative grid motion. For that approach grid line continuity was maintained along with keeping fixed index partners across the interface. Thus extraction-injection was used with no changing of index partners across the interface.

To simulate the signal measured by a microphone, the pressure time history was monitored at a sample point three-quarter chords downstream and one-quarter chord above the plate. The point was located ahead of the interface within the stationary subdomain containing the oscillating cascade. A comparison was made of the different techniques after three plate oscillations (240 time steps). Initial indications show that grid distortion has the least impact on wave travel and solution variation is thought to be due to reflections off of the interior boundary established between the two subdomains.

This effort complements those performed during previous ICOMP visits and has become sufficiently mature to garner support from those at NASA Lewis to see that conclusive results are obtained.

BO-NAN JIANG, University of Texas, Austin

The least-squares finite element method (LSFEM) based on the velocity-pressure-vorticity formulation is applied to 3D steady incompressible Navier-Stokes problems. This method can accommodate equal-order interpolation and results in a symmetric, positive definite algebraic system. An additional compatibility equation, i.e., the divergence of vorticity vector which should be zero, is included to make the first-order system elliptic. Newton's method is employed to linearize the partial differential equations, LSFEM is used to obtain discretized equations, and the system of algebraic equations is solved using the Jacobi preconditioned conjugate gradient method which avoids formation of either element or global matrices (matrix-free) to achieve high efficiency. In this method there is neither upwinding, adjustable parameters, numerical boundary conditions, splitting, projection, nor artificial compressibility. Besides the finite element interpolation and the linearization, no other approximation is introduced into this method. Therefore, solutions by this method are more reliable than that of any other existing method. The flow in a 3D cubic cavity is calculated at Re = 100, 400, and 1,000 with 50 X 52 X 50 trilinear elements. We found that the 3D driven cavity flow was unstable

for Re ≥ 1000, and the presence of Taylor-Görtler-like vortices was observed for the case of Re = 1000. Previous researchers found these phenomena only for Re ≥ 2000.

KAI-HSIUNG KAO (Postdoctoral), University of Colorado at Boulder

The development of a detailed method to study the aerodynamic behavior of complex configurations, such as forebody/inlet combinations, propulsion system integration, high speed aircraft, and so forth, is proposed for this research. Numerical procedures to compute the flowfield around complicated geometries will be developed and discussed. The computational domain will consist of a number of overset subgrids to simplify the surface definition and grid generation procedure. By using composite overlapping meshes, an objective of the study is also to develop a grid adaption method that not only enhances the accuracy of flow solutions using fine grid in high-gradient regions, but also is capable of handling complex geometries with cells of minimal skewness. An accurate and efficient numerical scheme will be used in the Navier-Stokes code in generalized coordinates.

Advanced algorithms exist, yet their limitations for complex flow fields are still not totally understood. It is known that proper selection of a numerical algorithm is a key issue in order to resolve complicated mechanisms such as shock wave/boundary layer interaction and high speed viscous flow with complex configurations. There are various methods and numerical schemes for solving the system equations. Many of them may display high accuracy and fast convergence rate in simple geometries. However, for complex configurations, they may not be capable of resolving the complex flow as expected. A new class of flux splitting scheme (AUSM) developed by Dr. Liou (1) has been incorporated into the Navier-Stokes code. It is notable that the capability of the AUSM scheme for accurately predicting shock waves and viscous layers has been confirmed. Good qualitative numerical results are also obtained in M5 waverider and Space Shuttle Orbiter simulations.

The grid construction for the proposed work would initially seem tedious; however, a composite grid system will be employed to greatly simplify the grid generation. Adoption of the CHIMERA scheme (2) allows discretization of the volume about the complex configuration as a collection of structured grids. The overlapped grid approach was chosen over a patched scheme to reduce the labor time involved in domain decomposition. The overall philosophy behind the use of overlapped grids is that individual component grids can be generated easily if one does not have to match to other grids or pieces of geometry. This advantage greatly reduces the tedium of trying to generate a complex 3D grid to define both external and internal flows.

The separate grid system will be generated using the GRIDGEN package (3) on the measured surface data. The GRIDGEN codes were developed to provide efficient, user-friendly and robust grid generation tools that would allow for routine applications of multiple block, computational fluid dynamics flow solvers to complex aircraft configurations.

At present, the Chimera grid scheme has been successfully applied in combination with the Navier-Stokes solver. Solution-adaption enhancement is also performed by using a secondary fine grid system which oversets a base grid in the high-gradient region, but without requiring the mesh boundaries to join in any special way. Applications to the Euler equations for shock reflections and to a shock wave/boundary layer interaction problem are being tested. With the present method, the salient features are well resolved.

By using the proposed numerical techniques, the preliminary design of complex bodies can be accelerated and much more efficient. Design improvements will be suggested upon thorough examination of the computational results.

REFERENCES

Liou, M.-S.; and Steffen, C. J.: A New Flux Splitting. NASA TM 104452, 1991.

Benek, J.A.; Steger, J.L.; Dougherty, F.C.; and Buning, P.G.: Chimera: A Grid Embedding Technique. AEDC-TR-85-64 (AD-A167466), April 1986.

Steinbrenner, J.P.; Chawner, J.R.; and Fouts, C.L.: The GRIDGEN 3D Multiple Block Grid Generation System. WRDC-TR-90-3022, February 1991.

JOHN KOSMATKA, University of California, San Diego

A newly developed nonlinear structural analysis computer program was used to study the extension bend-twist coupling behavior present in spinning laminated composite turbine and unducted fan blades. Twelve different laminated composite plates having initial twist were analyzed to span the range of parameters found in turbine blades and unducted fan blades including, initial twist, initial twist axis location, and symmetric and asymmetric ply definition. Furthermore, the coupling behavior was investigated for four different blade pitch and sweep settings. Blade displacements, rotations, strains, stresses, and natural frequencies of vibration were calculated as a function of rotational speed. Preliminary observations include: (1) presence of pretwist has a minor effect on the first bending natural frequency, but greatly effects all other bending frequencies and all torsion frequencies, (2) initial twist axis location has a minor effect on the blade frequencies, but it significantly effects the mode shapes and steady state displacements and stresses, (3) while stacking sequence greatly effects the bending frequencies, it is overshadowed by the pretwist effects as a means of controlling the torsion frequencies, and (4) asymmetric stacking sequences can be used to control the extension-bend-twist coupling behavior inherently present in turbine and unducted fan blades.

The results of this work will be written in manuscript form and submitted for publication, where the support of the ICOMP program will be properly acknowledged. This work was sponsored by the Structural Dynamics Branch of the Structures Division with O. Mehmed and G. Stefko acting as the technical representatives.

B. P. LEONARD, The University of Akron

During 1992, my ICOMP research has been concentrated in three areas. These might be categorized as (1) fundamental CFD algorithm development, (2) engineering-accuracy turbulence modeling, and (3) "technical politics" (relating to numerical uncertainty). Most of my own technical work has been in category (1). Item (2) has been in conjunction with my colleague, Dr. J. E. Drummond, at Akron, and NASA Lewis personnel, Julie Conley and Nick Georgiadis. The third item, although less technical, is probably the most important, if progress is to be made (and not reversed) in CFD and turbulence modeling for momentum, heat, and mass transport under (the most commonly occurring) highly convective conditions. The following outline gives somewhat more detail.

(1) CFD algorithm development.

The superficially simple problem of highly (or purely) convective transport of a scalar in a simple, prescribed, velocity field remains one of the most challenging problems in computational mechanics. Genuinely multidimensional one-step (two time-level) explicit convection schemes require certain cross-difference terms to maintain stability and isotropy. This has been attained to third-order with UTOPIA and UTOPIA-plus. Diffusion terms have been designed to the same order. However, such schemes are not positivity preserving. Devising truly multidimensional flux-limiters has occupied a major portion of my research effort. A 2D generalization of my ULTIMATE scheme has been constructed. As in the 1D case, the limiter introduces some distortion of the profile (while maintaining positivity). Results (in 1D and 2D) are less than satisfactory. A radically new approach has recently been embarked upon. I call this the NIRVANA project (nonoscillatory, integrally reconstructed, volume-averaged numerical advection). The idea is to introduce a discrete integral variable (in 1D this is the cumulative sum of the finite-volume averages of the advected scalar); the crucial step is the appropriate interpolation of this variable; analytic differentiation then gives sub-grid detail so that highly accurate fluxes can be calculated (in 1D, the flux-difference is a computationally inexpensive two-point difference in the continuous integral variable itself). This gives an explicit scheme with no stability restriction on the time step; i.e., the conventional wisdom (the CFL-condition) requiring the Courant number to be less than unity is wrong! Nonoscillatory results require convexity preservation in the integral-variable interpolation. These ideas can be extended to 2D (and 3D), but the interpolation problem is much more difficult.

(2) Practical turbulence modeling.

Although I'm not a turbulence-model "expert", my philosophy for some time has been that there is a strong need for an unsophisticated "engineering-accuracy" turbulence model that can be applied to practical

problems. This would be something like a Cebeci-Smith boundary-layer model; but that requires knowledge of a well-defined boundary-layer thickness, so what do you do if there's some separation? The Baldwin-Lomax model tries to address this (but sometimes fails in exactly the case it was supposedly designed to handle - as is well known!). My colleagues and I at Akron and Lewis have been trying to develop a model in the spirit of Baldwin-Lomax, while avoiding its pitfalls. We call this the "modified mixing-length" (MML) strategy. It is a "brute-force" empirical approach; it does not require a well-defined boundary-layer thickness, relying instead on the wall shear (with some simple strategies for handling the singularities near separation and reattachment points). Results have been surprisingly successful (accurately predicting massive separation - where Baldwin-Lomax fails miserably) and, in some cases performing as well as (or better than) k-ε for one-seventh the cost! Although this is not a very "elegant" approach to turbulence modeling, we are very excited about the possibilities of generating practical (inexpensive) results of engineering accuracy. This of course, has particular relevance to computationally intensive projects such as the HSCT, where higher-order closures would be exorbitantly expensive.

(3) **Numerical uncertainty**.

Finally, I continue to be concerned about the (increasing) use of low-accuracy CFD schemes containing implicit (or explicit) artificial viscosity (diffusivity). The main "culprits" are Spalding's Hybrid scheme and Patankar's Power-Law Difference Scheme (PLDS). Researchers increasingly use these methods out of context. They can (but need not) be used for quasi-1D flows when the flow is aligned with one of the grids; but must not be used when the flow is oblique (or skew) to the grid lines. The artificial numerical diffusion this introduces has been known about for seventeen years (from deVahl-Davis and Mallinson's well-known paper in *Computers and Fluids*). What is often not realized is that the turbulence model (although computed at great expense) is actually switched-off (Hybrid) or totally suppressed (PLDS) under practical flow conditions. How can you test a turbulence model when you're switching it off and replacing it with artificial numerical viscosity and diffusivity? I have been trying to get major CFD/heat-transfer journals to adopt an editorial policy on "numerical uncertainty" similar to that adopted by the ASME *Journal of Fluids Engineering* in 1986, effectively outlawing first-order-based schemes (such as Hybrid and PLDS). Some progress has been made. Jerry Drummond and I are presenting a paper entitled "Why You Should Not Use "Hybrid", "Power-Law", or "Related Schemes for Convective Modeling - There Are Much Better Alternatives" at the *Eighth International Conference on Numerical Methods in Thermal Problems* in Swansea in July 1993 (this is where some of the worst offenders are likely to present their artificially diffusive results). The alternatives, by the way, include my ULTRA-SHARP schemes developed, with ICOMP and NSF support, in previous years.

JOSEPH T. C. LIU, Brown University

The work on closure models for unsteady turbulent shear flow for small scale modulated stresses was continued. Rapid-distortion theory is re-examined for adaptation to unsteady closure: The ratio of turbulent shear stress to kinetic energy becomes a function of effective strain, which, in turn, accounts for the history of the flow. The aim is to pave the way towards broader issues associated with LES (large eddy simulation) and subgrid scale models and their present difficulties in principal and in application in problems of engineering interest.

M. T. LANDAHL, MIT

The work has focused on the evolution of a 3D eddy in the near-wall region of a boundary layer with strong mean shear. An expansion in terms of three different and separate time scales, a shear interaction one, a viscous one, and a nonlinear one, shows that a localized eddy tends into a purely convected one (a Taylor "frozen flow" structure) which develops a streaky structure, spanwise localized shear layers, and streamwise vorticity with strength growing linearly in time, as the time becomes large compared to the shear interaction time scale. During the viscous stage the eddy decays as $t^{-3/2}$. During the nonlinear stage the flow may develop local singularities resembling in structure those seen during sublayer bursting. The aim of this work (Landahl, 1993) is to serve as a basis for the construction of a new near-wall turbulence model which will avoid the need for the empirical damping functions presently used with κ-ε and similar models.

Landahl, M. T. 1993, "Model for the Wall Layer Structure of a Turbulent Shear Flow," *European Mechanics Journal B* (to appear).

WILLIAM W. LIOU (Postdoctoral), Pennsylvania State University

My research project involves the development of turbulence models for compressible flows and their validations/applications in flows of engineering interest. The goal is to develop compressible second-order Reynolds stress models. An important factor is the identification of the dominant effects of compressibility on turbulence. Several modeling methodologies are being examined. Based on physical reasoning and the results of DNS of compressible turbulence, a multiple-scale model has been developed in collaboration with T.-H. Shih and B.D. Duncan. In high speed free shear layers where the turbulent mixing is affected significantly by compressibility effects, the model predicts quite well some experimentally observed behavior. The plan now is to assess the model in problems of complex flow phenomena. During the reporting period, work has also been performed in the development of a weakly nonlinear wave model. A length scale is related to the wavelength of the dominant modes in turbulent free shear flows. This enables an inviscid characterization of the energy transfer from large-scale to small-scale turbulence. It is hoped that this new modeling approach will help identify the significance of flow compressibility on the dynamics of turbulence structures.

JAMES LOELLBACH (Postdoctoral), University of Illinois

I have been working with Dr. Chunill Hah on developing grid generation capabilities for axial and centrifugal turbomachinery configurations. This work is part of an ongoing effort by the Consortium for CFD Application in Propulsion Technology centered at NASA Marshall Space Flight Center. The goals of the Consortium are to first validate a set of numerical solution packages against experimental data for a series of turbomachinery problems, and then to apply these packages to the design of advanced turbopump components of liquid-fueled rocket engines. Due to the geometrical complexity of these components, the effort expended on grid generation is often a large part of the overall solution cost. I have been using a combination of algebraic and elliptic equation techniques to develop grid generation packages for axial compressor and turbine blade rows and centrifugal pump inducers and impellers. Emphasis has been placed on increasing the robustness and ease of use of these packages and on reducing grid skewness to improve the accuracy and convergence rates of flow solutions. Work is continuing on increasing the range of configurations to which these packages can be applied.

S. MASLOWE, McGill University

My primary effort has been directed toward the analysis of resonant triad interactions in incompressible free shear layers. The goal is to formulate a rational theory capable of explaining vortex-pairing phenomena in mixing layers. Previous theories based on the subharmonic resonant interaction between two plane waves due to Kelly (1967), and amended by Monkewitz (1988), seem to explain some features of the relevant experiments. However, the assumptions invoked in such approaches (e.g., near neutrality of the interacting modes) are not valid for plane waves satisfying the resonance conditions.

These difficulties can be overcome by choosing the subharmonic component of the interaction to be a pair of oblique waves. Various asymptotic developments are possible according to whether the amplitude of the oblique waves is larger, smaller or equal in order of magnitude terms to that of the plane wave. When all modes are initially of the same order of magnitude, the oblique waves exhibit very rapid (exponential of an exponential) amplification. Although such behavior has not yet been observed experimentally, this may be due to geometrical constraints of the experimental apparatus.

A formulation more consistent with the experimental observations reported to date is presently being developed by Dr. Lennart Hultgren and myself. The oblique waves will initially have smaller amplitudes than the plane wave so that the latter will saturate, as it does when evolving alone. During this stage of the evolution, the critical layer thickness associated with the oblique waves will be larger (due to their rapid amplification) than that of the plane (2D) wave. Evolution equations appropriate to the next stage of the process will then describe the onset of vortex-pairing according to our analysis (which parallels a similar investigation of the adverse pressure gradient boundary layer described in a paper presently being prepared by Wundrow, Hultgren and Goldstein).

J. MATHEW, Princeton University

The action of externally imposed, weak, streamwise vorticity on an incompressible, mixing layer was studied in collaboration with Dr. M. E. Goldstein.

The formulation and computations of both the outer inviscid region and the viscous shear layer were completed during the first year. The solution of the inviscid problem broke down at a finite downstream location. Several approaches to determine the nature of this breakdown were taken. As the appearance of a singularity and consequent breakdown is not unexpected, I looked for approximate analytical forms of the singularity to determine its structure and strength. However, no convincing match between analytical forms and numerical solutions in the vicinity of the breakdown was found.

In the related, special case of Rayleigh-Taylor instability of an interface between a fluid and vacuum, driven by gravity, a real singularity does not appear at a finite time. Singularities are present in the continued complex plane of the functions and they approach the real axis causing usual numerical schemes to breakdown. As the present problem is equivalent except that the interface evolution is driven by vorticity in the fluid and not gravity, it is possible that the breakdown is similar and the singularities remain in the complex plane. To determine that this is indeed the case, a more accurate scheme needs to be developed.

A paper discussing this work was written and will appear in the Physics of Fluids.

