Japanese Aerospace Literature
This month: *Computational Fluid Dynamics*

A93-16977 Investigation of turbulence model in turbulent channel flow. MASANORI KANO, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 35, No. 108, Aug. 1992, pp. 56–71. 18 Refs.

Two kinds of turbulence models, a k-epsilon model of Launder et al. using an anisotropic expression for Reynolds stress tensor, and a simple improved form of a classical Reynolds-stress model, have been applied to a turbulent channel flow. The performance of both models has been investigated. A feature of the Reynolds-stress model is in the form in which the pressure-strain correlation terms give values on the wall compatible with experimental data. The computed results are compared with experimental data. The results show that the calculated mean velocity profiles and turbulence intensities by the k-epsilon model are in good agreement with experimental data, while the calculated turbulence intensity in the x1 direction by the Reynolds-stress model is predicted insufficiently. Therefore the pressure-strain correlation terms in the Reynolds-stress model are investigated.

A93-15498 Application of three-level one-stage Runge-Kutta scheme for numerical solution of incompressible flow. TOSHIYA MIYAKE, YUJIRO SAKAMOTO, HIROSHI TOKUNAGA, and NOBUYUKI SATOFUKA, *Japan Society of Mechanical Engineers, Transactions B* (ISSN 0387-5016), Vol. 58, Jan. 1992, pp. 167–173. 6 Refs.

A three-level one-stage Runge-Kutta scheme has been proposed for time integration of the incompressible Navier-Stokes equations. This scheme contains three free parameters and includes the conventional one-stage schemes, forward Euler scheme and Adams-Bashforth scheme, as special cases. The novel feature of the scheme is that the stability region can be extended further than that of the conventional one-stage scheme, and time accuracy can be controlled up to second order by tuning parameters. Numerical solutions for the transient Poiseuille flow and the two-dimensional lid-driven cavity flow are presented and compared with the analytical solution and those of the one-stage schemes. It is shown that the present scheme allows a larger time step and requires less computing cost than does the conventional one-stage scheme.

A93-15497 Numerical analyses of reflection and diffraction of weak shock wave. YASUNARI TAKANO, *Japan Society of Mechanical Engineers, Transactions B* (ISSN 0387-5016), Vol. 58, Jan. 1992, pp. 159–166. 16 Refs.

The accuracy of finite difference methods for weak shock waves is investigated via comparisons of the reflection and the diffraction of weak shock waves around wedges and corners between simulations. Analytical solutions are obtained for the wave equations by employing the conical flow method of Keller and Blank (1951). Finite difference simulations are conducted by applying such finite difference methods as the total variation diminishing method, the flux-corrected transport method, the flux vector splitting method, and the method of Roe (1981) to numerically solve the Euler equations. The approximate solutions obtained from finite difference calculations deviate from the analytical exact solutions near the apexes of the wedges or the corners that are singular points in grid systems.

A93-15494 Numerical analysis of two-dimensional turbulent flows through an oscillating cascade. YOSHIO SHIKANO and KIYOSHI NAMURA, *Japan Society of Mechanical Engineers, Transactions B* (ISSN 0387-5016), Vol. 58, Jan. 1992, pp. 139–144. 11 Refs.

A numerical technique for the computation of two-dimensional unsteady turbulent flows through an oscillating cascade is presented. To consider the interblade phase angle, a time phase shifted boundary condition is introduced on the periodic boundaries. A finite volume method is used to obtain the spatially discretized governing equations, while the second-order accurate Adams-Bashforth method is employed for the time integration. In the present analysis, a two-equation model of turbulence is introduced to estimate the turbulence effect. In order to assure the effectiveness of the present method, computations are carried out for the flow through cascades of flat plate blades (inviscid flow analysis), lens-type blades and compressor blades. The present method gives unsteady periodic flow fields including the aerodynamic forces and moments acting on the blade clearly, and the negative damping force is obtained from the computational result of flat plate cascade flow.

A92-55427 Experimental and numerical studies of focusing process of weak shock waves. KOJI IZUMI, TERUYUKI NAKAJIMA, SHIGERU ASO, and MICHIO NISHIDA, *Kyushu University, Faculty of Engineering, Memoirs* (ISSN 0023-6160), Vol. 52, No. 2, June 1992, pp. 233–242. 12 Refs.

This paper describes numerical and experimental studies of a weak shock wave reflected from various shapes of the parabolic reflector. Experiments have been carried out using a conventional shock tube with air as test gas at an incident shock Mach number of 1.1. In the experiments, the process of the shock focusing have been visualized by a Schlieren technique. Numerical studies have been conducted using a two-dimensional inviscid flow model. Thus obtained numerical results have been compared with the experimental ones for various reflector shapes and satisfactory agreement have been obtained.

A93-15486 Experimental and numerical study on the basic performance of a two-dimensional right-angled intake flow. NOBUYUKI FUJISAWA and MASAKAZU SHIRAKAWA, *Japan Society of Mechanical Engineers, Transactions B* (ISSN 0387-5016), Vol. 58, Jan. 1992, pp. 59–63. 11 Refs.

