

NASA Technical Memorandum 102519
ICOMP-90-01

Institute for Computational Mechanics in Propulsion (ICOMP)

Fourth Annual Report—1989
produced by the ICOMP Steering Committee

(NASA-TM-102519) INSTITUTE FOR
COMPUTATIONAL MECHANICS IN PROPULSION
(ICOMP) FOURTH ANNUAL REVIEW, 1989
59 p

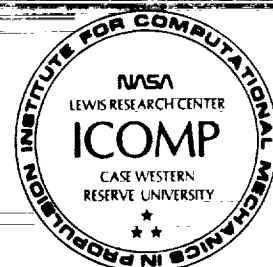
(NASA)
CSCL 12A

N90-20769

Unclas
63/64 0275332

March 1990

NASA





CONTENTS

	Page
INTRODUCTION	1
THE ICOMP STAFF OF VISITING RESEARCHERS	2
RESEARCH IN PROGRESS	3
REPORTS AND ABSTRACTS	27
SEMINARS	39
WORKSHOPS	51

INSTITUTE FOR COMPUTATIONAL MECHANICS

IN PROPULSION (ICOMP)

FOURTH ANNUAL REPORT

1989

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University 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 1989.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) is jointly operated at the NASA Lewis Research Center by Case Western Reserve University and NASA Lewis Research Center under a Space Act Agreement. 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 organization and operation of ICOMP have been described in ICOMP Report No. 87-8 (NASA TM-100225), "The Institute for Computational Mechanics in Propulsion (ICOMP), First Annual Report," Nov., 1987, 14 pages. The activities of 1987 and 1988 are described in ICOMP Reports No. 88-1 (NASA TM-100790), and No. 89-1 (NASA TM-101961) respectively.

The scope of the ICOMP research 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 flow; structural mechanics and dynamics; multivariable control theory and applications; and chemistry and material science.

The report summarizes the activities at ICOMP during 1989. It lists the visiting researchers, their affiliations and time of visit followed by reports of RESEARCH IN PROGRESS, REPORT AND ABSTRACTS, and SEMINARS presented. The results of a two-day workshop entitled "Lifetime Prediction of Flawed Structures," sponsored by ICOMP and held at the NASA Lewis Research Center in May, 1989, are also discussed.

THE ICOMP STAFF OF VISITING RESEARCHERS

The composition of the ICOMP staff during 1989 is shown in figure 1. Forty-six 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 June, 1989. Figure 3 lists the universities or other institutions represented and the number of people from each. The figure lists thirty-five organizations. Figure 4 shows the growth of ICOMP during its first four years in terms of staff size, organizations presented 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

Various vibration problems generally arise in propulsion systems: blade flutter, unbalanced rotor whirl, excessive forced response amplitudes, etc. At ICOMP, my work was directed at getting a fundamental understanding of the impact of mistuning on the forced response of blades in turbine engines.

It turns out that the effect of mistuning on forced response is extremely complicated and, thus far, very difficult to predict accurately. Cumulative research results by various workers have shown that a very small amount of mistuning has the potential to induce very large vibration amplitudes, which may then lead to catastrophic failure, under certain circumstances. Yet, under a slightly different excitation condition, the same amount of mistuning will not induce any catastrophe; in fact, mistuning may even be beneficial. Thus, the global effect of mistuning can be seen as a problem which is ill-conditioned and, thus, leads to different results under different perturbations.

The problem to be addressed is this:

Under what conditions will a small amount of mistuning lead to catastrophic blade failure; and, how can we know in advance whether such such catastrophes will arise or not?

In order to solve the problem, I am applying elements from a branch of mathematics formerly called "Catastrophe Theory," but now known as "Singularity Theory." Singularity theory, as propounded by Arnold, explains how small perturbations of degenerate systems can lead to various unfoldings at the bifurcation points on a manifold.

By applying the local bifurcation theory, I have established in an ICOMP Report that the tuned bladed disk is a degenerate system because of its circular profile. It is this cyclicity that places a mistuned system exactly on a bifurcation set. Thus, two perturbations which are qualitatively different, but quantitatively similar can lead to forced response curves with very different topologies. When a severe distortion of the forced response curve is induced by mistuning, then certain blades in an assembly will vibrate at much larger amplitudes than others. Such a situation could deteriorate in some rare cases to such an extent that some rogue blades could fail.

In addition to finishing the report, work is almost completed on another. I also presented a seminar: "Qualitative Characteristics of Mistuned Cyclic Systems," on June 29, 1989.

REFERENCES

- Arnold, V. I., et al., "Singularities of Differentiable Maps," Birkhauser, Boston, MA, 1989
 Arnold, V. I., "Lectures on Bifurcations in Versal Families," Russian Mathematical Surveys, Vol. 27, No. 5, pp. 54-123, 1972
 Arnold, V. I., "On Matrices Depending on Parameters," Russian Mathematical Surveys, Vol. 26, No. 2, pp. 29-44, 1968
 Afolabi, D., "Effects of Mistuning and Matrix Structure on the Topology of Frequency Response Curves," ICOMP Report: ICOMP 89-17, NASA TM-102290, July 1989
 Afolabi, D., "On the Geometric Stability of Certain Modes of Vibration," ICOMP Report (in preparation)

Suresh K. Aggarwal, University of Illinois at Chicago

While at NASA Lewis under the ICOMP program, I interacted with Drs. E. J. Mularz, D. L. Bulzan, and J. S. Shuen. The computations of turbulent evaporating sprays were completed. In this work, the detailed structure of an evaporating spray in a parabolic flow configuration is obtained. Favre-averaged gas-phase equations are used and a k-e-g- turbulence enclosure model is employed. For the liquid-phase equations, a stochastic separated flow model is used to represent the effects of gas-phase turbulence on droplet trajectory and vaporization, the variable-property effects are considered in detail, and three vaporization models are used to assess the importance of transient liquid-phase processes in turbulent spray computations. The results indicate that the vaporization behavior of turbulent sprays is generally quite sensitive to the vaporization models. The degree of sensitivity depends upon the boiling temperature and the heat of vaporization of liquid fuel considered. A paper on this work has been submitted to the Journal of Propulsion and Power and is currently under review.

The spray code developed in the above work has been extended to turbulent combusting sprays. Results have been obtained for a methanol combusting spray. I am writing a paper on these results, which will be submitted to Combustion and Flame.

F. A. Akl, Ohio University

The objective of this research on Vibration Analysis in Parallel Microcomputing Architecture was to study the computational characteristics associated with the mapping of the generalized eigensolution of linear elastic finite element models onto a parallel microcomputing architecture. Backward substitution has been added to previously implemented parallel processes dealing with the creation and assembly of the stiffness matrices of finite element models in binary architecture. Communication links to handle backward substitution processes have also been developed using the transputer development system in Occam. Testing and verification of the software on a transputer network have been successful on a limited number of processes. Further testing, debugging and verification of the work completed to date are in progress. The last phase of this project requires the implementation of an auxiliary eigensolver within each iterative cycle to extract the eigenpairs of the projected matrices. I will also attempt to integrate this algorithm with the graphics routines developed by G. K. Ellis, former ICOMP researcher.

Farid M. L. Amirouche, University of Illinois at Chicago

The objective of the visit was to interact with the structural dynamics branch at NASA Lewis and expose my techniques in the area of multibody dynamics and control. My primary motives were to identify technical areas of research for which I could provide some expertise.

During my week visit, I gave a seminar on "Constrained Flexible Multibody Dynamics" where I emphasized both the computational aspects of the problem as well as its application to the deployment of satellites, space robotics, and mechanisms. Dr. Chuck Laurence who was my primary technical contact introduced me to some ongoing projects related to my research. During the course of our discussions we identified three areas where we anticipate some collaboration in the future. The first project involves adding the control capabilities to the existing general purpose program "DYUMAS" for the analysis of complex dynamical

systems. A second project which we believe will benefit other interest groups within NASA will be to study the time varying modes in structures with elastic bodies to enable us to investigate the effects of space robots moving on flexible tracks. The last project is one in which we are currently investigating an alternative solution to zero base reaction in redundant space robotic manipulators operating under the microgravity effects.

Andrea Arnone, (Postdoctoral), University of Florence, Italy

During my six-months stay at ICOMP I have continued my work in the field of transonic cascade flow calculations. The final goal of this activity is the development of a fast, robust and accurate solver for viscous/inviscid flows to be used in turbomachinery design. My activity consists of three parts: (a) generation of grids capable of picking up details of the flow field; (b) computation of two-dimensional flows; (c) computation of three-dimensional flows.

Rotor and stator cascades of modern turbomachinery are often characterized by a high turning geometry and/or by strong flow deviations from the axial so that the generation of meshes capable of picking up the flow details is not so straightforward. Procedures that use periodicity generally give rise to highly distorted meshes. In the present work, this problem is solved by the introduction of non-periodic elliptic C-type grids. The removal of periodicity allows the grid to be only slightly distorted even for cascades having a large camber and a high stagger angle. On these new kinds of grids, the Euler and Navier-Stokes equations are efficiently solved using a Runge-Kutta scheme in conjunction with accelerating techniques like multi-gridding and variable coefficient implicit residual smoothing. The two-layer eddy-viscosity model of Baldwin and Lomax is used for the turbulence closure.

The above procedure has been applied first to two-dimensional flows and later extended to the three-dimensional case. The predicted flows agree well with experiments and both robustness and computational time are competitive with respect to other existing codes.

Richard J. Bodonyi, Ohio State University

During my three week visit at ICOMP, a research effort was begun to study the boundary-layer receptivity of a two-dimensional flow due to uniform suction through a slotted wall beneath an incompressible boundary-layer flow using the triple-deck theory formulation of the receptivity problem. This approach was originally developed by Goldstein [Journal of Fluid Mechanics V. 154, 1985] for linearized mean flows and extended by Bodonyi, Welch, Duck and Tadjfar [Journal of Fluid Mechanics V. 209, 1989] to include the effects of nonlinear mean flows. In both cases the disturbance flow is governed by the unsteady linearized triple-deck equations.

An existing code for the nonlinear steady triple-deck equations was modified to solve for the slotted wall problem and solutions for several values of the wall suction parameter were found. Also, modifications of a computer code for the receptivity problem were begun during this visit and numerical solutions were initiated for different values of the Strouhal number and wall suction parameter. Computations have been continued and a paper is currently being prepared to report the results of the study.

Concurrently, a study, in conjunction with Drs. M. E. Goldstein and J. W. Elliott, was initiated to study the receptivity characteristics of the incompressible breakaway separation solution of Smith [Proc. Roy. Soc. Lond. V.

A356, 1977]. This effort is still underway and results will be reported at an appropriate time.

Shih-Hung (Fred) Chang, Cleveland State University

We continue the study and development of ENO-type numerical schemes for shock capturing. Our goal is to extend the basic ENO algorithms to multidimensional problems. Last summer, we reported our progress on the extension to two-dimensional problems (NASA TM-101355, 1988). Although the ENO scheme is robust and quite dependable, there is room for improvement. One of these concerns is the need to improve the resolution of contact discontinuities. Harten (J. Comput. Phys. 1989) proposed the use of subcell resolution to obtain a modification of the basic ENO scheme, denoted ENO/SR, and showed significant improvement in the resolution of contact discontinuities.

In the meantime, another interesting development emerged. In the study of the behavior of numerical methods on reacting flow problems, LeVeque and Yee (NASA TM-100075, 1988) proposed the use of a certain model problem consisting of a one-dimensional scalar conservation law with stiff source terms. It was discovered that for the very stiff case most of the current finite difference methods, including TVD schemes, developed for non-reacting flows, would produce wrong solutions when there is a propagating discontinuity. This poses an important question on extending numerical methods developed for non-reacting flows to reacting flows.

We tested the ENO and ENO/SR schemes on the model problem of LeVeque and Yee. The numerical results are in general better than those published in LeVeque and Yee, but both schemes become unstable for the very stiff case. We then adopted the idea of subcell resolution but modified the ENO/SR scheme. Instead of using the ENO flux to accomplish the time evolution, we do it by advancing along the characteristics. The resulting scheme is denoted ENO/SRCD. Numerical experiment using this scheme shows excellent results on the model problem of LeVeque and Yee. The results and comparisons with ENO and ENO/SR schemes will appear in/a ICOMP Report (NASA Technical Memorandum) entitled "On the Application of Subcell Resolution to Conservation Laws with Stiff Source Terms."

This scheme ENO/SRCD is not our basis for further study and development because it can improve the resolution of contact discontinuities for non-reacting flow problems and also produce excellent solutions to the model problem of LeVeque and Yee for reacting flows. Extension to multidimensional problems based on ENO-SRCD should be more promising than those based on ENO. However, more study and extensive numerical experiment are necessary to explore the capability of this basic scheme.

C. J. Chapman, University of Cambridge, England

A theoretical investigation was made of diffraction patterns in the acoustic field of a supersonically rotating propeller. The main result is that the near field contains regions in which the pressure amplitude varies in a complicated way from point to point; nevertheless, this behavior can be accurately described by analytical formulae containing a small number of terms, typically two or three. The technique used was asymptotic approximation of oscillatory double integrals by the method of stationary phase. Detailed calculations were performed for a propeller of simple shape, and a diffraction pattern of remarkable structure was revealed.

It would appear that this is the first time in the study of propeller acoustics that a diffraction field has been obtained. Yet in other parts of acoustics, and even more so in optics, diffraction theory has been extensively investigated. The method, therefore, holds great promise for future application to propellers, especially as detailed accounts of the technique may be found in the research literature. Since the final formulae may be evaluated numerically, using negligible computer time, the method lends itself to incorporation in computer codes, and may well prove a useful addition to current work on the acoustics of propfans and contrarotating propellers.

Seo Won Choi, (Postdoctoral), Stanford University

The activity in 1989 has been mainly carried out with Dr. M. E. Goldstein. We continued our previous study of interacting oblique waves at nonlinear evolution stage, and have extended it to compressible shear layer and boundary layers.

A pair of spatially-growing oblique waves with the same frequency and streamwise wavenumber but equal and opposite spanwise wavenumbers are studied where they go through nonlinear growth just upstream of the linear neutral point with nonlinear non-equilibrium critical layer analysis. The analysis is a rational extension of upstream linear solution, i.e., it can be matched on to the linear solution in the sense of matched asymptotic expansion. The interaction between the two oblique waves causes the critical layer nonlinearity to take place at a smaller amplitude than either single oblique wave or two-dimensional wave analyses, i.e., this analysis applies to further upstream locations. The resulting amplitude growth is determined by an integro-differential equation with cubic-type nonlinearity. The integral is of a convolution type indicating a history effect. In other words, the nonlinear contribution to the amplitude growth is nonlocal. The numerical solutions to this equation always end in a singularity at a finite downstream distance. This suggests a rapid growth of the amplitude there, and may be due to vortex-stretching caused by the fully three-dimensional disturbances used in our analysis.

According to Gropengiesser (1970), and Jackson and Grosch (1988), oblique modes exhibit the most rapid linear growth in shear layers at sufficiently high Mach numbers. Therefore this interaction will be prominent in supersonic shear layer experiments.

REFERENCES

Goldstein, M. E. and Choi, S. W., J. Fluid Mech., October (1989).

Stephen Cowley, Imperial College, England

During my visit to ICOMP this year, I started some joint work with Dr. Marvin E. Goldstein on aspects of nonlinear, non-equilibrium, viscous critical layers in hypersonic boundary layers when the viscosity is governed by Sutherland's law. The aim is to extend previous work which assumed a Chapman's viscosity law for the basic flow, and assumed that the critical layers were inviscid. I also continued writing up a paper on some periodic boundary layer problems. This work is relevant to the sub boundary layers which develop under very large Tollmien-Schlichting waves.

A. O. Demuren, University of Lagos, Nigeria

Current research work focuses on developing second-moment turbulence closure models for compressible flows. The most promising ones appear to be the full Reynolds Stress Models (RSM) based on the numerical solution of partial differential equations for the turbulent (Reynolds) stresses and associated turbulence fluxes, and a representative equation for the length scale. In the present work, the incompressible RSM, proposed by Launder, Reece and Rodi (Journal of Fluid Mechanics, 1975) is first extended for compressible flows up to Mach 5. We then address various modeling issues which go beyond that work, such as ensuring realizability, and the proper asymptotic behavior in the two-component limit as the wall is approached.

