A89-45418 Accuracy of the marching method for parabolized Navier-Stokes equations. V. V. RUSANOV, O. N. BELOVA, and V. A. KARLIN, *Proceedings of the 11th International Conference on Numerical Methods in Fluid Dynamics,* Williamsburg, VA, June 27–July 1, 1988, (A89-45351 19-34). Berlin and New York, Springer-Verlag, 1989, pp. 512–517. 7 Refs.

The accuracy of viscous flow computations in the mixed steady parabolized Navier-Stokes (PNS) problem is studied, and the results are presented. The mixed steady PNS problem corresponding to a supersonic two-dimensional flow problem is formulated, and the effects of upstream propagation of perturbations and its regularization on the accuracy of the numerical solution is examined in detail.

A89-42561 Quasi-one-dimensional approximation in two-dimensional problems of gas dynamics (Kvaziodnomernoe priblizhenie v dvumernykh zadachakh gazovoi dinamiki). I. S. MEN'SHOV, *Akademiia Nauk SSSR, Izvestiia, Mekhanika Zhidkosti i Gaza* (ISSN 0568-5281), Mar.–Apr. 1989, pp. 136–143. 9 Refs.

The possibility of extending the quasi-one-dimensional approach to the analysis of two-dimensional gasdynamic processes is investigated. The quasi-one-dimensional model proposed here is applied to several problems in gas dynamics, including flow past a cone, flow past a blunt body, and jet flows. The results obtained are compared with solutions to the corresponding two-dimensional problems.

A89-42519 Calculation of stationary subsonic and transonic nonpotential flows of an ideal gas in axisymmetric channels (Raschet statsionarnykh do- i transzvukovykh nepotentsial'nykh techenii ideal'nogo gaza v osesimmetrichnykh kanalakh). IU. S. KOSOLAPOV, *Zhurnal Vychislitel'noi Matematiki i Matematicheskoi Fiziki* (ISSN 0044-4669), Vol. 29, May 1989, pp. 765–774. 11 Refs.

A generalization of a method for calculating stationary subsonic and transonic nonpotential flows of an ideal gas is proposed. The method is based on the numerical solution of a current function equation written in arbitrary coordinates. To determine density in transonic flow calculations, a marching procedure is proposed for solving one of the Euler equation projections. Results of calculations are presented.

A89-38420 Properties of the solvability of the unsteady Euler equations in terms of H, psi (Svoistva razreshimosti nestatsionarnykh uravnenii Eilera v terminakh H, psi). V. M. SOLOPENKO, *Akademiia Nauk SSSR, Doklady* (ISSN 0002-3264), Vol. 305, No. 3, 1989, pp. 567–570. 6 Refs.

An analysis is made of the solvability properties of the Euler equations in terms of H (total pressure head), psi (stream function), which are the main physically measurable quantities of the flow problem considered. A nonlinear operator equation of the first kind is derived whose solvabilty in the case of arbitrary initial data requires additional smoothness conditions.

A89-35450 A method for calculating potential transonic flows in turbomachinery cascades (Metod rascheta potentsial'nykh transzvukovykh techenii v reshetkakh turbomashin). P. M. BYVAL'TSEV and M. IA. IVANOV, *Zhurnal Vychislitel'noi Matematiki i Matematicheskoi Fiziki* (ISSN 0044-4669), Vol. 29, March 1989, pp. 447–459. 25 Refs.

A fast method has been developed for calculating stationary subsonic, transonic, and supersonic potential flows in plane cascades and in cascades located on a rotation surface in a variable-thickness layer. The method implements a version of the approximate factorization method which retains second-order accuracy in the supersonic regions of the flow. The method is based on the numerical integration of the complete equation for the velocity potential written in divergent form in arbitrary curvilinear coordinates. The advantages of this approach over the commonly used versions of the approach that uses the method of artificial compressibility in the supersonic regions are demonstrated.

A89-34043 Energy dissipation rate for a viscous fluid with a tangential stress condition at the boundary flow line (Skorost' dissipatsii energii viazkoi zhidkosti s usloviem dlia kasatel'nogo napriazheniia na granichnoi linii toka). A. G. PETROV, *Akademiia Nauk SSSR, Doklady* (ISSN 0002-3264), Vol. 304, No. 5, 1989, pp. 1082–1086. 8 Refs.

The Helmholtz theorem yields an exact lower bound on the dissipation rate of flow for fluids of different viscosities with the same velocity at the boundary. It is shown here that, for the same tangential stress at the boundary, it is possible to obtain an exact upper bound on the dissipation rate for fluids of different viscosities. A proof for the corresponding theorem is presented, and flow of a viscous fluid inside a spherical drop in the path of viscous gas flow is considered as an example.

