Soviet and Japanese Aerospace Literature

Throughout 1989 the *AIAA Journal* will carry selected abstracts on leading research topics from the Soviet aerospace literature and, as space permits, from similar Japanese literature. The topics will be chosen and the abstracts reviewed for pertinency by *AIAA Journal* editors. This month features Computational Fluid Dynamics from the USSR and Japan.

Support for assembling and publishing the selected abstracts has been provided by the Innovative Science and Technology Directorate of the Strategic Defense Initiative Organization (SDIO), with the sponsorship and technical management of the abstract service by the Office of Naval Research (ONR) under ONR Grant N0014-87-6-0137.

Abstracts in this listing have been taken from the semimonthly abstract journal International Aerospace Abstracts (IAA), published by the American Institute of Aeronautics and Astronautics (AIAA) in cooperation with the National Aeronautics and Space Administration (NASA) under Contract No. NASW-4373. Additional material can be obtained through searching the Aerospace Database – available online via DIALOG or NASA RECON.

Paper copies and microfiche of the original documents cited are available from AIAA Library, Technical Information Service, American Institute of Aeronautics and Astronautics, Inc., 555 W. 57th St., New York, NY 10019 (212) 247-6500, ext. 231. Use the "A" number to identify material you want. Please be advised that most of the original documents are in the original language. Direct questions concerning this abstract section of the *AIAA Journal* to John Newbauer, AIAA Administrator, Technical Publications.

Japanese Aerospace Literature
This month: *Computational Fluid Dynamics*

A89-36003 Vorticity field in a cascade model of turbulence. TOHRU NAKANO, *Fluid Dynamics Research* (ISSN 0169-5983), Vol. 3, No. 1-4, Sept. 1988, pp. 327-330. 11 Refs.

A model is developed that is capable to interpret simultaneously the behavior of the spontaneous singularity and the intermittent structure in fully developed turbulence. Assuming that a turbulent flow has a vortexlike coherent structure, the vortical structure in turbulence is presented from a statistical point of view. It is shown that the model is justified in the framework of a cascade model of turbulence.

A89-35968 Vortex motion; Proceedings of the IUTAM Symposium on Fundamental Aspects of Vortex Motion, Tokyo, Japan, Aug. 31-Sept. 4, 1987. HIDENORI HASIMOTO and TSUTOMU KAMBE, *Fluid Dynamics Research* (ISSN 0169-5983), Vol. 3, No. 1-4, Sept. 1988, p. 454. For individual items see A89-35969 to A89-36016.

The conference presents papers on two-dimensional vortices, ring and three-dimensional vortices, reconnection of vortices, vortex breakdown, stability and turbulence, vortex and sound, high-speed flow, and stratified and rotating fluids. Particular attention is given to the elementary aspects of vortex motion, waves on vortex cores, nonlinear analysis for the evolution of vortex sheets, chaos and collapse of a system of point vortices, and the shedding of vorticity from a smooth surface. Other topics include the bifurcation of an elliptic vortex ring, vortex filament motion in terms of Jacobi theta functions, reconnections of vortex filaments, numerical prediction of vortex breakdown, isoenstrophy points and surfaces in turbulent flow and mixing, the vortex pair in a compressible ideal gas, and vortex motions in stratified wake flows.

A88-40743 Numerical analysis of multiple element high lift devices by Navier Stokes equation using implicit TVD finite volume method. EIJI SHIMA, AIAA Applied Aerodynamics Conference, 6th, Williamsburg, VA, June 6-8, 1988, Technical Papers (A88-40701 16-02). Washington, DC, American Institute of Aeronautics and Astronautics, 1988, pp. 399-406. 18 Refs. (AIAA Paper 88-2574).

This paper deals with the analysis of multiple element high lift devices by solving the Navier-Stokes equations using the TVD (Total Variation Diminishing) finite difference method. In order to generate a computational grid around the multiple element airfoils automatically, the grid generator using the elliptic method, in which Poisson equations are by the finite difference method, combined with 2-D panel method is develcped. As to the flow solver, some improvements are added to the TVD scheme to calculate low Mach number flows efficiently. Numerical calculations are carried out for the single slotted flap configuration.