Two versions of the final model are devised. One uses a wall function method which approximates the behavior of the flow in the near-wall region with analytical expressions, and the other integrates the differential equations through the sublayer right down to the wall. The former is computationally less expensive since many grid points are saved by not having to resolve the near-wall flow in which many properties have high gradients. However, there is uncertainty as to the generality of the wall functions and the sensitivity of the computed results to the location at which they are applied. The second approach does not have these problems, but requires considerably more grid points in resolving the near wall flow, and tends to produce a stiff system of algebraic equations which is more difficult to solve. In cooperation with the Computational Fluid Dynamics Branch these models will be incorporated in existing LU and ADI codes for both two- and three-dimensional flows. The performance of both versions will be tested against experimental data and computations with zero-equation (algebraic) and two-equation ($k - \epsilon$) turbulence models.

Peter Duck, University of Manchester, England

The bulk of my work carried out at ICOMP was involved with the problem of unsteady three-dimensional marginal separation. In this problem, we study a boundary layer which is just on the verge of separating at a point (situated along a line of symmetry). Around this point we allow a small amount of "interaction" to occur, and introduce an unsteady disturbance to the flow. The problem reduces to the solution of a non-linear partial integro-differential system (in two spatial variables, together with time). In order to solve this problem, a combination of a pseudo-spectral scheme (applied spatially) together with a fully implicit Crank-Nicolson scheme (applied temporally) has been used. A number of the computations performed very strongly suggesting that the solution can break down at a finite time, at a finite spatial location. This breakdown has been analyzed, and an asymptotic description for the associated three-dimensional singularity has been obtained. This problem is believed to be of direct relevance to the problem of dynamic stall, such as may occur near the leading edge of an airfoil.

Additional work has been carried out with Dr. John Goodrich, involving the flow inside a cavity. A number of recent computations by Dr. Goodrich (involving flow inside a "driven cavity") have indicated that under certain regimes, colliding wall jets can give rise to intermediate regions of recirculatory flow. In an attempt to study these regions more closely, we have been computing cavity flows where fluid is injected through the vertical walls (partly in opposing directions), thereby enforcing a collision of fluid on an "order-one" length-scale.

Further work, in collaboration with Dr. John Elliott has been underway on the reversing flow over the trailing edge of a flat plate airfoil. This work is described in Dr. Elliott's report.

Peter J. Eiseman, Columbia University

The consulting activities at ICOMP comprised developmental work on guiding the sequence of TURBO codes, the first part of a lecture sequence on adaptive grid generation, and the initiation of research for generating pointwise distributions on curves. The TURBO codes represent a continuing effort at NASA Lewis under the direction of Dr. Yung K. Choo. Each code is based upon the control point form (CPF) of algebraic grid generation that was developed by the consultant. Moreover, several technical papers co-authored with Yung K. Choo and others have resulted. The first of a sequence of lectures on adaptive grid generation was successfully given and covered many of the basic elements in the somewhat uncluttered mathematical framework of one dimension. The next lectures will fill out the one-dimensional story, extend those results into multiple dimensions in successive stages and deal with the fundamental techniques for coupling the grid to a PDE solver. The research on pointwise distributions for curves represents a continuation of much earlier work with the hope of gaining a number of advantages by establishing very valuable and useful algorithm, that is being done with Aaron Snyder.

John W. Elliott, University of Hull, England

This report covers the period July-August 1989 in which the author was a visitor to ICOMP. Essentially two major investigations were initiated and these are discussed below.

Firstly, a high-Reynolds number, $Re \gg 1$, investigation into the receptivity of separating flow (past a smooth body) due to an imposed harmonic disturbance was undertaken. The frequency regime investigated was for a Strouhal number, St , given by $St = \Omega Re^{1/4}$, where Ω is an $O(1)$ parameter. Here the spatial and temporal length-scales of the innermost region for the linear, unsteady, inviscid investigation of Goldstein (1984) are those for the "triple deck" structure. In order to investigate this regime two computer codes were written. The first solved for the mean flow. Essentially the solutions of Smith (1977) were repeated although here a finer grid was used for greater accuracy and Veldman's interactive technique was employed. The second code numerically solved for the time-harmonic linearized problem. Here the time derivative $\partial/\partial t$ is replaced by $-i\Omega$ and the resulting problem is solved using the ideas of Duck and Bodonyi (1988) to interactively couple the numerical solutions to both the lower-deck and upper-deck problems. These calculations are still in progress. Further a start was made to investigate further aspects to this problem. These include an alternative solver for the initial-value problem required both as a check on the current solutions and to anticipate problems regarding causality since early indications are that there are waves present in the numerical solutions far upstream. Also a start has been made on an "Orr-Sommerfeld" solver for short-scale disturbances to the highly non-parallel mean-flow following the ideas of Smith and Bodonyi. Here it is hoped that we can decide whether such disturbances are either absolutely or convectively unstable. Finally it is hoped to start a study of the nonlinear problem by first considering the "hump" problem

Secondly, an investigation into the quasi-steady flow near the trailing-edge of a finite flat plate due to a decelerating mainstream at the time of flow-reversal was continued. Earlier numerical solutions of the classical boundary-layer equations indicated no appearance of the Van Dommelen singularity and these were further considered. However a study of the near-wake at the time where the flow first reverses indicated an inconsistency between numerical and analytic results for the downstream behavior unless there was a second solution to the Goldstein-continuation problem for a separating profile. This second solution was confirmed using a shooting routine. This in turn has led to a novel "triple-deck" problem for the flow at the trailing edge. Here the problem is again quasi-steady with time entering only parametrically. Numerical solutions for a wide range of values of the scaled time indicate that the structure may be capable of passing through the critical time of flow-reversal. Further analysis of the ultimate behavior of this state is required. The above investigations have been undertaken in conjunction with M. E. Goldstein, R. J. Bodonyi, and P. W. Duck.

Nessan Fitzmaurice, Case Western Reserve University

The problem of homogeneous turbulent shear flow in a rotating frame constitutes a nontrivial test of turbulence models since it incorporates arbitrary combinations of shear and rotation which can have either a stabilizing or destabilizing effect. Since the flow is statistically homogeneous, two-equation turbulence models and second-order closure models give rise to an initial value problem for a coupled set of nonlinear ordinary differential equations. This allows for a fairly detailed mathematical analysis of the models based on bifurcation theory and for the generation of highly accurate numerical solutions. Combining these features with the wealth of published results from physical and numerical experiments gives us a powerful tool for testing and screening turbulence models. In collaboration with Charles Speziale and Tom Gatski, I have used this tool on a variety of two equation and second order closure models. These range from the standard K- ϵ model to the more recent second order closures of Shih-Lumley and Fu, Launder and Tselepidakis. This work is detailed in a number of reports that appeared or were submitted this year: ICASE Report 89-43; Phys. Fluids A, 1, (Feb. 89); Seventh Symposium on Turbulent Shear Flows, 27.3, (Aug. 89); and Jour. Fluid Mech., 209, (Dec. 89).

The next phase of this project will be the development of significantly improved models based on invariance arguments and a dynamical systems approach. The work will benefit greatly from the network facilities that should be available at Case early in 1990. It should also benefit from the renewed interest in turbulence modeling that is apparent in the Lewis Research Center.

Work also continues on the Kolmogorov flow project outlined in last years report. This is something of a test-bed problem that seeks to characterize a turbulent flow by means of its so called proper orthogonal decomposition. Significant numerical simulations have been carried out and the results of the initial data analysis looks promising. A couple of reports have been prepared and were submitted to Phys. Fluids.

Jitesh Gajjar, Exeter University, England

The work that I was engaged in during the time I spent at ICOMP in 1989 was to finish off some calculations and analysis of nonlinear neutral modes in

compressible boundary layer flows. This work is a generalization of that of Benney and Bergeron (1969), and Bodonyi, Smith and Gajjar (1983), to compressible flow, and concerning strongly nonlinear critical layers. The critical layer is strongly nonlinear in the sense that the disturbance amplitudes are much larger and the viscous effects occur at much higher order. The difficulties of analyzing the structure of the critical layer are compounded also when the additional effects of the basic density and temperature variations are taken into consideration. Nevertheless the analysis can be carried through as for the incompressible case, and the eigenvalue problem for the neutral modes is reduced to solving the compressible Rayleigh equation with suitable jump conditions across the critical layer, coupled with a nonlinear amplitude equation determining the nonlinear neutral modes. A special case of the work is when the generalized inflexion point criterion holds. Neutral modes for a range of phase speeds and Mach numbers were computed and their limiting cases analyzed. As in the linear case there are considerable differences depending on whether the flow is relatively subsonic or supersonic.

This work is described in a paper which is to appear as part of the proceedings of the "Stability and Transition Workshop" ICASE/Langley 1989, and in much more detail in a paper by Gajjar and Cole (1990) which is in preparation.

Karl Gustafson, University of Colorado

Recently we investigated in depth the computational vortex dynamics of cavity flows. These flows are rich in the fundamental features of internal fluid dynamics. In particular, the cavity geometry is amenable to computational grids and the testing of new numerical schemes, while at the same time permitting a study of basic internal fluid flow features such as behavior in corners, behavior near driving force singularities, internal recirculation dynamics, the onset of internal vortex sheddings at sufficiently high critical Reynolds numbers.

Of particular interest are a better understanding of the bifurcation diagrams for such flows. At present I have begun a collaboration with Dr. John Goodrich, NASA Lewis, to study when sustained periodic oscillations develop in impulsively started internal flows. These are self-oscillations, not due to special frequencies in any external driving forces, i.e., they are Hopf bifurcations, inherent in the vessel geometry and the basic flow. The demonstration numerically beyond a doubt of the periodicity is a tricky and interesting problem, and one basic to a better understanding of the onset of fluid oscillations in internal flow geometries.

K. Gustafson and K. Halasi, J. Comp. Physics 64 (1986), 279-319
 K. Gustafson and K. Halasi, J. Comp. Physics 70 (1987), 271-283
 J. Goodrich, K. Gustafson, K. Halasi, in preparation.

Thomas Hagstrom, SUNY at Stony Brook

The major portion of my research effort while resident at ICOMP was devoted to the problem of designing stable, accurate boundary conditions for time accurate simulations of fluid flows on unbounded domains. Among the particular cases considered were the Euler equations of inviscid compressible fluids as well as the incompressible Navier-Stokes equations.

The work on the Euler equations, done in collaboration with S. I. Hariharan of ICOMP, was focussed on exterior domains with uniform subsonic flows in the far field. The conditions we have developed involve the specification of vorticity and entropy at inflow as well as a radiation condition for the pressure which is imposed at all points of the artificial boundary. The latter is constructed from a linearized asymptotic analysis of propagating acoustic waves. We carried out a series of numerical experiments to test the utility of these conditions. These experiments involved the simulation of transonic flow past an impulsively started cylinder. The capability to produce accurate results on domains of modest size was clearly demonstrated. Our plans for further development of this procedure include:

- Inclusion of nonlinear effects in our asymptotic analysis of the acoustic radiation.
- Inclusion of viscous effects and an undertaking of numerical simulation of viscous, compressible flow past a cylinder.
- Applications to other problems in unsteady fluid dynamics such as the analysis of oscillating cascade sections.

The work on the Navier-Stokes equations is a continuation of research carried out during previous visits to ICOMP. Among the accomplishments of this time period were a better understanding and utilization of the scalings employed in our asymptotic analysis and generalizations of our procedure to Reynolds averaged equations. A series of numerical experiments has been planned in collaboration with John Goodrich of the Computational Fluid Dynamics Branch. Under consideration is the generalization of these results to compressible flows.

A third problem which was considered involves the nonlinear stability of supersonic shear layer and long wavelength acoustic waves. In this I have greatly benefited from the concentration of experts in hydrodynamic stability at ICOMP and Lewis.

C. A. Hall and T. A. Porsching, University of Pittsburgh

The computer code ALGAE (ALgorithms for the GAs Equations) was developed at the Institute for Computational Mathematics and Applications, University of Pittsburgh during the past 13 years. ALGAE treats incompressible, thermally expandable or locally compressible flows in complicated two-dimensional flow regions, and the technology developed in ALGAE regarding the solution method, finite differencing schemes and basic modelling of the thermal and fluid flow equations is applicable to engineering design settings of the type found in Stirling cycle engines.

During this summer of 1989, a test was made to determine if the code could be used to model multiple components of the SPRE Stirling engine. At the same time certain features were identified that would allow for more realistic two-dimensional simulations of these components. A video tape was generated containing a color simulation of the transient behavior of the working gas (helium) in the heater-regenerator-cooler complex of the SPRE. This video tape and supporting hard copy demonstrated that ALGAE did in fact model the type of geometries involved and provided information on two-dimensional phenomena in multiple component sections of the SPRE.

The investigation of the two-dimensional modelling of the thermal and hydraulic behavior of Stirling engines has precipitated several basic research problems. These include issues concerning: (1) The appropriate modelling of engine components that are of radically differeng geometric scales, (2) The

correct boundary conditions for two dimensional models, (3) The determination of mesh configurations that will adequately resolve flow detail in complex geometries, (4) The need for turbulence in some components of the SPRE, (5) The desirability and feasibility of three-dimensional simulations, (6) How color graphics can best be used in analyzing the results of computer simulation; and, (7) The development of new solver technologies to economize on the cost of computer simulation of such thermal-fluid calculations.

After appropriate code modifications are made, it is expected that ALGAE will be a viable tool for the two dimensional simulation of Stirling engine thermal and hydraulic operating conditions.

Awatef Hamed, University of Cincinnati

The focus of this work was to explore the numerical simulation of the probabilistic aerothermodynamic response of propulsion system components to randomness in their environment. The reusable rocket engine turbopumps were selected as an example because of the severe cryogenic environment in which they operate. The thermal and combustion instabilities, coupled with the engine thrust requirements from start up to shut down, lead to randomness in the flow variables and uncertainties in the aerodynamic loading.

The probabilistic modeling of the turbopumps aerodynamic response was accomplished using the panel method coupled with Fast Probability Integration methods. The aerodynamic response in the form of probabilistic rotor blades and splitter loading were predicted and the results presented for specified flow coefficient and rotor preswirl variance. Possible future applications of the aerothermodynamic probabilistic modeling in engine transient simulation, condition monitoring and engine life prediction are briefly discussed.

S. I. Hariharan, University of Akron

A variety of important problems in computational gas dynamics, in particular in aerodynamics and acoustics, are posed in exterior domains. A natural approach to their numerical solution is to introduce an artificial boundary at which appropriate boundary conditions must be imposed. For certain problems the accuracy of these conditions is crucial as errors at the boundary can propagate into the interior. In our work we use the conditions based on asymptotic expansions of the solution in the far field. The accuracy of these may, in some instances, be explicitly determined, an advantage in comparison with other techniques.

Our derivation is based on the use of "wave front" coordinates related to the geometrical acoustics approximation. An infinite series has been obtained recursively from the radiation field. Using this representation we have derived a family of boundary conditions for problems in unbounded regions governed by the Euler equations. A first order boundary condition obtained through our approach coincides with the boundary condition (untested) of Bayliss and Turkel. We tested the effectiveness of our boundary conditions when applied to realistic flow problems in conjunction with the work described in the next two sections.

Asymptotic Artificial Boundary Conditions for Computational Gas Dynamics (with T. M. Hagstrom, SUNY at Stony Brook. We consider the physically more interesting case of a uniform flow at infinity. We approached by studying the linearization of the two-dimensional Euler equations about a uniform, subsonic

state. For the purpose of computing an asymptotic expansion, we found that entropy, pressure, vorticity and divergence are the most convenient variables. This is consistent with the observations of Gustafsson and Roe. The key step is the derivation of an expansion for the pressure, which satisfies a convective wave equation. Numerical experiments are proposed to demonstrate the accuracy of this approach.

Unsteady Aerodynamics (with J. R. Scott). Calculation of unsteady vortical disturbances around a flat plate which is modeled after an airfoil is considered. The resulting problem is governed by a hyperbolic equation in an open domain, namely the convective wave equation. For computational purposes the open domain must be truncated by finite artificial boundaries. Approximate boundary conditions derived based on our asymptotic theory directly apply here. A numerical solution procedure for the time domain calculation has been derived. Preliminary comparisons made with solutions of a corresponding frequency domain problem indicate good agreement. Computational efficiency through the time domain treatment was demonstrated; however, it was at the expense of loss of accuracy in the low frequency content due to the input of time dependent signals. This work is jointly being done with Yu Ping, Department of Mathematical Sciences, University of Akron.