A89-32281 Entry of a free expanding gas jet into a round opening in a transverse obstacle (Vkhod svobodno rasshiriaiushcheisia gazovoi strui v krugovoe otverstie v poperechnoi pregrade). A. M. BISHAEV, E. F. LIMAR, S. P. POPOV, and E. M. SHAKHOV, *Zhurnal Vychislitel'noi Matematiki i Matematicheskoi Fiziki* (ISSN 0044-4669), Vol. 29, Feb. 1989, pp. 277–285. 6 Refs.

Results of a numerical solution are presented for an axisymmetric problem concerning the impingement of a free expanding jet of a monoatomic gas on a plane transverse obstacle separating the gas jet from a vacuum and having a circular opening at the jet axis. The obstacle may be either infinitely thin or have a finite thickness. The problem is solved on the basis of Euler equations and a model kinetic equation. In the limit of a continuum, a steady oscillation regime is observed for moderate distances between the obstacle and the nozzle exit section.

A89-30108 Vortex generation in computational aerodynamics (O vikhreobrazovanii v vychislitel'noi aerodinamike). A. A. GLADKOV, *Zhurnal Vychislitel'noi Matematiki i Matematicheskoi Fiziki* (ISSN 0044-4669), Vol. 29, Jan. 1989, pp. 135–137. 14 Refs.

Three implicit difference schemes of splitting type are used to calculate the unsteady flows of a viscous heat-conducting gas with highly inhomogeneous regions (boundary layer, shock wave, or mixing zone). A numerical solution is obtained to the problem of an arbitrary discontinuity near a limiting plane. The results obtained demonstrate the effectiveness of the proposed approach.

A89-40501 Numerical modeling of separated flows near the boundaries (Chisel'ne modeliuvannia vidrivnikh techii poblizu granits' potoku). I. M. GORBAN', V. O. GORBAN', and M. V. SALTANOV, *Akademiia Nauk Ukrains'koi RSR, Dopovidi, Seriia A—FizikoMatematichni ta Tekhnichni Nauki* (ISSN 0002-3531), Feb. 1989, pp. 26–30. In Ukrainian. 7 Refs.

A numerical algorithm for modeling separated flows near the boundaries is proposed which combines the discrete-vortex approach with the conformal mapping method. The problem of the formation of a separation zone in flow past a step is analyzed, and the importance of considering vorticity dissipation is demonstrated. Results of calculations of vortex bunch formation in the case of a jet issuing into transverse flow are presented. It is shown that the parameters of vortex structures formed at the jet boundary are significantly affected by small jet flow rate fluctuations.

A89-35495 Potential models of transonic flows (O potentsial'nykh modeliakh transzvukovykh techenii). IU. B. LIFSHITS and A. A. SHAGAEV, *Akademiia Nauk SSSR, Doklady* (ISSN 0002-3264), Vol. 304, No. 6, 1989, pp. 1315–1319. 6 Refs.

The paper considers the development of potential models for transonic flows, where the Zhukovskii-Chaplygin condition guarantees uniqueness of the solution. The proposed method is used to calculate transonic flow past wing profiles. In particular, the pressure-coefficient distribution is shown for the NACA 0012 profile at a freestream Mach number of 0.8 and an angle of attack of 1.25 deg.

Japanese Aerospace Literature
This month: *Computational Fluid Dynamics*

A91-28830 An upwind finite element scheme for high-Reynolds-number flows. MASAHISA TABATA and SHOICHI FUJIMA, *International Journal for Numerical Methods in Fluids* (ISSN 0271-2091), Vol. 12, Feb. 20, 1991, pp. 305–322. 28 Refs.

A new upwind finite element scheme for the incompressible Navier-Stokes equations at high Reynolds number is presented. The idea of the upwind technique is based on the choice of upwind and downwind points. This scheme can approximate the convection term to third-order accuracy when these points are located at suitable positions. From the practical viewpoint of computation, the algorithm of the pressure Poisson equation procedure is adopted in the framework of the finite element method. Numerical results of flow problems in a cavity and past a circular cylinder show excellent dependence of the solutions on the Reynolds number. The influence of rounding errors causing Karman vortex shedding is also discussed in the latter problem.

A91-24729 A Chebyshev collocation method for the compressible Navier-Stokes equations in generalized coordinates. JIAN PING WANG, YOSHIAKI NAKAMURA, and MICHIRU YASUHARA, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 33, Nov. 1990, pp. 120–134. 13 Refs.