A. F. MESSITER, University of Michigan

Possibilities have been considered, following a suggestion of M. E. Goldstein, for describing 3D flow separation that results from boundary-layer distortion by free-stream vorticity normal to a flat plate (Goldstein, Leib and Cowley, J. Fluid Mech. 1992). The general idea was to study a 2D cross-flow far downstream, including a local boundary-layer interaction, characterized by a "triple-deck" structure, linked with a free-streamline solution, with rotational external flow. Various uncertainties suggest that the better approach for the moment is to continue and extend boundary-layer calculations near the first appearance of singular behavior, as in the current work of D. W. Wundrow.

I have also continued work initiated last year concerning nonlinear instability of a supersonic vortex sheet, especially in relation to long-wave instability of a shear layer having nonzero thickness as currently under study by T. F. Balsa. While the time scale for wave steepening to become appreciable in the external flow is larger than the scale for nonlinearity in the viscous critical layer, it appears that a minor extension of Balsa's work to a larger wave amplitude leads to a decrease in the relative magnitude of critical-layer disturbances, until they become of higher order than the disturbances in the external flow.

ROY NICOLAIDES, Carnegie Mellon University

This research is centered around the computation of unstructured meshes of the Voronoi-Delaunay type. There are a number of difficulties which have to be overcome when generating these meshes for realistic aerodynamic configurations. An important issue is the triangulation of nonconvex domains and that is one of the problems addressed in the research. We have developed a novel approach, based on the concept of a constrained Delaunay tessellation for handling nonconvexity in an automatic way. Our new algorithm is optimally efficient having 0(n) time and space complexity. It has been applied to several 2D problems of practical interest. The correctness of the algorithm has been rigorously proved.

ALEXANDER ORON, Technion-Israel Institute of Technology

During my 5 weeks visit at ICOMP, I worked in collaboration with Dr. R.J. Deissler of ICOMP and Dr. J.C. Duh of Sverdrup Tech. on two problems.

The first is the problem of the preferred wavelength in the Marangoni convection in a fluid layer after onset of instability, the so-called supercritical Marangoni convection. The main motivation is to explain the experimental results where the characteristic cell size is shown to decrease first and then to increase past the criticality. The reasons for that behavior are unclear yet. The approach to the problem solution which has been adopted in the current work, is to develop, from the governing equations and the boundary conditions, an

amplitude equation of the Ginzburg-Landau type $\frac{\partial A}{\partial \tau} = \varepsilon^2 \alpha A + \beta \frac{\partial^2 A}{\partial \chi^2} - \gamma |A|^2 A$ where χ, τ are respectively spatial

and temporal variables, α, β, γ are positive parameters determined via the physical properties of the system at equilibrium, ε is a small parameter describing the difference between the actual and the critical values of the Marangoni number, and A is an amplitude function related to the flow field. The features of the flow pattern are to be recovered by investigation of the steady solutions of the Ginzburg-Landau equation mentioned above.

The second problem is that of the onset of Marangoni convection in an extended fluid layer with two free surfaces. This model can simulate instability in a layer flowing on top of a heavier fluid layer. Thus, generally the values of the Marangoni number are different at the two interfaces.

A condition for the first onset of the longwave instability out of the quiescent state for a layer with nearly insulating free boundaries heated from below is derived: where M_u, M_i are respectively the values of the Marangoni number at the upper and the lower interfaces.

A nonlinear evolution equation describing the spatiotemporal behavior of the flow slightly above the criticality has been derived. A study of the nonlinear evolution of the velocity and temperature fields is now underway.

CHRISTOPHE PIERRE, University of Michigan

The basic purpose of this research is the development of computational tools for understanding and predicting the effects of unavoidable blade-to-blade dissimilarities, or mistuning, on the dynamics of nearly cyclic bladed-disk assemblies. This topic is of importance as mistuning has been shown to increase the forced response amplitudes of some blades significantly, and even to lead to single blade failure. Furthermore, the trend toward high performance propulsion turbomachinery designed for finite service life demands an accurate prediction of system performance and dynamics.

This past year, research supported by ICOMP has focused on the following topics. First, we have developed a stochastic measure of sensitivity to mistuning for blade assemblies with aerodynamic interblade coupling. This simple predictive tool allows one to evaluate the effects of mistuning without requiring obtaining solutions for mistuned systems. Thus it has the potential to be effective at the design stage. In particular, the need for high-cost mistuned dynamic analyses is alleviated if a particular rotor design if found to be insensitive to random blade mistuning. The results of this study are reported in [1].

Second, we have developed a transfer matrix methodology that allows us to cast the dynamics of blade assemblies in terms of the coupling coordinates between blades, which has important conceptual as well as computational advantages. In particular, using a wave propagation analysis, we were able to formulate an easily calculated and universal measure of the strength of the phenomenon of vibration localization in mistuned blade assemblies with one coupling coordinate between blades. (Blade assemblies which are highly sensitive to mistuning are subject to the localization phenomenon, in which nearly all the vibrational energy is concentrated within a few blades rather than being distributed along the rotor. Localization is believed to be one of the causes of single blade failure.) A paper has been written which describes these results [2].

Third, we have examined the dynamics of mistuned assemblies with several component modes per blade. In particular, the interaction of two blade modes which have close frequencies (for example, a bending mode and a torsion mode, or two modes of a plate-like blade) and the resulting effect on the sensitivity of the assembly to mistuning have been explored. An ICOMP paper is currently being written [3].

PUBLICATIONS

1. Murthy, D.V.; and Pierre, C.: Stochastic Sensitivity Measure for Mistuned High-Performance Turbines. NASA Technical Memorandum 105821, ICOMP 92-13; Also, Proceedings of the Fourth International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC-4), Honolulu, Hawaii, April 5-8, 1992.

2. Ottarsson, G.; and Pierre, C.: A Transfer Matrix Approach to Vibration Localization in Mistuned Blade Assemblies. ICOMP Report in progress; Also submitted to the 34th AIAA/ASME Structures, Structural Dynamics, and Materials Conference, San Diego, California, April 18-21, 1993.

3. Pierre, C.; and Murthy, D.V.: Aeroelastic Dynamics of Mistuned Blade Assemblies with Closely Spaced Modes. Submitted to the 34th AIAA/ASME Structures, Structural Dynamics, and Materials Conference, San Diego, California, April 18-21, 1993.

R. H. PLETCHER, Iowa State University

Work continued with Philip Jorgenson on the simulation of internal viscous flows using unstructured grids. The research has been motivated by the need to develop improved methods for dealing with flows in complex geometries. An implicit, finite-volume based scheme has been developed for the compressible form of both the Euler and Navier-Stokes equations in 2D. The scheme includes preconditioning to permit efficient solutions to be obtained even at very low Mach numbers. At present the algebraic system is being solved by a four-color block Gauss-Seidel scheme, although work continues on the evaluation of other candidate schemes including a generalized minimum residual procedure. Computations are in progress to establish the merits of the formulation. Test cases include the subsonic and transonic inviscid flow over a circular arc airfoil, developing flow in a channel at several Reynolds numbers, flow over a rearward-facing step (sudden expansion), flow over periodic tandem cylinders in crossflow in a channel and flow through a four port valve. An effort was also initiated to extend the unstructured simulation capability to 3D. An "allspeed," time-accurate formulation of the Navier-Stokes equations is planned.

Further work was done on the low Mach number conditioning of the discretized compressible time-dependent Navier-Stokes equations. The most recent work indicates that the convergence rate for the conditioned system remains remarkably independent of Mach number for Mach numbers below about 0.2 for the flows considered to date. It appears that time dependent problems can be solved by this procedure.

Collaboration with Lewis personnel is also in progress on the further development of efficient numerical schemes for combustion applications.

KENNETH G. POWELL, University of Michigan

A tree-based grid-generation procedure for solution of the 2D Navier-Stokes equations at high Reynolds number was implemented during my stay, and some preliminary tests of the methods were carried out. The method is made-up of the following steps:

1. A single large square is constructed that covers the entire flow domain — this cell serves as the root of the tree;

2. A cell that contains any portion of the bodies spawns four children cells — this process is done recursively until the largest body face is below some user-specified limit;

3. For each cell in this mesh, the distance from the cell centroid to the nearest body point is calculated;

4. The intersections of an isoline of this distance function with the mesh are calculated;

5. Each cell of the mesh which is cut by the isoline spawns two children cells;

6. The last two steps are repeated for several (small) values of the distance function, until suitable resolution in the boundary layer is obtained.

The resulting mesh looks like an overlay of a "collar-grid" around the bodies in the flow on a background cut-Cartesian mesh, but is stored in a manner consistent with an unstructured-grid solver. Of the steps listed above, the first two were previous work, steps 3-5 were carried out during my stay, in collaboration with Bill Coirier. The final step requires a slight modification of the method we developed during my stay, and should be completed quite soon. Follow-up work will concentrate on making this grid generation robust for general geometries (we have tested it on analytically defined shapes, airfoils and multi-element airfoils) and on discretization of the viscous terms on the resulting meshes.

THOMAS H. RAMIN (Graduate Student), Iowa State University

The purpose of the visit was to conduct research on 3D unstructured grids, ultimately leading to the development of a complete flow code. Preliminary work was done on unstructured grid generation and experience was gained in the use of existing 2D and 3D grid generators. Existing grid generators are regarded as sufficient for the initial phases of the project and code validation. In the framework of algorithm development, various schemes such as flux splitting and central difference with added artificial viscosity were investigated using established 1D testcases for comparison of results.

S. G. RUBIN, P. K. KHOSLA AND D. BROWN (Graduate Student), University of Cincinnati

A 2D (in space) inlet code developed at the University of Cincinnati has been transferred to, and debugged and run, for designated test cases on the NASA Lewis YMP. The code, which is valid for steady state and transient calculations, and for incompressible to large Mach number supersonic flows, is based on a reduced Navier-Stokes or RNS methodology. A deferred-corrector default is available for full Navier-Stokes solutions, and another default specification results in a reduction to an initial value (in space) PNS solver. The full RNS implementation allows for shock-boundary layer and shock-shock interaction, and for flow with axial and/or secondary flow (in 3D) recirculation. The 3D version of the code has been validated for several subsonic and low supersonic cases. Appropriate 3D shock limiters or artificial viscosity shock capturing techniques for the secondary flow behavior are under development for very strong, large Mach number, oblique shocks. This version will then be validated against available NASA Lewis databases through an ongoing interaction program with Dr. D. R. Reddy and Dr. H. S. Lai of NASA Lewis.

AAMIR SHABBIR (Postdoctoral), SUNY Buffalo

The best way of verifying turbulence models is to perform a direct comparison between the various terms and their models. This approach has been used by several researchers. The success of this approach depends upon the availability of the data for the exact correlations (both from experiments and direct numerical simulations). The other approach for testing these models is to numerically solve the differential equations and then compare the results with data. The results of such a computation will depend upon the accuracy of all the modeled terms. Because of this it is sometimes difficult to find the cause of the poor performance of a model. However, such a calculation is still meaningful as it shows the performance of a complete turbulence model. The current work had focussed on evaluating the second order closures using three different models for the pressure strain correlation. A total of thirteen homogeneous flows are numerically computed using the second order closure models. The work concentrated only on those models which use a linear (or quasi-linear) model for the rapid term. This, therefore, includes the Launder, Reece and Rodi (LRR) model, the isotropization of production (IP) model, and the Speziale, Sarkar and Gatski (SSG) model. (The work involving nonlinear models is currently under way.) The objective of the study was to explore the performance of these three models in homogeneous flows and also to determine their limitations. A total of thirteen homogeneous shear flows were computed which included; homogeneous shear flow with and without rotation, distortion of turbulence by plane strain, straining of turbulence through axisymmetric expansions and contractions. Except for two, all of the data for these came from DNS. It was found out that overall, the re-calibrated LRR model worked better than both the IP and the SSG models. This work is to be presented at the 31st AIAA Aerospace Sciences Meeting and Exhibit to be held in Reno in January of 1993. Some of the other work in progress is concerned with the evaluation/development of models for the turbulent diffusion terms in the second moment equations and the modeling of scalar turbulence.

T.-H. SHIH, Stanford University (CTR)

In developing turbulence models for CFD related to aerospace propulsion systems, I have been working, together with my colleagues at CMOTT, on eddy viscosity models, Reynolds-stress algebraic equation models, second order closure models, multiple scale models, compressible models, near wall turbulence models and boundary layer transition models. For all of the modeling schemes mentioned, we have developed successful CMOTT versions, some of which have been reported in my 12 papers in 1992.

To enhance the turbulence research activities and the collaboration between different groups in turbulence modeling for aerospace propulsion systems, ICOMP organized a three-month long lecture series entitled "Turbulence - Fundamentals and Computational Modeling". I successfully wrote the course book, lecture notes and performed the teaching. Here, I would like to thank Dr. A. Shabbir, Dr. C. Feiler, and Professor T. Keith for their advice and comments on the lecture notes, and Mr. D. Conrad and Ms. K. Balog for their help in organizing the lectures.

AVRAM SIDI, Technion-Israel Institute of Technology

The research in extrapolation methods for time-periodic steady states that was begun two years ago was continued. The linear model that was developed then has been extended and shown to be valid for time dependent P.D.E.'s that are driven by a source term and/or boundary conditions that depend on time in a periodic fashion. The basic result obtained from the model is that the numerical solution computed by marching in time with a fixed time step has two parts to it: (1) the time-periodic part; (2) the transient. It is assumed that only the period of the time periodic part is known. Both of these contributions have well defined structures that enable us to employ extrapolation methods to approximate the discrete Fourier transform of the steady state and give precise convergence theorems that involve rates of convergence as well.

The performance of the extrapolation methods can be enhanced substantially by applying suitably designed linear filters to the sequence of solution vectors computed by time marching. These filters are obtained by using the fact that the period of the steady state is known. A recursive method for applying these filters has also been developed.

The procedure thus developed was applied to a set of pressure data obtained from a linear stability analysis of a turbomachinery problem (provided by David Whitfield). It was also applied by Ehtesham Hayder to a nonlinear problem involving a jet flow with a time-periodic disturbance. In both cases the extrapolation method used was the scalar epsilon algorithm which is the simplest to program and apply.

The theoretical developments of this research make it clear that the procedure developed thus far is quite general and is also applicable to scalar as well as vector time series.

ERLENDUR STEINTHORSSON (Postdoctoral), Carnegie Mellon University

The objective of my research has been to develop a computer program (flow solver) that is capable of detailed simulations of flows in turbine-blade coolant passages. Initially, three existing flow solvers were tested and evaluated with respect to applicability to the coolant passage flow problems. These flow solvers were LeRC3D (Howe, et at., 1992; Steinthorsson, et al., 1992), RPLUS (Hsieh, et al., 1990) and TRAF3D (Arnone, et al., 1991 and 1992). The three solvers were found to give essentially identical results for the problems on which they were tested. TRAF3D, however, was found to be much more computationally efficient than both LeRC3D and RPLUS, using an order of magnitude less CPU time than the other two codes. TRAF3D was, therefore, selected for further development for the coolant passage application.

The first phase of the development of TRAF3D was to transform the code from a specially-designed tool for cascade-flow simulations, to a general, single-block flow solver. This work involved removal of miscellaneous features from the code that were built in specifically to facilitate computation of cascade flows, as well as modularizing the implementation of boundary conditions. Upon completion of this work, the code was tested and validated by computing 2D and 3D laminar flow over a backward-facing step, and comparing the results of experimental data. A report describing the results of these tests is currently in preparation.

The second phase in the development of TRAF3D, which is currently in progress, is the implementation of a composite grid capability into the code so that it can handle the complicated geometries of turbine-blade coolant passages. The composite grids that the code will be able to handle are partially continuous multiblock grids. Grids of this type have the advantage that they appear as single grids to most flow solvers and no interpolation is needed to transfer information from one block of a grid system to another. In the past, reliance on partially continuous multiblock grids has often been rejected on the premise that, for complex geometries they are too difficult to generate. Lately, however, several methods and software have emerged that aid in the decomposition of complex geometries into blocks in such a manner that high quality partially continuous grid systems can be generated for the entire geometry. The use of such domain decomposition tools greatly reduces the effort needed to generate high quality partially continuous grids. Upon the completion of this second phase in the development of TRAF3D, the code will be capable of computing coolant flow and heat transfer inside coolant passages of turbine blades.

Future work on the development of TRAF3D includes the implementation of an advanced turbulence model. The improved turbulence model should improve predictions of heat transfer by the code. Other options for further development of TRAF3D include the implementation of an adaptive grid-refinement capability. The adaptive grid refinement capability will allow grid points to be easily added where they are needed, such as in the neighborhood of walls, and around pin fins and other small scale features in the geometry of coolant passages.

In the work described above, input and collaboration has been given by various individuals including the following: Dr. Louis A. Povinelli (Deputy Chief, IFMD) on the overall direction of the research; Dr. Meng-Sing Liou (Senior Scientist, IFMD) on the evaluation of the flow solvers and the development of TRAF3D; Dr. Yung K. Choo (CFD Branch, IFMD), Dr. Peter R. Eiseman and Zheming Cheng (Program Development Corp., White Plains, New York) on the generation of partially continuous grid systems for coolant passage geometries; and Dr. Kestutis C. Civinkas (Deputy Chief, Turbomachinery Technology Branch, PSD) on various issues related to coolant passages in turbine blades.

In addition to the development of TRAF3D, some research effort has been directed at the development of an "unstructured grid" flow solver for coolant passage geometries. This flow solver is based on a recently developed algorithm for unstructured, staggered grids (Nicolaides, 1991). This work is still in the very early stages. Thus far, a grid generation program has been developed that can generate a Cartesian grid inside any 2D geometry. This work is done in collaboration with Dr. Roy A. Nicolaides (Dept. of Mathematics, Carnegie Mellon University). Useful input has also been received from William J. Coirier (CFD Branch, IFMD).