A89-29701 Numerical analysis of a flow induced in a rarefied gas between noncoaxial circular cylinders with different temperatures for the entire range of the Knudsen number. KAZUO AOKI, YOSHIO SONE, and TAKERU YANO, *Physics of Fluids A* (ISSN 0899-8213), Vol. 1, Feb. 1989, pp. 409-419. 46 Refs.

A numerical analysis is presented for a flow induced by the temperature field in a rarefied gas contained between two noncoaxial circular cylinders at rest with different uniform temperatures. The analysis is based on the linearized Boltzmann-Krook-Welander equation with diffuse reflection boundary condition for the entire range of the Knudsen number. It is demonstrated that two symmetric circulating flows are established, the direction of which does not depend on the Knudsen number. The steady behavior of the gas, the force acting on the cylinders, and the energy transmitted to them, for arbitrary values of the Knudsen number, are obtained on the basis of kinetic theory. At a moderate Knudsen number, the magnitude of the force is at its maximum, and the reversal of the force direction takes place at a small Knudsen number.

A89-30943 Numerical simulation of turbulent channel flow at low and high Reynolds numbers. KIYOSI HORIUTI, Transport phenomena in turbulent flows: Theory, experiment, and numerical simulation; Proceedings of the Second International Symposium, Tokyo, Japan, Oct. 1987 (A89-30901 12-34). New York, Hemisphere Publishing Corp., 1988, pp. 743-755. 21 Refs.

Large eddy simulation (LES) is used to study turbulent flow in a straight plane channel. High and low Reynolds numbers are used and computations of passive scalar transport are carried out. Up to 128 x 256 x 129 grid points are used; estimates are made of the numerical constants involved in the k-epsilon model via LES data bases.

A89-30939 A refined numerical procedure for turbulent heat transfer with coarse grids of finite difference calculation. ISAO SATOH and YASUO KUROSAKI, Transport phenomena in turbulent flows: Theory, experiment, and numerical simulation; Proceedings of the Second International Symposium, Tokyo, Japan, Oct. 1987 (A89-30901 12-34). New York, Hemisphere Publishing Corp., 1988, pp. 687-697. 10 Refs.

A refined numerical procedure for predicting turbulent heat transfer using coarse grid nodes in finite difference calculations is proposed. Results obtained with the present method are compared with those obtained with the conventional k-epsilon model with wall functions. It is shown that the present method can be used to calculate turbulent heat transfer both in a thermally developing region and in a region where the turbulence Reynolds number is low.

A89-34743 Three-dimensional turbulent flow analysis in a cleanroom by the finite element method. M. IKEGAWA, C. KATO, and K. TSUZUKI, Advances and applications in computational fluid dynamics; Proceedings of the Symposium, ASME Winter Annual Meeting, Chicago, IL, Nov. 27-Dec. 2, 1988 (A89-34726 14-02). New York, American Society of Mechanical Engineers, 1988, pp. 161-167. 8 Refs.

A new finite element technique for two- and three-dimensional turbulent flow analysis based on the two-equation model of turbulence is presented. A sophisticated time integration scheme with second-order accuracy and a simultaneous iteration method for both pressure and velocity field are proposed to obtain numerically stable and accurate results based on the algorithm. To achieve high-speed computing by the effective use of a supercomputer, extensive code vectorization is performed and high-speed computing about 20 times as fast as the usual code is attained. The method's effectiveness is verified by computing the flow behind an impulsively started circular cylinder and a flow in a cleanroom. Comparison of the computed results with experimental ones shows good agreement.

A89-31350 An implicit time-marching scheme for solving the compressible Navier-Stokes equations. SATORU YAMAMOTO, HISAAKI DAIGUJI, and HIROSHI ISHIGAKI, Computational fluid dynamics; Proceedings of the International Symposium, Sydney, Australia, Aug. 23-27, 1987 (A89-31301 12-34). Amsterdam, North-Holland, 1988, pp. 773-784.
10 Refs.

An implicit time-marching finite-difference scheme for solving the compressible Navier-Stokes equations is presented. The fundamental equations of this scheme are the Navier-Stokes equations of contravariant velocities in general curvilinear coordinates. The numerical method is based on the Beam-Warming (1978) delta-form approximate-factorization scheme, and stabilized by adopting the flux vector splitting and the TVD scheme. In the calculation of turbulent flows, the two-equation k-epsilon turbulence model is used together with the law of the wall. The present scheme is applicable to two-dimensional and three-dimensional steady flows. As numerical examples, some computed results of two-dimensional transonic and supersonic flows are shown.