A new Chebyshev spectral collocation scheme was developed for the compressible Navier-Stokes equations transformed from the Cartesian coordinates to generalized ones. Both the spatial derivatives of variables and the metrics of coordinate transformation were calculated by the spectral collocation method (SCM). An explicit integration scheme was used for time-marching. The supersonic flow around a two-dimensional cylinder was solved as a test problem. Comparisons between the SCM code and a finite difference method code showed the superiority of the SCM in accuracy, though more CPU time was needed. Good agreement was also obtained between the numerical results and the experimental data.

A91-27263 Propagation of disturbances locally imposed on a vortex ring. OSAMU INOUE, *JSME International Journal, Series II* (ISSN 0914-8817), Vol. 34, Feb. 1991, pp. 24–29. 13 Refs.

The instability of a vortex ring is investigated numerically by a three-dimensional vortex method. Disturbances with small amplitude are imposed initially over a local region of a vortex ring, and the development of large-scale instability waves and small-scale irregular motions of vortices along the vortex core is studied. Three different types of disturbances are considered: (1) sinusoidal disturbances with wave number m = 6 (hexagonal), (2) random disturbances, and (3) random disturbances superimposed on sinusoidal (m = 6) disturbances. It is found that locally imposed sinusoidal disturbances propagate along a vortex core and produce large-scale instability waves over the whole region of a vortex ring. On the other hand, the propagation velocity of locally imposed random disturbances along a vortex core is negligibly small, and the development of small-scale irregular motions of vortices is almost confined within the local region.

A91-27262 Construction of a flow-simulating method with finite volume based on a Voronoi diagram. NOBUYUKI TANIGUCHI, CHUICHI ARAKAWA, and TOSHIO KOBAYASHI, *JSME International Journal, Series II* (ISSN0914-8817), Vol. 34, Feb. 1991, pp. 18–23. 5 Refs.

In flow simulations performed with the finite difference method or the finite volume method, it is a serious limitation that the calculating points must be ordered on the coordinates. Using the Voronoi diagram for the cell division of the finite volume method creates a new discretization form which permits an arbitrary distribution of points. This paper constructs a new method for the flow simulations by a Voronoi diagram and shows the calculation results of two-dimensional flows.

A91-27259 Three-dimensional distortions of a vortex filament with axial velocity. YASUHIDE FUKUMOTO and TAKESHI MIYAZAKI, *Journal of Fluid Mechanics* (ISSN 0022-1120), Vol. 222, Jan. 1991, pp. 369–416. MOESC-supported research. 47 Refs.

The three-dimensional motion of a thin vortex filament with axial velocity, embedded in an inviscid incompressible fluid, is studied. The equation of the vortex motion is derived which is correct to the second order in the ratio of the core radius to that of curvature, and an asymptotic formula for the linear dispersion relation is obtained up to the second order. A new integrable nonlinear evolution equation generalizing the localized induction equation is derived, and the N-soliton solution is obtained which is identical to that of the localized induction equation if the pertinent dispersion relation is used. The nonlocal influence of the entire perturbed filament on the linear stability of a helicoidal vortex is explored. The axial velocity is found to discriminate between right- and left-handed helices and the long-wave instability mode is found to disappear in a certain parameter range when the successive turns of the helix are not too close together.

A91-25334 Experimental and computational studies on unsteady aerodynamic heating phenomena in shock reflection processes. SHIGERU ASO, YOSHIHARU TANAHASHI, ANZHONG TAN, and MASANORI HAYASHI, *Kyushu University, Faculty of Engineering, Memoirs* (ISSN 0023-6160), Vol. 50, Sept. 1990, pp. 295–308. 7 Refs.

Surface temperature rises and surface heat flux at the ramp surface were measured in order to study the unsteady aerodynamic heating induced by shock reflection. Numerical simulations of unsteady shock reflections by a TVD scheme are also conducted. The second peak heating due to the impingement of slip layer is seen in complex Mach reflection and double Mach reflection, and the primary peak heating due to the Mach stem and the second peak heating due to the slip layer are observed. The detailed structure of unsteady aerodynamic heating of shock reflection is correctly calculated. In particular the second peak heating due to the impingement of the slip layer is predicted in good agreement with experiments.

A91-25331 Theoretical study on the stability of the longitudinal vortex street. TERUNORI OHMOTO and MUNEO HIRANO, *Kyushu University, Technology Reports* (ISSN 0023-2718), Vol. 63, Oct. 1990, pp. 495–502. 20 Refs.

The boundary effect on the stability of the street of longitudinal vortices with alternately changing directions is investigated by using the vortex filament model. The analysis shows that the street is unstable to an infinitesimal disturbance in both the infinite and the semi-infinite regions over a flat bed, whereas it is stable in the finite region bounded by the free surface and the flat bed. These results correspond with the behavior of longitudinal vortices experimentally found in both the inner and the outer layers of an open channel flow.

