reference waves are incoherent, the third-order nonlinear susceptibility (10 to the -7th esu with a relaxation time of about 10 to the -11 s) makes a dominant contribution to the reflection under the four-wave interaction. In the case of the coherent interaction, the fifth-order nonlinear susceptibility (10 to the -8th esu with a relaxation time of 10 to the -8th s) plays a leading role in the reflection.

A88-29838 Polarized light transfer in a medium containing perfectly oriented elongated particles: General theory—'Rayleigh' scattering (Perenos poliarizovannogo izlucheniia v srede, sostoiashchei iz polnost'iu orientirovannykh sil'no vytianutykh chastits. Obshchaia teoriia. 'Releevskoe' rasseianiia) M. I. MISHCHENKO and E. G. IANOVITSKII, *Kinematika i Fizika Nebesnykh Tel* (ISSN 0233-7665), Vol. 4, Jan.-Feb. 1988, pp. 19-29. 11 Refs.

Transfer of polarized light is studied in a plane parallel medium containing perfectly oriented elongated particles (infinite cylinders). External linearly polarized radiation is assumed to fall perpendicularly to the direction of particles orientation. The Sobolev method is applied in order to determine polarized radiation field in a semiinfinite homogeneous medium. Numerical results are represented for the case of very thin cylinders ('Rayleigh' scattering) which are compared with corresponding results for purely gaseous planetary atmosphere.

Japanese Aerospace Literature
This month: *Fluid Dynamics/Computational Fluid Dynamics*

A88-29718 Numerical simulation of vortex-induced flow fields in a turbine cascade (Numerische Simulation durch Wirbel zu bestimmender Stroemungsfelder im Schaufelgitter) H. NISHIMURA, *Zeitschrift fuer andewandte Mathematik und Physik* (ISSN 0044-2275), Vol. 39, Jan. 1988, pp. 50-64. 7 Refs.

Vortex-induced flow fields in (1) a parallelogram-shaped zone enclosed by four nonporous walls; (2) a channel enclosed by upper and lower walls; and (3) a half-open region enclosed by upper, lower, and left-side walls are investigated by means of numerical simulations based on the potential-vortex model of Lakshminarayana (1970). The derivation of the governing equations is discussed, and the results are presented graphically and characterized in detail. The effectiveness of the present approach in describing secondary cascade flows with vortices is demonstrated.

A88-29465 Analytical study of the structure of radiation controlled flame YOSHIO YOSHIZAWA, KIYOSHI SASAKI, and RYOZO ECHIGO, *International Journal of Heat and Mass Transfer* (ISSN 0017-9310), Vol. 31, Feb. 1988, pp. 311-319. 16 Refs.

This study is aimed at clarifying the effects of radiative heat transfer on the flame structure and burning velocity in gas-solid two-phase systems. Based on a strict treatment of radiation, a detailed numerical analysis has been performed for a one-dimensional model of premixed combustion in a porous medium, and the effects of the absorption coefficient and total optical thickness of the porous medium, as well as the position of the reaction zone within the porous medium on the flame structures and burning velocity have been elucidated.

A88-29375 Effects of wind distribution over aircraft on the longitudinal equations of motion in wind shear conditions HARUO KIMURA and HAMID BASSIRI, *Kyushu University, Faculty of Engineering, Memoirs* (ISSN 0023-6160), Vol. 47, Sept. 1987, pp. 193-205. 8 Refs.

This paper derives the longitudinal equations of motion for an aircraft incorporating the wind shear terms in which the effect of wind distribution over the aircraft is taken into account. The aerodynamic forces and moment being dependent on the motion relative to the atmosphere, pertinent use of wing and tail relative velocities is emphasized. For simplicity, only the variation of horizontal atmospheric winds with altitude is considered. It is shown by a simple numerical example that - depending on the severity of atmospheric movement - the wind distribution over the aircraft has a distinguishable effect on the moment, and therefore the stability.

A88-29275 On the Kolmogorov's spectrum for turbulence - A review of the statistical mechanical theory of turbulence NORIO OHTOMO, TADASHI SEIDOU, and YUKIO TANAKA, *Hokkaido University, Faculty of Engineering, Bulletin* (ISSN 0385-602X), Feb. 1988, pp. 69-75. 23 Refs.