REFERENCES

Arnone, A.; Liou, M.-S.; and Povinelli, L. A.: Multigrid Calculation of 3D Viscous Cascade Flows. AIAA-91-3238, September 1991.

Arnone, A.; Liou, M.-S.; and Povinelli, L.A.: Navier-Stokes Solution of Transonic Cascade Flows Using Non-Periodic C-Type Grids. Journal of Propulsion and Power, Vol. 8, 1992, pp. 410-417.

Howe, G.W.; Li, Z.; Shih, T.I.P.; and Nguyen, H.L. Simulations of Mixing in the Quick Quench Region of a Rich Burn-Quick Quench Mix-Lean Burn Combustor. AIAA-91-0410, 1991.

Hsieh, K.-C.; Shuen, J.-S.; Tsai, Y.-L.P.; and Yu, S.-T. RPLUS2D/3D User's Manual. Sverdrup Technology, Inc., NASA Lewis Research Center, October 1990.

Nicolaides, R.A.: Covolume Algorithms. Proc., Fourth International Symposium on Computational Fluid Dynamics, September 1991, pp. 861-866.

Steinthorsson, E.; Shih, T.I.-P.; and Roelke, R.J. Computation of the 3D Flow and Heat Transfer Within a Coolant Passage of a Radial Turbine. AIAA-91-2238, June 1991.

MARK STEWART (Postdoctoral), Princeton University

During the past year, my CFD research at ICOMP has been concerned with the physical models and numerical solution techniques necessary to develop a simulation capability for jet engine configurations. This work builds on work in the previous year on grid generation and numerical solution techniques which were used to generate Euler solutions for an unbladed jet engine configuration.

From this foundation, an engine model was developed which includes blade force effects and the heat addition of the combustor. In the current model blade forces are modelled for the operating point based on design air angles. This is an interim model which allows one to simulate the design point. The effect of the combustor is modelled by adding heat corresponding to the fuel mass flow.

Numerical methods have been devised and refined in an effort to properly represent these models and give convergent solutions. The engine model and numerical techniques are being tested on a single stage 1.15 Pressure Ratio Fan engine model, as well as the full Energy Efficient Engine geometry.

TIMOTHY W. SWAFFORD, Mississippi State University

The majority of effort has been involved with the so-called NPHASE code, which was originally developed for the computation of unsteady inviscid flow fields (using the Euler equations) associated with 2D cascades undergoing forced sinusoidal oscillation. This code is now being used by several researchers at NASA Lewis (and elsewhere) for studies involving both aero-acoustics and aero-elasticity. Because the use of this code is becoming more wide-spread, and because of the need to address flow regimes outside the present version's applicable range, several improvements and enhancements are needed within NPHASE, which was the focus of efforts over the past two weeks.

The numerical method in Version 1.0 of NPHASE is Whitfield's so-called "Two-Pass" scheme. Numerical stability problems have been encountered using this scheme which motivated present efforts regarding incorporating a scheme with improved stability properties, called the "Modified Two-Pass" scheme (again,

proposed by Whitfield). In addition, to improve the time-accuracy and convergence of numerical simulations, a Newton iteration capability was also incorporated.

Because of the need to address viscous flows (such that stall flutter can be simulated), Version 1.0 of NPHASE was modified (by Dr. David Huddleston and Mr. David Loe, both of MSU) to include viscous terms prior to the writer's ICOMP visit. The new numerical scheme was thus added to the viscous version.

A flat plate test case has been executed and it appears that the conversion (to Version 2.0) has been successful. However, we are still in the preliminary check-out stages and more evaluation is forthcoming.

GRETAR TRYGGVASON, University of Michigan

Work continued on the development and application of the Tracked, Immersed Boundary (TIB) technique to simulate multifluid flows. The basic method is described in Unverdi and Tryggvason: A Front Tracking Method for Incompressible Flows (J. Comput. Phys., 100 (1992), p. 25-37). The collision of axisymmetric drops has now been studied in some detail (with a graduate student) and part of that was consolidated and written up during my stay. When the drops collide, they bounce or coalesce depending on whether the thin film of ambient fluid left between the drops is ruptured or not. The rupturing of such a film in numerical simulations has two aspects. One is the purely technical aspect of changing the topology of the drop boundary (which is explicitly tracked in the TIB method); the second is the modeling of the physics that determines the rupture time. This summer, a topology change algorithm was written for our 2D TIB code. Several simulations showed that the procedure yields "realistic" evolution, including a drainage time for the film and then rapid relaxation to a new drop configuration following the rupture. In these simulations an *ad hoc* criterion was used to determine when to rupture, but the results showed that sometimes it is important to predict the rupture time accurately. A physically based rupture model is therefore necessary. Such a model is currently being developed here at Lewis by D. Jacqmin and M. Foster, and we plan to incorporate it into the codes. A topology-change algorithm was also developed for the 3D code, but has not been used in actual simulations as of yet. I also worked on the 3D front regridding procedure to make it simpler and more robust and carried out a few fully 3D simulations of drop collisions.

ELI TURKEL, Tel Aviv University

Work continued with M. E. Hayder on the fourth order MacCormack type scheme for aeroacoustics. A way was found to keep the scheme fourth order accurate for the viscous terms also. In addition an effort was begun to evaluate several alternatives for treating the outflow boundary condition. In order to test the code a number of cases were compared with the linear stability theory for a parallel jet both for planar and axisymmetric geometries. A paper for the AIAA aerosciences meeting in Reno is being prepared together with R. Mankbadi. Work was also begun on inserting a nozzle exit into the code so that one can analyze the excitations caused by the nozzle lip.

Further work has been done on a new implicit solver for spectral methods. It is based on an LU decomposition of the Chebyshev derivative matrix. For the 1D Euler equations we are able to obtain high convergence rates to the steady state. Different extensions to two space dimensions are being explored.

In collaboration with A. Arnone a central difference code for the incompressible flow over a cascade is being investigated. Improvements to an older code of Arnone have been made to allow for a general preconditioner to accelerate the convergence to a steady state. Comparisons of preconditioners suggested by Turkel and by Van Leer are in process. It is generally found that it is better to use constant values for preconditioning variables rather than the variable parameters suggested by the theory.

JACOBUS J. VAN DER VEGT, Stanford University, CTR

The main activity during 1992 has been the further development and testing of a code for direct simulations of transition and turbulence in compressible boundary layers. Excellent agreement has been obtained with growth rates for ribbon induced flat plate transition in comparison with the Parabolic Stability Equations (PSE) results from Bertolotti at a Mach number .5 and incompressible DNS results from Fasel and non-parallel theory of Gastner. Results have been submitted to the fluid dynamics conference in Orlando 1993 and were also presented at the 1992 APS conference.

The code which has been developed uses a multi-block discretization and is fully implicit in time. Time accuracy is maintained using a Newton-Raphson scheme to solve the non-linear equations for the implicit time integration. The Osher approximate Riemann solver is used for the inviscid part of the compressible Navier-Stokes equations because the Osher scheme has good shock capturing properties, minimal dissipation in boundary layers and a continuously differentiable flux which is important for implicit calculations. The numerical scheme is a conservative finite difference method, fourth order accurate in space and second order accurate in time. The higher order accurate Osher scheme is constructed using the flux-ENO reconstruction method discussed in Van Der Vegt. This method maintains higher order accuracy on a non-uniform grid which is generated with a general purpose grid generator such as GridGen2d. Most higher order accurate upwind schemes are only higher order accurate on analytically defined grids which also frequently have to be the product of 1D stretchings. In addition the geometric conservation law is satisfied which means that a uniform flow is also uniform after discretization.

A separate topic has been the study of numerical schemes which have the above mentioned properties, but can also capture shocks. Standard TVD schemes, which capture shocks without oscillations, are only first order accurate at non-sonic local extreme. Work has been done on the development of schemes which maintain higher order accuracy outside discontinuities and results of using Essentially Non-Oscillatory (ENO) schemes up to fifth order on a series of shock tube problems are presented in Van Der Vegt. ENO schemes do not have the limitation of TVD schemes and are a possible candidate for direct simulations of shock turbulence interaction but further study is needed to make them more efficient and robust.

Current activities are further study of flat plate transition in the highly non-linear stages, especially the effects of subharmonic instabilities. Effects of curvature will also be investigated because an unique feature of the DNS code is that it allows for general geometries. To facilitate the study of bypass transition and turbulent boundary layers a significant effort has to be made to numerically generate turbulence and work on higher order accurate shock capturing schemes will have to be continued.

REFERENCES

Bertolotti, F.P.: Linear and Nonlinear Stability of Boundary Layers with Streamwise Varying Properties. Dissertation, Ohio State University, Columbus, Ohio (1990)

Fasel, H.F.; Rist, U.; and Konzelmann, U.: Numerical Investigation of the Three-Dimensional Development in Boundary Layer Transition. AIAA J., 28, pp. 29-37 (1990)

Gaster, M.: On the Effect of Boundary Layer Growth on Flow Stability. JFM, 66, pp. 465-480 (1974)

Van Der Vegt, J.J.: Higher Order Accurate Osher Schemes with Application to Compressible Boundary Layer Transition. Submitted to Fluid Dynamics Conference, Orlando, 1993

Van Der Vegt, J.J.: ENO-Osher Schemes for Euler Equations. ICOMP Report 92-21, NASA TM-105928, November, 1992 (AIAA Paper 93-0335)

BRAM VAN LEER, University of Michigan

I extended earlier work on local preconditioning of the Euler equations by investigating the eigenvector structure of the Jacobians resulting after preconditioning. It appears that the flexibility in the form of the preconditioning matrix allows variations in the eigenvectors that may be beneficial for numerical convergence of the preconditioned method. In particular, it appears possible to make the eigenvectors more nearly orthogonal by introducing asymmetry and/or extra non-zero elements in the matrix. Numerical tests will have to determine whether orthogonality is indeed helpful for convergence.

DAVID WHITFIELD, Mississippi State University

A research area receiving increased attention of late is the numerical solution of linearized Euler equations. An area receiving much less attention is that of linearized Navier-Stokes equations. The objective of this work is to make a preliminary investigation of the type of results that can be obtained from a linearized form of the Navier-Stokes equations and compare these results with those obtained from solutions of the nonlinear Navier-Stokes equations. The approach is to use an incompressible Navier-Stokes code because it was currently being used for algorithm development, and consequently, available. There are some difficulties in

using the incompressible equations as opposed to the compressible equations. However, it was felt that if the incompressible results proved promising then a compressible approach would likely be more promising.

The results obtained thus far should be considered preliminary for the following reason. Because the compressible Euler equations are homogeneous of degree one in the dependent variable vector it is easy to linearize the equations where the linearized flux vector turns out to be the product of the flux Jacobian matrix evaluated with the mean flow solution times the dependent variable vector where, due to linearization, this can be either the complete (mean plus perturbed) flux vector or the perturbed flux vector. However, for incompressible flow this homogeneity property is not always valid and any results based on this approach are not strictly correct. Nevertheless, because an incompressible code was readily available and comparisons could be made with nonlinear results a test cease involving a cascade of infinitely thin blades with no stagger, in laminar flow at a Reynolds number of 500,000 at a reduced frequency of 0.5 oscillating in plunge with 400 time steps per oscillation was considered. The end result was that the periodic linear and nonlinear solutions were in reasonably good agreement with regard to periodic blade pressures. The agreement was certainly close enough that this approach should be considered further, particularly with the compressible equations.

DANIEL WINTERSCHEIDT (Postdoctoral), University of Kansas

Methods used to improve the quality of finite element solutions include h-methods, p-methods and h-p methods. Researchers have generally employed the h-version of the method, where the accuracy of the solution is improved by refining the mesh while using fixed low order element interpolation. This approach can be contrasted with spectral methods, which use high order global approximation functions. Spectral methods have been successfully applied to a variety of fluid dynamics problems.

During the past two decades p- and h-p versions of the finite element method have been developed which combine the geometric flexibility of standard low-order finite element techniques with the rapid convergence properties of spectral methods. In the p-version of the method, accuracy is achieved by increasing the element interpolation rather than refining the mesh. The h-p version involves a combination of mesh and polynomial refinement. These finite element developments have been primarily applied to structural mechanics analysis. In recent years, 'spectral element' methods which are similar to the p- and h-p version finite element methods have been developed specifically for fluid dynamics problems.

I have developed an adaptive p-refinement code for the steady incompressible Navier-Stokes equations which uses the least squares formulation. The formulation is based on minimization of the integral of the squared residuals. The formulation produces a symmetric matrix and permits the use of equal order interpolation for all field variables. Element interpolation is based on p-version approximation functions which are derived from the Lagrange interpolation functions. The program monitors the element residuals and automatically increases the p-level in elements where the errors (residuals) exceed a prescribed tolerance. Inter-element continuity is enforced by placing constraints on appropriate degrees of freedom. Future research involves determining the best way to apply p-version methods to transient and compressible flow problems.

XUESONG WU (Imperial College) AND STEPHEN COWLEY (Cambridge University)

During our visit, we mainly studied resonant-triad interactions of Tollmien-Schlichting waves within Blasius boundary layers. By building on earlier work by Wu, Lee & Cowley (1992 to appear as an ICOMP report), an important interaction mechanism was identified. The flow was found to be described by a complicated seven-zoned asymptotic structure (c.f. the five-zoned structure for the linear problem). The dominant nonlinear interactions were shown to arise in the critical layer and the so-called "diffusion layer". These interactions not only affect the evolution of the waves, but also drive a "large" mean-flow; this mean flow has its maximum magnitude in the wall region.

The evolution equations describing the resonant-triad interactions are of novel integro-differential type, and correct earlier equations derived by Mankbadi. The mechanism and structure of this type of interaction apply to a much broader class of shear flows of practical importance than just Blasius flow, e.g. acceleration and deceleration boundary layers.

In addition to the above we had scientific discussions with M. E. Goldstein, S. S. Lee, R. Mankbadi, L. Hultgren, S. Leib, D. Wundrow, J. Mathew, and many of the visitors to ICOMP.

ZHIGANG YANG (Postdoctoral), Cornell University

In the past year, I worked on the following projects: modeling of near wall turbulence, modeling of bypass transition due to freestream turbulence, and stability analysis of swirling flows.

1. k-ε model for near wall turbulence (with T.-H. Shih of ICOMP)

We have proposed a k-ε model for wall bounded turbulent flows. In this model, the eddy viscosity is characterized by a turbulent velocity scale and a turbulent time scale. The time scale is bounded from below by the Kolmogorov time scale. The dissipation equation is reformulated using this time scale and no singularity exists at the wall. The damping function used in the eddy viscosity is chosen to be a function of

$$R_y = \frac{k^{1/2} y}{v}$$ instead of y^+. Hence, the model could be used for flows with separation. The model constants

used are the same as in the high Reynolds number standard k-ε model. Thus, the proposed model will be also suitable for flows far from the wall. Turbulent channel flows at different Reynolds numbers and turbulent boundary layer flows with and without pressure gradient are calculated. The model predictions are in good agreement with direct numerical simulation and experimental data. This work was reported in NASA TM-105768 and is to appear in the AIAA Journal.

2. A transport equation for eddy viscosity (with P. A. Durbin of CTR)

We have proposed an eddy viscosity transport model for wall bounded turbulent flows. The proposed model reduces to a quasi-homogeneous form far from surfaces. Near to a surface, the nonhomogeneous effect of the wall is modeled by an elliptic relaxation model. All the model terms are expressed in local variables and are coordinate independent; the model is intended to be used in complex flows. Turbulent channel flow and turbulent boundary layer flows with and without pressure gradient are calculated using the present model. Comparisons between model calculations and direct numerical simulation or experimental data show good agreement. A paper on this work is to appear in Proceedings of the 1992 Summer Program on Studying Turbulence Using Numerical Simulation Database.

3. Modeling of bypass transition (with T.-H. Shih of ICOMP)

We have proposed a model for transitional boundary layers where transition is due to freestream turbulence. The k-ε model for near wall turbulence (as reported in 1 above) is modified by a weighting factor to account for the effect of intermittency in the transitional boundary layer. The weighting factor is related to both the intermittency factor in the boundary layer and the freestream turbulence level. Transitional flat plate boundary layers with different freestream turbulence levels are calculated using the proposed model. It is found that the model calculations agree well with the experimental data and give a better transition prediction compared with other low Reynolds number k-ε models which do not incorporate the effect of intermittency. This work was reported in the 4th NASA Lewis Bypass Transition Workshop. A paper detailing this work is in preparation.

4. Stability analysis of swirling flows

The viscous linear stability of a trailing line vortex (Batchelor vortex) was studied. The flow is characterized by two parameters, the Reynolds number Re and the rotation rate q. The marginal stability curve which separates the stable domain from the unstable domain was searched over the (Re, q) plane. It is found that on the marginal stability curve, q increases with the Reynolds number and does not approach a constant even when the Reynolds number is as large as 10^5. The values of q for large Reynolds numbers are higher than their inviscid counterpart. These findings suggest that modes giving rise to marginal stability are viscous and do not approach the inviscid limit as the Reynolds number goes to infinity. These modes have an azimuthal wavenumber n = -1 when the Reynolds number is larger than 200 in contrast to n = -2 for smaller Reynolds numbers. As the Reynolds number is increased, the eigen-functions of the marginal stability modes become more and more concentrated near the axis of the vortex suggesting that these modes are viscous center modes in the limit of large Reynolds number. This work was presented in the 18th Internal Congress of Theoretical and Applied Mechanics. A paper on this work is in process.