A91-18069 A new method to solve Poisson equation with Neumann boundary conditions. YOSHIAKI NAKAMURA, WEI JIA, and MICHIRU YASUHARA, *Japan Society for Aeronautical and Space Sciences Journal* (ISSN 0021-4663), Vol. 38, No. 441, 1990, pp. 541–550. 12 Refs.

In this paper a new numerical method to solve a pressure Poisson equation with Neumann boundary conditions is presented. The Poisson equation with Neumann boundary conditions is divided into two equations. The solution of the original equation is obtained as the sum of each solution. This method is applied to the primitive variable procedure to solve the incompressible flow around a circular cylinder at the Reynolds number of 100,000, which produces a converged pressure solution at every time step and the time-averaged drag coefficient close to the experiment.

A91-25084 Energy and flatness spectra in a forced turbulence. SHIGEO KIDA, KOJI OHKITANI, MICHIO YAMADA, and YOUICHI MURAKAMI, *Physical Society of Japan Journal* (ISSN 0031-9015), Vol. 59, Dec. 1990, pp. 4323–4330. Research supported by MOESC. 21 Refs.

The statistical properties of forced Navier-Stokes turbulence are investigated numerically using the high-symmetry flow method. A spectral simulation with a resolution of 340 cubed realizes the K exp -5/3 power law with a Kolmogoroff constant of 1.8 in the one-dimensional longitudinal and lateral spectra over one decade of wavenumber. The normalized form of the energy spectrum is the same as for the freely decaying turbulence reported by Kida and Murakami (1987) in both the inertial and dissipation-ranges. The flatness factor of an individual Fourier component of velocity increases montonically with wavenumber in the inertial range and saturates at approximately 3.7 in the dissipation range.

A91-24756 A numerical study of the transition of jet diffusion flames. H. YAMASHITA, G. KUSHIDA, and T. TAKENO, *Proceedings of the Royal Society (London), Series A—Mathematical and Physical Sciences* (ISSN 0080-4630), Vol. 431, No. 1882, Nov. 8, 1990, pp. 301–314. 18 Refs.

The transition to turbulence in a two-dimensional methane jet flame in a coflowing air stream at injection Reynolds numbers (Re) up to 2500 is investigated by means of numerical simulations based on a flame-surface model with Lewis number = 1 and an infinite reaction rate.The temperature dependence of the diffusion coefficient and the viscosity is taken into account in solving the time-dependent Navier-Stokes equations for compressible flow, and the results are presented in a series of color computer graphics. The flame and flow are found to remain laminar for Re = 1000 or less; at higher Re, the flame and flow downstream of a transition point exhibit small-scale fluctuations near the jet axis and large-scale fluctuations outside the flame surface, both related to a Kelvin-Helmholtz instability which is strongly affected by the radial distribution of density and transport coefficients.

A91-24514 Applicability of Euler analysis to prop-fan aerodynamic design. MAKOTO KOBAYAKAWA, RYOJI TAKAKI, YOSHIFUMI KAWAKAMI, and FREDERICK B. METZGER, *Proceedings of the 17th ICAS, Congress,* Stockholm, Sweden, Vol. 2 Sept. 9–14, 1990, (A91-24301 09-01). Washington, DC, American Institute of Aeronautics and Astronautics, Inc., 1990, pp. 2073–2081. 8 Refs.

Applicability of a numerical code to aerodynamic design of a prop-fan is guaranteed by precise agreement of numerical results with experimental data; i.e., not only integrated performance indices such as power coefficient and net efficiency but also pressure distribution on the blade surface should agree well between computed and experimental results. An Euler code using the TVD scheme is developed for this purpose. The numerical calculations are performed for the SR-7L prop-fan at freestream Mach number 0.5. The computed power coefficient, Cp = 1.734, shows comparatively good agreement with the experimental data, Cp = 1.440 + or 0.080 if the measurement error of the blade twisted angle is considered.

A91-24424 Blunt trailing edge analysis of supercritical airfoils by a Navier-Stokes code. NAOKI HIROSE and NOBUHIKO KAMIYA, *Proceedings of the 17th ICAS, Congress,* Stockholm, Sweden, Vol. 1, Sept. 9–14, 1990, (A91-24301 09-01). Washington, DC, American Institute of Aeronautics and Astronautics, Inc., 1990, pp. 1184–1193. 23 Refs.