A89-28955 Reduction of computing time and improvement of convergence stability of the Monte Carlo method applied to radiative heat transfer with variable properties. M. KOBIYAMA, *ASME, Transactions, Journal of Heat Transfer* (ISSN 0022-1481), Vol. 111, Feb. 1989, pp. 135-140.

A modified Monte Carlo method is suggested to reduce the computing time and improve the convergence stability of iterative calculations without losing other excellent features of the conventional Monte Carlo method. In this method, two kinds of radiative bundle are used: energy correcting bundles and property correcting bundles. The energy correcting bundles are used for correcting the radiative energy difference between two successive iterative cycles, and the property correcting bundles are used for correcting the radiative properties. The number of radiative energy bundles emitted from each control element is proportional to the difference in emissive energy between two successive iterative cycles.

A88-48949 A diagonalizing formulation of general real gas-dynamic matrices with a new class of TVD schemes. Y. WADA, S. OGAWA, T. ISHIGURO, and H. KUBOTA, AIAA Paper 88-3596 presented at the AIAA, ASME, SIAM, and APS 1st National Fluid Dynamics Congress, Cincinnati, OH, July 25-28, 1988, Technical Papers. Part 3 (A88-48776 20-34). Washington, DC, American Institute of Aeronautics and Astronautics, 1988, pp. 1455-1462. 21 Refs.

The eigenvalues and eigenvectors are analytically derived for general real gas dynamic equations in generalized curvilinear coordinates. In the diagonalizing formulation, the total mass conservation equation is taken into account, and arbitrary nonequilibrium effects, such as chemical reactions or vibrational nonequilibrium, can be treated in the same fashion. This diagonalizing formulation opens the way for chemically reacting flows to the construction of finite difference schemes based on characteristic relations, or the simplification of the inversion work of block-tridiagonal systems that arise in implicit time-split algorithm. Furthermore, a new class of TVD schemes in space discretization is proposed, which has a possibility to retain the second-order accuracy even near shock waves. For sample calculations, it has been chosen two kinds of reacting flows. The one is shock-induced ignition and the other is re-entry flow. Both results show the efficiency and high resolution of the scheme.

A88-44769 Flow field around a propeller by Navier-Stokes equation analysis. MAKOTO KOBAYAKAWA and ICHIRO HATANO, AIAA Paper 88-3150 presented at the 24th AIAA, ASME, SAE, and ASEE, Joint Propulsion Conference, Boston, MA, July 11-13, 1988. 9 pp. 15 Refs.

A flow field around a propfan is obtained through finite difference procedure. Viscous effects are incorporated with a thin layer approximation of the Navier-Stokes equations. An implicit approximate factorization technique is used to solve this boundary value problem. An analytical grid generation by the Poisson's equations is employed. A flow field around an eight bladed propfan in cruise condition with Mach number of 0.8 is calculated for two Reynolds numbers in which one corresponds to laminar boundary layer and the other to turbulent boundary layer. The viscous effects diminish the shock wave on the blade surfaces and moderate the steep change of pressure as was seen in the Euler analysis.

A88-22533 Numerical solution of viscous flows over cascades with sidewalls. KOJI MORINISHI and NOBUYUKI SATOFUKA, AIAA 26th Aerospace Sciences Meeting, Reno, NV, Jan. 11-14, 1988. 9 pp. 10 Refs. (AIAA Paper 88-0708).

Numerical solution of viscous transonic flows through cascades with sidewalls is obtained. The RRK scheme used in the previous two-dimensional study has been extended to the three-dimensional version. The algebraic two-layer eddy-viscosity model proposed by Baldwin and Lomax with a modified distance is used to simulate viscous turbulent flows. The preliminary calculation are carried out for the flow through the plane NACA 65-(12)10 compressor cascade with sidewalls. Three cases with different aspect ratios of 1, 2, and 3 are calculated. For each case, a large separated flow region is found on the suction surface near sidewalls, which indicates strong effect of sidewalls on the flow fields. The results are compared with those of the previous two-dimensional code and experiments. The results obtained for the aspect ratio 3 qualitatively agree with the experimental data taken without sidewall boundary-layer removal and porous endwall suction, while the previous results agree with those taken with endwall suction.