AKIRA YOSHIZAWA, University of Tokyo

The mathematical structure of a three-equation compressible turbulence model with the density-variance transport equation [Phys. Rev. A 46/6 (1992)], which was derived using a two-scale DIA (TSDIA), was investigated through qualitative comparison with direct numerical simulation (DNS) results for shockwave/turbulence interactions . A feature of this model is that Reynolds averaging is adopted in place of mass-weighted averaging which is widely used in aerodynamical studies so that effects of density change can be incorporated more explicitly. Specifically, those effects are related to the non-dimensional coefficient proportional to the ratio of the relative magnitude of the density variance to the turbulent Mach number. The model was shown to give an interesting explanation of the following important phenomena in shock-wave/turbulence interactions: (1) the mechanism for the maintenance of thin structures of a shock-wave region under the enhanced effects of turbulent intensities and density fluctuation, (2) the importance of the pressure-transport term in the equation for the turbulent kinetic energy (the effect is usually neglected in the solenoidal turbulence modeling).   This model was applied by W. W. Liou to a compressible mixing layer and was found to give a reasonable estimate of the growth rate.

SHAYE YUNGSTER (Postdoctoral), University of Washington

The objective of the present research is to develop computational fluid dynamics (CFD) prediction techniques to support external burning and nozzle performance experiments aimed at reducing the transonic drag of NASP type vehicles. The effects of external combustion on NASP-like nozzles at transonic speeds were investigated numerically using a 2D fully implicit total variation diminishing (TVD) code that solves the Reynolds-averaged Navier-Stokes equations including finite-rate chemistry. Three-dimensional calculations (nonreacting) have also been conducted using the MAWLUS code of T. Chitsomboon.

A computational model for the external fuel injection and combustion processes was developed and results were obtained for complete nozzle configurations with hot and cold exhaust flows, and with or without external burning. Results obtained on the baseline cowl nozzle configuration including external burning were in very good qualitative and quantitative agreement with the experiments. A grid refinement study, including the use of adaptive grids, was completed, and computations for other nozzle configurations that include an extended cowl and a flame holder are being currently performed at several inflow conditions and with several combustion models.

A large effort was concentrated on the full optimization of the CFD code, which now executes at a sustained rate of more than 100 MFLOPS. In addition, it can be combined with the vector extrapolation techniques of A. Sidi for faster convergence. The code is also set up to be easily adapted to any given combustion model in terms of number of species, number of reaction steps, etc. Extension of the code to three dimensions, the inclusion of a κ-ε turbulence model, and a more detailed simulation of the injection process are tasks planned for the future.

J. ZHU, University of Karlsruhe

Two backward-facing step flows were used to validate four low Reynolds-number κ-ε turbulence models: Launder-Sharma, Two-layer, Yang-Shih, and Shih-Lumley. A common feature of these models is that none involve y_+ and thus can be applied directly to separated flows. The standard κ-ε model with wall functions was also used for comparison. The test results have shown that only the two-layer model performs better than the standard model with wall functions. The deficiencies of the other low Reynolds-number models were traced to an over-predicted damping function f_μ under severe adverse pressure gradient.

The two new high Reynolds-number models recently developed at ICOMP were also validated against the two backward-facing step flows. The first is an eddy viscosity model in which C_μ is set to a simple function of the turbulent to mean strain time scale, and the second is a realizable and anisotropic Reynolds stress algebraic equation model. Both models have been shown to result in significant improvements over the standard κ-ε model. Under way are further tests of the models for confined coaxial jets with an without swirl and 3D flow in an S-shaped channel.

REPORTS AND ABSTRACTS

The ICOMP Steering Committee: "Institute for Computational Mechanics in Propulsion (ICOMP), Sixth Annual Report - 1991," ICOMP Report No.92-01, NASA TM–105612, 48 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is a combined activity of Case Western Reserve University, Ohio Aerospace Institute (OAI) and 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 at ICOMP during 1991.

CMOTT: "Workshop on Engineering Turbulence Modeling," ICOMP Report No. 92-02, CMOTT-92-02, NASA CP-10088, March 1992, 530 pages.

The purpose of this meeting is to discuss the present status and the future direction of various levels of engineering turbulence modelling related to CFD computations for propulsion. For each level of complication, there are a few turbulence models which represent the state-of-the-art for that level. However, it is important to know their capabilities as well as their deficiencies in CFD computations in order to help engineers select and implement the appropriate models in their real world engineering calculations. This will also help turbulence modelers perceive the future directions for improving turbulence models. The focus of this meeting will be one-point closure models (i.e. from algebraic models to higher order moment closure schemes and pdf methods) which can be applied to CFD computations. However, other schemes helpful in developing one-point closure models, such as RNG, DIA, LES and DNS, will be also discussed to some extent.

Leonard, B. P. (ICOMP) and Mokhtari, Simin (University of Akron): "ULTRA-SHARP Solution of the Smith-Hutton Problem," ICOMP Report No. 92-03, NASA TM-105435, February 1992, 40 pages.

Highly convective scalar transport involving near-discontinuities and strong streamline curvature was addressed in a paper by Smith and Hutton in 1982, comparing several different convection schemes applied to a specially devised test problem. First-order methods showed significant artificial diffusion, whereas higher-order methods gave less smearing but had a tendency to overshoot and oscillate. Perhaps because unphysical oscillations are more obvious than unphysical smearing, the intervening period has seen a rise in popularity of low-order artificially diffusive schemes, especially in the numerical heat-transfer industry. The present paper describes an alternative strategy of using non-artificially diffusive higher-order methods, while maintaining strictly monotonic transitions through the use of simple flux-limited constraints. Limited third-order upwinding is usually found to be the most cost-effective basic convection scheme. Tighter resolution of discontinuities can be obtained at little additional cost by using automatic adaptive stencil expansion to higher order in local regions, as needed.

Hariharan, S. I. (ICOMP): "Long Time Behavior of Unsteady Flow Computations," ICOMP Report No. 92-04, NASA TM-105584, March 1992, 24 pages.

This paper addresses a specific issue of time accuracy in the calculations of external aerodynamic problems. The class of problems that is discussed here consists of inviscid compressible subsonic flows. These problems are inherently governed by a convective equation. This is readily seen by linearizing the Euler equation which results in a convective wave equation for the pressure. A key mathematical issue that is not well understood in literature for these problems is the long time behavior of the solution. This is an important aspect if one desires transient calculations of problems governed by the Euler equations or its derivatives such as the small disturbance equations or the potential formulations of the gust problem. In particular, difficulties arise for 2D problems. In 2D the time decay rate of solutions of the wave equation is known to be rather slow. This applies to the convective wave equation as well. The consequences are rather

severe if one focuses on the time accuracy of solutions of problems governed by the Euler equations. In concert with the above mentioned problem, exterior flows require proper modelling of boundary conditions. In particular, computations of these flow problems require truncation of infinite regions into finite regions with the aid of artificial boundaries. On these boundaries one must impose boundary conditions that are consistent with the physics as well as guarantee consistency with the original problem posed in the unbounded region. Moreover, these boundary conditions must have accuracy in time as well as space. Some of the well-known procedures do address the issues of spatial accuracy and have remedy for these conditions. Unfortunately, these procedures do not address the time accuracies, which are crucial for the transient problems. Our treatment is discussed in detail and examples are presented to verify the results.

Iannelli, G. S. (ICOMP): "Conservative-Variable Average States for Equilibrium Gas Multi-Dimensional Fluxes," ICOMP Report No. 92-05, NASA TM-105585, March 1992, 46 pages.

Modern split component evaluations of the flux vector jacobians are thoroughly analyzed for equilibrium-gas average-state determinations. It is shown that all such derivations satisfy a fundamental eigenvalue consistency theorem. A conservative-variable average-state is then developed for arbitrary equilibrium-gas equations of state and curvilinear-coordinate fluxes. Original expressions for eigenvalues, sound speed, Mach number, and eigenvectors are then determined for a general average jacobian, and it is shown that the average eigenvalues, Mach number, and eigenvectors may not coincide with their classical pointwise counter-parts. A general equilibrium-gas equation of state is then discussed for conservative-variable CFD Euler formulations. The associated derivations lead to unique compatibility relations that constrain the pressure jacobian derivatives. Thereafter, alternative forms for the pressure variation and average sound speed are developed in terms of two average pressure jacobian derivatives. Significantly, no additional degree of freedom exists in the determination of these two average partial derivatives of pressure. Therefore, they are simultaneously computed exactly without any auxiliary relation, hence without any geometric solution projection or arbitrary scale factors. Several alternative formulations are then compared and key differences highlighted with emphasis on the determination of the pressure variation and average sound speed. The relevant underlying assumptions are identified, including some subtle approximations that are inherently employed in published average-state procedures. Finally, a representative test case is discussed for which an intrinsically exact average-state is determined. This exact state is then compared with the predictions of recent methods, and their inherent approximations are appropriately quantified.

Barton, J. M. (Sverdrup) and Rubinstein R. (Sverdrup): "Renormalization Group Analysis of the Reynolds Stress Transport Equation," ICOMP Report No. 92-06, CMOTT 92-3, NASA TM-105588, March 1992, 26 pages.

The pressure-velocity correlation and return to isotropy term in the Reynolds stress transport equation are analyzed using the Yakhot-Orszag renormalization group. The perturbation series for the relevant correlations, evaluated to lowest order in the ε-expansion of the Yakhot-Orszag theory, are infinite series in tensor product powers of the mean velocity gradient and its transpose. Formal lowest order Padé approximations to the sums of these series produce a fast pressure strain model of the form proposed by Launder, Reece, and Rodi, and a return to isotropy model of the form proposed by Rotta. In both cases, the model constants are computed theoretically. The predicted Reynolds stress ratios in simple shear flows are evaluated and compared with experimental data. The possibility is discussed of driving higher order nonlinear models by approximating the sums more accurately.

Steffen, Christopher J., Jr. (NASA Lewis): "An Investigation of DTNS2D for Use as an Incompressible Turbulence Modelling Test-bed," ICOMP Report No. 92-07, CMOTT 92-4, NASA TM-105593, March 1992, 18 pages.

This paper documents an investigation of a 2D, incompressible Navier-Stokes solver for use as a test-bed for turbulence modelling. DTNS2D is the code under consideration for use at the Center for Modelling of Turbulence and Transition (CMOTT). This code was created by Gorski at the David Taylor Research Center and incorporates the pseudo compressibility method. Two laminar benchmark flows are used to measure the performance and implementation of the method. The classical solution of the Blasius boundary layer

is used for validating the flatplate flow, while experimental data is incorporated in the validation of backward facing step flow. Velocity profiles, convergence histories, and reattachment lengths are used to quantify these calculations. The organization and adaptability of the code are also examined in light of the role as a numerical test-bed.

Yang, Z. (ICOMP) and Shih, Tsan-Hsing (ICOMP): "A κ-ε Calculation of Transitional Boundary Layers," ICOMP Report No. 92-08, CMOTT-92-05, NASA TM-105604, March 1992, 12 pages.

A recently proposed κ-ε model for low Reynolds-number turbulent flows (Yang and Shih, 1991) was modified by introducing a new damping function f_μ. The modified model is used to calculate the transitional boundary layer over a flat plate with different freestream turbulence levels. It is found that the model could mimic the transitional flow. However, the predicted transition is found to be sensitive to the initial conditions.

Sidi, Avram (ICOMP) and Shapira, Yair (Technion-Israel Institute of Technology): "Upper Bounds for Convergence Rates of Vector Extrapolation Methods on Linear Systems with Initial Iteration," ICOMP Report No. 92-09, NASA TM-105608, 1992, 58 pages.

The application of minimal polynomial extrapolation (MPE) and the reduced rank extrapolation (RRE) to a vector sequence obtained by the linear iterative technique, is considered. Both methods produce a 2D array of approximations to the solution of the system $(I - A)x = b$. Here is obtained from the vectors . It was observed in an earlier publication by the first author that the sequence , for , but fixed, possesses better convergence properties that the sequence . A detailed theoretical explanation for this phenomenon is provided in the present work. This explanation is heavily based on approximations by incomplete polynomials. It is demonstrated by numerical examples when the matrix A is sparse that cycling with , but fixed, produces better convergence rates and costs less computationally than cycling with . It is also illustrated numerically with a convection-diffusion problem that the former may produce excellent results where the latter may fail completely. As has been shown in an earlier publication, the results produced by are identical to the corresponding results obtained by applying the Arnoldi method or GMRES to the system $(I - A)x = b$.

Shih, Tsan-Hsing (ICOMP) and Lumley, John L. (Cornell University): "Kolmogorov Behavior of Near-Wall Turbulence and Its Application in Turbulence Modeling", ICOMP Report No. 92-10, CMOTT-92-06, NASA TM–105663, 16 pages.

The near-wall behavior of turbulence is re-examined in a way different from that proposed by Hanjalic and Launder and followers. It is shown that at a certain distance from the wall, all energetic large eddies will reduce to *Kolmogorov* eddies (the smallest eddies in turbulence). All the important wall parameters, such as friction velocity, viscous length scale, and mean strain rate at the wall, are characterized by *Kolmogorov* microscales. According to this *Kolmogorov* behavior of near-wall turbulence, the turbulence quantities, such as turbulent kinetic energy, dissipation rate, etc. at the location where the large eddies become "*Kolmogorov*" eddies, can be estimated by using both direct numerical simulation (DNS) data and asymptotic analysis of near-wall turbulence. This information will provide useful boundary conditions for the turbulent transport equations. As an example, the concept is incorporated in the standard κ-ε model which is then applied to channel and boundary layer flows. Using appropriate boundary conditions (based on *Kolmogorov* behavior of near-wall turbulence), there is no need for any wall-modification to the κ-ε equations (including model constants). Results compare very well with the DNS and experimental data.

Yang, Zhigang (ICOMP) and Shih, Tsan-Hsing (ICOMP): "A New Time Scale Based k-ε Model for Near Wall Turbulence," ICOMP Report No. 92-11, CMOTT 92-07, NASA TM–105768, January 1992.

A k-ε model is proposed for wall bounded turbulent flows. In this model, the eddy viscosity is characterized by a turbulent velocity scale and a turbulent time scale. The time scale is bounded from below by the Kolmogorov time scale. The dissipation equation is reformulated using this time scale and no singularity exists at the wall. The damping function used in the eddy viscosity is chosen to be a function of

$R_y = \dfrac{k^{1/2}\,y}{v}$ instead of y^+. Hence, the model could be used for flows with separation. The model constraints used are the same as in the high Reynolds number standard k-ε model. Thus, the proposed model will be also suitable for flows far from the wall. Turbulent channel flows at different Reynolds numbers and turbulent boundary layer flows with and without pressure gradient are calculated. Results show that the model predictions are in good agreement with direct numerical simulation and experimental data.

CMOTT: "Center for Modeling of Turbulence and Transition (CMOTT)," ICOMP Report No. 92-12, CMOTT-92-08, NASA TM-105834, September 1992, 192 pages.

This research brief contains the progress reports of the Research Staff of the Center for Modeling of Turbulence and Transition (CMOTT) from May 1991 to May 1992. It is intended as an annual report to the Institute of Computational Mechanics in Propulsion and NASA Lewis Research Center. A separate report entitled, "Workshop on Engineering Turbulence Modeling," covering some of the 1991 CMOTT Summer research activities was released earlier this year. The main objective of the CMOTT is to develop, validate and implement the turbulence and transition models for practical engineering flows. The flows of interest are 3D, incompressible and compressible flows with chemical reaction. During this period, the research covered two-equation (e.g., κ-ε) and algebraic Reynolds-stress models, second moment closure models, probability density function (pdf) models, Renormalization Group Theory (RNG), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). Last year was CMOTT's second year in operation. CMOTT now has eleven members from ICOMP, NASA LeRC, and Sverdrup Technology Inc., working on various aspects of turbulence and transition modeling in collaboration with NASA-Lewis scientists and Case Western Reserve University faculty members. The CMOTT members have been continuously and actively involved in international and national turbulence research activities. A biweekly CMOTT seminar series has been conducted with speakers invited from within and without the NASA Lewis Research Center, including foreign speakers. The current CMOTT roster and its organization are listed in Appendix A. Listed in Appendix B are the abstracts and the scientific and technical issues discussed in biweekly CMOTT seminars. Appendix C gives a list of references which are the papers contributed by CMOTT members in the last two years.

Murthy, Durbha V. (NASA Research Associate) and Pierre, Christophe (ICOMP) : "Stochastic Sensitivity Measure for Mistuned High-Performance Turbines," ICOMP Report No. 92-13, NASA TM-105821, August 1992.

A stochastic measure of sensitivity is developed in order to predict the effects of small random blade mistuning on the dynamic aeroelastic response of turbomachinery blade assemblies. This sensitivity measure is based solely on the nominal system design (i.e., on tuned system information), which makes it extremely easy and inexpensive to calculate. The measure has the potential to become a valuable design tool that will enable designers to evaluate mistuning effects at a preliminary design stage and thus assess the need for a full mistuned rotor analysis. The predictive capability of the sensitivity measure is illustrated by examining the effects of mistuning on the aeroelastic modes of the first stage of the oxidizer turbopump in the Space Shuttle Main Engine. Results from a full analysis of mistuned systems confirm that the simple stochastic sensitivity measure predicts consistently the drastic changes due to mistuning and the localization of aeroelastic vibration to a few blades.

Ameri, Ali A. (NASA Research Associate) and Arnone, Andrea (ICOMP): "Navier-Stokes Turbine Heat Transfer Predictions Using Two-Equation Turbulence Closures", ICOMP Report No. 92-14, NASA TM-105817, August 1992, 12 pages.