REFERENCES

- A. Bayliss and E. Turkel, "Far Field Boundary Conditions for Compressible Flows," J. Comp. Phys., Vol. 48, (1982), 182-199.
- B. Gustafsson, "Far Field Boundary Conditions for Time-dependent Hyperbolic Systems," SIAM J. Sci. Stat. Comp., Vol. 9, (1988), 812-828.
- P. Roe, "Remote Boundary Conditions for Unsteady Aerodynamic Computations," ICASE Report No. 86-75, (1986).
- J. R. Scott and H. M. Atassi, "Numerical Solution of Periodic Vortical Flows About a Thin Airfoil," (1989), AIAA Paper 89-1691.

Mark Janus, Mississippi State University

In addition to consulting with several of the NASA Lewis staff who are using a turbomachine flow solver developed at Mississippi State University for NASA, I spent my two weeks developing software to enhance the flow solver. The software developed includes three routines, two which adequately locate control points for defining Bezier cubic curves to connect computational domains in relative motion, and one routine which defines each curve and returns the azimuthal position of an arbitrary point on the curve.

The latter routine defines a Bezier cubic curve based on two end points (one in each of the two computational domains to be connected) and two control points which are input arguments. Thus when given an axial position along the curve the corresponding parameterization value can be iterated for (using a Newton-Raphson scheme), then that value is used to compute the corresponding azimuthal position of the point. Note the axial and radial positions of points within the connection region remain unchanged from their original values (i.e., a body of revolution is the assumed hub geometry).

Presently, two routines have been developed to locate control points for each grid line connecting the computational domains. One method treats each grid line independently providing safeguards to ensure nonintersecting grid lines. The other method links each grid line's curve definition via the use of

a common incoming slope and a common outgoing slope. The latter method does not provide safeguards for the possibility of intersecting grid lines.

Aesthetically the latter method provides a smoother connecting grid at all time levels for computational domains moving relative to one another. Although this is the case, there is no guarantee the grid lines will not cross one another (i.e., twist). The former location scheme does provide this kind of protection yet tends to resort back to a linear connection scheme under certain circumstances. Numerical studies need to be performed to assess the optimum scheme for locating control points, in addition to evaluating the computational expense of providing an enhanced domain connection procedure.

Bo-Nan Jlang, (Postdoctoral), University of Texas, Austin

A least-squares finite element method based on the velocity-pressure-vorticity formulation was developed for solving steady incompressible Navier-Stokes problems. This method leads to a minimization problem rather than to a saddle-point problem by using the classic mixed method, and can thus accommodate equal-order interpolations. This method has no parameter to tune. The associated algebraic system is symmetric and positive definite. Numerical experiments for cavity flow at Reynolds number up to 10 000 and backward-facing step flow at Reynolds number up to 900 were conducted. This method is being extended to solving three-dimensional problems.

S.-W. Kim, (Postdoctoral), University of Texas, Arlington

Incorporation of a $k-\epsilon$ and the multiple-time-scale turbulence models into the RPLUS code (a compressible flow solver based on the lower-upper approximate factorization method) is in progress.

In the effort, new versions of the RPLUS code were made to facilitate the incorporation of the advanced turbulence models. The new versions are different than the original code in its grid layout and the way the near-wall boundary conditions are handled. Otherwise, the new versions are the same as the RPLUS code in their capability to solve chemically reacting flows as well as non-reacting compressible turbulent flows. Verification of the new codes, theoretical formulation of flow equations including the advanced turbulence models, and implementation of the flow equations into the LU factorization method is in progress.

Lala Krishna, University of Akron

Parallel iterative linear solvers were developed. This included parallelized preconditioned conjugate gradient (PCG), Gauss-Seidel and Successive Overrelaxation (SOR) methodologies. Large scale benchmark tests were performed on the ALLIANT FX-80 to quantify the numerical schemes. Additionally, significant analytic effort was given to justify the formal robustness of the various methodologies. The numerical benchmarks were devised so as to establish the impact of processor architectures on the efficiency of the parallelism. Further studies are ongoing with particular emphasis given to the PCG.

Sang Soo Lee, (Postdoctoral), Brown University

My ICOMP activity, described as follows, has been mainly in collaboration with Dr. M. E. Goldstein. The effect of a nonlinear nonequilibrium critical

layer on the resonant triad interaction, suggested by Raetz (1959), Craik (1971), and others is analyzed for an adverse pressure gradient boundary layer. Our results show that the critical layer causes this interaction to take place at much smaller amplitudes than could have been anticipated from Craik's original analysis and that the oblique instability waves exhibit faster than exponential growth and that the growth rate of the two-dimensional mode is independent of the oblique mode when the modal amplitudes are all of the same order. The growth rates become fully coupled at very large oblique mode amplitudes but the plane wave coupling terms are quartic rather quadratic as in the Craik (1971) analysis. The overall growth of the oblique modes is significantly enhanced by this weaker coupling.

B. P. Leonard, University of Akron

Work proceeded in two areas of high-convection modeling: (1) the ULTRA-SHARP project for steady-state flows and (2) the VANGUARD project for time-accurate unsteady simulation. ULTRA-SHARP is based on a universal limiter for tight resolution and accuracy making use of a simple high-accuracy resolution program based on local adaptive stencil expansion. For multidimensional flow, the universal limiter guarantees nonoscillatory resolution of discontinuities. The basic algorithm uses third-order upwinding (QUICK) in "smooth" regions (identified by monitoring local changes in gradient); then, in regions requiring higher-order resolution (but only in such regions), the algorithm automatically branches to a locally higher-order scheme. This is a very cost-effective strategy, since the larger stencil is required only in isolated narrow regions in the vicinity of the discontinuities. The VANGUARD strategy is based on a vectorial approach to nonoscillatory generalization of upwinding for accurate resolution of discontinuities for time-accurate transient simulation. The overall philosophy is similar to the steady-state case, using a vectorial third-order time-accurate scheme (QUICKEST) in smooth regions and local adaptive stencil expansion in high-curvature regions. In this case, the location of the wider-stencil computation automatically moves with the local region requiring high resolution. The algorithm is fundamentally multidimensional in that it is based on vector transient interpolation modelling: $\phi(\underline{x}, \Delta t) = \phi(\underline{x} - \underline{v}\Delta t, 0)$. This technique automatically generates important "cross-difference" terms in the flux-based control-volume formulation, thereby avoiding the anisotropic distortion associated with schemes which rely on component-wise one-dimensional fluxes. Initial results of this work were presented at the Sixth International Conference on Numerical Methods in Thermal Problems in Swansea, Wales, and at CTAC-89, the Computational Techniques and Applications Conference in Brisbane, Australia. An expanded version of the Swansea paper will appear in a special issue of the International Journal for Numerical Methods in Fluids devoted to selected papers from the conference. A similar arrangement has been made with Computers and Fluids for selected expanded papers from the CTAC-89 meeting.

Avi Lin, Temple University

This report summarizes briefly the first three stages in our long term activity in parallel computations of turbulence. We have studied the finest grain of parallelism that is expected in this development, namely, the inverse of matrices of moderate size (between 3 and 18). The final algorithm for these cases is up and running in the hypercluster environment over three processors.

The parallel solution of a block-tridiagonal linear system is expected to be in the second level of parallelism of the final code. The parallel algorithm, which is based on the domain decomposition operator decomposition approach is running now on the same system using three nodes. A test of six coupled boundary value problems has shown excellent results in this setup. We are now in a process of solving six coupled two-dimensional elliptic equations, and will convert them later on to the Navier-Stokes equations.

The scalar (serial) version of this approach has been tested and the results are good. We are now in a process of formulating the Reynolds stress equations in an appropriate form so that the above two kernels of parallelism will be easily applied.

Reda R. Mankbadi, University of Cairo

My research is concerned with excited shear flows and is summarized as follows.

Two-wave Interactions. The conditions for resonance interaction between two instability waves in an axisymmetric jet were investigated. Analysis of the energy exchanges between the two-waves indicate that the phase angle between the wave-induced stresses and wave-induced strains plays a crucial role in the resonance interactions. The subharmonic resonance was found to be highly dependent on this angle. Favorable agreement was found between the phase angles prescribed by the theory and those measured.

Multiwave Interaction in Turbulent Jets. Nonlinear wave-wave interactions in turbulent jets were investigated based on the integrated energy of each scale of motion in a cross section of the jet. The analysis indicates that two frequency components in the axisymmetric mode can amplify an enormous number of other frequencies. Two frequency components in a single helical mode cannot, by themselves, amplify other frequency components. But combinations of frequency components of helical and axisymmetric modes can amplify other frequencies in other helical modes. The present computations produce several features consistent with observations. In a multifrequency excited jet, mixing enhancement was found to be a result of the turbulence enhancement rather than simply the amplification of forced wave components. The high-frequency waves enhance the turbulence close to the jet exit, while the low-frequency waves are most effective further downstream.

REFERENCES

- Mankbadi, R. R.: "Numerical Simulation of Nonlinear Development of Instability Waves," Presented at the NASA/AIAA Computational Fluid Dynamics Conference, Ames Research Center, Moffett Field, CA, March 7-9, 1989.
 Mankbadi, R. R.; Raman, G.; and Rice, E. J.: "Development of the Phase Angles of Instability Waves in Turbulent Jets," NASA TM-101477, ICOMP 89-4, AIAA Paper No. 90-503.
 Mankbadi, R. R.: "Multiwave Interactions in Turbulent Jets," NASA TM-101985, ICOMP 89-12, AIAA Paper No. 90-0365.

Subodh K. Mital, (Postdoctoral), Case Western Reserve University

The effects of fiber/matrix fracture and fiber-matrix interface debonding in a metal matrix composite (MMC) were studied. These studies were part of a

research activity to predict load redistribution, continued fracture propagation and fracture toughness. The three-dimensional finite element model used in the simulation consisted of a group of nine fibers in three by three unit cells, all unidirectional fibers ("nine cell model"). The fracture propagation and toughness were quantified in terms of strain energy release rate (SERR). The composite system used in the present work was a SiC-Ti15 metal matrix composite with 35% fiber volume ratio. Current results indicate that debonding does not initiate by itself. It propagates along an interface as a follow-up of fiber fracture. Even if debonding were present to begin with due to some flaw in manufacturing process, it does not propagate itself without a fiber fracture. Based on these findings, it is concluded that interface debonding is a benign failure mode with respect to composite longitudinal (along the fiber) load direction (S. Mital, J. Caruso and C. Chamis). An abstract was submitted to the Symposium on Computation Technology for Flight Vehicles, Nov. 5-7, 1990, in Washington D.C.

A second study simulated a single fiber push-through from a group of nine fibers in three by three unit cells, all unidirectional fibers ("nine cell model"). Fiber push-through experiments, where a single fiber is pushed through with a diamond indenter and high temperature microhardness tester, are being computationally simulated. The idea is to find predict a load or an applied nominal shear stress at which the fiber is pushed through. Effects of residual stresses on the interface induced by temperature changes, fiber volume ratio, fiber roughness and indenter size are also being studied (S. Mital and C. Chamis).

Christophe Pierre, University of Michigan

The basic purpose of this research is the development of computational methods for understanding and predicting the effects of blade-to-blade dissimilarities, or mistuning, on the dynamics of nearly cyclic bladed-disk assemblies. The topic is of importance as mistuning has been shown to increase the forced response amplitudes of some blades drastically and even to lead to blade failure.

Our research has focused on developing a true measure of sensitivity to mistuning for generic models of turbomachinery rotors. Because mistuning is known in a statistical fashion only, our measure of sensitivity had to be probabilistic. The primary result of our research is that we showed the mean of the second-order eigenvalue perturbation to be a very good descriptor of sensitivity. Specifically, we proved that first-order perturbation terms completely overlook high sensitivity to mistuning and that second-order terms must be included. Furthermore, we obtained analytical expressions for the sensitivity of simple models of blade assemblies. This makes our proposed sensitivity descriptor a very cost-effective tool for identifying parameter ranges where small mistuning has large effects. Indeed, we found the cost of our measure of sensitivity to be negligible compared to that of Monte Carlo simulations.

At least one, perhaps two, refereed journal publications will result from the research supported by ICOMP. The first manuscript is currently in preparation. Future research will focus on extending this measure of sensitivity to flutter and forced response problems. (The work on flutter was initiated near the end of the ICOMP stay; preliminary results are encouraging.) We also need to develop probabilistic techniques that predict the forced response of systems identified to be highly sensitive to small mistuning by our measure of sensitivity. These goals could be achieved through support by ICOMP next summer.

Kenneth Powell, The University of Michigan

The eigensystem associated with the numerical solution of quasi-one-dimensional compressible flow was studied, as a means of understanding the effects of different boundary condition formulations on convergence. For a given inflow Mach number, geometry and boundary condition formulation, a converged solution (to 10^{-12}) was obtained using Roe's scheme. The Jacobian matrix $\partial F/\partial U$ was constructed numerically. At each point, the converged state was perturbed slightly (by a factor 10^{-6}) and the resulting residual was calculated. Fourth-order centered differences of the residual with respect to the change in state were then used to construct the Jacobian. The resulting matrix was then decomposed using a QR algorithm to find its eigenvalues and eigenvectors. The eigensystem was found to have the proper behavior for those cases that can easily be checked analytically: three circles of eigenvalues, tangent at the point (1,0) in the complex plane, for the case of a straight channel with periodic boundary conditions; and a degenerate system with an eigenvector corresponding to a spike at the outlet, for a straight channel with freestream boundary conditions. For other cases, the eigenvalues and dominant eigenvectors showed more complex, but still highly ordered, patterns.

Avram Sidi, Technion-Israel Institute of Technology, Haifa, Israel

The possibility of using acceleration of convergence methods for time-periodic steady-state problems was investigated. It turns out that these problems can be related to eigenvalue-eigenvector problems in numerical linear algebra. As a result of this, the numerical treatment of eigenvalue-eigenvector problems was considered from the point of view of convergence acceleration for iterative procedures. In the course of the research some new vector valued rational approximations were developed. These were studied with respect to their convergence properties which are relevant to time periodic steady-state computations.

Patrick Smolinski, University of Pittsburgh

This study involved the use of a system of transputer computers for the solution of structural dynamics problems. A transputer is a chip level computer with local memory and four asynchronous communication channels that can be linked to other transputers. The transputers can be arranged in a variety of configurations, however, each transputer can have at most four neighbors. In this study a pipeline configuration was used. The transputers were programmed using the OCCAM language which simplifies the programming of parallel algorithms.

The central difference rule, a direct time integration method, was used to solve the governing structural dynamics equations. With direct integration methods the time period of interest is divided into intervals or steps and the solution is progressively computed at each of these time steps. The central difference method is said to be an explicit method since it does not require the solution of algebraic equations. Because of this fact, the displacement at a node can be computed independently of the other nodes. For parallel processing the nodes of the mesh are partitioned into groups which are assigned to different processors for updating and information is exchanged between processors after each update. In this study only nearest neighbor communication was necessary.

The test problem chosen for study is a two-dimensional plate which is fixed at one end and has an applied load at the other end as shown in figure 1. The size and geometry of the problem were varied by changing the number of nodes in the x and y directions. For example, the notation 20×10 indicates that the plate was divided so that there is a row of 20 nodes in the x direction ($nsegx=20$) and 10 nodes in the y-direction ($nsegy=10$). In which case the total number of nodes in the problem would be 200 ($nsegx*nsegy$). To partition the problem for parallel processing, the domain is divided into subdomains by vertical lines and the nodes in each subdomain are assigned to a processor as shown in figure 1. In all cases an equal number of nodes were assigned to each processor.

The results for the first series of test problems for various numbers of time steps are given in table 1. Here the number of processors used was two and the problems had 100, 200, and 400 nodes, respectively. It can be seen from the data that once a sufficient number of time steps are run to overcome the overhead involved in setting up the problem to run in parallel, the solution time is proportional to the size of the problem.

In the second problem, the problem size was varied so that the number of nodes per processor was kept fixed at 50 and the number of processors was varied from two to eight. Note that the eight processor problem is four times as large as the two processor problem. Using only one processor the solution of a 50 node problem takes 31.7 seconds for one thousand time steps. If the parallel algorithm were perfectly efficient, the solution times for the three cases would be equal to the one processor solution time. From table 2 it is evident that some efficiency is lost due to the interprocessor communication. However, it appears that for the one thousand time step case additional processors could be used, thereby increasing the problem size, with little increase in the solution time.

The third problem investigates how the solution time varies as the amount of interprocessor communication is increased. This is done by keeping the number of nodes per processor fixed at 100 while varying the lengths of the problem in the x- and y-directions. Since the problems are partitioned by vertical lines, the interprocessor communication is proportional to the number of nodes along the interface. As can be seen from the data in table 3 the solution time increases with increasing amounts of interprocessor communication.