Several contributions to the recent development of a statistical mechanical theory of turbulence are briefly reviewed. The first group of these is represented by Wyld's theory, which adopts a perturbation method for solving the Navier-Stokes equation, analogous to the perturbation theory using Feynman's diagrams of quantum field theory. The Kolmogorov '-5/3' spectrum is derived by Shut'ko on the basis of Wyld's theory.The next is Hopf's theory based on the functional formulation, in which the so-called Hopf equation is derived. The Hopf equation is formally identical to the Tomonaga-Schwinger equation of quantum field theory. The Kolmogorov '-5/3' spectrum is derived by Edwards and McComb using the maximum-entropy principle.

A88-28931 Mean-field magnetohydrodynamics associated with random Alfven waves in a plasma with weak magnetic diffusion HIROMITSU HAMABATA and TOMIKAZU NAMIKAWA, *Journal of Plasma Physics* (ISSN 0022-3778), Vol. 39, Feb. 1988, pp. 139-149. 7 Refs.

Using first-order smoothing theory, Fourier analysis and perturbation methods, a new equation is derived governing the evolution of the spectrum tensor (including the energy and helicity spectrum functions) of the random velocity field as well as the ponderomotive and mean electromotive forces generated by random Alfven waves in a plasma with weak magnetic diffusion. The ponderomotive and mean electromotive forces are expressed as series involving spatial derivatives of mean magnetic and velocity fields whose coefficients are associated with the helicity spectrum function of the random velocity field. The effect of microscale random Alfven waves, through ponderomotive and mean electromotive forces generated by them, on the propagation of large-scale Alfven waves is also investigated by solving the mean-field equations, including the transport equation of the helicity spectrum function.

A88-24847 Pressure losses and flow field distortion induced by tip clearance of centrifugal and axial compressors YASUTOSHI SENOO, *Kyushu University, Research Institute of Industrial Science, Reports* (ISSN 0368-6841), no. 82, 1987, pp. 1-13. 40 Refs.

The flow field near the tip of compressor rotor blades is distorted by leakage through the tip clearance, and the performance of the compressor deteriorates. Empirical equations expressing the pressure loss and the efficiency drop are varied. They are related to the lift coefficient in different ways such as proportional to C(L), C(L) exp 1.5, C(L) sq, or the sum of two terms, depending upon the ways of understanding the mechanics of pressure losses. These methods are examined and compared. Also included is a brief discussion on the optimum value of the tip clearance.

A88-24508 Shock wave/turbulent boundary-layer interactions induced by a semicone NOBUMI SAIDA and TOMONARI OOKA, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 30, Nov. 1987, pp. 173-185. 12 Refs.

This paper presents an experimental study of shock wave/turbulent boundary-layer interactions induced by a semicone placed on the floor of a wind tunnel. The experiments were carried out in an 8 x 10 sq cm supersonic wind tunnel at free-stream Mach numbers of 1.98 and 2.48. Corresponding unit Reynolds numbers at the test section were in both cases 3.8 x 10 to the 7th/m. Semicone models with half angles varying from 20 to 90 deg were used in this study. Surface static pressure measurements, oil flow studies, and Schlieren photographs of the flow field were made. It was found that, on a flat plate, the shape of the separation line is insensitive to the cone angle of over 40 deg. Furthermore, a secondary separation region embedded in the shock-induced primary separated flow exists along the semicone and plate junction.

A88-24507 Theoretical and experimental studies of a turbulent wall jet along a highly convex surface NOBUYUKI FUJISAWA and HIROYUKI SHIRAI, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 30, Nov. 1987, pp. 162-172. 22 Refs.

Mean flow and turbulence characteristics of a two-dimensional wall jet along a highly convex surface of constant radius are studied theoretically and experimentally. The calculations are based on a Reynolds stress model of turbulence with modifications of streamline curvature and velocity acceleration. The transport equations for the turbulence energy, Reynolds shear stress and dissipation rate, coupled with the fluid dynamical conservation equations, are solved by a parabolic numerical method. Streamwise pressure gradients needed for the calculations are evaluated from the experiment. Main features observed in the present experiments such as streamwise variations of half width of the jet, mean velocity profiles, turbulence energy, and Reynolds shear stress are fairly well reproduced in the calculated results.