A preliminary analysis of flow about the blunt trailing edge of NACA 0012 and supercritical airfoils in transonic speed was made utilizing a two-dimensional time-averaged Navier-Stokes code with turbulence model of Baldwin and Lomax. A very fine mesh distribution was focused at the trailing edge region where conventional codes treat as sharp trailing edge with zero thickness. Computation was made for NACA 0012 airfoil with three kinds of trailing edge thicknesses: cusp-type sharp, standard and 1percent thickness and compared with the result of conventionally-treated trailing edge. A 15-percent-thickness supercritical airfoil with trailing edge thickness of 0.5 percent was also analyzed. It was found that a vortex shedding similar to the Karman vortices is formed and surface pressure near the trailing edge shows unsteady oscillations due to vortices. The magnitude and periodicity of the oscillation is governed by the bluntness. Also, it was shown that 'Kutta condition' is not necessarily satisfied for the blunt trailing edge.

A91-24407 A computational and experimental analysis of joined-wing aerodynamics. MASAKATA HASHIMOTO, MASAYUKI ISHIKAWA, NAOKI HIROSE, and TAKESHI OHNUKI, *Proceedings of the 17th ICAS, Congress,* Stockholm, Sweden, Vol. 1, Sept. 9–14, 1990, (A91-24301 09-01). Washington, DC, American Institute of Aeronautics and Astronautics, Inc., 1990, pp. 1017–1027. 14 Refs.

The aerodynamic characteristics of a joined-wing were investigated by computational methods and by a low-speed wind tunnel test to show its advantages over the conventional configurations and the feasibility of adapting it for practical application. An initial analysis by the extended lifting-line theory and a second analysis by the three-dimensional potential-flow panel method were performed to clarify the effects of geometrical parameters on the joined-wing aerodynamics, such as solid/planar types, sweep angles, span ratio, and area ratio of the rear and front wings. A third analysis involves finite-difference computation of the three-dimensional Euler equations to obtain the lift and drag characteristics in the high subsonic to transonic regime. It is shown that the three-dimensionally diamond-shaped joined-wing configuration has a higher value of lift-to-drag ratio than the planely joined one.

A91-24115 Double linearization theory for a rotating subsonic annular cascade of oscillating blades. I—Mathematical expressions of disturbance flow field. II—Numerical study of unsteady aerodynamic forces. KAZUHIKO TOSHIMITSU, MASANOBU NAMBA, and PING LI, *Kyushu University, Faculty of Engineering, Memoirs* (ISSN 0023-6160), Vol. 50, June 1990, pp. 161–199. 17 Refs.

Mathematical expressions for steady and unsteady disturbance flow fields are obtained on the basis of the double linearization theory. A numerical study of unsteady aerodynamic forces is carried out. Three-dimensional effects in pure n- and s-wise bending and in pure torsional vibrations are investigated.

A91-21745 Parabolic multi-grid method for incompressible viscous flows using a group explicit relaxation scheme. SHIGERU MURATA, NOBUYUKI SATOFUKA, and TADASHI KUSHIYAMA, (CTAC-89—International Conference on Computational Techniques and Applications, Brisbane, Australia, July 10–12, 1989) *Computers and Fluids* (ISSN 0045-7930), Vol. 19, No. 1, 1991, pp. 33–41. 7 Refs.

This paper provides an efficient numerical method on supercomputers for two-dimensional incompressible unsteady viscous flows. The unsteady vorticity-velocity Navier-Stokes equations are discretized by means of the implicit Euler formula and the resulting elliptic system of equations is solved by the parabolic multi-grid method with the group explicit relaxation scheme. Application of the present method to unsteady flows in a square cavity shows the high computational efficiency on vector and parallel supercomputers.

A91-21599 Construction of modified third-order upwind schemes on nonuniform meshes. I. SUSUMU SHIRAYAMA, *29th AIAA, Aerospace Sciences Meeting,* Reno, NV, Jan. 7–10, 1991. 7 pp. 9 Refs. (AIAA Paper 91-0727).

Some upwind schemes on nonuniform meshes are introduced. The concept for numerical scheme is based on two relations: (1) a polynomial and the derivatives and (2) a polynomial and the integrations. In this paper, the Taylor series expansion of a function q about a certain point and the Lagrangian interpolation formula is utilized in order to construct the upwind scheme on a physical space. Three results in two dimensions are shown: a motion of viscous vortex with a uniform background flow, a driven cavity, and flow past a circular cylinder. The numerical solutions by the method are expected to be independent of the grid spacing.

A91-19150 Numerical investigation of supersonic inlet with realistic bleed and bypass systems. AKIRA FUJIMOTO, NOBUO NIWA, and KEISUKE SAWADA, *29th AIAA, Aerospace Sciences Meeting,* Reno, NV, Jan. 7–10, 1991. 9 pp. 14 Refs. (AIAA Paper 91-0127).