A89-10661 Numerical approach of advanced turboprop with three-dimensional Euler equations. SHIGERU SAITO, HIROSHI KOBAYASHI, YASUHIRO WADA, and YUICHI MATSUO, *Proceedings of the International Pacific Air and Space Technology Conference,* Melbourne, Australia, Nov. 13-17, 1987, (A89-10627 01-01). Warrendale, PA, Society of Automotive Engineers, Inc., 1988, pp. 415-424. 31 Refs. (SAE Paper 872448).

Numerical analysis by solving the three-dimensional Euler equations has been performed in order to investigate the complicated flow patterns or aerodynamic characteristics of the advanced turboprop (ATP) propeller with two types of spinner configuration. The governing equations are written for a rotating Cartesian coordinate system in terms of absolute flow variables. The solution algorithm used is an implicit approximate factorization method and resultant matrices are efficiently solved by a LUADI scheme. This solver has been applied to study the effect of interference between highly swept blades and axisymmetrical spinner on the aerodynamic performance of the ATP propeller. Numerical results show that the selection of an area-ruled spinner is important for the aerodynamic design of an efficient turboprop.

A89-15652 Application of multigrid methods for the coupled and decoupled solution of the incompressible Navier-Stokes equations. CH. ARAKAWA, A. O. DEMUREN, W. RODI, and B. SCHOENUNG, *Proceedings of the 7th GAMM-Conference on Numerical Methods in Fluid Mechanics,* Louvain-la-Neuve, Belgium, Sept. 9-11, 1987, (A89-15651 04-34). Brunswick, Federal Republic of Germany, Friedr. Vieweg und Sohn, 1988, pp. 1-8. Research sponsored by DFG and Alexander von Humboldt-Stiftung. 11 Refs.

This paper presents the application of multigrid (MG) methods to the solution of the Navier-Stokes equations for incompressible two-dimensional laminar flow problems. The full approximation storage full multigrid algorithm is utilized with relaxation schemes which solve the primitive variables in either a coupled or a decoupled manner. The performance of the MG methods is compared with those of several single-grid (SG) methods, using two flow problems as test cases. The results show that the MG methods always converged faster than equivalent SG methods. More importantly, however, the best MG method was found to converge 10-80 times faster than the best SG method.

A89-26696 Numerical prediction of flow characteristics and retardation of mixing in a turbulent swirling flow. SHUICHIRO HIRAI, TOSHIMI TAKAGI, and TERUYOSHI HIGASHIYA, *International Journal of Heat and Mass Transfer* (ISSN 0017-9310), Vol. 32, Jan. 1989, pp. 121-130. 24 Refs.

Numerical predictions are compared with the experiments of turbulent swirling flow and mixing in a stationary pipe. Two kinds of turbulence models, the k-epsilon two-equation model and the stress/flux equation model, are employed in the present calculations. The axial and tangential velocity profiles and the characteristics of the retardation of mixing in the presence of the swirl can be predicted by the stress/flux equation model, whereas the k-epsilon two-equation model fails in this respect. Comprehensible interpretations of the phenomena, especially of the retardation of turbulent transport of momentum and species (He) due to swirl, are presented.

A88-43703 Computation of three-dimensional chemically reacting viscous flow around rocket body. K. V. REDDY, T. FUJIWARA, T. OGAWA, and K. ARASHI, AIAA Paper 88-2616 presented at the AIAA, Thermophysics, Plasmadynamics and Lasers Conference, San Antonio, TX, June 27-29, 1988. 22 pp. MOESC-supported research. 21 Refs.

An hybrid scheme that treats the strong peripheral shock as sharp shock is developed to compute the hypersonic, viscous and chemically reacting three-dimensional flow around a rocket body. A multicomponent mixture of thermally perfect but calorically imperfect gas is used. Diffusion velocities at all grid points are computed by solving multicomponent diffusion equations. All the molecular diffusion transport terms are retained. The governing equations in conservation law form are solved using a noniterative, approximately factored implicit finite-difference scheme. Both fully catalytic and noncatalytic walls are treated. Laminar flowfields over a rocket nose configuration are generated for different gas models and boundary conditions.