A88-23321 Estimate of constants in the k-epsilon model of turbulence by using large eddy simulation FUJIHIRO HAMBA, *Physical Society of Japan, Journal* (ISSN 0031-9015), Vol. 56, Oct. 1987, pp. 3405-3408. 7 Refs.

Constants in the k-epsilon model are estimated using the statistics obtained from a large eddy simulation of a turbulent mixing layer. Estimated values are compared with those usually adopted in the k-epsilon model and those derived by statistical analysis. It is shown that the estiamte using large eddy simulation can give an important clue to the optimization of the constants and improvement of the model.

A88-23272 Research in computational fluid dynamics at National Aerospace Laboratory, Japan H. NAGASU, *Computer applications in aircraft design and operation—Proceedings of the First International Conference on Computer Aided Design, Manufacture and Operation in the Aeronautics and Space Industries,* Paris, France, June 16-18, 1987, Computational Mechanics Publications, 1987, pp. 195-208. 7 Refs.

The development of computational fluid dynamics (CFD) software and its applications at the Japanese National Aerospace Laboratory (NAL) are discussed. Two-dimensional and three-dimensional CFD codes for design-

ing aircraft, cascades and nozzles, and transonic wings are addressed. The use of a new numerical simulator system at NAL is briefly discussed.

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

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.

A88-18647 **A method of predicting the pressure amplitude of the combustion oscillation** YORIHIDE SEGAWA, *JSME International Journal* (ISSN 0913-185X), Vol. 30, Sept. 1987, pp. 1443-1449.

A theoretical study was performed to enable prediction of the pressure amplitude of the combustion oscillation in a combustor. The change in the heat release rate at the flame zone was assumed to be represented by polynomials of the velocity and the pressure fluctuations. Then the equilibrium condition between the thermodriving and damping energy of the oscillation was applied to determine the stability of the system. For the case of a single flame, solutions of the combustion oscillation were obtained and the condition for the suppressing the combustion oscillation was identified.

A88-18645 **Application of an inverse cascade design method to an axial fan** AKIRA GOTO, *JSME International Journal* (ISSN 0913-185X), Vol. 30, Sept. 1987, pp. 1414-1422. 9 Refs.

In order to demonstrate the applicability of an inverse cascade design method for blade design for an axial turbomachine, rotor and stator blades of a single stage axial fan are designed and tested. This design concept, proposed in the previous reports, is based on an inverse boundary layer method and an inverse cascade method. The overall performance and the rotor exit flow are measured and are compared with those of a conventional fan designed using an NACA 65 series cascade. The inversely designed fan obtains higher efficiency and a wider operating range compared with the conventional one. The rotor blade midspan wake in the trailing edge region reveals an extremely low velocity defect, a low wake shape factor, and low momentum thickness. Boundary layer separation on the rotor blade surface seems to be avoided and the losses are low. The experimental results show the validity of the practical application of the inverse design concept, especially for rotating blades.

A88-18644 **Numerical analysis of an axisymmetric jet using a streamline coordinate system** TETSUHIRO TSUKIJI and KOJI TAKAHASHI, *JSME International Journal* (ISSN 0913-185X), Vol. 30, Sept. 1987, pp. 1406-1413. 22 Refs.

The complete equations which describe a viscous axisymmetric flow are formulated using a streamline coordinate system (phi, psi), where psi is the stream function. The independent variable phi is constant along the orthogonal trajectories of streamlines in a plane including a symmetric axis. The axisymmetric laminar jet leaving a Poiseuille tube for an inviscid outer fluid phase is solved numerically using parabolic differential equations which are simplified by a boundary-layer-type analysis. The prediction of the velocity distributions and shape of the free surface profile, under the influence of both gravity and surface tension, is undertaken. The results obtained by the present method are found to agree very closely with experimental results. The behavior of a laminar jet issuing from a convergent nozzle with a small convergence angle is also investigated by the same method.

A88-13782 **Large eddy simulation of a turbulent mixing layer** FUJIHIRO HAMBA, *Physical Society of Japan, Journal* (ISSN 0031-9015), Vol. 56, Aug. 1987, pp. 2721-2732. 21 Refs.