The study shows that the transputer can be used to solve complex structural dynamics problems. The best performance is obtained for problems where the processor computation to communication ratio is large. However, some issues that require further investigation are, given the geometry of the problem: (1) how can the processors be arranged and the problem be partitioned to minimize interprocessor communication; and (2) can non-nearest neighbor communication be used without a substantial loss in efficiency?

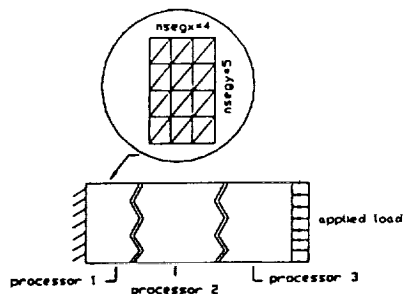


Figure 1. - Problem statement
for finite element test problem.

TABLE 1. - SOLUTION TIMES FOR VARIOUS
SIZE PROBLEMS USING TWO PROCESSORS

Number of time steps	Time, sec		
	10 x 10 problem	20 x 10 problem	40 x 10 problem
1	.3	1.0	1.8
10	.6	1.7	5.0
100	4.0	8.4	18.5
1000	37.4	75.2	153.5

TABLE 2. - SOLUTION TIME FOR VARIOUS NUMBERS OF
PROCESSORS WHERE THE NUMBER OF SEGMENTS PER
PROCESSORS IS FIXED

Number of time steps	Time, sec		
	2 processors 10 x 10 problem	4 processors 20 x 10 problem	8 processors 40 x 10 problem
1	0.30	0.58	1.02
10	.64	.99	1.43
100	3.98	5.06	5.50
1000	37.44	45.78	46.29

TABLE 3. - SOLUTION TIMES USING EIGHT
PROCESSOR FOR PROBLEMS WITH DIF-
FERENT NUMBERS OF SEGMENT IN THE

Number of time steps	Time, sec			
	80 x 10 problem	40 x 20 problem	20 x 40 problem	10 x 80 problem
1	3.28	3.83	4.96	7.54
10	4.02	4.68	5.99	8.92
100	11.44	13.23	16.33	22.69
1000	85.64	98.73	119.71	160.37

Timothy Swafford, Mississippi State University

During these two weeks in July 1989, the majority of time was focused upon the computation of supersonic inviscid flow through oscillating flat plate cascades using Whitfield's high resolution wave-split upwind scheme for solving the Euler equations. Much of these efforts were devoted to converting an existing version of this code (denoted APhase) such that the computational grid could be moved (oscillated) in the same manner as that used by Dennis L. Huff of NASA Lewis Research Center; (the new version has been denoted NPhase). All computations were performed on the NAS (CRAY YMP) located at NASA Ames. As a result of performing this conversion, an error was discovered in the APhase version of the code, written to be capable of handling arbitrary inter-blade phase angles. This error was remedied, which should improve the quality of computed results.

In addition, discussions were held throughout these two weeks with Dennis Huff and T. S. Reddy (University of Toledo) regarding the overall scope and the consolidation of efforts and approach for this computational problem. Future efforts will concentrate on extending the existing two-dimensional capability to three-dimensional geometries for both transonic and supersonic regimes.

Nai-Kuan Tsao, Wayne State University

In this period we have finished the following reports with titles and summaries quoted below:

(1) On the Accuracy of Solving Triangular Systems in Parallel (ICOMP Report No. 88-19)

An error complexity analysis of two algorithms for solving a unit-diagonal triangular system is given. The results show that the usual sequential algorithm is optimal in terms of having the minimal maximum and cumulative error complexity measures. The parallel algorithm described by Sameh and Brent is shown to be essentially equivalent to the optimal sequential one. Some numerical experiments are also included.

(2) On the Equivalence of Gaussian Elimination and Gauss-Jordan Reduction in Solving Linear Equations (ICOMP Report No. 89-2)

A novel general approach to round-off error analysis using the error complexity concepts is described. This is applied to the analysis of the Gaussian Elimination and the Gauss-Jordan scheme for solving linear equations. The results show that the two algorithms are equivalent in terms of our error complexity measures. Thus the inherently parallel Gauss-Jordan scheme can be implemented with confidence if parallel computers are available.

(3) On the Equivalence of a Class of Inverse Decomposition Algorithms for Solving Systems of Linear Equations (ICOMP Report No. 89-11)

A class of direct inverse decomposition algorithms for solving systems of linear equations is presented. Their behavior in the presence of round-off errors is analyzed. It is shown that under some mild restrictions on their implementation, the class of direct inverse decomposition algorithms presented is equivalent in terms of our error complexity measures.

(4) An Accurate Tridiagonal Equation Solver

In this note a tridiagonal equation solver is devised based on divide-and-conquer technique which decouples the original system into ever smaller independent systems which are easily solved. The elimination of the usual back-substitution stage in the new algorithm makes it a better choice for solving tridiagonal systems when compared with the usual elimination-backsubstitution algorithm. Some numerical experiments are also presented.

(5) Error Complexity Analysis of Swarztrauber's Parallel Algorithm for Tridiagonal Systems (Submitted to J. Applied Numerical Mathematics)

The error complexity analysis of two algorithms based on Cramer's rule are given. The results show that the parallel algorithm, although efficient, is more likely to incur round-off errors than the straight-forward sequential algorithm. Two alternate better algorithms are also derived. Numerical experiments supporting the theoretical results are also included.

(6) On the Optimal Implementation of LU Decomposition Methods for Solving Linear Equations (Presented at the International Conference on Computational Techniques and Application at Brisbane, Australia, July 10-12, 1989)

In this note the optimal order in which the inner-product type of computation should be executed in the LU decomposition stage is described for solving systems of linear equations. Numerical experiments supporting the theoretical results are also presented.

Reports (1), (2), (3) together with (6) deal with the basic questions in solving dense n by n systems of equations: among the many methods that one can choose, which one or which class of methods should be chosen to minimize round-off error propagation? We have provided the answer for the class of methods based on triangular decomposition - they are all equivalent.

Reports (4) and (5) deal with the accuracy issue in solving tridiagonal systems of equations. Basically triangular systems can be solved more accurately by employing methods derived by Cramer's rule - implicitly or explicitly. The former is the method derived in Report (4), the latter is Swartrauber's algorithm which is easily parallelized, although it is numerically a little inferior to its sequential brethren as is reported in Report (4).

Eli Turkel, Tel Aviv University

Work continued with Alvin Bayliss on adaptive strategies for pseudospectral methods. The partial differential equations are transformed to a new coordinate and the mapping depends on two free parameters. These parameters are chosen so as to minimize a functional that bounds the maximum error. We choose a mapping that can be expressed in terms of arctangent functions. Tests indicate that this mapping works not only for problems with large gradients, but also localized behavior as in a Gaussian and even for smooth sinusoidal functions. Applications to a combustion code demonstrate that one can substantially reduce the number of nodes required for a given accuracy.

The Navier-Stokes code has been extended to use a matrix valued artificial viscosity. Numerous tests for external flow show the increased accuracy that can be obtained with this new formulation. Applications to internal flow are presently being pursued. Progress has also been made on the use of a better shock locator with applications to hypersonic flow.

William J. Usab, Jr., Purdue University

The goal of the present research is the development of improved multi-grid acceleration formulations for the Euler and Navier-Stokes equations which can be incorporated into Drs. Adamczyk's multi-stage averaged-passage analysis. The focus of this research is in the proper formulation of coarse mesh boundary conditions in the multi-grid acceleration scheme and in the improvement of multi-grid acceleration for highly stretched meshes commonly used to resolve viscous shear layers. The approach taken here is to first formulate the multi-grid coarse mesh equations on a differential level through the introduction of a filter operator. Then by specifying a discrete approximation to the coarse mesh equations and the filter operator a consistent formulation of both the multi-grid acceleration scheme and the coarse mesh boundary conditions follow. This work is a continuation of work which was initiated over the last two summers.

During the present summer a general two-dimensional Euler and Navier-Stokes multi-grid program written at Purdue was modified to compute compressor and turbine cascade solutions on h -type meshes generated by Tim Beach's mesh generation program (a two-dimensional version of the mesh generation program used in the average-passage analysis). Several different multi-grid boundary condition formulations have been constructed and studied for typical cascade configurations. As shown with previous model problems, a consistent formulation of the coarse mesh boundary conditions is required to maintain the fine mesh accuracy. Based on concepts demonstrated in this program I helped Dr. Kevin Kirtley formulate and implement a basic multi-grid acceleration scheme

into a multi-stage analysis. Also, an ICOMP seminar was given on the formulation of boundary conditions for multi-grid acceleration of the Euler and Navier-Stokes equations.

A study of the effect of different multi-grid cycling approaches was made. It was found that both the dissipation and viscous terms may be lagged in the multi-grid residual transfer operation resulting in a significant reduction in the computational work while maintaining the over all convergence rate of the multi-grid acceleration scheme.

A study of the effects of high mesh stretching on the numerical solution of the Euler and Navier-Stokes equations using a four-stage Runge-Kutta scheme was conducted. It was found that high mesh stretching, which results in very large aspect ratio cells, (on the order of 1000 or greater for typical viscous flow problems) causes both extremely slow convergence rates and very high numerical errors. A model subsonic duct flow problem was constructed to study this problem. It was found that the same errors are present for two-dimensional Euler solutions computed on viscous type meshes. The source of the numerical errors was found to be due to the modified wave number scaling commonly used in the artificial dissipation. A new scaling model for the artificial dissipation which eliminates these errors has been formulated and tested.

Bram Van Leer, The University of Michigan

The research with Liou and Shuen on the formulation of upwind fluxes for real gases with equilibrium chemistry continued and led to an extension of the real-gas Roe-solver reported on in Reno, January 1989. The extension is included in a paper submitted to and accepted by the Journal of Computational Physics. A second paper submitted to the JCP deals with upwind fluxes for non-equilibrium gases.

Another research effort concerns the formulation of multi-dimensional wave models for describing spatial residuals of the Euler equations. The goal here is speeding up convergence to a steady state by enhancing the wave speeds in the discrete model. A two-dimensional model based on four characteristic waves moving along a streamwise coordinate, and two acoustic waves moving across streamlines, was adopted. The six wave strengths can be derived from the four components of the residual and the two components of the pressure gradient.

David Walker, Lehigh University

During my visit at ICOMP I worked on two problems. The first was associated with computation of boundary-layer flows which develop sudden normal updrafts. At high Reynolds numbers, the viscous flow near the wall develops a limit structure in which the boundary layer bifurcates and a viscous shear layer rapidly rises toward the inviscid region away from the wall. As the phenomenon develops, the boundary-layer fluid focusses into a band which progressively narrows in the streamwise direction. It is not possible to compute such flows using conventional techniques based on the Eulerian description of the fluid motion. Calculations for the limit problem, $Re \rightarrow \infty$, were carried out using Lagrangian methods, wherein the trajectories of a large number of fluid particles are computed. The second problem I worked on was the development of computational methods for three-dimensional turbulent boundary layers using embedded function algorithms. In this approach, a numerical solution for the outer flow in the boundary layer is systematically matched to a set of functionals which represent the velocity distribution in the near wall region.

David Whitfield, Mississippi State University

Implicit solid wall and farfield characteristic variable boundary conditions were formulated and incorporated into the two-pass block triangular implicit upwind scheme used in some of the rotating machinery codes at NASA Lewis. The use of flux Jacobian freezing for steady state solutions is not as effective when using implicit boundary conditions as it is when using the current formulation. The reason is that the flux Jacobians are directly in the implicit formulation, and current Jacobians are needed. On the other hand, for unsteady problems, there are advantages to implicit boundary conditions because the Jacobians must be updated each time cycle. Unfortunately, implicit boundary conditions tend to complicate the coding of general three-dimensional composite blocked grid flow solvers.

Yau Shu Wong, University of Alberta

The current work is a continuation of the research initiated last year. Our early report (ICOMP 89-7) presented a uniform approach to construct absorbing boundary conditions for linear hyperbolic partial differential equations. The report has also been accepted for publication in the Journal of Computational Physics. Our approach is derived from the group velocity theory of wave propagation, and the absorbing boundary conditions are developed so that the wave-like solutions of the hyperbolic equations are permitted to propagate through the artificial boundary as if there is no boundary present. It has been shown that the resulting boundary conditions satisfy a canonical factorization form in which each factor annihilates a wave packet traveling at a given velocity. The technique is currently being extended to nonlinear problems and the application of the method for Computational Fluid Dynamics (CFD) codes such as the unsteady transonic flows around airfoil and wing has been pursued.

In addition to the work on absorbing boundary conditions, preconditioned gradient algorithms for large systems of equations in CFD applications have been investigated. Once the mathematical equations and boundary conditions are formulated, finite-difference or finite-element methods are usually applied for discretization. The solution of the continuum problem is then obtained by solving the resulting algebraic equations. For a nonlinear problem, the solution is obtained via solving a sequence of systems of linear equations. For many CFD applications, the resulting linear systems are large and sparse, and finding the solution of these large systems of equations is frequently the most time consuming part in the entire CFD code. The proposed gradient algorithms have proven to provide a rapid rate of convergence and it can also be implemented efficiently on a vector computer. Application of the method to problems of interest to NASA Lewis researchers will be pursued in the future.

A. Yoshizawa, Institute of Industrial Science, University of Tokyo

I worked on turbulence modeling using a two-point closure method, especially, a two-scale direct-interaction approximation (TSDIA) in collaboration with Dr. Meng-Sing Liou, NASA Lewis. The work consists of two parts. One is reexamination of the framework of the TSDIA. In relation to this, a new effect of the mean velocity shear on the dissipation rate was pointed out. This effect may explain the discrepancy in dissipation rates between direct simulation and turbulence models. Another is the study of compressible turbulence

modeling of the three-equation type. Its prominent feature is inclusion of the density fluctuation terms. Those terms can explain the effect of decreasing turbulence length scales in shock boundary-layer interactions.

Shaye Yungster, (Postdoctoral), University of Washington

Advanced numerical studies of the superdetonative propulsion modes of the ram accelerator concept were started at ICOMP. These propulsion modes, which utilize oblique detonation waves or other shock induced combustion modes, are similar to cycles proposed for NASP. The ram accelerator is a ramjet-in-tube projectile accelerator whose principle of operation is similar to that of a supersonic airbreathing ramjet. The projectile resembles the centerbody of a ramjet and travels through a stationary tube filled with a premixed gaseous fuel and oxidizer mixture. The tube acts as the outer cowl of the ramjet, and the combustion process travels with the projectile. Theoretical and numerical projections consistent with experiments conducted at the University of Washington suggest that the ram accelerator concept can efficiently accelerate projectiles to velocities up to 10 km/s for projectile masses ranging from 0.1 to 1000 kg. This potential places the ram accelerator in competition with rail guns and other electromagnetic or electrothermal hypervelocity accelerators for applications requiring this velocity range, such as hypervelocity impact studies, direct launch to orbit of acceleration insensitive payloads and hypersonic testing. In addition, the flow and combustion processes in the ram accelerator are very similar to the supersonic combustion flow fields of interest to hypersonic airbreathing propulsion systems, as in the NASP program.

In the present study, several CFD codes (including RPLUS and my own code) will be applied to investigate the flow and combustion processes in the ram accelerator. These codes solve the complete Navier-Stokes equations and species transport equations in a fully coupled and efficient manner. The thermal and chemical nonequilibrium processes are simulated by means of a finite-rate chemistry model and a multitemperature thermodynamics model incorporated into these codes. In the first phase of the proposed project, an 8 species-14 reaction combustion model for hydrogen/oxygen mixtures will be implemented. Due to the similarity between the flow and combustion processes in the ram accelerator and those associated with hypersonic airbreathing engines, it is clear that the numerical studies proposed here are also of direct value to the NASP program.

REPORTS AND ABSTRACTS

The ICOMP Steering Committee: "Institute for Computational Mechanics in Propulsion (ICOMP), Third Annual Report-1988," ICOMP Report No. 89-1, NASA TM-101961, March 1989, 60 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University 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 1988.

Tsao, Nai-kuan (ICOMP): "On the Equivalence of Gaussian Elimination of Gauss-Jordan Reduction in Solving Linear Equations," ICOMP Report No. 89-2, NASA TM-101466, February 1989, 22 pages.

A novel general approach to round-off error analysis using the error complexity concepts is described. This is applied to the analysis of the Gaussian Elimination of the Gauss-Jordan scheme for solving linear equations. The results show that the two algorithms are equivalent in terms of our error complexity measures. Thus the inherently parallel Gauss-Jordan scheme can be implemented with confidence if parallel computers are available.