Navier-Stokes calculation were carried out in order to predict the heat transfer rates on turbine blades. The calculations were performed using TRAF2D which is a 2D, explicit, finite volume mass-averaged Navier-Stokes solver. Turbulence was modeled using Coakley's q-ε and Chien's k-ε two-equation models and the Baldwin-Lomax algebraic model. The model equations along with the flow equations were solved explicitly

on a non-periodic C grid. Implicit residual smoothing (IRS) or a combination of multigrid technique and IRS was applied to enhance convergence rates. Calculations were performed to predict the Stanton number distributions on the first stage vane and blade row as well as the second stage vane row of the Rocketdyne Space Shuttle Main Engine (SSME) high pressure fuel turbine. The comparison with the experimental results, although generally favorable, serves to highlight the weaknesses of the turbulence models and the possible areas of improving these models for use in turbomachinery heat transfer calculations.

Stewart, Mark E. M. (ICOMP): "A Multiblock Grid Generation Technique Applied to a Jet Engine Configuration" ICOMP Report No. 92-15, Reprint from NASA CP-3143, Workshop on Software Systems for Surface Modelling and Grid Generation, a conference held at NASA Langley Research Center, Hampton, Virginia, April 27-30, 1992.

Techniques are presented for quickly finding a multiblock grid for a 2D geometrically complex domain from geometrical boundary data. An automated technique for determining a block decomposition of the domain is explained. Techniques for representing this domain decomposition and transforming it are also presented. Further, a linear optimization method may be used to solve the equations which determine grid dimensions within the block decomposition. These algorithms automate many stages in the domain decomposition and grid formation process and limit the need for human intervention and inputs. They are demonstrated for the meridional or throughflow geometry of a bladed jet engine configuration.

Liou, William W.-W. (ICOMP): "A New Energy Transfer Model for Turbulent Free Shear Flow", ICOMP Report No. 92-16, CMOTT-92-09, NASA TM-105854, September 1992.

A new model for the energy transfer mechanism in the large-scale turbulent kinetic energy equation is proposed. An estimate of the characteristic length scale of the energy containing large structures is obtained from the wavelength associated with the structures predicted by a weakly nonlinear analysis for turbulent free shear flows. With the inclusion of the proposed energy transfer model, the weakly nonlinear wave models for the turbulent large-scale structures are self-contained and are likely to be independent flow geometries. The model is tested against a plane mixing layer. Reasonably good agreement is achieved. Finally, it is shown by using the Laipunov function method, the balance between the production and the drainage of the kinetic energy of the turbulent large-scale structures is asymptotically stable as their amplitude saturates. The saturation of the wave amplitude provides an alternative indicator for flow self-similarity.

Sidi, Avram (ICOMP): "Rational Approximations From Power Series of Vector-Valued Meromorphic Functions", ICOMP Report No. 92-17, NASA TM-105859, September 1992, 24 pages.

Let $F(z)$ be a vector-valued function, $F:C \rightarrow C^N$, which is analytic at $z = 0$ and meromorphic in a neighborhood of $z = 0$, and let its Maclaurin series be given. In this work we develop vector-valued rational approximation procedures for $F(z)$ by applying vector extrapolation methods to the sequence of partial sums of its Maclaurin series. We analyze some of the algebraic and analytic properties of the rational approximations thus obtained, and show that they are akin to Padé approximants. In particular, we prove a Koenig type theorem concerning their poles and a de Montessus type theorem concerning their uniform convergence. We show how optimal approximations to multiple poles and to Laurent expansions about these poles can be constructed. Extensions of the procedures above and the accompanying theoretical results to functions defined in arbitrary linear spaces is also considered. One of the most interesting and immediate applications of the results of this work is to the matrix eigenvalue problem. In a forthcoming paper we exploit the developments of the present work to devise bona fide generalizations of the classical power method that are especially suitable for very large and sparse matrices. These generalizations can be used to approximate simultaneously several of the largest distinct eigenvalues and corresponding eigenvectors and invariant subspaces of arbitrary matrices which may or may not be diagonalizable and are very closely related with known Krylov subspace methods.

Sidi, Avram (ICOMP): "Application of Vector-Valued Rational Approximations to the Matrix Eigenvalue Problem and Connections with Krylov Subspace Methods", ICOMP Report No. 92-18, NASA TM-105858, September 1992.

Let $F(z)$ be a vectored-valued function, $F:C\rightarrow C^N$ which is analytic at $z = 0$ and meromorphic in a neighborhood of $z = 0$, and let its Maclaurin series be given. In a recent work by the author, vector-valued rational approximation procedures for $F(z)$ that are based on its Maclaurin series, were developed, and some of their convergence properties were analyzed in detail. In particular, a Koenig type theorem concerning their poles and a de Montessus type theorem concerning their uniform convergence in the complex plane were given. With the help of these theorems it was shown how optimal approximations to the poles of $F(z)$ and the principle parts of the corresponding Laurent series expansions can be obtained. In the present work we use these rational approximation procedures in conjunction with power iterations to develop bona fide generalizations of the power method for an arbitrary N X N matrix that may be diagonalizable or not. These generalizations can be used to obtain simultaneously several of the largest distinct eigenvalues and corresponding eigenvectors and other vectors in the invariant subspaces. We provide interesting constructions for both nondefective and defective eigenvalues and the corresponding invariant subspaces, and present a detailed convergence theory for them. This is made possible by the observation that vectors obtained by power iterations with a matrix are actually coefficients of the Maclaurin series of a vector-valued rational function, whose poles are reciprocals of some or all of the nonzero eigenvalues of the matrix being considered, while the principle parts of the Laurent expansions of this rational function are vectors in the corresponding invariant subspaces. In addition, it is shown that the generalized power methods of this work are equivalent to some Krylov subspace methods, among them the methods of Arnoldi and Lanczos. Thus, the theory of the present work provides a set of completely new results and constructions for these Krylov subspace methods. This theory suggests at the same time a new mode of usage for these Krylov subspace methods that has been observed to possess computational advantages over their common mode of usage.

Leonard, B. P. (ICOMP): "Comparison of Truncation Error of Finite-Difference and Finite-Volume Formulations of Convection Terms", ICOMP Report No. 92-19, NASA TM-105861, September 1992.

Judging by errors in the computational-fluid-dynamics literature in recent years, it is not generally well understood that (above first-order) there are significant differences in spatial truncation error between formulations of convection involving a finite-difference approximation of the first derivative, on the one hand, and a finite-volume model of flux differences across a control-volume cell, on the other. The difference between the two formulations involves a second-order truncation-error term (proportional to the third-derivative of the convected variable). Hence, for example, a third (or higher) order finite-difference approximation for the first-derivative convection term is only second-order accurate when written in conservative control-volume form as a finite-volume formulation, and vice versa.

Wu, Xuesong (Imperial College), Lee, Sang Soo (Sverdrup), and Cowley, Stephen J. (ICOMP): "On the Nonlinear Three-Dimensional Instability of Stokes Layers and Other Shear Layers to Pairs of Oblique Waves", ICOMP Report No. 92-20, NASA TM-105918, December 1992.

The nonlinear evolution of a pair of initially linear oblique waves in a high-Reynolds-number Stokes layer is studied. Attention is focused on times when disturbances of amplitude e have $O(e^{1/3} R)$ growth rates, where R is the Reynolds number. The development of a pair of oblique waves is then controlled by nonlinear critical-layer effects (Goldstein & Choi, 1989). Viscous effects are included by studying the distinguished scaling. This leads to a complicated modification of the kernel function in the integro-differential amplitude equation. When viscosity is not too large, solutions to the amplitude equation develop a finite time singularity, indicating that an explosive growth can be induced by nonlinear effects; we suggest that such explosive growth can lead to the bursts observed in experiments. Increasing the importance of viscosity generally delays the occurrence of the finite-time singularity, and sufficiently large viscosity may lead to the disturbance decaying exponentially. For the special case when the streamwise and spanwise wavenumbers are equal, the solution can evolve into a periodic oscillation. A link between the unsteady critical-layer approach to high-Reynolds-number flow instability, and the wave vortex approach of Hall & Smith (1991), is identified.

Van Der Vegt, Jacobus J. (ICOMP): "Eno-Osher Schemes for Euler Equations", <u>ICOMP Report No. 92-21</u>, CMOTT 92-10, NASA TM-105928, November 1992, 12 pages.

In this paper the combination of the Osher approximate Riemann solver for the Euler equations and various ENO schemes is discussed for 1D flow. The three basic approaches, viz. the ENO scheme using primitive variable reconstruction, either with the Cauchy-Kowalewski procedure for time integration or the TVD Runge-Kutta scheme, and the flux-ENO method are tested on different shock tube cases. The shock tube cases were chosen to present a serious challenge to the ENO schemes in order to test their ability to capture flow discontinuities, such as shocks. Also the effect of the ordering of the eigen values, viz. natural or reversed ordering, in the Osher scheme is investigated. The ENO schemes are tested up to fifth order accuracy in space and time. The ENO-Osher scheme using the Cauchy-Kowalewski procedure for time integration is found to be the most accurate and robust compared with the other methods and is also computationally efficient. The tests showed that the ENO schemes perform reasonably well, but have problems in cases where two discontinuities are close together. In that case there are not enough points in the smooth part of the flow to create a non-oscillatory interpolation.

Ajmani, Kumud (ICOMP), Ng, Wing-Fai (Virginia Polytechnic Institute) and Liou, Meng-Sing (NASA Lewis): "Preconditioned Conjugate-Gradient Methods for Low-Speed Flow Calculations," <u>ICOMP Report No. 92-22</u>, NASA TM-105929, January 1993, 12 pages.

An investigation is conducted into the viability of using a generalized Conjugate Gradient-like method as an iterative solver to obtain steady-state solutions of very low-speed fluid flow problems. Low-speed flow at Mach 0.1 over a backward-facing step is chosen as a representative test problem. The unsteady form of the 2D, compressible Navier-Stokes equations is integrated in time using discrete time-steps. The Navier-Stokes equations are cast in an implicit, upwind finite-volume, flux split formulation. The new iterative solver is used to solve a linear system of equations at each step of the time-integration. Preconditioning techniques are used with the new solver to enhance the stability and convergence rate of the solver, and are found to be critical to the overall success of the solver. A study of various preconditioners reveals that a preconditioner based on the Lower-Upper Successive Symmetric Over-Relaxation iterative scheme is more efficient than a preconditioner based on Incomplete L-U factorizations of the iteration matrix. The performance of the new preconditioned solver is compared with a conventional Line Gauss-Seidel Relaxation (LGSR) solver. Overall speed-up factors of 28 (in terms of global time-steps required to converge to a steady-state solution) and 20 (in terms of total CPU time on one processor of a CRAY-YMP) are found in favor of the new preconditioned solver, when compared with LGSR solver.

Zhu, Gang (Wayne State University), Lai, Ming-Chia (Wayne State University) and Shih, Tsan-Hsing (ICOMP): "Second-Order Closure Modeling of Turbulent Buoyant Wall Plumes", <u>ICOMP Report No. 92-23</u>, CMOTT 92-11, NASA TM-105956, December 1992, 14 pages.

Non-intrusive measurements of scalar and momentum transport in turbulent wall plumes, using a combined technique of laser Doppler anemometry and laser-induced fluorescence, has shown some interesting features not present in free jets or plumes. First, buoyancy-generation of turbulence is shown to be important throughout the flow field. Combined with low-Reynolds-number turbulence and near-wall effect, this may raise the anisotropic turbulence structure beyond the prediction of eddy-viscosity models. Second, the transverse scalar fluxes do not correspond only to the mean scalar gradients, as would be expected from gradient-diffusion modeling. Third, higher-order velocity-scalar correlations which describe turbulent transport phenomena could not be predicted using simple turbulence models. A second-order closure simulation of turbulence adiabatic wall plumes, taking into account the recent progress in scalar transport, near-wall effect and buoyancy, is reported in the current study to compare with the non-intrusive measurements. In spite of the small velocity scale of the wall plumes, the results showed that low-Reynolds-number correction is not critically important to predict the adiabatic cases tested and cannot be applied beyond the maximum velocity location. The mean and turbulent velocity profiles are very closely predicted by the second-order closure models. But the scalar field is less satisfactory, with the scalar fluctuation level underpredicted. Strong intermittency of the low-Reynolds-number flow field is suspected of these discrepancies. The trend in second- and third-order velocity-scalar correlations, which describe turbulent transport phenomena, are also predicted in general, with the cross-streamwise correlations better than the

streamwise one. Buoyancy terms modeling the pressure-correlation are shown to improve the prediction slightly. The effects of equilibrium time-scale ratio and boundary conditions are also discussed.

Shabbir, Aamir (ICOMP) and Shih, Tsan-Hsing (ICOMP): "*Critical Assessment of Reynolds Stress Turbulence Models Using Homogeneous Flows*", ICOMP Report No. 92-24, CMOTT-92-12, NASA TM-105954, December 1992.

We numerically compute thirteen homogeneous flows using three different Reynolds stress closure models. We concentrate only on those models which at most use a quasi-linear model for the rapid pressure strain correlation. This includes LRR, IP and SSG models. The flows computed include the flow through axisymmetric contraction; axisymmetric expansion; distortion by plane strain; and homogeneous shear flows with and without rotation. Results of such a computation depend on models for the pressure strain term and the dissipation rate equation. This approach is useful for checking the predictive capabilities of a complete model. From the overall performance of these models it is found that LRR model (with a slightly modified model constant) performs better than both the IP and the SSG models.

Shabbir, Aamir (ICOMP) and George, William K. (State University of New York): "Experiments on a Round Turbulent Buoyant Plume", ICOMP Report No. 92-25, CMOTT 92-13, NASA TM-105955, December 1992.

This paper reports a comprehensive set of hot-wire measurements of a round buoyant plume which was generated by forcing a jet of hot air vertically up into quiescent environment. The boundary conditions of the experiment were measured and are documented in the present paper in an attempt to sort out the contradictory mean flow results from the earlier studies. The ambient temperature was monitored to insure that the facility was not stratified and that the experiment was conducted in a neutral environment. The axisymmetry of the flow was checked by using a planer array of sixteen thermocouples and the mean temperature measurements from these are used to supplement the hot-wire measurements. The source flow conditions were measured so as to ascertain the rate at which the buoyancy was added to the flow. The measurements conserve buoyancy within 10%. The results are used to carry out the balances of the mean energy and momentum differential equations. In the mean energy equation it is found that the vertical advection of the energy is primarily balanced by the radial turbulent transport. In the mean momentum equation the vertical advection of momentum and the buoyancy force balance the radial turbulent transport. The buoyancy force is the second largest term in this balance and is responsible for the wider (and higher) velocity profiles in plumes as compared to jets. Budgets of the temperature variance and turbulence kinetic energy are also carried out in which thermal and mechanical dissipation rates are obtained as the closing terms. Similarities and differences between the two balances are discussed. It is found that even though the direct effect of buoyancy on turbulence, as evidenced by the buoyancy production term, is substantial, most of the turbulence is produced by shear. This is in contrast to the mean velocity field where the effect of buoyancy force is quite strong. Therefore, it is concluded that in a buoyant plume the primary effect of buoyancy on turbulence is indirect, and enters through the mean velocity field (giving larger shear production).

Hayder, Ehtesham (ICOMP) Turkel, Eli (ICOMP) and Mankbadi, Reda R. (NASA Lewis): "Numerical Simulation of a High Mach Number Jet", ICOMP Report No. 92-26, NASA TM 105985, January 1993, 16 pages.

Two dimensional simulations of plane and axisymmetric jets are presented. These simulations were made by solving full Navier-Stokes equations using a high order finite difference scheme. Simulation results are in good agreement with the linear theory predictions of the growth of instability waves.

Shih, Tsan-Hsing (ICOMP), Zhu, Jiang (ICOMP) and Lumley, John L. (Cornell University): "A Realizable Reynolds Stress Algebraic Equation Model", <u>ICOMP Report No. 92-27</u>, CMOTT 92-14, NASA TM-105993, January 1993, 36 pages.

The invariance theory in continuum mechanics is applied to analyze Reynoids stresses in high Reynolds number turbulent flows. The analysis leads to a turbulent constitutive relation that relates the Reynolds stresses to the mean velocity gradients in a more general form in which the classical isotropic eddy viscosity model is just the linear approximation of the general form. On the basis of realizability analysis, a set of model coefficients are obtained which are functions of the time scale ratios of the turbulence to the mean strain rate and the mean rotation rate. The coefficients will ensure the positivity of each component of the mean rotation rate. These coefficients will ensure the positivity of each component of the turbulent kinetic energy-realizability that most existing turbulence models fails to satisfy. Separated flows over backward-facing step configurations are taken as applications. The calculations are performed with a conservative finite-volume method. Grid-independent numerical diffusion-free solutions are obtained by using differencing schemes of second-order accuracy on sufficiently fine grids. The calculated results are compared in detail with the experimental data for both mean and turbulent quantities. The comparison shows that the present proposal significantly improves the predictive capability of K-ε based two equations models. In addition, the proposed model is able to simulate rotational homogeneous shear flows with large rotation rates which all conventional eddy viscosity models fail to simulate.

SEMINARS

(* = CMOTT Seminars)

Aggarwal, Suresh K. (University of Illinois): "Droplet Dispersion with Large Eddy Simulation in Shear Flows"

The objective of this research is to study the dispersion and vaporization behavior of droplets in turbulent shear flows dominated by large vortical structures. In this presentation, the results will be discussed on the dynamics of large vortical structures and the dispersion of droplets in these structures. Two flow configurations considered are a transitional planar shear layer and an axisymmetric free jet. The results highlight the centrifugal mechanism which is responsible for the enhanced dispersion of intermediate size droplets. In addition, the effects of subharmonic forcing of the shear layer on droplet dispersion will be discussed.