A time-developing turbulent mixing layer is simulated numerically using large eddy simulation. Several statistics such as mean velocity, turbulent energy, velocity skewness and flatness factors, turbulent energy balance, and two-point correlation functions are computed to get a good agreement with experimental data. Especially, the production and dissipation terms in the turbulent energy equation are shown to be balanced near the center of the layer, and a streaky structure of the streamwise velocity component is also observed.

A88-13544 **Numerical solutions of the Euler equations for the flow field around counter-rotating propellers** MAKOTO KOBAYAKAWA and MASAHIRO NAKAO, *Japan Society for Aeronautical and Space Sciences, Journal* (ISSN 0021-4663), Vol. 35, no. 403, 1987, pp. 389-398. 7 Refs.

In order to investigate the flow field around the ATP through numerical methods for its optimal design, 3D-Euler equations are most prominent. The flow field around counterrotating propellers in advancing 0.8 Mach is obtained. Two spaces including front and rear blades are solved separately. The interaction between both blades are taken into the calculation by the connecting surfaces. The noniterative implicit ADI scheme is used in order to solve Euler equations. The periodic steady and averaged steady solutions are obtained. The latter is simplified by averaging the variables at the connecting surface. This shortens the calculation time to one-third compared with the periodic steady analysis which simulates the relative motion of the blades exactly. Numerical calculations are performed for two counter rotating ATP's with SR-1 and SR-3 blades. Both results show that they are similar to each other, and the propeller efficiencies increase compared with single rotating ATP with same blades.

A87-52380 Development of the Karman vortex due to buoyant force in opposing flow K. NOTO, H. TSUGUI and R. MATSUMOTO, *Flow visualization IV---Proceedings of the Fourth International Symposium,* Paris, France, Aug. 26-29, 1986 Hemisphere Publishing Co, 1987, pp. 649-654. 8 Refs.

Attention is given to the thermofluid behavior of Karman vortex streets in an opposing flow by means of time-dependent numerical simulations, flow visualizations, and the simultaneous measurements of velocity and temperature. A numerical technique for obtaining an asymmetrical thermofluid pattern is developed and verified by smoke-wire flow visualization, followed by simultaneous measurements of the time-dependent velocity and temperature by means of hot wire anemometry. Opposing buoyancy is noted to develop in the Karman vortex street.

A87-50025 Temperature field induced around a sphere in a uniform flow of a rarefied gas KAZUO AOKI and YOSHIO SONE, *Physics of Fluids* (ISSN 0031-9171), Vol. 30, July 1987, pp. 2286-2288. 11 Refs.

The temperature variation induced around a sphere with a uniform temperature in a slow uniform flow of a rarefied gas with the same temperature as the sphere is analyzed on the basis of the linearized Boltzmann-Krook-Welander equation with diffuse or specular reflection as the boundary condition. The analytic expressions of the temperature field are obtained for small and large Knudsen numbers.

A87-50019 Kolmogorov similarity in freely decaying turbulence S. KIDA and Y. MURAKAMI, *Physics of Fluids* (ISSN 0031-9171), Vol. 30, July 1987, pp. 2030-2039. Research supported by the Itoh Science Foundation. 29 Refs.

The three-dimensional Navier-Stokes equation without external force is solved numerically to simulate a freely decaying turbulence. Kolmogorov's similarity forms of the energy spectra in the universal range are observed in a decaying period after the energy dissipation rate takes the maximum value. During this period, the energy decays exponentially in time and the microscale Reynolds number changes from 100 to 60. At the lower part of the universal range, Kolmogorov's inertial range spectrum is observed over nearly one decade of the wavenumber, where the Kolomogorov constant is roughly 1.8. At the higher part of the universal range, on the other hand, it has an exponential tail with an algebraic correction.

A87-49323 Experimental investigation on the dynamic behavior of the liquid in spherical tanks of a spinning satellite KEIJI KOMATSU, SYUJI ONO, JUNICHIRO SHIMIZU, and RYUICHI NAGASHIMA, *Japan Society for Aeronautical and Space Sciences, Journal* (ISSN 0021-4663), Vol. 35, no. 400, 1987, pp. 266-269.

Two experiments are carried out to investigate the dynamic behavior of the liquid partially filled in spherical tanks of a spinning satellite. One is the simulation test at the spin up phase of the rocket. The other is the sloshing test to analyze the propellant sloshing effects on the satellite attitude control. These experiments are specially set for the Engineering Test Satellite V which has two spherical tanks 37 percent filled with hydrazine, but the results obtained can be applicable to the other problems of the liquid dynamics in spherical tanks.