Kim, S.-W. (ICOMP): "Numerical Computation of Shock Wave-Turbulent Boundary Layer Interaction in Transonic Flow Over an Axisymmetric Curved Hill," ICOMP Report No. 89-3, NASA TM-101473, February 1989, 30 pages.

A control-volume based finite difference computation of a turbulent transonic flow over an axisymmetric curved hill is presented. The numerical method is based on the SIMPLE algorithm, and hence the conservation of mass equation is replaced by a pressure correction equation for compressible flows. The turbulence is described by a k-e turbulence model supplemented by a near-wall turbulence model. In the method, the dissipation rate in the region very close to the wall is obtained from an algebraic equation and that for the rest of the flow domain is obtained by solving a partial differential equation for the dissipation rate. The other flow equations are integrated up to the wall. It is shown that the present turbulence model yields the correct location of the compression shock. The other computational results are also in good agreement with experimental data.

Mankbadi, Reda R. (ICOMP), Raman, Ganesh (Sverdrup), and Rice, Edward J. (NASA Lewis): "On the Conditions for Resonance Interactions of Instability Waves in the Axisymmetric Jet," ICOMP Report No. 89-4, NASA TM-101477, March 1989, 11 pages.

The conditions for resonance interaction between two instability waves in an axisymmetric jet were investigated. Considerations of the energy equation of the wave resulting from the interaction indicate that the phase angle between the wave-induced stresses and the wave-induced strains plays a crucial role in the resonance interaction. This fact is demonstrated experimentally by exciting a jet at fundamental and subharmonic frequencies. The phase angle between the waves stresses and strains was varied by varying the initial phase-difference between the two excitation waves. The subharmonic resonance was found to be highly dependent on this angle. Favorable agreement was found between the phase angles predicted by a nonlinear theory and the measured ones. The theory is used to explain the subharmonic's resonance in terms of the phase-angles.

Kim, S.-W. (ICOMP): "Control-Volume Based Navier-Stokes Equation Solver Valid at All Flow Velocities," ICOMP Report No. 89-5, NASA TM-101488, February 1989, 40 pages.

A control-volume based finite difference method to solve the Reynolds averaged Navier-Stokes equations is presented. A pressure correction equation valid at all flow velocities and a pressure staggered grid layout are used in the method. Example problems presented herein include: a developing laminar channel flow, developing laminar pipe flow, a lid-driven square cavity flow, a laminar flow through a 90-degree bent channel, a laminar polar cavity flow, and a turbulent supersonic flow over a compression ramp. A k- ϵ turbulence model supplemented with a near-wall turbulence model was used to solve the turbulent flow. It is shown that the method yields accurate computational results even when highly skewed, unequally spaced, curved grids are used. It is also shown that the method is strongly convergent for highly Reynolds number flows.

Duck, Peter W. (ICOMP): "The Response of a Laminar Boundary Layer in Supersonic Flow to Small Amplitude Progressive Waves," ICOMP Report No. 89-6, NASA TM-101965, March 1989, 38 pages.

In this paper the effect of a small amplitude progressive wave on the laminar boundary layer on a semi-infinite flat plate, due to a uniform supersonic freestream flow, is considered. The perturbation to the flow divides into two streamwise zones. In the first, relatively close to the leading edge of the plate, on a transverse scale comparable to the boundary layer thickness, the perturbation flow is described by a form of the unsteady linearized compressible boundary layer equations. In the free-stream, this component of flow is governed by the wave equation, the solution of which provides the outer velocity conditions for the boundary layer. This system is solved numerically, and also the asymptotic structure in the far downstream limit is studied. This reveals a breakdown and

a subsequent second streamwise zone, where the flow disturbance is predominantly inviscid. The two zones are shown to match in a proper asymptotic sense.

Jiang, Hong (University of Alberta) and Wong, Yau Shu (ICOMP): "Absorbing Boundary Conditions for Second-Order Hyperbolic Equations," ICOMP Report No. 89-7, NASA TM-102009, April 1989, 33 pages.

A uniform approach to construct absorbing artificial boundary conditions for second-order linear hyperbolic equations is proposed. The nonlocal boundary condition is given by a pseudodifferential operator that annihilates travelling waves. It is obtained through the dispersion relation of the differential equation by requiring that the initial-boundary value problem admits the wave solutions travelling in one direction only. Local approximation of this global boundary condition yields an n th-order differential operator. It is shown that the best approximations must be in the canonical forms which can be factorized into first-order operators. These boundary conditions are perfectly absorbing for wave packets propagating at certain group velocities. A hierarchy of absorbing boundary conditions is derived for transonic small perturbation equations of unsteady flows. These examples illustrate that the absorbing boundary conditions are easy to derive, and the effectiveness is demonstrated by the numerical experiments.

Van Dommelen, Leon L. (ICOMP) and Cowley, Stephen J. (ICOMP): "On the Lagrangian Description of Unsteady Boundary Layer Separation, I - General Theory," ICOMP Report No. 89-8, NASA TM-102026, May 1989, 50 pages.

Although unsteady, High-Reynolds number, laminar boundary layers have conventionally been studied in terms of Eulerian coordinates, a Lagrangian approach may have significant analytical and computational advantages. In Lagrangian coordinates the classical boundary-layer equations decouple into a momentum equation for the motion parallel to the boundary, and a hyperbolic continuity equation (essentially a conserved Jacobian) for the motion normal to the boundary. The momentum equations, plus the energy equation if the flow is compressible, can be solved independently of the continuity equation. Unsteady separation occurs when the continuity equation becomes singular as a result of touching characteristics, the condition for which can be expressed in terms of the solution of the momentum equations. The solutions to the momentum and energy equations remain regular. Asymptotic structures for a number of unsteady three-dimensional separating flows follow and depend on the symmetry properties of the flow (e.g., line symmetry, axial symmetry). In the absence of any symmetry, the singularity structure just prior to separation is found to be quasi two-dimensional with a displacement thickness in the form of a crescent shaped ridge. Physically the singularities can be understood in terms of the behavior of a fluid element inside the boundary layer which contracts in a direction parallel to the boundary and expands normal to it, thus forcing the fluid above it to be ejected from the boundary layer.

Van Dommelen, Leon L. (ICOMP): "On the Lagrangian Description of Unsteady Boundary Layer Separation, II - The Spinning Sphere," ICOMP Report No. 89-9, NASA TM-102027, May 1989, 26 pages.

A theory to explain the initial stages of unsteady separation has been proposed by Van Dommelen and Cowley (1989). In the present paper, this theory is verified for the separation process that occurs at the equatorial plane of a sphere or a spheroid which is impulsively spun around an axis of symmetry. A Lagrangian numerical scheme is developed which gives results in good agreement with Eulerian computations, but which is significantly more accurate. This increased accuracy, and a simpler structure to the solution, also allows verification of the Eulerian structure, including the presence of logarithmic terms. Further, while the Eulerian computations broke down at the first occurrence of separation, it is found that the Lagrangian computation can be continued. It is argued that this separated solution does provide useful insight into the further evolution of the separated flow. A remarkable conclusion is that an unseparated vorticity layer at the wall, a familiar feature in unsteady separation processes, disappears in finite time.

Goldstein, M. E. (NASA Lewis) and Choi, S. W. (ICOMP): "Nonlinear Evolution of Interacting Oblique Waves on Two-Dimensional Shear Layers," ICOMP Report No. 89-10, NASA TM-102030, June 1989, 32 pages.

We consider the effects of critical layer nonlinearity on spatially growing oblique instability waves on nominally two-dimensional shear layers between parallel streams. The analysis shows that three-dimensional effects cause nonlinearity to occur at much smaller amplitudes than it does in two-dimensional flows. The nonlinear instability wave amplitude is determined by an integro-differential equation with cubic type nonlinearity. The numerical solutions to this equation are worked out and discussed in some detail. We show that they always end in a singularity at a finite downstream distance.

Tsao, Nai-kuan (ICOMP): "On the Equivalence of a Class of Inverse Decomposition Algorithms for Solving Systems of Linear Equations," ICOMP Report No. 89-11, NASA TM-102036, May 1989, 26 pages.

A class of direct inverse decomposition algorithms for solving systems of linear equations is presented. Their behavior in the presence of round-off errors is analyzed. It is shown that under some mild restrictions on their implementation, the class of direct inverse decomposition algorithms presented are equivalent in terms of our error complexity measures.

Mankbadi, Reda R. (ICOMP): "Multiwave Interactions in Turbulent Jets," ICOMP Report No. 89-12, NASA TM-101985, September 1989, 46 pages.

Nonlinear wave-wave interactions in turbulent jets were investigated based on the integrated energy of each scale of motion in a cross section of the

jet. The analysis indicates that two frequency components in the axisymmetric mode can interact with other background frequencies in the axisymmetric mode, thereby amplifying an enormous number of other frequencies. Two frequency components in a single helical mode cannot, by themselves, amplify other frequency components. But combinations of frequency components of helical and axisymmetric modes can amplify other frequencies in other helical modes. The present computations produce several features consistent with experimental observations such as (1) dependency of the interactions on the initial phase differences, (2) enhancement of the momentum thickness under multifrequency forcing, and (3) the increase in background turbulence under forcing. In a multifrequency-excited jet, mixing enhancement was found to be a result of the turbulence enhancement rather than simply the amplification of forced wave components. The excitation waves pump energy from the mean flow to the turbulence, thus enhancing the latter. The high-frequency waves enhance the turbulence close to the jet exit, but, the low-frequency waves are most effective further downstream.

Powell, Kenneth G. (ICOMP) and van Leer, Bram (ICOMP): "A Genuinely Multi-Dimensional Upwind Cell-Vertex Scheme for the Euler Equations," ICOMP Report No. 89-13, NASA TM-102029, May 1989, 21 pages.

A scheme for solving the two-dimensional Euler equations is developed. It is based on a new scheme for the two-dimensional linear convection equation, and the Euler-equation decomposition developed by Hirsch et al. [1]. The scheme is genuinely two-dimensional. At each iteration, the data are locally decomposed into four variables, allowing convection in appropriate directions. This is done via a cell-vertex scheme with a downwind-weighted distribution step. The scheme is conservative, and third-order accurate in space. The derivation and stability analysis of the scheme for the convection equation, and the derivation of the extension to the Euler equations are given. Preconditioning techniques based on local values of the convection speeds are discussed. The scheme for the Euler equations is applied to two channel-flow problems. It is shown to converge rapidly to a solution that agrees well with that of a third-order solver.

Hariharan, S. I. (ICOMP) and Hagstrom, Thomas (ICOMP): "Far Field Expansion for Anisotropic Wave Equations," ICOMP Report No. 89-14, NASA TM-102112, June 1989, 12 pages.

A necessary ingredient for the numerical simulation of many time dependent phenomena in acoustics and aerodynamics is the imposition of accurate radiation conditions at artificial boundaries. The asymptotic analysis of propagating waves provides a rational approach to the development of such conditions. In this work we derive a far field asymptotic expansion of solutions of anisotropic wave equations. This generalizes the well known Friedlander expansion for the standard wave operator. We use the expansion to derive a hierarchy of radiation conditions of increasing accuracy. Two numerical experiments are given to illustrate the utility of our

approach. The first application is to the study of unsteady vortical disturbances impinging on a flat plate; the second is to the simulation of inviscid flow past an impulsively started cylinder.

Gajjar, J. S. B. (ICOMP) and Cole, J. W. (Iowa State University): "The Upper-Branch Stability of Compressible Boundary Layer Flows," ICOMP Report No. 89-15, NASA TM-102128, August 1989, 38 pages.

The upper-branch linear and nonlinear stability of compressible boundary layer flows is studied using the approach of Smith and Bodonyi (1982) for a similar incompressible problem. Both pressure gradient boundary layers and Blasius flow are considered with and without heat transfer, and the neutral eigenrelations incorporating compressibility effects are obtained explicitly. The compressible nonlinear viscous critical layer equations are derived and solved numerically and the results indicate some solutions with positive phase shift across the critical layer. Various limiting cases are investigated including the case of much larger disturbance amplitudes and this indicates the structure for the strongly nonlinear critical layer of the Benney-Bergeron (1969) type. Finally we show also how a match with the inviscid neutral inflexional modes arising from the generalized inflexion point criterion, is achieved.

Demuren, A. O. (ICOMP): "Calculation of Turbulence-Driven Secondary Motion in Ducts With Arbitrary Cross Section," ICOMP Report No. 89-16, NASA TM-102142, July 1989, 22 pages.

Calculation methods for turbulent duct flows are generalized for ducts with arbitrary cross sections. The irregular physical geometry is transformed into a regular one in computational space, and the flow equations are solved with a finite-volume numerical procedure. The turbulent stresses are calculated with an algebraic stress model derived by simplifying model transport equations for the individual Reynolds stresses. Two variants of such a model are considered in the present study. These procedures enable the prediction of both the turbulence-driven secondary flow and the anisotropy of the Reynolds stresses, in contrast to some of the earlier calculation methods. Model predictions are compared to experimental data for developed flow in triangular duct, trapezoidal duct and a rod-bundle geometry. The correct trends are predicted, and the quantitative agreement is mostly fair. The simpler variant of the algebraic stress model procured better agreement with the measured data.

Afolabi, Dare (ICOMP): "Effects of Mistuning and Matrix Structure on the Topology of Frequency Response Curves," ICOMP Report No. 89-17, NASA TM-102290, October 1989, 49 pages.

The stability of a frequency response curve under mild perturbations of the system's matrix is investigated. Using recent developments in the theory of singularities of differentiable maps, it is shown that the stability of a response curve depends on the structure of the system's

matrix. In particular, the frequency response curves of a cyclic system are shown to be unstable. Consequently, slight parameter variations engendered by mistuning will induce a significant difference in the topology of the forced response curves, if the mistuning transformation crosses the bifurcation set.

Kim, S.-W. (ICOMP): "Calculation of Reattaching Shear Layers in Divergent Channel With a Multiple-Time-Scale Turbulence Model" ICOMP Report No. 89-18, NASA TM-102293, August 1989, 26 pages.

Numerical calculations of turbulent reattaching shear layers in a divergent channel are presented. The turbulence is described by a multiple-time-scale turbulence model. The turbulent flow equations are solved by a control-volume based finite difference method. The computational results are compared with those obtained using $k-\epsilon$ turbulence models and algebraic Reynolds stress turbulence models. It is shown that the multiple-time-scale turbulence model yields significantly improved computational results than the other turbulence models in the region where the turbulence is in a strongly inequilibrium state.

Hariharan, S. I. (ICOMP), Ping, Yu (University of Akron) and Scott, J. R. (NASA Lewis): "Time Domain Numerical Calculations of Unsteady Vortical Flows About a Flat Plate Airfoil," ICOMP Report No. 89-19, NASA TM-102318, September 1989, 22 pages.

A time domain numerical scheme is developed to solve for the unsteady flow about a flat plate airfoil due to imposed upstream, small amplitude, transverse velocity perturbations. The governing equation for the resulting unsteady potential is a homogeneous, constant coefficient, convective wave equation. Accurate solution of the problem requires the development of approximate boundary conditions which correctly model the physics of the unsteady flow in the far field. A uniformly valid far field boundary condition is developed, and numerical results are presented using this condition. The stability of the scheme is discussed, and the stability restriction for the scheme is established as a function of the Mach number. Finally, comparisons are made with the frequency domain calculation by Scott and Atassi, and the relative strengths and weaknesses of each approach are assessed.

Kim, S.-W. (ICOMP): "Numerical Investigation of an Internal Layer in Turbulent Flow Over a Curved Hill," ICOMP Report No. 89-20, NASA TM-102230, October 1989, 40 pages.