Bayliss, Alvin (Northwestern University): "Nonlinear Dynamics and Spatial Pattern Formation in Gaseous Combustion"

In this talk we describe three examples of complex spatial and temporal pattern formation which can occur in the burning of a combustible gaseous mixture. We consider flames established either in the interior of a cylinder or in the region between two concentric cylinders, with the combustible mixture fed in through the inner cylindrical tube.

In the first example we solve a model in which the thermal expansion of the gas is assumed to be weak (diffusional thermal model) and consider the effect on the spatial and temporal patterns when the diffusivity of the limiting component of the reaction is increased as would occur in, for example, lean Hydrogen/air mixtures. We consider cellular flames, i.e. flames for which there are alternating regions of high and low temperature (cells) along the flame front. As the diffusivity is decreased we find transitions from axisymmetric stationary flames, to stationary cellular flames to rotating cellular flames. If the diffusivity is further decreased we find quasi-periodic dynamics and non-periodic cellular arrays.

In the second example we consider axisymmetric flames and a model which accounts for the thermal expansion of the gas. We study the role of the injection velocity on the dynamics and consider in particular the case when the injection velocity is reduced to near extinction conditions. We find a sequence of period doubling transitions leading to apparently chaotic behavior near the extinction limit.

In the third example we consider the behavior of rotating non-adiabatic cellular flames, described by the diffusional thermal model, when the diffusivity of the deficient component of the reactant is small, for example lean, heavy hydrocarbon/air mixtures. Near the extinction limit we find that subharmonic modes are generated along the flame front. This leads to a complex sequence of transitions, the nature of which depends critically on the number of cells which are initially presented. If the number of cells present initially is a power of two, we find a finite sequence of period doublings in space and time and ten a transition to non-periodic behavior. In the case that the number of cells present initially is odd, we find a transition to modulated traveling waves, with the cells exhibiting a periodic oscillation. The cell motion is found to undergo several period doublings and then a transition to apparently chaotic behavior.

*Bui, Trong T. (NASA Lewis): "Some Practical Turbulence Modeling Options for Full Reynolds-Averaged Navier-Stokes Calculations of 3D Flows"

New turbulence modeling options currently under development for Proteus, a general purpose compressible full Reynolds-averaged Navier-Stokes code, are discussed. The turbulence modeling capability in the 3D version of Proteus in the current work consists for four turbulence models: the Baldwin-Lomax, the Baldwin-Barth, the Chien k-e, and the Launder-Sharma k-e models. Five compressibility corrections and one length scale correction are also available for the k-e models. Features of the Proteus turbulence modeling package include: well documented and easy to use turbulence modeling modules, uniform integration of turbulence models from different classes, automatic starting options for turbulence calculations using any turbulence model, and fully vectorized L-U solver for one- and two-equation models. Validation test cases include the incompressible and compressible flat plate turbulent boundary layers, turbulent developing S-educt flow, and

glancing shock wave/turbulent boundary layer interaction. Sensitivity of the turbulence solutions with y+ computation and compressibility options are examined. The test cases show that the highly optimized one- and two-equation turbulence models can be used in routine 3D Navier-Stokes computations with no significant increase in CPU time as compared with the algebraic Baldwin-Lomax model.

*Davis, David O. (NASA Lewis): "Experimental Investigation of Turbulent Supersonic Developing Pipe Flow"

Turbulent, supersonic, developing pipe flow is being investigated in the NASA Lewis 5"X5" Supersonic Wind Tunnel. The objective of the study is to establish a baseline dataset for the validation of turbulence measurement techniques. The resulting dataset should also be useful for turbulence model and CFD code validation. The mean flow and turbulence field is being measured with pressure probes and hot-wire anemometry. Preliminary results have been obtained for a Mach 3 inlet condition and a flow development length of x/D=32 (D=2.0 inches). These results, along with a discussion of the problems of hot-wire anemometry in a compressible boundary layer will be presented. Input from turbulence modelers and code developers regarding future testing with the supersonic pipe flow facility will be solicited.

*Davis, Dominic (ICOMP): "Weakly Nonlinear Vortex/Wave Interactions in Incompressible Crossflow Boundary Layers in Transition"

The instability of an incompressible 3D boundary layer is considered theoretically and computationally in the context of vortex/wave interactions. Specifically the work centers on two low-amplitude, lower-branch Tollmien-Schlichting (TS) wave which mutually interact to induce a weak longitudinal vortex flow; the vortex motion, in turn, gives rise to significant wave-modulation via wall-shear forcing. The characteristic Reynolds-number is taken as a large parameter and, as a consequence, the TS waves and the vortex are governed primarily by triple-deck theory. The nonlinear interaction is captured by a viscous partial-differential system for the vortex coupled with a pair of amplitude equations for the wave pressures. Computations were performed for relatively small crossflow values. Three distinct possibilities were found to emerge for the nonlinear behavior of the flow solution downstream - an algebraic finite-distance singularity, far-downstream decay or repeated oscillations - depending on the various parameter values, the input amplitudes and the wave angles.

Deissler, Robert J. (ICOMP): "Noise-Sustained Structure in the Navier-Stokes Equations: Taylor-Couette Flow with Through-Flow"

I will first review my early work on the complex Ginzburg-Landau equation, where the concept of noise-sustained structure was first introduced. Briefly, in *convectively* unstable systems - meaning that a localized perturbation is convected with the flow such that it grows only in a moving frame, eventually damping at any fixed point - continuous external noise will be selectively and spatially amplified giving rise to a *noise-sustained structure*. If the noise is removed from the system. The structure will be convected out through the boundary and the system will return to its unperturbed state.

After this I will discuss a recent numerical solution of the Navier-Stokes equation for Taylor-Couette flow with an imposed axial flow which was done in collaboration with Dr. Wai-Ming To (Sverdrup Technology). For a sufficiently large axial flow rate this system will be convectively unstable and therefore can exhibit noise-sustained structure. We find that indeed noise-sustained structure results. A video of the flow will be shown for varying noise levels. Also, comparison will be made with recent experiments of this system by Babcock *et al*. In particular we are interested in investigating a recent study indicating that the noise-sustained structure in the experiments may be of thermal origin.

Demuren, A. O. (ICOMP): "Multigrid Acceleration for the Proteus Computer Code"

The lecture presents the progress made in incorporating multigrid acceleration into the 2D version of the Proteus computer code. Through von Neumann stability analysis methods, and actual computations of several test cases on three sets of grids with increasing levels of refinement, it is shown that the code has convergence properties which are typical of single-grid methods based on approximate factorization. The error reduction rate is of order $[1 - O(h^2)]$ so that it approaches unity as the grid is refined. The analyses also

show that it is a good candidate for multigrid acceleration, provided care is exercised in specifying artificial dissipation terms and in the choice of the CFL number. The goal of the multigrid method is to obtain convergence rates which are independent of grid refinement. A fixed, V-cycle multigrid procedure is implemented in the 2D version of the code and initial results indicate that while the goal has not been completely realized, sensitivity to grid refinement has been considerably reduced. Typically, speed-up factors of 2-5 have been achieved in selected computations via multigrid. All the multigrid routines are self-contained. It has been necessary to make slight modifications to only two routines in the code; *program main* and *subroutine exec*. Future work involves implementation and testing of the multigrid procedures for turbulent variables and for 3D flow.

*Duncan, Beverly (Sverdrup): "Validation of a Two-Scale Turbulence Model for Incompressible Shear Flows"

A two-scale eddy viscosity model has been developed which splits the energy spectrum into a high wave number regime and a low wave number regime. Dividing the energy spectrum into multiple regimes simplistically emulates the cascade of energy through the turbulence spectrum. This new model has been calibrated and tested for turbulent shear layers. Calculations of mean and turbulent properties show good agreement to experimental data for a plane jet, a round jet and two mixing layers. Preliminary results for boundary layers will also be presented.

Fix, George J. (University of Texas at Arlington): "Effects of Grid Irregularities on Iterative Methods"

Irregular and sometimes totally unstructured grids are finding greater use in the attempt to gain increased accuracy in both finite element and finite difference schemes with fewer degrees of freedom. For some iterative methods such as conjugate gradients, the effects of irregular grids can be predicted through the conditioning of the associated algebraic systems. With other iterative schemes, e.g., relaxation and multi-grid, the relation between rates of convergence for the iterations and condition numbers is not as direct. In this talk, a survey of the effects will be given through selected numerical examples. A theoretical analysis of model problems will also be given.

Frankel, Steven H. (SUNY at Buffalo): "Probabilistic and Deterministic Description of Reacting Turbulence"

A *potpourri* of mathematical techniques are presented for the modeling of turbulent reacting flows. Specifically, closed form analytical expressions are presented for predicting the maximum rate of reactant conversion in a binary reaction of the type $F + rO \rightarrow (1 + r)$ Products in non-premixed, homogeneous and non-homogeneous turbulence. These relations are determined through the use of the single point Probability Density Function (PDF) method. The amplitude mapping closure of Kraichnan-Pope (Chen et al. 1989; Pope 1991) leads to expressions involving the Parabolic Cylinder Function (PCF) for the general case of non-stoichiometric mixtures, and the use of the Pearson Family (PF-J) of densities yields expressions in terms of the Incomplete Beta Function (IBF). These mathematical expressions simplify considerably for a stoichiometric mixture. Data generated from Direct Numerical Simulations (DNS) are used to validate these formulations in both homogeneous and non-homogeneous turbulent reacting flows (Frankel et al. 1992a). Next a family of Probability Density Functions (PDF's) generated by Johnson-Edgeworth Translation (JET) is used for statistical modeling of the mixing of an initially binary scalar in isotropic turbulence (Miller et al. 1992). However, none of the models considered (JET, AMC, and PF) are capable of predicting the evolution of the conditional expected dissipation and/or the conditional expected diffusion of the scalar field in accordance with DNS. It is demonstrated that this is due to the incapability of the models to account for the variations of the scalar bounds as the mixing proceeds. The mapping closure results are also coupled to the Eddy-Dampled Quasi-Normal Markovian (EDQNM) spectral closure to produce an attractive hybrid model for the modeling of plug-flow reactors (Frankel et al. 1992b). Comparisons with experimental results are made for this hybrid mapping-EDQNM model and exhibit excellent agreements. The idea of implementing Pearson family PDFs for use as a subgrid scale closure (SGS) in Large Eddy Simulations (LES) of reacting flows is then postulated and investigated via a priori and a posteriori analysis in both homogeneous and non-homogeneous reacting turbulence. A discussion is also provided on the recently developed Linear Eddy Model (LEM) of turbulent mixing, and also the applications of LEM in treating more complex turbulent reacting flows.

*Garg, Vijay K. (NRC): "Heat Transfer in Film-Cooled Turbine Blades"

There is a growing tendency to use higher temperatures at the inlet to a gas turbine in an effort to improve the thermal efficiency and increase the specific power output. This calls for increasingly effective means of cooling the turbines blades. One cooling technique presently receiving wide application is film cooling. In order to study the effect of film cooling on the flow and heat transfer characteristics of actual turbines blades, Rod Chima's 3D Navier-Stokes code has been modified. The effects of film cooling have been incorporated into the code in the form of appropriate boundary conditions at the hole locations on the blade surface. Each hole exit (generally an ellipse) is represented by several control volumes. This provides the code an ability to study the effect of hole shape on the film-cooling characteristics. Different velocity and temperature profiles for the injected gas can be specified at the hole exit. These include uniform, laminar or turbulent (1/7th power-law) profiles. The code can also handle either a specified heat flux or a variable temperature condition on the blade surface. A periodic C-grid with nearly half a million grid points is used. Comparison with experimental data is fair.

*Georgiadis, Nick (NASA Lewis): "Evaluation of the Turbulence Models in PARC"

The PARC2D/3D Navier-Stokes codes are used to analyze a variety of complex turbulent propulsion flows. Both algebraic and two-equation turbulence models are available in PARC. This presentation will compare the capabilities of PARC's turbulence models to predict several flows including benchmark turbulent test cases (flow over a flat plate and over a backward-facing step), the Sajben transonic diffuser flow, and other propulsion flow cases. The algebraic turbulence models that are investigated are the P.D. Thomas model (the standard turbulence model in PARC which was optimized for free shear layer flows but also calculates wall boundary layers), the Bladwin-Lomax model, and a new combination model which uses the Modified Mixing Length (MML) for wall boundary layers and the Thomas model for free shear layers. The two-equation model that is also investigated is the Chien k-epsilon model with modifications for compressibility added by Nichols. The presentation will show that there are significant differences among flow solutions obtained with these turbulence models for even the simplest of flows.

*Goodrich, John (NASA Lewis): "Unsteady Time Asymptotic States: Incompressible Results and New Directions for Algorithms"

Unsteady time asymptotic flow states for high Reynolds-number viscous incompressible flow problems are presented. Discrete frequency flows are shown for the square driven cavity, with periodic cases for Re = 9000 and Re = 9600, and with aperiodic cases for Re = 9700 and Re = 10000. The algorithm for these calculations is based on the fourth order PDE for incompressible fluid flow which uses the stream function as the only dependent variable, it is second order accurate in space and time, and it has a stability constraint of $CFL \leq 1$. The algorithm is extremely robust with respect to Reynolds number.

The direct numerical simulation of transition and turbulence requires numerical methods to be more than second-order accurate in order to accurately represent the relevant scales of the physical processes. Recently developed finite difference algorithms are presented for unsteady convection equations, including the advection and inviscid Burgers equation in one space dimension, and the wave equation treated as a system, with remarks on diffusion equations and extension to higher space dimension. The new algorithms that will be discussed all use local stencils, they range from third to seventh-order in accuracy, they all have the same order of accuracy in both space and time, and they are all one step explicit methods (except for diffusion equations). Since all of the algorithms use a small local stencil, the number of degrees of freedom of known data required for higher order accuracy is obtained by higher information density than just the solution data. The use of a two point stencil (for some of the methods) allows for arbitrary grid spacing, though a convective stability constraint must be observed at each grid point. The use of local data for an explicit algorithm with high order accuracy avoids the requirement of using a global solution method such as compact differencing or spline based algorithms. There will be computational results for all of the algorithms that are presented.

*Kim, S.-W. (NASA Lewis), Zaman, K. (NASA Lewis) and Panda, J. (NASA Lewis): "Calculation of Unsteady Turbulent Flow over Oscillating Airfoil"

A numerical method to solve unsteady turbulent flows with moving boundaries and calculation of unsteady turbulent flow over an oscillating NACA 0012 airfoil at Reynolds number of 44,000 are presented. The Navier-Stokes equations defined on Lagrangian-Eulerian coordinates are solved by a time-accurate finite volume method that incorporates an incremental pressure equation for the conservation of mass. The turbulence filed is described by a multiple-time-scale turbulence model. The numerical method successfully predicts the large dynamic stall vortex and the trailing edge vortex that are periodically generated by the oscillating airfoil. The calculated turbulence field show that the transition from laminar to turbulent state occurs widely along the airfoil. The calculated streaklines and the ensemble-average mean velocity profiles are in good agreement with the measured data.

*Lee, Jinho (SVERDRUP): "Development of the RPLUS Code with Standard κ-ε Models and Its Applications"

The primary goal of this research effort is to develop a CFD tool which can be used in a variety of practical supersonic/hypersonic propulsion device development/analysis environments. One focus of this work has been to develop and validate the Reactive Propulsion code based on LU Scheme (RPLUS). This effort also includes the development of turbulence models which can be used in the predictions of highly complex flow environments inside of combustors.

This presentation will cover only a small part of a larger development effort and the focus will be primarily on the analysis, implementation, and development of the turbulence model capabilities of the RPLUS code.

Some of the issues which will be covered are; formulation of the turbulence models; the numerical technique used to solve the turbulence model equations; and modeling of compressibility effects. The primary numerical technique used in the RPLUS code is the LU-SSOR (LU scheme based on Successive Symmetric Over Relaxation) technique. Therefore, the turbulent kinetic energy transport and dissipation transport equations are also solved using the LU-SSOR numerical technique.

Both 2D and 3D turbulence models are being developed for the RPLUS code. However, the majority of the presentation will focus on the development of the 2D κ-ε models for the RPLUS code. Issues regarding compressible wall-function boundary conditions and compressibility effects will be addressed. Both low and high Reynolds number forms of the κ-ε models are being developed. The "standard" low Reynolds number model of Launder-Sharma and Chien has been used in this study. The problems of primary interests are supersonic turbulent boundary-layers, shock-wave/boundary-layer interactions, and shear-layers in 2D and 3D environments.

Leonard, B. P. (ICOMP): "The (Frustrating) Problem of Highly Advective Flow: The Search for Positivity-Preserving Two-Dimensional Flux Limiters"

Simulating highly advective multi-dimensional flow is a very challenging problem. Most CFD schemes in common use fail miserably when tested on a superficially extremely simple problem: pure advection by a constant velocity field at an angle to the grid. The uniformly third-order polynomial interpolation algorithm (UTOPIA) and an extension (UTOPIA+) do fairly well on smooth profiles but discontinuities generate overshoots and undershoots. Truly multi-dimensional positivity-preserving flux limiters can be found, but they introduce terrible anisotropic distortion. The search continues!