A87-49321 Subsonic flow region on blunted cones in supersonic flow TAKASHI TANI, NORIO ARAI, KOETSU TAKEHANA, HIDEO SEKINE, and NAOKI HIROSE, *Japan Society for Aeronautical and Space Sciences, Journal* (ISSN 0021-4663), Vol. 35, no. 400, 1987, pp. 253-259. 6 Refs.

The article investigates both experimentally and numerically the complicated flow around the blunted cone of semiapex angle 20-45 degrees at Mach number 1.4-3.0. Especially emphasized is the structure of the subsonic flow region on the cone surface that is caused by the transition from the overexpanded flow at the nose to the recompression on the cone. Both results are in good agreement with each other. Consequently, it is shown that the numerical analysis of inviscid flow is very practical for such a complicated flow.

A87-47720 Improved form of the k-epsilon model for wall turbulent shear flows Y. NAGANO and M. HISHIDA, *ASME, Transactions, Journal of Fluids Engineering* (ISSN 0098-2202), Vol. 109, June 1987, pp. 156-160. 17 Refs.

An improved k-epsilon turbulence model for predicting wall turbulence is presented. The model was developed in conjunction with an accurate calculation of near-wall and low-Reynolds-number flows to meet the requirements of the Evaluation Committee report of the 1980-1981 Stanford Conference on Complex Turbulent Flows. The proposed model

was tested by application to turbulent pipe and channel flows, a flat plate boundary layer, a relaminarizing flow, and a diffuser flow. In all cases, the predicted values of turbulent quantities agreed almost completely with measurements, which many previously proposed models failed to predict correctly, over a wide range of the Reynolds number.

A87-47313 Theoretical and experimental studies of turbulent wall jet along a strongly concaved surface NOBUYUKI FUJISAWA and HIROYUKI SHIRAI, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 30, May 1987, pp. 26-37. 22 Refs.

Mean flow and turbulence characteristics of two-dimensional wall jets along a strongly concaved surface are studied theoretically and experimentally. Theoretically, a new version of the Reynolds stress model of turbulence is proposed. This model, which is applicable to wall jets with a large curvature, is coupled with the transport equations for turbulence energy and dissipation, and solved numerically to obtain the flow properties such as the velocity distribution, turbulence intensities, and Reynolds stress. The calculated results are in reasonable agreement with previous experiments for wall jets along mildly concaved surfaces and the present ones for a highly concaved surface. Moreover, the effect of an initial condition on the development of such wall jets along constant radius surfaces is discussed by comparing with results for a logarithmic spiral surface. This effect is also well predicted by the present turbulence model.

A87-47312 Gain characteristics of a supersonic flow CO chemical laser WATARU MASUDA, NORIO OGAWAHARA, and MOTOSHI TOHYAMA, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 30, May 1987, pp. 1-12. 16 Refs.

A numerical analysis is done with a CO chemical laser, which utilizes the supersonic mixing of the dissociated products of CS2 diluted in Ar with O2. It is assumed that the flow is quasi-one-dimensional and the mixing takes place instantaneously. A chemical and vibrational kinetic model encompassing twenty-four reactions and twenty-seven vibrational transitions is developed to approximate the reacting and relaxing mixture contained within a combustor. The present simulation is used to study the effects of various parameters on the species concentrations, CO vibrational population distributions and the small signal gain coefficients of a supersonic flow CO chemical laser and the proper gasdynamic conditions to give good performance are discussed.

A87-45280 Extension of local circulation method to counter rotation propeller SHIGERU SAITO, HORISHI KOBAYASHI, YOSHIYA NAKAMURA, and KENICHI NASU, AIAA Paper 87-45280 presented at the AIAA, SAE, ASME, and ASEE, 23rd Joint Propulsion Conference, San Diego, CA, June 29-July 2, 1987. 10 pp. 15 Refs.