The development of an internal layer in turbulent boundary layer flow over a curved hill is investigated numerically. The turbulence field of the boundary layer flow over the curved hill is compared with that of a turbulent flow over a symmetric airfoil (which has the same geometry as the curved hill except that the leading and trailing edge plates were removed) to study the influence of the strongly curved surface on the turbulence

field. The turbulent flow equations are solved by a control-volume based finite difference method. The turbulence is described by a multiple-time-scale turbulence model supplemented with a near-wall turbulence model. Computational results for the mean flow field (pressure distributions on the walls, wall shearing stresses and mean velocity profiles), the turbulence structure (Reynolds stress and turbulent kinetic energy profiles), and the integral parameters (displacement and momentum thicknesses) compared favorably with the measured data. Computational results show that the internal layer is a strong turbulence field which is developed beneath the external boundary layer and is located very close to the wall. Development of the internal layer was more obviously observed in the Reynolds stress profiles and in the turbulent kinetic energy profiles than in the mean velocity profiles. In this regard, the internal layers is significantly different from wall-bounded simple shear layers in which the mean velocity profile characterizes the boundary layer most distinguishably. Development of such an internal layer, characterized by an intense turbulence field, is attributed to the enormous mean flow strain rate caused by the streamline curvature and the strong pressure gradient. In the turbulent flow over the curved hill, the internal layer begin to form near the forward corner of the hill, merges with the external boundary layer, and develops into a new fully turbulent boundary layer as the fluid flows in the downstream direction. For the flow over the symmetric airfoil, the boundary layer began to form from almost the same location as that of the curved hill, grew in its strength, and formed a fully turbulent boundary layer from mid-part of the airfoil and in the downstream region. Computational results also show that the detailed turbulence structure in the region very close to the wall of the curved hill is almost the same as that of the airfoil in most of the curved regions except near the leading edge. Thus the internal layer of the curved hill and the boundary layer of the airfoil were also almost the same. Development of the wall shearing stress and separation of the boundary layer at the rear end of the curved hill mostly depends on the internal layer and is only slightly influenced by the external boundary layer flow.

Goodrich, John W. (NASA Lewis), Gustafson, Karl (ICOMP), and Halassi, Kodosa (Kansas State University): "Hopf Bifurcation in Driven Cavity," ICOMP Report No. 89-21, NASA TM-102334, October 1989, 52 pages.

Incompressible two dimensional calculations are reported for the impulsively started lid driven cavity with aspect ratio two. The algorithm is based on the time dependent streamfunction equation, with a Crank-Nicolson differencing scheme for the diffusion terms, and with an Adams-Bashforth scheme for the convection terms. A multigrid method is used to solve the linear implicit equations at each time step. Periodic asymptotic solutions have been found for $Re = 10000$ and for $Re = 5000$. The $Re = 5000$ results are validated by grid refinement calculations. The solutions are shown to be precisely periodic, and care is taken to demonstrate that asymptotic states have been reached. A discussion is included about the indicators that are used to show that the asymptotic state is indeed periodic.

Chulya, A. (ICOMP), and Walker, K. P. (Engineering Science Software Inc.): "A New Uniformly Valid Asymptotic Integration Algorithm for Elasto-Plastic-Creep and Unified Viscoplastic Theories Including Continuum Damage," ICOMP Report No. 89-22, NASA TM-102344, November 1989.

A new scheme to integrate a system of stiff differential equations for both the elasto-plastic-creep and unified viscoplastic theories is presented. The method has high stability, allows large time increments, and is implicit and iterative. It is suitable for use with continuum damage theories. The scheme was incorporated into MARC, a commercial finite element code through a user subroutine called HYPELA. Results from numerical problems under complex loading histories are presented for both small and large scale analysis. To demonstrate the scheme's accuracy and efficiency, comparisons to a self-adaptive forward Euler method are made.

Jiang, Bo-nan (ICOMP), and Povinelli, Louis A. (NASA Lewis): "Least-Squares Finite Element Method for Fluid Dynamics," ICOMP Report No. 89-23, NASA TM-102352, August 1989, 38 pages.

This paper gives an overview of new developments of the least-squares finite element method (LSFEM) in fluid dynamics. Special emphasis is placed on the universality of LSFEM; the symmetry and positiveness of the algebraic systems obtained from LSFEM; the accommodation of LSFEM to equal-order interpolations for incompressible viscous flows; and the natural numerical dissipation of LSFEM for convective transport problems and high-speed compressible flows. The performance of LSFEM is illustrated by numerical examples.

Demuren, A. O. (ICOMP): "Application of Multi-grid Methods for Solving the Navier-Stokes Equations," ICOMP Report No. 89-24, NASA TM-102359, October 1989, 22 pages.

This paper presents the application of a class of multi-grid methods to the solution of the Navier-Stokes equations for two-dimensional laminar flow problems. The methods consist of combining the full approximation scheme--full multi-grid technique (FAS-FMG) with point-, line-, or plane relaxation routines for solving the Navier-Stokes equations in primitive variables. The performance of the multi-grid methods is compared to that of several single-grid methods. The results show that much faster convergence can be procured through the use of the multi-grid approach than through the various suggestions for improving single-grid methods. The importance of the choice of relaxation scheme for the multi-grid method is illustrated.

Sidi, Avram (ICOMP), and Ford, William F. (NASA Lewis): "Quotient-Difference Type Generalizations of the Power Method and Their Analysis," ICOMP Report No. 89-25, NASA TM-102361, September 1989, 18 pages.

The recursion relations that were proposed in "Recursive Algorithms for Vector Extrapolation Methods" by W. F. Ford and A. Sidi (Appl. Numer. Math, 4 (1988), pp. 477-489) for implementing vector extrapolation methods are used for devising generalizations of the power method for linear operators. These generalizations are shown to produce approximations to largest eigenvalues of a linear operator under certain conditions. They are similar in form to the quotient-difference algorithm and share similar convergence properties with the latter. These convergence properties resemble also those obtained for the basic LR and QR algorithms. Finally, it is shown that the convergence rate produced by one of these generalizations is twice as fast for normal operators as it is for nonnormal operators.

Walker, J. D. A. (ICOMP): "Wall-Layer Eruptions in Turbulent Flows," ICOMP Report No. 89-26, NASA TM-102362, September 1989, 11 pages.

The near-wall region of a turbulent flow is investigated in the limit of large Reynolds numbers. When low-speed streaks are present, the governing equations are shown to be of the boundary-layer type. Physical processes leading to local breakdown and a strong interaction with the outer region are considered. It is argued that convected vortices, predominantly of the hairpin type, will provoke eruptions and regenerative interactions with the outer region.

Chang, Shih-Hung (ICOMP): "On the Application of Subcell Resolution to Conservation Laws with Stiff Source Terms," ICOMP Report No. 89-27, NASA TM-102384, November 1989, 12 pages.

LeVeque and Yee recently investigated a one-dimensional scalar conservation law with stiff source terms modeling the reacting flow problems and discovered that for the very stiff case most of the current finite difference methods developed for non-reacting flows would produce wrong solutions when there is a propagating discontinuity. A numerical scheme, ENO/SRCD, is proposed in this report for solving conservation laws with stiff source terms. This scheme is a modification of Harten's ENO scheme with subcell resolution, ENO/SR. The locations of the discontinuities and the characteristic directions are essential in the design. Strang's time-splitting method is used and time evolutions are done by advancing along the characteristics. Numerical experiment using this scheme shows excellent results on the model problem of LeVeque and Yee. Comparisons of the results of ENO, ENO/SR, and ENO/SRCD are also presented.

Jiang, Bo-nan (ICOMP): "A Least-Squares Finite Element Method for Incompressible Navier-Stokes Problems," ICOMP Report No. 89-28, NASA TM-102385, October 1989, 20 pages.

A least-squares finite element method, based on the velocity-pressure-vorticity formulation, is developed for solving steady incompressible Navier-Stokes problems. This method leads to a minimization problem rather than to a saddle-point problem by the classic mixed method, and can thus accommodate equal-order interpolations. This method has no parameter to tune. The associated algebraic system is symmetric, and positive definite. Numerical results for the cavity flow at Reynolds number up to 10 000 and the backward-facing step flow at Reynolds number up to 900 are present.

Boretti, A. A. (ICOMP): "Two-Dimensional Euler and Navier Stokes Time Accurate Simulations of Fan Rotor Flows," ICOMP Report No. 89-29, NASA TM-102402, November 1989.

The paper presents two numerical methods useful to describe the unsteady flow field in the blade-to-blade plane of an axial fan rotor. These methods solve the compressible, time-dependent, Euler and the compressible, turbulent, time-dependent, Navier-Stokes conservation equations for mass, momentum and energy. The Navier-Stokes equations are written in Favre-Averaged form and closed with an approximate two equation turbulence model with low Reynolds number and compressibility effects included. The unsteady aerodynamic is obtained by superposing inflow or outflow unsteadiness to the steady conditions through time-dependent boundary conditions. The integration in space is performed by using a finite volume scheme, while the integration in time is performed by using k-stage Runge-Kutta schemes, $k = 2, 5$. The numerical integration algorithm allows the reduction of the computational cost of an unsteady simulation involving high frequency disturbances in both CPU time and memory requirements. Less than 200 sec CPU time are required to advance the Euler equations in a computational grid made up by about 2000 grid points during 10 000 time steps on a CRAY Y-MP computer, with a required memory less than 0.3 mw.

Goldstein, M. E. (NASA Lewis) and Wundrow, D. W. (State University of New York at Buffalo): "Spatial Evolution of Nonlinear Acoustic Mode Instabilities on Hypersonic Boundary Layers," ICOMP Report No. 89-30, NASA TM-102431, December 1989.

We consider the effects of strong critical layer nonlinearity on the spatial evolution of an initially linear "acoustic mode" instability wave on a hypersonic flat plate boundary layer. Our analysis shows that nonlinearity, which is initially confined to a thin layer, first becomes important when the amplitude of the pressure fluctuations become $O(1/M^4 \ln M^2)$, where M is the free stream Mach number. The flow outside the critical layer is still determined by linear dynamics and therefore takes the form of a linear instability wave--but with its amplitude completely determined by the flow within the critical layer. The latter flow is determined by a coupled set of nonlinear equations, which we had to solve numerically.

Akl, Fred A. (ICOMP), and Morel, Michael R. (Ohio University): "Eigensolution of Finite Element Problems in a Completely Connected Parallel Architecture," ICOMP Report No. 89-31, NASA TM-102450, November 1989.

The paper presents a parallel algorithm for the solution of the generalized eigenproblem in linear elastic finite element analysis $[K][\Phi] = [M][\Phi][\Omega]$, where: $[K]$ and $[M]$ are of order N , and $[\Omega]$ is of order q . The parallel algorithm is based on a completely connected parallel architecture in which each processor is allowed to communicate with all other processors. The algorithm has been successfully implemented on a tightly coupled multiple-instruction-multiple-data (MIMD) parallel processing computer, Cray X-MP. A finite element model is divided into m domains each of which is assumed to process n elements. Each domain is then assigned to a processor, or to a logical processor (task) if the number of domains exceeds the number of physical processors. The macro-tasking library routines are used in mapping each domain to a user task. Computational speed-up and efficiency are used to determine the effectiveness of the algorithm. The effect of the number of domains, the number of degrees-of-freedom located along the global fronts and the dimension of the subspace on the performance of the algorithm are investigated. For a 64-element rectangular plate, speed-ups of 1.86, 3.13, 3.18 and 3.61 are achieved on two, four, six and eight processors, respectively.

Hamed, Awatef (ICOMP): "Probabilistic Modeling for Simulation of Aerodynamic Uncertainties in Propulsion Systems," ICOMP Report No. 89-32, NASA TM-102472, December 1989.

The focus of this work was to explore the numerical simulation of the probabilistic aerothermodynamic response of propulsion system components to randomness in their environment. The reusable rocket engine turbopumps were selected as an example because of the severe cryogenic environment in which they operate. The thermal and combustion instabilities, coupled with the engine thrust requirements from start up to shut down, lead to randomness in the flow variables and uncertainties in the aerodynamic loading. The probabilistic modeling of the turbopumps aerodynamic response was accomplished using the panel method coupled with Fast Probability Integration methods. The aerodynamic response in the form of probabilistic rotor blades and splitter loading were predicted and the results presented for specified flow coefficient and rotor preswirl variance. Possible future applications of the aerothermodynamic probabilistic modeling in engine transient simulation, condition monitoring and engine life prediction are briefly discussed.

SEMINARS

Dare Afolabi, Purdue University: "Qualitative Characteristics of Mistuned Cyclic Systems"

The influence of mistuning on the forced response curves of cyclic systems is discussed. Using recent developments in the theory of singularities of differentiable maps, it is shown that the dynamic behavior of cyclic systems exhibit "geometric instability." Thus, mistuning due to small parameter variations has the potential to induce considerable changes in the topology of forced response curves, although this can only happen when the mistuning leads to a crossing of the bifurcation set. The practical implication of the foregoing for bladed disk and propfan response is discussed.

F. Akl, Ohio University and M. Morel, Sverdrup: "Eigensolution of Finite Element Problems in a Completely Connected Parallel Architecture"

This seminar presents a parallel algorithm for the solution of the generalized eigenproblem in linear elastic finite element analysis, $[K][\Phi] = [M][\Phi][\Omega]$, where: $[K]$ and $[M]$ are of order N , and $[\Omega]$ is of order q . The parallel algorithm is based on a completely connected parallel architecture in which each processor is allowed to communicate with all other processors. The algorithm has been successfully implemented on a tightly coupled multiple-instruction-multiple-data (MIMD) parallel processing computer, Cray X-MP. A finite element model is divided into m domains each of which is assumed to process n elements. Each domain is then assigned to a processor, or to a logical processor (task) if the number of domains exceeds the number of physical processors. The macrotasking library routines are used in mapping each domain to a user task. Computational speed-up and efficiency are used to determine the effectiveness of the algorithm. The effect of the number of domains, the number of degrees-of-freedom located along the global fronts and the dimension of the subspace on the performance of the algorithm are investigated. For a 64-element rectangular plate, speed ups of 1.86, 3.13, 3.18 and 3.61 are achieved on two, four, six and eight processors, respectively.

Farid Amirouche, University of Illinois at Chicago: "Dynamics and Control of Multibody Systems"

Professor Amirouche will speak on the simulation and analysis of large structure dynamics. With the advent of computer hardware and software, it is becoming apparent that much is needed to be investigated before the new technology of high speed articulated structures, more efficient and intelligent systems is to be developed. Algorithms for simulation of complex mechanical systems require the automatic inclusion of geometric stiffening and complete nonlinear interactions between the elastic and rigid body motion, as well as being expressed in minimum dimension form and applicable at singular configurations.

A recursive formulation of the equations of motion for general constrained flexible multibody systems will be presented using finite element, Kane's equations and modal analysis techniques. The elastic body deformation and the

rigid body motion are solved simultaneously with the complete nonlinear interactions between them. Closed loops and prescribed motions will be included into the system via constraint equations. A coordinate reduction technique called the PUTD will be introduced. Numerical examples of robotic manipulators at spin-up and deployment motions with rigid and flexible links will be presented.

A discussion on the computational aspects of the dynamic simulation of complex mechanical systems will also be addressed.

Alvin Bayliss, Northwestern University: "Numerical Study of Unsteady Behavior in Combustion"

We present a numerical study of unsteady solutions for problems in both gaseous and solid fuel combustion. Examples of periodic, quasi-periodic and chaotic solutions will be presented for various parameter regimes. These solutions will be obtained by utilizing an adaptive pseudo-spectral method in which the problem is dynamically transformed to one in which the solution is more slowly varying. A two-dimensional version of this method will be described. A family of mappings appropriate for the computation of problems with rapid variation will be discussed and shown to be very effective in resolving the rapid spatial variations occurring in combustion problems.

Richard J. Bodonyi, Ohio State University: "Boundary Layer Receptivity"

A numerical study of the generation of Tollmien-Schlichting waves due to the interaction between a small freestream disturbance and a small localized variation of the surface geometry has been carried out using both finite difference and spectral methods. The nonlinear steady flow is of the viscous-inviscid interactive type while the unsteady disturbed flow is assumed to be governed by the Navier-Stokes equations linearized about this flow. Numerical solutions illustrate the growth or decay of the T-S waves generated by the interaction between the freestream disturbance and the surface distortion, depending on the value of the scaled Strouhal number. An important result of this receptivity problem is the numerical determination of the amplitude of the Tollmien-Schlichting waves.

C. J. Chapman, University of Cambridge, England: "Diffraction Patterns in the Acoustic Field of a Propeller or What Does the Near Field Really Look Like?"

Recent evidence suggests that the near field of a supersonic propeller may be strongly influenced by diffraction patterns, i.e., complex structures of peaks and troughs in the variation of pressure amplitude with position. A method will be presented for obtaining simple analytical formulae which determine these patterns, using powerful techniques developed originally for optics.

The prospect arises of easy analytical determination of those aspects of the sound field which, because of excessive cancellation, do not lend themselves to numerical calculation by computer codes. In the talk, a summary will be given of results already obtained by the diffraction method, and the scope for its future development will be assessed.

A. O. Demuren, University of Lagos: "Turbulence Modeling in Passages with Arbitrary Geometry"

Realistic prediction of turbulent flow and heat transfer in passages with complex geometries are possible only through the use of non-isotropic turbulence models. The more conventional models based on the Prandtl-Kolmogorov eddy viscosity concept fail to simulate important features such as the generation of streamwise vorticity, redistribution of wall shear stresses and heat transfer and anisotropy of the normal stresses. A procedure based on the calculation of the individual Reynolds stresses is presented for three dimensional flow in passages with arbitrary geometry. When the mean strain rates are moderate, simplified variants yield accurate results. Otherwise the full equations must be solved.