*Liou, William W. (ICOMP): "Extension of a Two-Scale Turbulence Model to Compressible Free Shear Flows"

A two-scale model for compressible turbulent flows is described. The model incorporates a notion that the effect of compressibility on turbulence is mainly on the energetic large eddies. The small eddies are affected through the increased spectral energy transfer from the large eddies due to the effect of compressibility. The turbulent eddy-viscosity is determined by the total turbulent kinetic energy and the energy transfer rate, which is different from the energy dissipation rate. The model is tested against high-speed mixing layers. The results agree satisfactorily with measured data.

Madabhushi, Revi K. (University of Illinois): "Direct and Large-Eddy Simulations of Turbulent Flow in a Square Duct"

Turbulence-driven mean secondary flows are known to exist in straight ducts of non-circular cross-sections. These secondary flows distort the contours of mean streamwise velocity, temperature, turbulence, kinetic energy etc. toward the corners. The fully-developed turbulent flow in a square duct has been simulated using the Direct Numerical Simulation (DNS) and Large-Eddy Simulation (LES) techniques. The time-dependent, 3-D Navier-Stokes equations for an incompressible flow have been solved using mixed spectral-finite-difference and full spectral methods. Some of the results from these simulations are discussed in this presentation.

*Mankbadi, Reda R. (NASA Lewis): "Unsteady Turbulent Flows"

Current research activities emphasize computation/modelling of turbulent flows when basic flow is time-periodic. Wall-bounded flows and free shear flows exhibit different features when the basic flow is unsteady; and as such, different approaches are used to model them.

(A) In wall-bounded, oscillatory flows, two approaches are used to model turbulence: (1) Turbulence is assumed to behave in a quasi-steady manner, and steady-state models are directly extended to the unsteady case. This approach fails at high frequencies of oscillations. (2) Rapid Distortion Theory (RDT) is successfully adapted to aid in turbulence modelling of highly unsteady flows (high frequencies). The eddy viscosity hypothesis is replaced by the ratio of turbulent stresses/kinetic energy; which is given by RDT as a function of the accumulated rate of strain.

(B) In free shear flows (naturally unsteady, or excited to be unsteady), two approaches are investigated: (1) The large-scale (organized, coherent) component is modelled as instability waves interacting with each other as well as with the mean flow and the fine-scale (random, background) turbulence. Integrated kinetic-energy equations are then obtained for each scale of motion. The approach is successful in predicting results in good agreements with experiments in which excitation devices are used to control jet mixing and turbulence. (2) The other approach adopted is Large-Eddy Simulations (LES) with application to predicting the far-field noise of a supersonic jet.

Pierre, Christophe (University of Michigan) and Ottarsson, Gisli (University of Michigan): "A Measure of Sensitivity to Mistuning for Models of Bladed-Disk Assemblies"

A study of mode localization in mistuned bladed disks is performed using transfer matrices. The approach yields the free response of a general, mono-coupled, perfectly cyclic assembly in closed form. A mistuned structure is represented by random transfer matrices and the expansion of these matrices in terms of the small mistuning parameter leads to the definition of a measure of sensitivity to mistuning. An approximation of the localization factor, the exponential attenuation per blade-disk site, is obtained through perturbation techniques in the limits of high and low sensitivity. The method is applied to a common model of a bladed disk and the results verified by Monte Carlo simulations. The easily calculated sensitivity measure may prove to be a valuable design tool due to its system-independent quantification of mistuning effects such as vibration localization.

The mon-coupled model is then expanded by adding coupling between non-adjacent blade-disk sites. The conditions under which the dynamics of the system with two coupling coordinates can be approximated by the mono-coupled system are examined. Finally, preliminary results of a forced response analysis currently underway are presented in order to assess the validity of the sensitivity measure.

Powell, Kenneth G. (University of Michigan): "Computational MHD The Michigan Way - Godunov Meets Maxwell"

A computational approach for solving equations for a conducting fluid in a magnetic field will be described. The approach uses a Godunov-based scheme similar to that of modern CFD codes. The derivation of a Roe scheme for the MHD equations will be described, and the underlying Riemann problem on which the Roe scheme is based will be discussed. In particular, properties of the Riemann problem of MHD which are not

encountered in the Riemann problem of gas dynamics, and ways to surmount the problems caused by these properties, will be detailed. The incorporation of this Roe scheme for MHD into an adaptively-refined-mesh solution procedure will be described, and some preliminary results of the code will be shown. Finally, some discussion of work that is planned for extending the code to non-ideal MHD flows, and to applications in propulsion and space physics, will be discussed.

*Rigby, David (SVR): "The Effect of Spanwise Variations in Momentum on Leading Edge Heat Transfer"

A study of the effect of spanwise variation in momentum on leading edge heat transfer is discussed. Numerical and experimental results are presented for a circular leading edge and for a 3:1 elliptical leading edge. Direct comparison of the 2D results, that is with no spanwise variations, to the analytical results of Frossling is very good. The numerical calculation, using the PARC3D code, solves the 3D Navier-Stokes equations, assuming steady laminar flow on the leading edge region. Experimentally, increases in spanwise averaged heat transfer coefficient as high as 50% above the 2D value were observed. Numerically, the heat transfer coefficient was seen to increase by as much as 25%. In general, the circular leading edge, under the same flow conditions, produced a higher heat transfer rate than the elliptical leading edge. As a percentage of the respective 2D values, the circular and elliptical leading edges showed similar sensitivity to spanwise variations in momentum. By equating the root mean square of the amplitude of the spanwise variation in momentum to the turbulence intensity, a qualitative comparison between the present work and turbulent results was possible.

*Rubinstein, Robert (Sverdrup Technology Inc.): "Analytical Theories of Turbulence Applied to Second Order Closures and Time-Dependent Modeling"

The general program of deriving turbulence models from analytical theories like renormalization group and direct interaction approximation will be described. Renormalization group analysis of the Reynolds stress transport equation leads to the Launder-Reece-Rodi as a lowest order approximate model with theoretically computed constants in good agreement with accepted values. However, the analysis shows that this model is not exact and justifies its replacement either with higher order non-linear models or with a generalization of the stress transport equation based on a decomposition of the Reynolds stress. Application of the direct interaction approximation to time-dependent turbulence models will be discussed.

*Shih, Tsan-Hsing (ICOMP): "Kolmogorov Behavior of Near-Wall Turbulence and Its Application in Turbulence Modelling"

The near-wall behavior of turbulence is re-examined in a way different from that proposed by Hanjalic and Launder[1] and followers[2], [3], [4], [5]. It is shown that at a certain distance from the wall, all energetic large eddies will reduce to *Kolmogorov* eddies (the smallest eddies in turbulence). All the important wall parameters, such as friction velocity, viscous length scale, and mean strain rate at the wall, are characterized by *Kolmogorov* microscales. According to this *Kolmogorov* behavior of near-wall turbulence, the turbulence quantities, such as turbulent kinetic energy, dissipation rate, etc. at the location where the large eddies become *"Kolmogorov"* eddies, can be estimated by using both direct numerical simulation (DNS) data and asymptotic analysis of near-wall turbulence. This information will provide useful boundary conditions for the turbulent transport equations. As an example, the concept is incorporated in the standard κ-ε model which is then applied to channel and boundary layer flows. Using appropriate boundary conditions (based on *Kolmogorov* behavior of near-wall turbulence), there is no need for any wall-modification to the κ-ε equations (including model constants). Results compare very well with the DNS and experimental data.

Sidi, Avram (Technion-Israel Institute of Technology): "Convergence Rates of Vector Extrapolation Methods on Linear Systems with Initial Iterations"

The application of the minimal polynomial extrapolation (MPE) and the reduced rank extrapolation (RRE) to a vector sequence obtained by the linear iterative technique $x_{j+1} = Ax_j + b$, $j = 1,2,...,$ is considered. Both methods produce a 2D array of approximations $s_{n,k}$ to the solution of the system $(I - A)x = b$. Here $s_{n,k}$ is obtained from the vectors x_j, $n \leq j \leq n + k + 1$. It was observed earlier that the sequence $s_{n,k}$, $k = 1,2,...,$ for $n > 0$, but

fixed, possesses better convergence properties than the sequence $s_{o,k}$, $k = 1,2...$. A theoretical explanation for this phenomenon is provided. This explanation is based on approximations by incomplete polynomials. It is demonstrated by numerical examples when the matrix A is sparse that cycling with $s_{n,k}$ for $n > 0$, but fixed, produces better convergence rates and costs less computationally than cycling with $s_{o,k}$. It is also illustrated numerically with a convection-diffusion problem that the former may produce excellent results where the latter may fail completely. As has been shown earlier, the results produced by $s_{o,k}$ are <u>identical</u> to the corresponding results obtained by applying the Arnoldi method or GMRES to the system $(I - A)x = b$.

*So, Ronald M. C. (Arizona State University): "Near-Wall Modelling of Turbulent Heat Transfer"

A near-wall two-equation model for turbulent heat fluxes is derived from the temperature variance and its dissipation-rate equations and the assumption of gradient transport. The near-wall asymptotics of each term in the exact equations are examined and used to derive near-wall correction functions that render the modeled equations consistent with these behaviors. Thus modeled, the equations are used to calculate fully-developed pipe and channel flows with heat transfer. It is found that the proposed two-equation model yields asymptotically correct near-wall behavior for the normal heat flux, the temperature variance and its near-wall budget and correct limiting wall values for these properties compared to direct simulation data and measurements obtained under different wall boundary conditions.

Stuart, J. T. (Imperial College): "The Principle of Duality and Nonlinear Problems of Vorticity Evolution"

The aerodynamics of flows can sometimes be simplified by use of a so-called vortex sheet, but this can itself lead to new computational problems when the timewise development is considered. This lecture considers a new formulation, which enables a mathematical and computational treatment to be made in a more satisfactory way.

Tabata, Masahisa (University of Electro-Communications) and Fujima, Shoichi (University of Electro-Communications): "Two-Dimensional and Three-Dimensional Upwind Finite Element Schemes for High Reynolds-Number Flows"

Recently we have developed 2D and 3D upwind finite element schemes for the incompressible Navier-Stokes equations at high Reynolds-Numbers.
The idea of the upwind technique is based on the choice of upwind and downwind points. The scheme has the potential to approximate the convection term in third-order accuracy. Therefore, the included numerical viscosity is small, and the scheme can produce solutions showing a good dependence on the Reynolds-Number. We apply the scheme to typical 2D and 3D flow problems at high Reynolds-Numbers.

Tsang, Tate T. H. (University of Kentucky) and Tang, L. Q. (University of Kentucky): "A Least-Squares Finite Element Method for Time-Dependent Incompressible flows and Transport Processes"

A least-squares finite element method is proposed for solving time-dependent incompressible flow problems. Unlike the commonly used Galerkin finite element method, the least-squares approach can allow equal-order interpolations for velocity and pressure. Furthermore, the method gives rise to a symmetric positive definite matrix system. The least-squares method is based on the velocity-pressure-vorticity-temperature-heat flux-composition-mass flux formulation. We have tested the method by three standard flow problems of driven cavity flow, flow over a backward facing step and flow over a square obstacle. We further extend the method to simulate thermally driven flow, doubly-diffusive flow and advective transport. For each case study, the least-squares finite element method provides satisfactory results. Future applications to air pollution modeling, atmospheric turbulence and aspects of parallelization of the method will be discussed.

*Van der Vegt, Jaap (ICOMP): "The Development of an ENO-Osher Scheme for Direct Simulation of Compressible Flows"

Direct simulation of turbulence and transition in compressible wall bounded flows presents an alternative to investigate important physical phenomena which are difficult to measure or study otherwise. It also provides

data useful for turbulence modelling. A new program is being developed which solves the 3D compressible Navier-Stokes equations using a higher order, fully implicit and time accurate ENO scheme together with Osher flux splitting. In this presentation an overview will be given of the numerical scheme and several test cases, both for supersonic and subsonic flow, will be presented and further improvements will be discussed.

*Yang, Zhigang (ICOMP): "A κ-ε Model for Near-Wall Turbulence and its Application in Turbulent Boundary Layer with/without Pressure Gradient"

A κ-ε model is proposed for turbulent wall bounded flows. In this model, turbulent velocity scale and turbulent time scale are used to define the eddy viscosity. The time scale used is bounded from below by the Kolmogorov time scale. The dissipation equation is reformulated using this time scale, removing the singularity of the high Reynolds-number κ-ε equation at the wall and rendering the introduction of the pseudo-dissipation unnecessary. The damping function used in the eddy viscosity is chosen to be a function of

$$R_y = \frac{k^{1/2} \, y}{v}$$

instead of y^+. Thus, the model could be used for flows with separation. The model constants used are the same as the model constants in the commonly used high turbulent Reynolds standard κ-ε model. Thus, the proposed model would reduce to the standard κ-ε model when it is far away from the wall. Boundary layer flows at zero pressure gradient, favorable pressure gradient, adverse pressure gradient and increasingly adverse pressure gradient are calculated respectively. The comparisons of model predictions and the available experimental data are found to be good.

*Yang, Zhigang (ICOMP): "A Modeling of Bypass Transition"

A model for the calculation of bypass transitional boundary layers due to the freestream turbulence is proposed. The model combines a near wall κ-ε model proposed for the fully developed turbulent flows with the intermittency of the transitional boundary layers. The intermittency factor is assumed to be a function of both the free stream turbulence and the shape factor of the boundary layer. Transitional boundary layers over a flat plate with different free stream turbulence level are calculated using the proposed model. It is found that the model calculations agree well with the experimental data and give a better prediction compared with other low Reynolds-number κ-ε models, which do not incorporate the intermittency effect.

*Zaman, Khairul (NASA Lewis): "Effect on Tabs on the Evolution of Axisymmetric Jets"

Vortex generators, in the form of small tabs at the nozzle exit, can have a profound influence on the evolution of an axisymmetric jet. Using tabs of certain shapes, the jet cross section can be distorted almost arbitrarily. Such distortion is accompanied by elimination of screech noise from supersonic jets and a significant increase in jet mixing. Key results obtained so far, covering a jet Mach number range of 0.3 and 1.8, will be summarized in this presentation. Observations will be made on the mechanisms of the effect including the likely vorticity dynamics in the flow.

*Zhu, J. (ICOMP): "Finite-Volume Computations of Incompressible Flows with Complex Geometries"

A brief review is given of finite-volume procedures developed at the Institute for Hydromechanics, University of Karlsruhe, for calculating incompressible elliptic flows with complex boundaries. The procedures include: numerical grid generation, higher-order bounded convection schemes, zonal solution, simulation of two-phase flows, and near-wall turbulence modelling. Various application examples will be given.

TURBULENCE LECTURE SERIES

Under the auspices of the Center for Modeling and Transition (CMOTT), Dr. Tsan-Hsing Shih, Technical Leader for CMOTT, organized and conducted a special lecture series on turbulence modeling for the LeRC, CMOTT, and Sverdrup staff members active in this field. The lectures were all given by Dr. Shih with an average attendance of 65. The objective and a description of the course material follow:

OBJECTIVES

The main objective of this lecture series was to introduce the fundamentals of turbulence and computational turbulence modeling to the engineers working on aerospace propulsion systems and to enhance collaboration between the different groups at LeRC in turbulence research. The basic concepts of turbulence, turbulence theory and related tools were described in detail. The material in this lecture series reflects recent developments and the state-of-the-art in computational turbulence modeling. The lecture topics were as follows:

1. **Introduction**

   1.1 The characteristics of turbulence

   Randomness, Diffusivity, Dissipation, 3-D vorticity fluctuations.

   1.2 The origin of turbulence

   Natural transition, Bypass transition.

   1.3 Methods of analysis

   Dimensional analysis, Asymptotic invariance, Self-preservation, Invariant theory.

   1.4 Scales in turbulent flows

   The concept of turbulence eddies, Scales in turbulence.

   1.5 Turbulence closure problem in computational fluid dynamics

   Closure problem, Hierarchy of closures.

   1.6 Numerical simulation of turbulence

   Large-eddy simulation, Direct numerical simulation.

2. **Fundamentals of Fluid Motion**

   2.1 The basic equations

   Compact notation, Surface force F_i and stress tensor Ω_{ij}, Deformation rate tensors $u_{i,j}$, S_{ij} and T_{ij}, Continuity equation, Irreversible momentum equation, Irreversible energy equation, Constitutive relations, Equation of state, Boundary conditions.

2.2   Vorticity dynamics

Vorticity vector ω_i and rotation rate tensor Ω_{ij}, Contribution of vorticity to the velocity field, Vorticity equation.

2.3   Basic equation in non-inertial frame

2.4   A decomposition of the compressible velocity field

3. **Turbulence Equations**

3.1   Averaging methods

Time averaging, Space averaging, Ensemble averaging, Turbulence decomposition.

3.2. Governing equations for computational fluid dynamics (CFD)

Mean equations for computational fluid dynamics, One-point turbulence correlations, Alternative form of the mean momentum equation, Density weighted averaging and compressible flows.

3.3   Turbulence equations for CFD

Turbulent stress transport equations, Turbulent scalar flux transport equations, Turbulence variance transport equations, Dissipation equations, Equations for high-order turbulent correlations.

3.4   Two-point turbulence correlations

Two-point correlations, Dynamical equation for two-point correlations, Homogeneous turbulence, Some properties of two-point correlations.

4. **The Dynamics of Turbulence**

4.1   The dynamics of turbulent kinetic energy

Kinetic energy of the turbulence, Kinetic energy of the mean, Production and dissipation, Spectral energy transfer, Order of magnitude analysis.

4.2   Vorticity dynamics

Vorticity and Reynolds stress, Vorticity equations in turbulent flows, The dynamics of $\Omega_i \Omega_i$ and $\overline{\omega_i \omega_i}$.

4.3   The dynamics of scalar fluctuations

The dynamics for T^2 and $\overline{\Theta^2}$, Microscales in the scalar field.