A computational method for rotary wing airloading calculation called the local circulation method (LCM) has been extended to a counter rotation propeller (CRP). The new method is a combination of LCM for single rotor and a coaxial helicopter rotor model used in the local momentum theory (LMT) with some modifications. The interaction of two rotors via swirl velocity is introduced in addition to axial induced velocity. Results are compared with those of experiment for subsonic CRP performance and showed a good agreement. It is applied to transonic propeller employing experimental two-dimensional lift slope data at high Mach number. It also resulted in a good agreement with Euler calculation in single rotation advanced turbo-prop (ATP). Some results of calculation are given both for performance and time dependent airloading.

A87-42622 Numerical calculation of flow about wing-fuselage combination on the basis of Euler equations TOMIKO ISHIGURO and SATORU OGAWA, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 29, Feb. 1987, pp. 230-241. 16 Refs.

A procedure to calculate a transonic flow about a wing-fuselage combination on the basis of the Euler equations is presented. The equations transformed to a computational coordinate system are solved in a three-dimensional rectangular domain by the finite volume method using a Runge-Kutta type scheme of second-order accuracy and local time-step technique. The artificial viscosity used takes account of the eigenvalues of amplification matrix of the scheme. It is proved that this procedure is efficient and applicable for numerical analysis of inviscid flow.

A87-42621 Application of discrete vortex method to analysis of separated flow around aerofoils MASASHI SHIGEMI, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 29, Feb. 1987, pp. 207-219. 14 Refs.

The discrete vortex method was applied to the calculation of a separated flow around aerofoils. A new approach was formulated, in which the strength of the vortex, which is shed off at the separation point, was included into the unknown variables. In order to save the computational cost, simplifications were introduced in the formulation: e.g., clusters of drifting vortices were treated as concentrated single vortices. Because the total pressure in the separated region is less than that in the other regions, the pressure distribution around the aerofoil, which is obtained from the velocity distribution through Bernoulli's formula, has to be evaluated individually for the nonseparated and separated regions. To define the relativity between the two pressure distributions, a method to estimate the amount of the total pressure loss in the separated region was introduced. The Blasius formula, which is generally used to derive the forces acting on an obstacle, was modified to include the influence of the total pressure loss upon the lift and the drag forces.

A87-42465 Numerical investigation of nozzle shape effects on CO2 gas dynamic laser performance Y. WADA, M. YAMAGUCHI, and H. KUBOTA, AIAA Paper 87-1452 presented at the AIAA, 19th Fluid Dynamics, Plasma Dynamics, and Lasers Conference, Honolulu, HI, June 8-10, 1987. 11 pp. 18 Refs.

The effect of three types of nozzle shapes of CC-nozzle, S-nozzle and SFML-nozzle for the CO2 gasdynamic laser (CO2 GDL) component on GDL performance are investigated by two-dimensional simulation of the vibrational nonequilibrium inviscid flow field including the existence of shock wave. A point implicit scheme is used to solve these stiff equations. It is shown that the SFML-nozzle is superior to other nozzles from the point of view of GDL characteristics such as the small signal gain and the maximum power available. Design criteria for efficient GDL nozzles are also discussed.

A87-42448 Computation of flow around an NACA0012 airfoil at high angle of attack YOSHIFUMI SHIDA and KUNIO KUWAHARA, AIAA Paper 87-1425 presented at the AIAA, 19th Fluid Dynamics, Plasma Dynamics, and Lasers Conference, Honolulu, HI, June 8-10, 1987. 7 pp. 5 Refs.

Transonic flow around an NACA0012 airfoil at high angle of attack is simulated by solving the two-dimensional Navier-Stokes equations. The block pentadiagonal matrix scheme is employed. Periodic phenomena of shock-wave vortex interaction are observed. For comparison, computation of subsonic flow has been done. Small vortices are observed between the leading edge and the center of the chord.

A87-42346 Study of developing process of coherent structures in the turbulent boundary layer YU FUKUNISHI, HIROSHI SATO, and OSAMU INOUE, AIAA Paper 87-1253 presented at the AIAA, 19th Fluid Dynamics, Plasma Dynamics, and Lasers Conference, Honolulu, HI, June 8-10, 1987. 9 pp.