Peter W. Duck, University of Manchester: "The Inviscid Stability of Supersonic Flows Past Axisymmetric Bodies"

The supersonic flow (1) past a thin straight circular cylinder and (2) past a sharp cone is investigated. In both cases the associated boundary layer flow (i.e., velocity and temperature field) is computed, and compared with asymptotic solutions developed for the far downstream flow.

The inviscid, linear axisymmetric and nonaxisymmetric (temporal) stability of this general class of boundary layer is also studied. A so-called "triply generalized" inflexion condition is derived, which is a condition for so-called "subsonic" neutral modes. The eigenvalue problem (for the complex wavespeed) for the cylinder and cone flow is computed for a freestream Mach number of 3.8. These calculations reveal that in the case of axisymmetric modes, curvature has a profound stabilizing effect on the flow. In the case of non-axisymmetric modes, a mode of instability peculiar to flows with lateral curvature is found to be possible, and this is also investigated asymptotically.

Peter Eiseman, Columbia University: "Adaptive Grid Generation"

An informal seminar on adaptive grid generation will be given to start a sequence on that topic. The pace of development will be determined by the audience. The discussion will start with the basic elements of adaptivity. This will entail the fundamental aspects of extracting and preparing solution data for the purpose of grid adjustments. The techniques for such adjustments will be given first in one spatial dimension and then in higher dimensions.

John W. Elliott, Iowa State University: "On Structure of Three-Dimensional Flow: Viscous-Inviscid Interactions"

Three-dimensional steady viscous-inviscid interactions are examined first for compressible flows on the triple-deck scales and then on shorter spanwise length scales. Three-dimensional stall cells are found to be possible over a wide range of spanwise wavelengths although for subsonic flows there is a definite termination of the interaction at a critical wavelength. For weakly three-dimensional supersonic flow, the two-dimensional free-interaction is

recovered but in a form that can still admit a fully three-dimensional supersonic flow, the two-dimensional free-interaction is recovered but in a form that can still admit a fully three-dimensional separation structure. For small spanwise wavelengths, in contrast, both supersonic and subsonic problems are found to reduce to the case of a prescribed zero-displacement interaction as in the flow in rigid tubes. Numerical solutions are presented for both large and small spanwise wavelengths. At very small spanwise wavelengths we firstly recover the three-dimensional equivalent of Lighthill's (1953) investigation where the spanwise lengthscale is of the order of the boundary-layer thickness, and then on even shorter scales we are presented with a full Navier-Stokes problem to solve. For both of these cases stall cells are found to be possible.

The unsteady incompressible counterpart of the Lighthill Euler region is a small-amplitude version of the three-dimensional equivalent of Smith and Burggraf's Stage 3. A whole variety of large amplitude structures based on this form are examined which have both regular and singular behavior as the viscous wall-layer is approached. These solutions are compared with the Reynolds-number dependence of the turbulent skin-friction and it turns out that one type correlates to a high degree of accuracy. In addition asymptotic structures for disturbances outside of the boundary-layer, so called hyper-boundary layer critical layers, are examined.

Nessan Fitzmaurice, Case Western Reserve University: "The Lyapunov Spectrum of the Kolmogorov Flow"

Classically it was held that solutions to deterministic partial differential equations (i.e., ones with smooth coefficients and boundary data) only became random through one mechanism, namely by the activation of more and more of the infinite number of degrees of freedom that is available to a such system.

It is only recently that researchers have come to suspect that many infinite dimensional nonlinear systems may in fact possess finite dimensional chaotic attractors. Lyapunov exponents provide a tool for probing the nature of these attractors. We have implemented the tangent space method for calculating the Lyapunov spectrum for incompressible fluid flows. The implementation of the method will be discussed and results presented for flows driven by a simple, spatially periodic, forcing term--the Kolmogorov forcing. We will also briefly discuss recent work applying the orthogonal decomposition to these flows.

J. S. B. Gajjar, Exeter University: "Amplitude-Dependent Neutral Modes in Compressible Boundary-Layer Flows"

The amplitude-dependent nonlinear stability properties of compressible boundary layers are investigated theoretically for large Reynolds numbers. The disturbance amplitude taken is sufficiently large so that the flow structure takes on a form similar to that occurring in incompressible flow with a strongly nonlinear critical layer present. However because of the large density and temperature variations across the critical layer, in addition to the large transverse pressure gradients present, there are significant differences from the incompressible theory. We show how the ideas of Benney-Bergeron

(1969) and Davies (1969) on strongly nonlinear critical layers can be generalized to compressible flow. Several results for the nonlinear neutral wavelengths and wavespeeds for different Mach numbers and for different flow conditions will be presented. In addition we will discuss the limiting structures for increased disturbance amplitudes and for large Mach numbers.

Karl Gustafson, University of Colorado: "Vortex Dynamics in Aerodynamic Flows"

A numerical technique for the study of vortex dynamics in unsteady flows past airfoils has been formulated. This investigation was motivated by recent experimental work at the University of Colorado employing flow visualization techniques to give first views of intrinsic vortex events such as splitting and shredding. Our numerical method employs a grid generation technique which maps an infinite physical domain to a finite auxiliary domain and then into a computational cavity domain in which the full Navier-Stokes equations are solved. The solution procedure employs both multigrid and ADI methods.

Applications include constant accelerating flow from rest around a NACA 0015 airfoil. Comparisons of the numerical solutions to experimental studies show a remarkable resolution of such interesting flow phenomena such as vortex splitting, shredding, and higher order vortical detail. Splitting of vortices allows vorticity which would have advected downstream to return to the proximity of the airfoil, terminating the loss of lift associated with the stalling of the airfoil.

Thomas Hagstrom, SUNY at Stony Brook: "Simulating Time Dependent Compressible Flows: Some Problems and Results"

Recent years have seen significant advances in the theory and application of computational fluid dynamics to problems in aerospace engineering. The bulk of this development, however, has been concentrated on the simulation of steady-state or time averaged flows. Certain applications, for example to problems in aeroacoustics, are fundamentally time dependent and will require time accurate computations. Prominent among the difficulties are:

- Numerical radiation boundary conditions: Accurate conditions in the far field are necessary to eliminate the propagation of spurious waves into the near field, to evaluate the acoustic radiation produced by the given flow and, in receptivity analyses, for the precise imposition of incoming waves.
- Discretization schemes: High order discretizations are known to be most efficient for the study of wave propagation, while low order methods are often used to avoid oscillations at discontinuities. What methods will give the best results for multidimensional problems involving both complex wave interactions and shocks?

In this talk we will discuss some of these difficulties, emphasizing our work on asymptotic boundary conditions for the Euler equations. Simulations of compressible, inviscid, transonic flow past an impulsively started cylinder will be presented.

Donald Frederick Hawken, University of Toronto Institute for Aerospace Studies:
"Adaptive Node Movement in Finite-Element and Finite-Difference Solutions of Partial Differential Equations with Applications to Gasdynamics"

This thesis presents an in-depth study of two adaptive numerical methods of solving partial differential equations. These methods involve the adaptive movement of nodes so as to obtain a low level of solution of nonstationary flow problems that contain moving regions of rapid change in the flow variables, surrounded by regions of relatively smooth variation (shock waves, contact surfaces, slip streams, phase-change interfaces, boundary layers). Significant economies of execution can be attained if nodes are moved so that they remain concentrated in regions of rapid variation of the flow variables.

The Moving Finite Element (MFE) method, originally introduced by Miller and Miller, has been implemented and is compared to the new Moving Finite Difference (MFD) method developed in this thesis. Both methods solve simultaneously for the flow variables and the node locations at each time-step. These adaptive methods move the nodes so as to minimize an 'error' measure that contains a function of the time derivatives of the solution. This error measure is manipulated to obtain a matrix equation for node velocities. Both methods make use of penalty functions to prevent node crossing. The penalty functions result in extra terms in the matrix equations that promote node repulsion by becoming large when node separation becomes small.

Extensive work applying the MFE and MFD methods to one-dimensional gasdynamic problems has been performed. The test problems include Burgers' equation, ideal viscous planar flow within a shock-tube, analogous spherically symmetric flow in an explosion implosion process, the propagation of shock and rarefaction waves through area changes in ducts, and viscous transition through a contact surface and a shock. The MFD method is typically twice as fast as the MFE method, and the solution is comparable or better in quality.

Sang-Wook Kim, University of Texas, Arlington: "Numerical Investigation of an Internal Layer in Turbulent Flow over a Curved Hill"

Turbulent shear layers over curved surfaces are highly sensitive even to a small amount of streamline curvature (Bradshaw 1969 & 1973). Bradshaw (1969) proposed a curvature correction method based on the assumption that such turbulent flows can be characterized by a "curvature parameter." Application of the method to a number of turbulent flows with streamline curvature revealed that the curvature correction parameter needs to be adjusted widely to obtain physically meaningful results. Thus the validity of the method has been questioned in the open literature. Development of an internal layer in a turbulent boundary layer flow over a curved hill and development of a turbulent boundary layer in a turbulent flow over a symmetric airfoil are investigated numerically. The turbulence is described by a multiple-time-scale model. The turbulent flow equations are solved by a control-volume based finite difference method. Computational results for the mean flow field (pressure distribution on the walls, wall shearing stresses, and mean velocity profiles), the turbulence structure (turbulent kinetic energies and the Reynolds stress profiles), and the integral parameters (displacement and momentum thicknesses) compared favorably with the measured data. The pressure distributions on the walls were almost the same for both flows. Computational results show that the internal layer is a new

boundary layer which is developed beneath the external boundary layer and is located very close to the wall. The turbulence intensity of the internal layer is bigger than that of the external boundary layer. Development of such an intense turbulence field and formation of the internal layer is attributed to the enormous mean flow strain rates caused by the streamline curvature and the strong pressure gradient. The internal layer of the curved hill and the boundary layer of the airfoil were almost the same. Development of the wall shearing stress and separation of the boundary layer at the rear end of the curved hill mostly depend on the internal layer and is slightly influenced by the external boundary layer flow. The results suggest that any turbulence model incorporating curvature correction methods or wall function methods may not be able to describe the turbulent flow over the curved hill adequately due to the complex turbulence structure of the internal layer which is located very close to the wall.

B. P. Leonard, University of Akron: "Vectorial Approach to Nonoscillatory Generalization of Upwinding for Accurate Resolution of Discontinuities"

Multidimensional algorithms are constructed for the convection-diffusion equation under high-Peclet-number conditions. The convective component is based on explicit vector transient interpolation modelling: time-evolution is converted into a multidimensional spatial interpolation problem at the earlier time level, written in conservative CV form. Consistent treatment of diffusion terms uses a multidimensional Taylor expansion of the complex amplitude ratio (amplification factor). One-dimensional algorithms are thereby consistently generalized to two and three dimensions. The nonoscillatory ULTIMATE strategy is applied at each CV face. High resolution of discontinuities (without distorting smooth profiles) is achieved in a cost-effective manner, using adaptive stencil expansion--from the base multidimensional 3rd-order scheme to 5th-, 7th, or 9th-order upwinding, locally. Narrow peak resolution is maintained, using an automatic discriminator. Two-dimensional results are shown to be essentially the same as their one-dimensional counterparts. Inadequacies of so-called "high-resolution" TVD schemes demonstrate the need for higher order upwinding, as proposed here.

Christophe Pierre, The University of Michigan: "Localization Phenomena in Mistuned Blade Assemblies"

An investigation of the effects of small unavoidable blade-to-blade dissimilarities, or mistuning, on the dynamics on blade assemblies is presented. The system studied may be regarded as a simple model of a continuously shrouded blade assembly accounting for one or more structural modes per blade. Since mistuning is usually known in a statistical fashion only, a probabilistic approach is chosen whenever possible.

The main result of the study is that the sensitivity to mistuning depends primarily upon the ratio of mistuning strength to interblade coupling strength. Specifically, weakly coupled blade assemblies are shown to be highly sensitive to small mistuning through the strong localization of the free vibration modes to a small geometric region of the assembly, a significant increase in the

forced response amplitude, and the localization of the forced response amplitude pattern. Such localized vibrations are believed to be very damaging, as they lead to localized thus large stresses and possibly single blade failure.

The study also discussed the relative effects of mistuning on the various groups of modes by using a multi-component mode number through the severe localization of the mode shapes. It is shown that strong localization is unavoidable if the mode number is large enough, even for strong interblade coupling.

Kenneth Powell, The University of Michigan: "Design of Optimally Smoothing Multi-Stage Schemes for the Euler Equations"

A method is developed for designing multi-stage schemes that give optimal damping of high-frequencies for a given spatial-differencing operator. The objective of the method is to design schemes that combine well with multi-grid acceleration. The schemes are tested on a nonlinear scalar equation, and compared to Runge-Kutta schemes with the maximum stable time-step. The optimally smoothing schemes perform better than the Runge-Kutta schemes, even on a single grid. The analysis is extended to the Euler equations in one space-dimension by use of "characteristic time-stepping," which preconditions the equations, removing stiffness due to variations among characteristic speeds. Convergence rates independent of the number of cells in the finest grid are achieved for transonic flow with and without a shock. Characteristic time-stepping is shown to be preferable to local time-stepping, although use of the optimally damping schemes appears to enhance the performance of local time-stepping.

Tsan-Hsing Shih, Stanford University: "Modeling of Near-Wall Turbulence"

Many practical closure models (including two-equation models and second-order closure models) use the wall function to obtain their boundary conditions, rather than deal with a thick, viscous sublayer. However, "the law of the wall," and wall functions are based on the assumption of local equilibrium which is not always the case. For example, flows with separation, reattachment, body forces, strong secondary flows or streamwise pressure gradient can cause the behavior of the near-wall sublayer to depart from "the law of the wall." To solve these problems, the modeled turbulence equations must be carried out in the sublayer in order to capture the non-equilibrium characteristics of the near wall region. However, the near-wall behavior of the current closure models has not been correctly captured. Predictions of the normal stresses near the wall are quite poor.

The objection of this work is to develop better models (including two-equation model and second-order closure model) for the near-wall turbulence, using direct numerical simulation and existing methodologies. The models are tested using data from direct simulations, experiments and analysis. Specifically, we have done the following things:

- 1) Examine the performance of the existing two-equation eddy viscosity models and develop better models for the near-wall turbulence using direct numerical simulations of plane channel flow.

2) Using analysis of the near-wall asymptotic behavior of turbulence, examine the problems of the current second-order closure models and develop new models with the correct near-wall behavior using the direct numerical simulation data.

Avram Sidi, Technion-Israel Institute of Technology: "A Generalized Power Method and its Analysis for Normal Operators"

The power method is used in approximating the largest eigenvalue of a matrix. We present a generalized power method for obtaining approximations to several large eigenvalues. Like the power method, this new method does not require the matrix to be stored. All that it requires is a subroutine for performing matrix-vector multiplications. It is related to a convergence acceleration method for vector sequences, and can also be used in assessing the spectral properties of the Jacobian matrix in any iterative process. We will present results pertaining to the convergence rates of the different approximations to the corresponding eigenvalues. When the matrix is normal (e.g., real symmetric) these convergence rates are doubled. This new method has distinct advantages over the well-known subspace iteration method that is commonly used in Structures. We will discuss these advantages in detail.

Patrick Smolinski, University of Pittsburgh: "Multi-Time Step Integration Using Nodal Partitioning"

An algorithm is presented for first order finite element equations which integrates different nodal groups in the mesh with different time steps. Since the nodal groups are updated independently, no unsymmetric systems need be solved. Stability of the method is demonstrated by showing that a norm of the solution decreases after every update if the time step is less than a critical value. The element eigenvalue inequality theorem is used to give the critical time step in terms of element eigenvalues. Several numerical examples will be presented to examine the accuracy of the method.

Eli Turkel, Tel Aviv University: "Runge-Kutta Schemes for Subsonic, Transonic and Supersonic Speeds"

We discuss some of the recent improvements to the central difference code for Euler and Navier-Stokes equations. In order to reach a steady state it is necessary to add an artificial viscosity. However, for more difficult cases the standard artificial viscosity introduces too much smearing. To reduce this we introduce a matrix valued artificial viscosity. This improves the accuracy of the code. To further improve the robustness of the algorithm a preconditioning is introduced for slow subsonic or supersonic flows. These methods can be viewed as introducing different time steps for different waves.