A supersonic mixed-compression inlet was designed for Mach 2.5 and aerodynamically analyzed using CFD approach. Its ramp bleed and throat bypass systems were simulated in order to examine the physical nature of a flow around the systems. From the calculation, two types of 'unstart' were predicted. One is a shock/boundary-layer interaction induced 'unstart.' The other is usual 'unstart' due to failure of pressure balance. The boundary-layer control bleed was found to play an important role for avoiding the former 'unstart.' For avoiding or delaying the latter 'unstart,' a throat bypass system plays an important role. The shock capturing and stabilizing effect by the throat bypass was also confirmed numerically. Furthermore, the mechanism of improving pressure recovery ahead of an inlet 'unstart' was examined in detail. As a result, the role of a throat gap, or a slot, was revealed quantitatively for the first time. A streamline passing over the gap automatically constructs an optimum equivalent wall boundary in accordance with the flow conditions around the system. This noteworthy result could be obtained by the great advantage of the CFD approach.

A91-17998 Three-dimensional jet issuing from the cruciform nozzle. SHIGETAKA FUJITA, GORO UENO, and HIDEO OSAKA, *JSME International Journal, Series II* (ISSN 0914-8817), Vol. 33, Nov. 1990, pp. 712–721. 17 Refs.

Statistical characteristics of a three-dimensional jet issuing from a cruciform nozzle have been examined experimentally to clarify the features of the present flow field due to the secondary flows. In the extent of x/d greater than about 50 where the mean velocity profile U/U(ox) shows the similarity, the ratio of the integral length scales L(u)/L(v) on the x axis assumes a constant value. In the relatively upstream portion, both the skewness factors S(u) and S(v) show a negative value, while both the flatness factors F(u) and F(v) take a positive value. The turbulent large eddy structure conjectured from the space-time correlation is significantly different at each location throughout the cross section.

A91-15034 Accurate computations of compressible Navier-Stokes equations by the spectral collocation method. JIAN-PING WANG, YOSHIAKI NAKAMURA, and MICHIRU YASUHARA, *Japan Society for Aeronautical and Space Sciences Journal* (ISSN 0021-4663), Vol. 38, No. 436, 1990, pp. 232–240. 14 Refs.

The numerical code to solve axisymmetric compressible Navier-Stokes equations by the spectral collocation method was developed and applied to a supersonic flow about a sphere. A Residual-Dependent filter was delevoped, so that the accuracy could be considerably improved. The influence of filter, time step size, and grid number on accuracy is discussed. The results are compared with experiments, and excellent agreement is shown.

A91-17997 A numerical analysis of flow around rectangular cylinders. ATSUSHI OKAJIMA, TAROH NAGAHISA, and AKIRA ROKUGOH, *JSME International Journal, Series II* (ISSN 0914-8817), Vol. 33, Nov. 1990, pp. 702–711. Research supported by the Kajima Foundation. 12 Refs.

By the finite difference method, flows around cylinders with rectangular cross sections of various width-to-height ratios of 0.4 to 8 have been computed in the range of Reynolds numbers from 100 to 1200. The flow pattern critically changes when the B/H ratio is 2.8 and 6 at Reynolds numbers of 500 to 1200. It is clarified by the numerical simulation that the component with a high Strouhal frequency is induced by the vortices separated from the trailing edges and that the low Strouhal component is due to the oscillation of flow over the side surfaces accompanied by the movement of separation bubbles. Finally, the computed results of flow patterns, base pressure, drag force and Strouhal number show a good agreement with the experimental results.

A91-17996 An experiment on a Taylor vortex flow in a gap with a small aspect ratio—Bifurcation of flows in a symmetric system. IKUO NAKAMURA, SHINTARO YAMASHITA, YORINOBU TOYA, and YOSHINORI UEKI, *JSME International Journal, Series II* (ISSN 0914-8817), Vol. 33, Nov. 1990, pp. 685–691. 9 Refs.

This experimental study is concerned with the instability of several Taylor vortex flows between concentric cylinders for a symmetric end condition. The flow has several characteristic patterns: primary mode, normal secondary mode, and anomalous mode which has outward flow on either the upper or the lower end plate and/or on both end plates of cylinders. It is found that the anomalous mode changes to another anomalous mode or the secondary normal mode or primary mode when the Reynolds number is gradually decreased. Consequently, a bifurcation set of complex flows is presented with the bifurcation diagram in a symmetric system.

A91-17995 A zonal approach for solving the compressible Navier-Stokes equations using a TVD finite volume method. MASATO FURUKAWA, MASAHIRO INOUE, and MASAO YAMASAKI, *JSME International Journal, Series II* (ISSN 0914-8817), Vol. 33, Nov. 1990, pp. 665–673. 19 Refs.