Coherent structures in the turbulent boundary layer is investigated by wind tunnel experiments and numerical simulations. In the experiment, coherent structures which contribute to the production of Reynolds stress are extracted from the turbulent field by conditional sampling technique using instantaneous Reynolds stress as the sampling condition. Experimental results show that the coherent structure is a pair of counter rotating flows and has a long life span. The structures change their shape only gradually as they are convected downstreqm. The process of three-dimensional disturbance developing into a counter-rotating-flow-struc ture similar to the one found in the experiment is shown in the numerical simulation.

A87-42104 Calculations of unsteady Navier-Stokes equations around an oscillating 3-D wing using moving grid system JIRO NAKAMICHI, AIAA Paper 87-1158 presented at the 8th Computational Fluid Dynamics Conference, Honolulu, HI, June 9-11, 1987. 11 Refs.

The purpose of the present paper is to show some solutions of unsteady Navier-Stokes equations around a low-aspect-ratio wing undergoing pitching motions. The present program is based upon the thin layer Navier-Stokes equations and the Beam-Warming ADI diagonal form is employed for the equations. The scheme is combined with a moving grid system presented below. The aerodynamics obtained by the present code are compared with experimental data. The effects of the leading edge vortex on the unsteady aerodynamics, which can not be predicted by inviscid model, are computed.

A87-42086 A zonal approach for computation of unsteady incompressible viscous flow SUSUMU SHIRAYAMA and KUNIO KUWAHARA, AIAA Paper 87-1140 presented at the 8th Computational Fluid Dynamics Conference, Honolulu, HI, June 9-11, 1987. 8 Refs.

A new approach for solving a flow field with a complex geometry is developed, which has geometric flexibility of finite element method. This method is divided into two procedures. At first, the physical domain is subdivided into simply connected regions. Then interface regions are constructed for the communication between the subregions. Unsteady flow fields can be handled by using a relaxation method. The present method is applied to solve the far wake flow fields past a bluff body and multibodies upon the ground.

A87-40917 Pressure losses and flow field distortion induced by tip clearance of centrifugal and axial compressors YASUTOSHI SENOO, *JSME International Journal* (ISSN 0913-185X), Vol. 30, March 1987, pp. 375-385. 41 Refs.

The flow field near the tip of compressor rotor blades is distorted by leakage through the tip clearance and the performance of the compressor is deteriorated. The literature regarding the tip clearance of compressor blades consists of computational fluid mechanics and experimental studies on the flow field and the pressure loss. Empirical equations expressing the pressure loss and the efficiency drop are varied. They relate to the lift coefficient in different ways, depending upon the ways of understanding the mechanics of pressure losses. These methods are examined and compared. Also, a brief discussion is made on the optimum value of the tip clearance.

A87-40846 Development of a numerical method for the prediction of turbulent flows in dump diffusers YASUNORI ANDO, MASAFUMI KAWAI, YUKINORI SATO, and HIDEMI TOH, *Ishikawajima-Harima Engineering Review* (ISSN 0578-7904), Vol. 27, Jan. 1987, pp. 7-12. 13 Refs.

In order to obtain an effective tool to design dump diffusers for gas turbine combustors, a finite-volume numerical calculation method has

been developed for the solution of two-dimensional/axisymmetric incompressible steady Navier-Stokes equation in general curvilinear coordinate system. This method was applied to the calculations of turbulent flows in a two-dimensional dump diffuser with uniform and distorted inlet velocity profiles as well as an annular dump diffuser with uniform inlet velocity profile, and the calculated results were compared with experimental data. The numerical results showed a good agreement with experimental data in case of both inlet velocity profiles; eventually, the numerical method was confirmed to be an effective tool for the development of dump diffusers which can predict the flow pattern, velocity distribution and the pressure loss.

A87-35408 Nozzle flows of gas-particle mixtures R. ISHII, Y. UMEDA, and K. KAWASAKI, *Physics of Fluids* (ISSN 0031-9171), Vol. 30, March 1987, pp. 752-760. 13 Refs.

A numerical analysis of supersonic nozzle flows of gas-particle mixtures is described. A time-dependent technique is applied to solve a two-phase inviscid flow through an axially symmetric nozzle. The two-step MacCormack algorithm is used for the gas-phase flow and the method of characteristics is applied to the particle-phase flow. Attention is mainly focused on the location of the limiting particle streamline. The numerical results are compared with the theoretical results obtained previously and also with experiments.