Eli Turkel, Tel Aviv University: "Improving the Accuracy and Convergence Rates of Central Difference Schemes"

Central difference schemes to the fluid dynamic equations require an artificial viscosity in order to converge to a steady state. This artificial viscosity serves two purposes. One is to suppress high frequency noise which is not damped by the central differences. The second purpose is to introduce entropy-like conditions so that a shock can be properly captured. These viscosities need a coefficient to measure the amount of viscosity to be added. In the standard scheme, a scalar coefficient is used based on the spectral radius of the Jacobian of the convective fluxes. However, this adds too much viscosity to the slower waves. Hence, we suggest a matrix viscosity. We also discuss the necessary switches and connections with TVD. Examples of inviscid and viscous problems in two- and three-dimensional will be presented. We further discuss the use of multigrid to accelerate the convergence of the solution especially with regard to hypersonic flow.

William J. Usab, Jr., Purdue University: "Formulation of Boundary Conditions for Multi-Grid Acceleration of the Euler and Navier-Stokes Equations"

Multi-grid acceleration is currently a widely used, and very efficient, technique for accelerating the convergence of Euler and Navier-Stokes solvers to steady state. While clearly important, to date very little discussion has been made in the literature about the proper formulation of boundary conditions on the coarse mesh levels of the multi-grid acceleration schemes for these types of problems. In the present research, multi-grid acceleration is studied in terms of the filtering process which takes place as the fine mesh solution is transferred from one mesh to the next. By clearly defining this process both within the domain and at the domain boundaries, it is then possible to formulate boundary conditions which are consistent with the filtered (or transferred) solution.

Boundary conditions for the Euler and Navier-Stokes equations have been formulated for Jameson's 4-Stage Runge-Kutta scheme with FAS multi-grid acceleration. These new boundary conditions include a forcing term, which for viscous flows amounts to a partial slip condition at solid walls. The slip velocity in this formulation is a function of the length scales that can be resolved on each given mesh and is determined by the transferred solution at the domain boundary. In this talk the formulation of these boundary conditions will be presented, followed by solutions for a range of inviscid and viscous model problems.

J. D. A. Walker, Lehigh University: "Dynamical Features of Turbulent Wall Layers"

The dynamics of the near wall region of a turbulent flow are considered in the large Reynolds number limit. When low-speed streaks are present, the governing equations are shown to be of the boundary-layer type. Physical processes leading to local breakdown and bursting are considered. It is argued that convected vortices, predominantly of the hairpin type, will provoke eruptions and regenerative interactions with the outer region. The bursting process generally involves focussing instabilities, in which the flow erupts along

a progressively narrowing band in the streamwise direction. It is not possible to capture this phenomena using conventional numerical algorithms or current direct simulation methods. Recently developed Lagrangian methods are discussed as well as some current experimental work. It is shown that similar phenomena also occur in the transition zone.

A. T. Degani and J. D. A. Walker, Lehigh University: "Structure of a 3D Turbulent Boundary Layer--Asymptotic Theory and Its Application"

A self-consistent asymptotic analysis of a three-dimensional boundary layer has been carried out in the limit of large Reynolds number. It is shown that a complex, two-layered structure exists in which two terms in the asymptotic expansion are needed in the outer layer to produce useful results. The asymptotic forms of the streamwise and cross-stream velocities in the matching regions have been obtained, and algebraic formulae to determine the wall shear stress, skew angle at the wall, and the Stanton number have been derived. Some important features of the structure of a three-dimensional turbulent boundary layer are described and discussed.

A wall-function algorithm has been devised based on the results of the asymptotic analysis. In this method, the velocity and enthalpy in the inner wall layer are represented by analytical functions of the form of the law-of-the-wall profiles. The effectively inviscid flow in the outer layer of the boundary layer is then solved for numerically with the no-slip and heat-transfer condition at the wall being replaced with the requirement that the numerical solution asymptote in a manner consistent with the boundary-layer-flow structure. It is shown that a 40 percent reduction in the total number of mesh points is possible with no degradation in accuracy. Results are presented for a typical flow configuration.

Yau Shu Wong, University of Alberta: "Preconditioned Gradient Algorithms for Large Systems of Equations in CFD Applications"

In many computational fluid dynamics (CFD) codes, the resulting systems of algebraic equations are usually large and sparse. Solving the large system of equations is frequently the most time consuming part of the entire computer simulation program. A fully implicit numerical procedure has been investigated for a three-dimensional transonic flow around an isolated wing, in which the governing partial differential equation is of the mixed elliptic-hyperbolic type. The resulting large set of finite-difference equations is solved by a gradient type algorithm used in conjunction with an incomplete LU factorization or a polynomial preconditioning technique. Not only does the proposed iterative scheme provide a fast convergence rate, but it could also be implemented efficiently on a vector computer. Numerical results and comparison with a three-dimensional simulation code developed by Boppe (NASA CR-3243) will be presented.

Akira Yoshizawa, Institute of Industrial Science, University of Tokyo: "Turbulence Modeling Based on a Statistical Theoretical Method"

A method of deriving current incompressible turbulence models and constructing new models with the aid of a two-scale direct-interaction (line-renormalization) approximation (TSDIA) is presented. Specifically, the following points are discussed.

- 1) Outline of the TSDIA
- 2) Generalization of the eddy-viscosity representation for the Reynolds stress
- 3) Modeling of scalar diffusion in shear turbulence
- 4) Derivation of dissipation-rate equations

Akira Yoshizawa, Institute of Industrial Science, University of Tokyo: "Compressible Turbulence Modeling of Three-Equation Type"

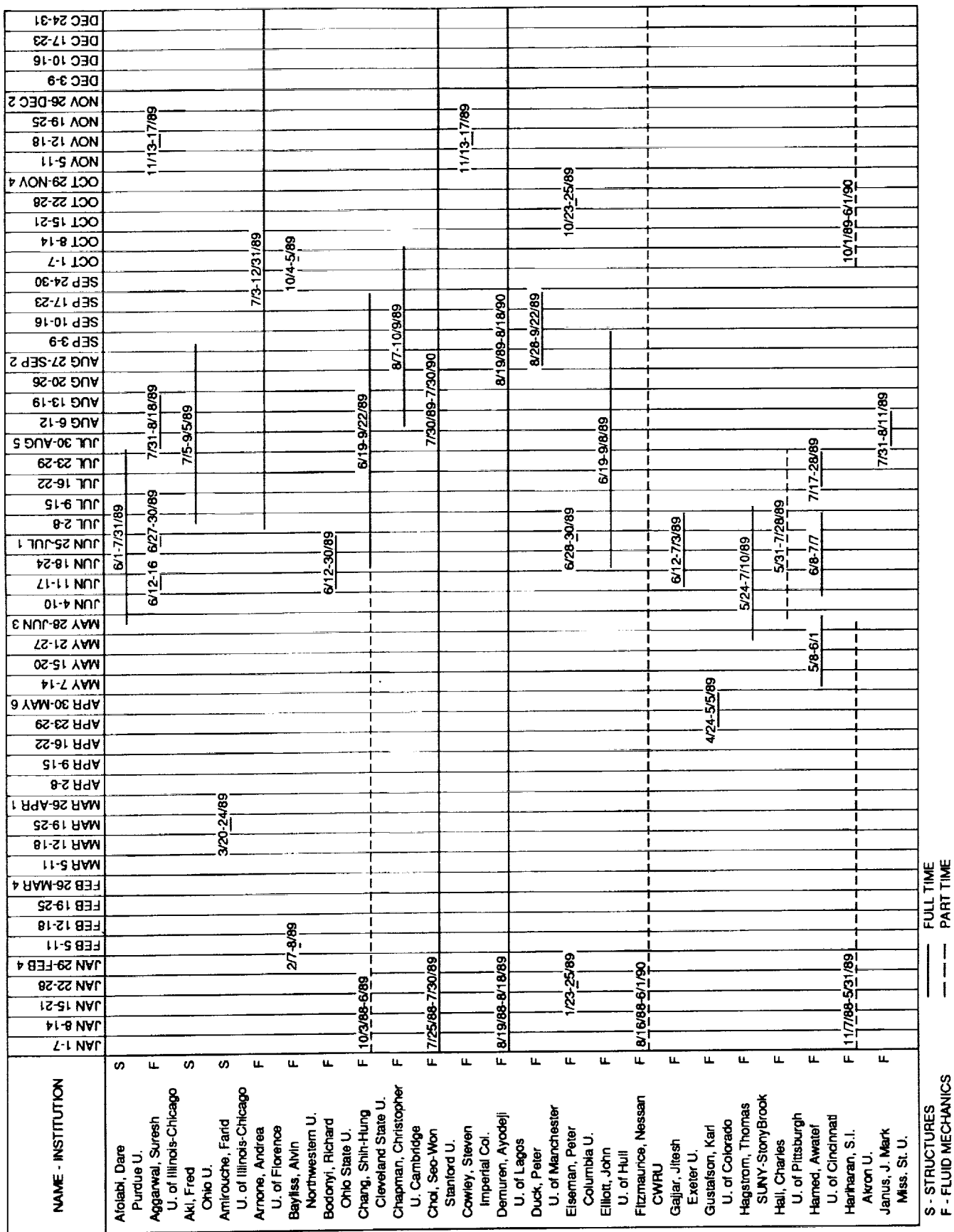
A compressible turbulence model is presented to investigate high-speed flows with shocks. This model consists of three equations for the intensity of momentum fluctuation, its dissipation rate, and the intensity of density fluctuation, besides the three conservation laws for mass, momentum, and internal energy. The Favre mean procedure cannot be applied straightforwardly to statistical theoretical methods, but the present choice of turbulence statistics retains the merit of the Favre mean.

WORKSHOPS

A workshop entitled "Lifetime Prediction of Flawed Structures," organized by Ted Belytschko, Northwestern University, Thomas Cruse, Southwest Research Institute and Lester Nichols, NASA Lewis, was held on May 16-17, 1989 at NASA Lewis Research Center. There were 57 attendees from 23 different organizations. The workshop was convened to discuss the benefit and level of accuracy of computational lifetime prediction methodology. Specific questions were: How close are we to achieving this kind of capability, what are the major obstacles, what are basic research needs, and can we design simple experiments to verify the methods?

The program for this workshop is shown in figure 5. There were 14 presentations covering research in computational methods, life expectancy, and experimental results. In addition there was some discussion on the requirements placed upon structural analysis and material behavior methods in the context of a reliability-based design system.

The workshop discussion generally concluded that there is benefit of computational lifetime prediction. However, there was divided opinion upon how close we are to achieving it. The division is a question of how accurate and how inclusive are the present methods compared to how accurate and inclusive one needs. With respect to major obstacles, there was considerable discussion as to how accurately one needs to know flaw growth behavior in order to have sufficiently accurate lifetime prediction. The disagreement centered around which point of view should prevail: the local (i.e. flaw) scale or the global (i.e., structure) scale. Clearly, further research in this area is required. A second point of discussion centered upon whether or not random processes are included in the present random variable probabilistic methodology. Again, it was agreed that further research is required. With respect to experimental verification we can design experiments not only to verify the computations, but also to guide development of the methods. However, the experiments must be defined with an eye toward both the needs of the model developer as well as the ability of the experimentalist to carry out the experiment with known accuracy.



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



Figure 2. - ICOMP Steering Committee and visiting researchers in June 1989. 1st row seated (left to right): Charles Feiler, Frances Pipak, Awatef Hamed, Isaac Greber, Richard Bodonyi, Suresh Aggarwal. 2nd row: Marvin Goldstein, Jitesh Gajjar, Reda Mankbadi, Eli Reshotko, John Elliott, S. I. Hariharan, Vittorio Michelassi, Lala Krishna, Subodh Mital. 3rd row: Louis Povinelli, Ayodeji Demuren, Patrick Smolinski, Thomas Hagstrom, Sang-Wook Kim, Dare Afolabi. 4th row: Robert Mullen, Lester Nichols, Seo-Won Choi, William Usab, Peter Eiseman, Yau Shu Wong, Bo-nan Jiang, Neal Saunders.

UNIVERSITY OR INSTITUTION	NUMBER
1 ALBERTA	1
2 AKRON	3
3 BROWN	1
4 CAIRO	1
5 CAMBRIDGE	1
6 CASE WESTERN RESERVE	2
7 CINCINNATI	1
8 CLEVELAND STATE	1
9 COLORADO	1
10 COLUMBIA	1
11 EXETER	1
12 FLORENCE	1
13 HULL	1
14 ILLINOIS-CHICAGO	2
15 IMPERIAL COLLEGE	1
16 LAGOS	1
17 LEHIGH	1
18 MANCHESTER	1
19 MICHIGAN	3
20 MISSISSIPPI STATE	3
21 NORTHWESTERN	1
22 OHIO	1
23 OHIO STATE	1
24 PITTSBURGH	3
25 PURDUE	2
26 STANFORD	1
27 SUNY-STONY BROOK	1
28 TECHNION-ISRAEL INSTITUTE OF TECH.	1
29 TEL-AVIV	1
30 TEMPLE	1
31 TEXAS-ARLINGTON	1
32 TEXAS-AUSTIN	1
33 TOKYO	1
34 WASHINGTON	1
35 WAYNE STATE	1
	<hr/>
	46

Figure 3. - Composition of 1989 ICOMP staff - organizations represented.

	<u>1986</u>	<u>1987</u>	<u>1988</u>	<u>1989</u>
PEOPLE	23	43	50	46
SEMINARS	10	27	39	30
REPORTS	2	9	22	32
WORKSHOPS	1	0	2	1
PRESENTATIONS	7	0	21	14
UNIVERSITIES OR ORGANIZATIONS	20	28	43	35

Figure 4. - ICOMP statistics (1986 to 1989).

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION

ICOMP WORKSHOP
ON
LIFETIME PREDICTION OF FLAWED STRUCTURES
MAY 16th and 17th, 1989
AD BLDG. AUDITORIUM

AGENDA

Tuesday, May 16

9:00 a.m.	Welcome and Opening Remarks Workshop Process and Goals An Overview of Reliable Lifetime Prediction Issues for Gas Turbine Engines Evolving Probabilistic Design Approach and Methodology at Rocketdyne Failure Probability and Risk Assessment for Select Aerospace Components Design of Ceramic Components with the NASA/ CARES Computer Program	L. Nichols T. Belytschko T. Cruse J. Newell M. Shiao J. Gyekenyesi
12:30 p.m.	LUNCH	
1:30 p.m.	Probabilistic Approach to Fatigue of Composite Materials Mechanistic and Probabilistic Aspects of Composite Stress-Life Prediction Computational Simulation of Structural Fractures in Composites	P. Spanos E. Wu C. Chamis
5:30 p.m.	Adjourn, Social Hour	

Wednesday, May 17

9:00 a.m.	Multiaxial Fatigue Under Complex Loading Micromechanics of Fatigue Crack Growth A Stochastic Approach for Modeling Fatigue Crack Growth Lifetime Prediction of Metal Matrix Composites Based on a Random Process Model for Cumulative Damage	D. Socie D. Davidson W. Spencer L. Boyce
12:00 p.m.	LUNCH Probabilistic Finite Elements Hierarchical Poly Tree Type Local-Global Analysis of Fracture in Engineering Structure	T. Belytschko J. Padovan
1:00 p.m.	Discussion, Wrap-up	
4:00 p.m.	Adjourn	

Figure 5. - Agenda for ICOMP workshop on Lifetime Prediction
of Flawed Structures.



National Aeronautics and
Space Administration

Report Documentation Page

1. Report No. NASA TM-102519 ICOMP-90-01		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle Institute for Computational Mechanics in Propulsion (ICOMP) Fourth Annual Report—1989				5. Report Date March 1990	
				6. Performing Organization Code	
7. Author(s)				8. Performing Organization Report No. E-5323	
				10. Work Unit No. 505-62-21	
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191				11. Contract or Grant No.	
				13. Type of Report and Period Covered Technical Memorandum	
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546-0001				14. Sponsoring Agency Code	
15. Supplementary Notes Report produced by the Steering Committee, Institute for Computational Mechanics in Propulsion, NASA Lewis Research Center (work funded under Space Act Agreement C-99066-G). Space Act Monitor: Louis A. Povinelli.					
16. Abstract The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University 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 1989.					
17. Key Words (Suggested by Author(s)) Numerical analysis Computer science Mathematics			18. Distribution Statement Unclassified—Unlimited Subject Category 64		
19. Security Classif. (of this report) Unclassified		20. Security Classif. (of this page) Unclassified		21. No. of pages 58	22. Price* A04