A new zonal approach for compressible viscous flow computation has been developed. The two-dimensional, Reynolds-averaged Navier-Stokes equations are discretized spatially by a cell-centered finite-volume formulation. The inviscid fluxes at cell interfaces are evaluated with a MUSCL-type upwind scheme. The viscous fluxes are determined by central differencing. The computational domain is divided into nonoverlapping zones; the zonal boundaries are constructed from the cell interfaces, and communication from one zone to another is accomplished by numerical fluxes through the zonal boundaries. The use of the finite-volume formulation ensures the uniqueness of the zonal boundaries and the complete conservation of the numerical fluxes across the zonal boundaries, which results in a highly accurate zonal approach. Numerical results are presented for viscous flows through a transonic turbine cascade.

A91-15053 Comment on construction and accuracy evaluation of finite-difference-operators. SHIGENORI ANDO, *Japan Society for Aeronautical and Space Sciences Journal* (ISSN 0021-4663), Vol. 38, No. 439, 1990, pp. 449–451. 18 Refs.

Some improvements of numerical methods for solving incompressible Navier-Stokes equations in psi-omega variables are presented. The flow around a cylinder is used as a test problem. The numerical instability due to a central difference approximation for convective terms is discussed. It is found to be strongly related to the pressure gradient in the flow field.

A91-11728 GSMAC finite element method for unsteady incompressible Navier-Stokes equations at high Reynolds numbers. T. TANAHASHI, H. OKANAGA, and T. SAITO, *International Journal for Numerical Methods in Fluids* (ISSN 0271-2091), Vol. 11, Oct. 5, 1990, pp. 479–499. Research supported by the Iwatani Naoji Foundation. 22 Refs.

A new finite element technique is developed for predicting the velocity and the pressure in transient incompressible viscous fluid flows at high Reynolds numbers. The new method is based on the generalized and simplified marker-and-cell method (GSMAC) and has two characteristics: one is an application of the Bernoulli function and the implicit pressure solution algorithm to the explicit fractional time step method; the other is a high-order flux calculation to prevent the pressure field from oscillating. Two examples, driven cavity flows at high Reynolds numbers and vortex shedding behind a circular cylinder, are presented. Satisfactory agreement with experiment is demonstrated.

A91-10350 A numerical study of turbulent square-duct flow using an anisotropic k-epsilon model. S. NISIZIMA, *Theoretical and Computational Fluid Dynamics* (ISSN 0935-4964), Vol. 2, No. 1, 1990, pp. 61–71. 31 Refs.

A turbulent square-duct flow is studied numerically using an anisotropic k-epsilon model, in which the deviation of the Reynolds stress from its isotropic eddy-viscosity representation plays a central role. The no slip boundary condition on the wall is imposed with the aid of wall damping functions. Various computed turbulent quantities of a square-duct flow are compared with experimental and numerical results. The comparison shows that the present anisotropic k-epsilon model gives reasonable results to major characteristic properties in a duct flow, such as the anisotropy of turbulent intensities and the secondary flow.

A90-47855 Studies on estimating the performance of impellers with cut-down of the blade edge of the centrifugal pump by the surface singularity method. AKINORI FURUKAWA, CI-CHANG CHENG, and YASUO TAKAMATSU, *JSME International Journal, Series II* (ISSN 0914-8817), Vol. 33, Aug. 1990, pp. 525–530. 7 Refs.

Pump performance depends on the outlet flow of the impeller. A method of surface singularities for core flow in the centrifugal impeller, combined with an integral method for a boundary layer, would explain the mechanism of the performance change caused by cutting the outlet edge of the impeller blades down. This method is applied to flows in the impellers with various cut-downs of the blade edge, and then the calculated results are compared with the experimental ones. Both results are shown to be quantitatively in good agreement. On the influence of cutting the blade edge on the outlet flow, it is indicated that the cut of the pressure surface results in the decrease of relative flow angle with the decrease of radial velocity in the core flow, while that of the suction surface results only in a decrease in radial velocity. The change in the flow separation region due to the cut on the suction surface, however, contributes to the deterioration of pump performance.

A90-47300 Numerical study of interaction of a jet with a supersonic cross flow. WEIGUANG HUANG, NOBUHIKO YAMASAKI, and MASANOBU NAMBA, *Kyushu University, Technology Reports* (ISSN 0023-2718), Vol. 63, June 1990, pp. 243–285. 7 Refs.

Two flux vector splitting methods of Steger-Warming and Van Leer are investigated to check their capabilities for the calculations of the flows near sonic point and discontinuity. The results show that Van Leer method is more powerful for those singularities. The transonic flow around a wing and the shock reflection processes by a ramp are simulated by the Van Leer method. For transonic flow a wing section of NACA 0012 is calculated under the conditions of freestream Mach number of 0.8 and attack angle of 1.25 degrees. The embedded shock wave on the upper surface is captured within only three mesh points. For shock reflection processes generated by a ramp, various shock-reflection patterns of regular reflection, single Mach reflection, complex Mach reflection, and double Mach reflection are simulated. Calculated results show excellent agreements with experimental results.