5.   **Turbulence Closure**

5.1   Turbulence closure problem

The closure problem in the equations for turbulent flows, Hierarchy in turbulence modeling.

5.2   Turbulence constitutive relations

Constitutive relation, Constitutive relations for Reynolds-stress and scalar flux, Constitutive relations for higher order moments.

6. **Turbulence Modeling (Part I)**

6.1   Zeroth-equation eddy viscosity models

6.2   One and Two-equation eddy viscosity models

6.3   Reynolds-stress algebraic equation models

7. **Turbulence Modeling (Part II)**

NAME/INSTITUTION	S/F
Afolabi, Dare — Purdue U. - Indianapolis	S
Aggarwal, Suresh — U. of Illinois-Chicago	F
Ajmani, Kumud — Virginia Tech	F
Arnone, Andrea — University of Florence	F
Balsa, Thomas — University of Arizona	F
Beard, Lisa — University of Michigan	F
Brereton, Giles — University of Michigan	F
Brown, David — U. of Cincinnati	F
Chitsomboon, Tawit — Old Dominion U.	F
Chung, Joongkee — U. of California-Berkley	F
Cowley, Stephen — University of Cambridge	F
Davis, Dominic — University College-London	F
Deissler, Robert J. — Los Alamos Nat. Lab.	F
Demuren, Ayodeji — Old Dominion U.	F
Gajjar, Jitesh — Manchester University	F
Greenspan, Bernard — Cornell University	F
Gunzburger, Max — Virginia Tech.	F
Hagstrom, Thomas — U. of New Mexico	F
Hariharan, S.I. — University of Akron	F
Hayder, M.E. — Princeton University	F
Hou, Lin-Jun — Georgia Tech	F
Janus, J. Mark — Mississippi State U.	F

S - Structures
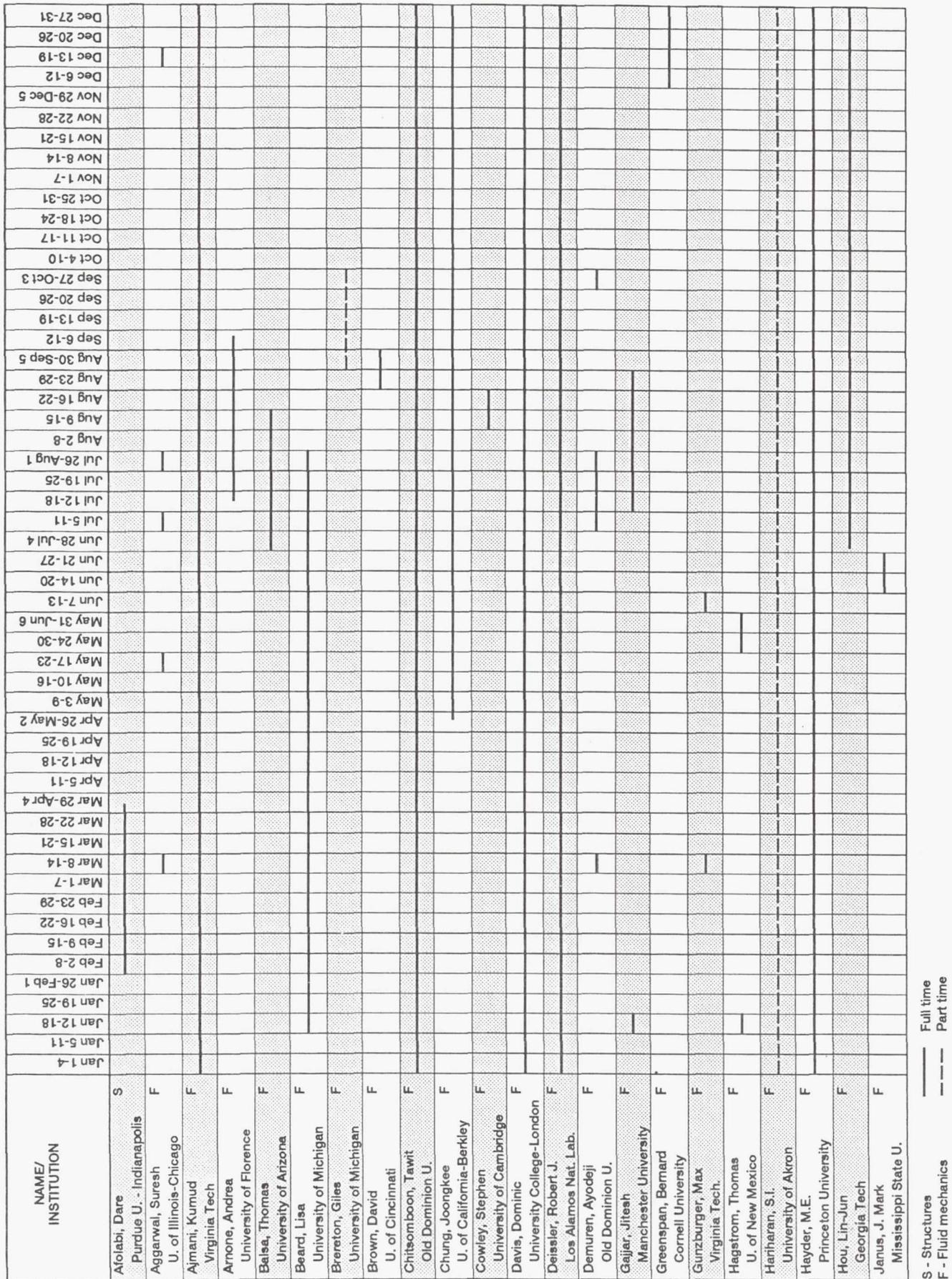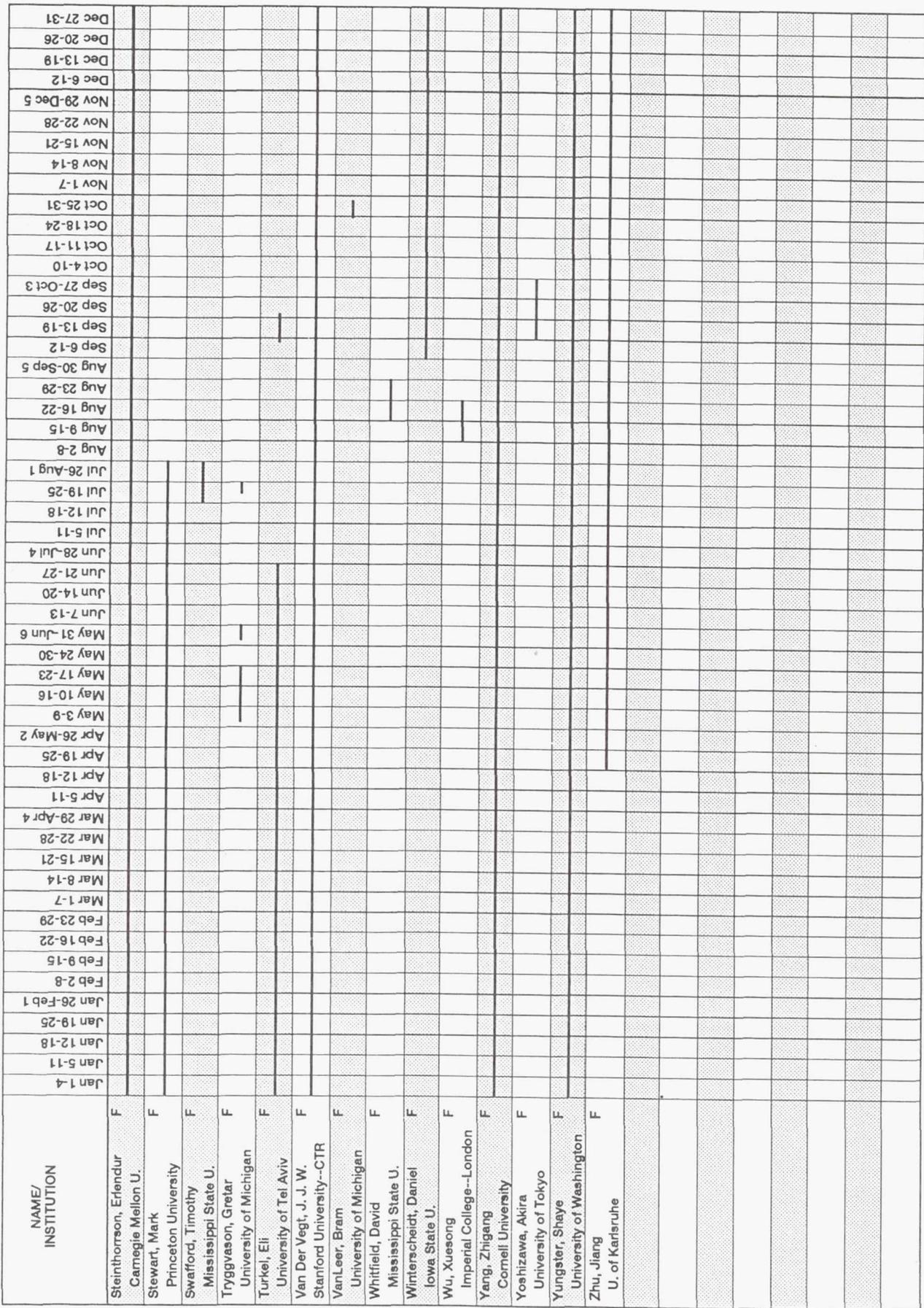F - Fluid mechanics

—— Full time
--- Part time

Figure 1.—ICOMP visiting researchers, 1992.

NAME/INSTITUTION	
Jiang, Bo-Nan / U. of Texas-Austin	F
Kao, Kai / U. of Colorado	F
Khosla, Preim / U. of Cincinnati	F
Kosmatka, John / U. of California-San Diego	S
Landahl, Marten / M.I.T.	F
Leonard, B.P. / University of Akron	F
Liou, William / Penn State University	F
Liu, J.T.C. / Brown University	F
Loellbach, James / University of Illinios	F
Maslowe, Sherwin / McGill University	F
Mathew, Joseph / M.I.T.	F
Messiter, Arthur / University of Michigan	F
Nicolaides, Roy / Carnegie-Mellon U.	F
Oron, Alexander / Technion	F
Pierre, Christophe / University of Michigan	S
Pletcher, Richard / Iowa State University	F
Powell, Kenneth / University of Michigan	F
Ramin, Thomas / Iowa State University	F
Rubin, Stanley / U. of Cincinnati	F
Shabbir, Aamir / SUNY-Buffalo	F
Shih, Tsan-Hsing / Stanford University-CTR	F
Sidi, Avram / Technion	F

Column headings (weeks): Jan 1-4, Jan 5-11, Jan 12-18, Jan 19-25, Jan 26-Feb 1, Feb 2-8, Feb 9-15, Feb 16-22, Feb 23-29, Mar 1-7, Mar 8-14, Mar 15-21, Mar 22-28, Mar 29-Apr 4, Apr 5-11, Apr 12-18, Apr 19-25, Apr 26-May 2, May 3-9, May 10-16, May 17-23, May 24-30, May 31-Jun 6, Jun 7-13, Jun 14-20, Jun 21-27, Jun 28-Jul 4, Jul 5-11, Jul 12-18, Jul 19-25, Jul 26-Aug 1, Aug 2-8, Aug 9-15, Aug 16-22, Aug 23-29, Aug 30-Sep 5, Sep 6-12, Sep 13-19, Sep 20-26, Sep 27-Oct 3, Oct 4-10, Oct 11-17, Oct 18-24, Oct 25-31, Nov 1-7, Nov 8-14, Nov 15-21, Nov 22-28, Nov 29-Dec 5, Dec 6-12, Dec 13-19, Dec 20-26, Dec 27-31

S - Structures
F - Fluid mechanics

——— Full time
– – – Part time

Figure 1.—Continued.

NAME/INSTITUTION

Name	Institution	Type
Steinthorsson, Erlendur	Carnegie Mellon U.	F
Stewart, Mark	Princeton University	F
Swafford, Timothy	Mississippi State U.	F
Tryggvason, Gretar	University of Michigan	F
Turkel, Eli	University of Tel Aviv	F
Van Der Vegt, J. J. W.	Stanford University--CTR	F
VanLeer, Bram	University of Michigan	F
Whitfield, David	Mississippi State U.	F
Winterscheidt, Daniel	Iowa State U.	F
Wu, Xuesong	Imperial College--London	F
Yang, Zhigang	Cornell University	F
Yoshizawa, Akira	University of Tokyo	F
Yungster, Shaye	University of Washington	F
Zhu, Jiang	U. of Karlsruhe	F

Time period columns (weeks): Jan 1-4, Jan 5-11, Jan 12-18, Jan 19-25, Jan 26-Feb 1, Feb 2-8, Feb 9-15, Feb 16-22, Feb 23-29, Mar 1-7, Mar 8-14, Mar 15-21, Mar 22-28, Mar 29-Apr 4, Apr 5-11, Apr 12-18, Apr 19-25, Apr 26-May 2, May 3-9, May 10-16, May 17-23, May 24-30, May 31-Jun 6, Jun 7-13, Jun 14-20, Jun 21-27, Jun 28-Jul 4, Jul 5-11, Jul 12-18, Jul 19-25, Jul 26-Aug 1, Aug 2-8, Aug 9-15, Aug 16-22, Aug 23-29, Aug 30-Sep 5, Sep 6-12, Sep 13-19, Sep 20-26, Sep 27-Oct 3, Oct 4-10, Oct 11-17, Oct 18-24, Oct 25-31, Nov 1-7, Nov 8-14, Nov 15-21, Nov 22-28, Nov 29-Dec 5, Dec 6-12, Dec 13-19, Dec 20-26, Dec 27-31

S - Structures   ——— Full time
F - Fluid mechanics   - - - Part time

Figure 1.—Concluded.

Figure 2.—ICOMP Steering Committee and visiting researchers in July, 1992. From left to right: Dominic Davis, Shaye Yungster, Jiang Zhu, Ayodeji Demuren, Charles Feiler, Ehtesham Hayder, Erlendur Steinthorsson, Thomas Ramin, Andrea Amone, Avram Sidi, Douglas Greenwald, Jaap Van Der Vegt, Marvin Goldstein, Aamir Shabbir, Kumud Ajmani, Louis Povinelli, Joongkee Chung, Michael Salkind, Lin-Jun Hou, William Liou, Glenda Gamble, Thomas Balsa, Bo-nan Jiang, Theodore Keith, Joseph Mathew, Karen Balog, Alexander Oron, Arthur Messiter, Tsan-Hsing Shih, Robert Deissler, and Jitesh Gajjar.

51

	UNIVERSITY OR INSTITUTION	NUMBER
1	AKRON	2
2	ARIZONA	1
3	BROWN	1
4	CALIFORNIA, BERKLEY	1
5	CALIFORNIA, SAN DIEGO	1
6	CAMBRIDGE	1
7	CARNEGIE-MELLON	2
8	CINCINNATI	3
9	COLORADO	1
10	CORNELL	2
11	FLORENCE	1
12	GEORGIA TECH	1
13	ILLINOIS	1
14	ILLINOIS, CHICAGO	1
15	IMPERIAL COLLEGE, LONDON	1
16	IOWA STATE	3
17	KARLSRUHE	1
18	LOS ALAMOS NAT. LAB	1
19	MANCHESTER	1
20	McGILL	1
21	M.I.T.	2
22	MICHIGAN	7
23	MISSISSIPPI STATE	3
24	NEW MEXICO	1
25	OLD DOMINION	2
26	PENN STATE	1
27	PRINCETON	2
28	PURDUE, INDIANAPOLIS	1
29	STANFORD (CTR)	2
30	SUNY, BUFFALO	1
31	TECHNION - Israel Institute of Tech.	2
32	TEL AVIV	1
33	TEXAS, AUSTIN	1
34	TOKYO	1
35	UNIVERSITY COLLEGE, LONDON	1
36	VIRGINIA TECH	2
37	WASHINGTON	1

58

Figure 3.—Composition of 1992 ICOMP staff. Organizations represented.

	1986	1987	1988	1989	1990	1991	1992
PEOPLE	23	43	50	46	47	49	58
SEMINARS	10	27	39	30	37	26	32
REPORTS	2	9	22	32	25	29	27
WORKSHOPS/LECTURE SERIES	1	0	2	1	1	1	1
PRESENATIONS	7	0	21	14	15	21	15
UNIVERSITIES OR ORGANIZATIONS	20	28	43	35	36	38	37

Figure 4.—ICOMP STATISTICS (1986 TO 1992)

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	3. REPORT TYPE AND DATES COVERED
	May 1993	Technical Memorandum

4. TITLE AND SUBTITLE

Institute for Computational Mechanics in Propulsion (ICOMP)
Seventh Annual Report — 1992

6. AUTHOR(S)

5. FUNDING NUMBERS

WU–505–90–5K

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–7839

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

National Aeronautics and Space Administration
Washington, D.C. 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA TM–106155

11. SUPPLEMENTARY NOTES

Report compiled and edited by Charles E. Feiler, ICOMP Executive Officer, and approved by Louis A. Povinelli, ICOMP Director, NASA Lewis Research Center (work funded under NASA Cooperative Agreement NCC3–233). ICOMP Program Director, Louis A. Povinelli, (216) 433–5818.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category 64

12b. DISTRIBUTION CODE

13. ABSTRACT (Maximum 200 words)

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in Cleveland, Ohio to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1992.

14. SUBJECT TERMS

Numerical analysis; Computer science; Mathematics; Fluid mechanics

15. NUMBER OF PAGES
54

16. PRICE CODE
A04

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