A87-31161 Turbulent channel and Couette flows using an anistropic k-epsilon model SHOITI NISIZIMA and AKIRA YOSHIZAWA, *AIAA Journal* (ISSN 0001-1452), Vol. 25, March 1987, pp. 414-420. 26 Refs.

Turbulent channel and Couette flows are studied numerically by using an anisotropic k-epsilon model. A feature of this model is an anisotropic expression for the Reynolds stress. The deviation of the Reynolds stress from an isotropic eddy-viscosity representation is incorporated. Only one kind of wall damping function is introduced to impose the no-slip boundary condition on solid walls. The results obtained show that turbulence quantities of channel and Couette flows are in good agreement with experimental data and numerical results from large-eddy simulation. The anisotropy of turbulent intensities, which the usual k-epsilon model cannot predict, is well reproduced.

A87-22733 Three-dimensional flow past a two-dimensional body SUSUMU SHIRAYAMA, TAKASHI OHTA, and KUNIO KUWAHARA, AIAA Paper 87-0605 presented at the AIAA, 25th Aerospace Sciences Meeting, Reno, NV, Jan. 12-15, 1987. 9 pp. 7 Refs.

By solving the three-dimensional incompressible Navier-Stokes equations, the development of three-dimensional structure was investigated in a flow around a circular cylinder placed between side walls. Also the case of a circular cylinder with an infinite spanwise length was treated using a periodic boundary condition. Initial vortex development and the development of spanwise vortical pattern are intensively visualized to see the three-dimensional structure. Various three-dimensional structures are observed such as the wavy structure of vortex filaments on the cylinder and in the Karman vortex street, the necklace vortex in front of the cylinder and arch vortex between the cylinder and the side wall. It was found that coarser three-dimensional computation often gives better results than very refined two-dimensional computation.

A87-22732 FDM-FEM zonal approach for viscous flow computations over multiple-bodies KAZUHIRO NAKAHASHI and SHIGERU OBAYASHI, AIAA Paper 87-0604 presented at the AIAA, 25th Aerospace Sciences Meeting, 25th, NV, Jan. 12-15, 1987. 10 pp. 12 Refs.

A hybrid method between a finite-difference method (FDM) and a finite-element method (FEM) is developed for computations of two- and three-dimensional viscous flowfields over multiple-bodies. In this scheme, an implicit finite-difference method is applied to viscous flow regions near bodies with body-fitted grids, and those FDM-zones are patched together by solving the remaining region using an explicit finite-element method. With this zonal approach, the computational efficiency and the solution quality of the FDM are retained, and the geometric flexibility is given by the FEM. The most important advantage the present technique yields is that the flow solver with this FDM-FEM zonal approach can be applied to various kinds of multiple-body flow problems without modifying the basic solution procedure. Flowfields of a two-dimensional turbine cascade and a three-dimensional nacelle/inlet are numerically simulated by the method in order to validate its capabilities. Effects on convergence rate owing to different CFL values and the variable time step are also explored.

A87-22652 Patterns of three-dimensional boundary layer separation SUSUMU SHIRAYAMA and KUNIO KUWAHARA, AIAA Paper 87-0461 presented at the AIAA, 25th Aerospace Sciences Meeting, Reno, NV, Jan. 12-15, 1987. 11 pp. 9 Refs.

Flows past a sphere and a spheroid are investigated by solving the unsteady incompressible Navier-Stokes equations numerically. The structure of three-dimensional boundary layer separation is studied by analyzing the computed flow patterns extensively. It was found that a pair of secondary vortices with positive and negative circulation near the body surface play an essential role in determining the type of the separation.

A87-22371 Navier-Stokes simulation of side-wall effect of two-dimensional transonic wind tunnel KUNIO KUWAHARA and SHIGERU OBAYASHI, AIAA Paper 87-0037 presented at the AIAA, 25th Aerospace Sciences Meeting, Reno, NV, Jan. 12-15, 1987. 10 pp. 6 Refs.

A transonic wind tunnel test of a flow around an NACA0012 airfoil is simulated by using both two-dimensional and three-dimensional Navier-Stokes codes. The effect of the side wall is focused on. The results revealed strong three-dimensionality introduced by the side-wall effect. To simulate the flow fields, 1.5 million grid points were used on a supercomputer VP200 having 256 MBytes main memory. The computation took about 25 hours for one case.