A90-40612 A numerical investigation of supersonic inlet using implicit TVD scheme. JUNJI SHIGEMATSU, KAZUOMI YAMAMOTO, KAZUO SHIRAISHI, and ATSUSHIGE TANAKA, *Joint Propulsion Conference of the 26th AIAA, SAE, ASME, and ASEE,* Orlando, FL, July 16–18, 1990. 10 pp. 11 Refs. (AIAA Paper 90-2135).

The compressible Navier-Stokes equations were solved to investigate supersonic inlet flowfield. In two-dimensional analysis, the mass flowplug which control the back pressure was modeled numerically for close modeling of experimental case. The three-dimensional computation was done to predict three-dimensional flow behavior such as corner vortex and interaction between oblique shock wave and side wall turbulent boundary layer. The two-dimensional numerical analysis could predict the complicated flowfield on various mass flowplug condition respectively. The results by three-dimensional computation could show the viscous effects caused by the interaction between ramp oblique shock wave and the side wall turbulent boundary layer. The computations were compared with the experimental results of wind tunnel test. The numerical solutions successfully agreed with the experimental results.

A90-47852 Numerical analysis of gas-solid two-phase nonequilibrium nozzle flows. MICHIO NISHIDA and SHOGO ISHIMARU, *JSME International Journal, Series II* (ISSN 0914-8817), Vol. 33, Aug. 1990, pp. 494–500. 10 Refs.

Numerical simulation of the steady gas-particle two-phase flowfield inside a JPL nozzle is presented. The calculations utilize a time-dependent technique that uses the TVD-MacCormack algorithm for both gas and particle phases. Nozzle flows under consideration are inviscid, and spherical particles 1 micron in diameter are dealt with. Mach number contours for the two-phase nozzle flow are found to be bent upstream in the vicinity of the wall downstream of the throat. The maximum particle density is observed near the wall upstream of the throat.

A90-47835 Some characteristics of exact strained solutions to the two-dimensional Navier-Stokes equations. MASANORI TAKAOKA, *Physical Society of Japan Journal* (ISSN 0031-9015), Vol. 59, July 1990, pp. 2365–2373. 14 Refs.

Exact strained solutions to the two-dimensional Navier-Stokes equations are obtained which have two-dimensional spatial periodic structures. The time developments of the circulation, the energy, the enstrophy and the palinstrophy are examined for several viscosities. In the inviscid limit, the enstrophy dissipation rate becomes independent of viscosity. The power spectrum of the energy behaves as k exp—3 in the inertial subrange, for constant strain, and different power laws are obtained depending on the time dependence of the strain.

A90-46852 Toward the validation of unsteady solutions by using highly accurate schemes on non-uniform grid system. SUSUMU SHIRAYAMA and KUNIO KUWAHARA, *Proceedings of the 3rd International Congress of Fluid Mechanics,* Cairo, Egypt, Vol. 1, Jan. 2–4, 1990, (A90-46836 21-34). Mansoura, Egypt, Mansoura University, 1990, pp. 327–333. 16 Refs.

The concept of constructing accurate designs on a nonuniform grid system is proposed. The schemes are based on a finite difference methodology with generalized curvilinear coordinates and the truncation errors for numerical transformation play an important role in obtaining the solution for a nonuniform grid system. A higher-order spatially accurate scheme on a nonuniform mesh is proposed. Several numerical experiments are carried out on unsteady flow problems and several visualization techniques are used to validate in two and three dimensions.

A90-42024 Equilibrium and nonequilibrium radiation heat transfer over a reentry blunt body. AKIHIRO SASOH, TOSHI FUJIWARA, and XINYU CHANG, *Joint Propulsion Conference of the 26th AIAA, SAE, ASME, and ASEE,* Orlando, FL, July 16–18, 1990. 9 pp. 13 Refs. (AIAA Paper 90-2113).

A numerical study on radiation heat transfer over a reentry blunt body has been conducted. The band method employed here saves the computer memory size necessary to the spectral radiative transfer calculation. The dominant species for the radiative heat transfer are found to be the atomic species and N2(+) (first negative band). The emission power is sensitive to the density and temperature of the electron. As a result, thermo-fluid-dynamical nonequilibrium enhances the emission power, which is specifically caused by the atomic species. Also, the wall condition considerably affects the radiative heat flux since the densities of the radiative species and the electron near the wall depends strongly on the boundary condition.