The flow in and around two-dimensional right-angled intake has been studied both experimentally and numerically. Pressure distributions on the intake surfaces are measured at various velocity ratios of the intake flow to the main flow, and the basic intake performances, such as the total pressure loss, the cavitation and the drag, are evaluated. It is shown that they are reproduced well by the numerical results with a standard k-epsilon model of turbulence, while the agreement becomes poorer as the velocity ratio becomes large. It is also demonstrated by the smoke-wire visualization study that the recirculating region formed upstream of the intake flow is reduced as the velocity ratio increases, contributing to the relative decrease in the total pressure loss at large velocity ratios. (Author)

A93-15485 Numerical analysis of unsteady separated flows by means of a parabolic multigrid method. SHIGERU MURATA, NOBUYUKI SATOFUKA, and HIROSHI KISE, *Japan Society of Mechanical Engineers, Transactions B* (ISSN 0387-5016), Vol. 58, Jan. 1992, pp. 44–50. 12 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 Crank-Nicholson formula and the resulting elliptic system of equations is solved by the parabolic multigrid method with the group explicit relaxation scheme. Application of the method to oscillating flows and impulsively started flows around a circular cylinder shows high computational efficiency on a parallel processor. The computed results are in good agreement with other numerical and experimental results and those based on the vorticity-stream function equations.

A93-15443 Turbulent structure in a vortex wake shed from an inclined circular cylinder). TSUTOMU HAYASHI, FUMIO YOSHINO, RYOJI WAKA, SEIICHI TANABE, and TETSUYA KAWAMURA, *Japan Society of Mechanical Engineers, Transactions B* (ISSN 0387-5016), Vol. 58, No. 546, Feb. 1992, pp. 297–304. 18 Refs.

This paper proposes a method by which to infer the turbulent structure in a vortex wake shed from a finite-length inclined circular cylinder in a uniform flow using the power spectrum and cross correlation. It is inferred from the results of cross correlation that the shedding vortex from the inclined circular cylinder forms a step like vortex line or a sawtooth-shaped vortex line. It was observed through the power spectrum that the end condition of the inclined circular cylinder gives a sensitive influence to the spanwise distribution of the turbulent structure in a vortex wake. It is also found that the inferred turbulent structure agrees with an aspect inferred from the results of a numerical calculation.

A92-56050 Generalized aerodynamics analysis by the boundary element method. MITSUNORI MATSUSHITA, KENJI FUJII, MITSUNORI YANAGIZAWA, *Proceedings of the 29th Aircraft Symposium,* Gifu, Japan, Oct. 7–9, 1991, (A92-56001 24-01). Tokyo, Japan Society for Aeronautical and Space Sciences, 1991, pp. 226–229. 5 Refs.

The computer code for analyzing rigid-body oscillating aerodynamic forces by the boundary element method (the panel method) is extended to compute the generalized aerodynamic forces for oscillating aircraft. The boundary condition portions of the original program are modified to adapt to the results of the FEM vibration analysis. Some example calculations for the flexible wing are shown.

A92-56004 High-speed aerodynamic design of space vehicle and required hypersonic wind tunnel facilities. S. NOMURA, K. HOZUMI, S. SAKAKIBARA, and K. SOGA, *Proceedings of the 29th Aircraft Symposium,* Gifu, Japan, Oct. 7-9, 1991, (A92-56001 24-01). Tokyo, Japan Society for Aeronautical and Space Sciences, 1991, pp. 16–19.

Considerations and problems associated with the aerodynamic design of space vehicles at hypersonic speeds are presented, as well as the aerodynamic design logic and optimization techniques to develop and refine the configurations in a traditional phased approach based on the programmatic design of space vehicles. Current methodology for the determination of aerodynamic characteristics for designing space vehicles is also considered. In particular, the role of new NAL M10 hypersonic wind tunnel and the NAL gun tunnel and the interrelationship of the tunnels and CFD methods in an actual aerodynamic design and analysis is discussed. (Author)

A92-35915 Combustion research and design technology of aircraft engines. TAKASHI TAMARU, *Japan Society for Aeronautical and Space Sciences Journal* (ISSN 0021-4663), Vol. 40, No. 459, 1992, pp. 209–213. 47 Refs.

Computational fluid technology is used for the combustor design of aircraft engines. The fuel vaporizer for a gas turbine engine is analyzed and Knight and Walker methods are used for the component pressure losses in combustion chambers.

A92-55452 Investigation of the propagation of the heavy monoatom beam. KAZUNORI SHIODA, YOSHIO HASHIDATE, and SHIGEO NONAKA, *JSME International Journal, Series II* (ISSN 0914-8817), Vol. 35, No. 3, Aug. 1992, pp. 361–369. 8 Refs.

In this paper, the numerical analyses on the behavior of a heavy monoatom beam are presented. Some numerical methods for the analysis of rarefied gas flow have been proposed thus far, which have been successfully applied to the flow of Kn less than 1, such as in the semiconductor manufacturing processes and other vacuum technologies. However, since the atomic number density of the beam is extremely low, the flow here is categorized as molecular (Kn greater than 1), in which the atomic free path is rather large compared with the representative scale of the beam. Hence, the probability density function of the atomic free path in the nonequilibrium field was taken into account in the numerical simulation of the beam propagation. Consequently, the calculated results of the beam diffusion due to mutual atomic scattering agreed well with the measured data obtained using gadolinium vapor.

A92-54498 Three-dimensional calculation of radiative field in hypersonic air shock layers. AKIHIRO SASOH, XIN-YU CHANG, TOSHIYUKI MURAYAMA, and TOSHI FUJIWARA, *Nagoya University, Faculty of Engineering, Memoirs* (ISSN 0027-7657), Vol. 43, No. 2, 1991, pp. 179–224. 31 Refs.

The method of numerical calculation of three-dimensional radiative transfer from nonequilibrium air shock layers over a body is presented with some reviews on radiative transfer and molecular physics. A numerical technique, which reduces the necessitating memory size of a computational resource, thereby enabling one to conduct three-dimensional calculation, has been developed. This method is applied to radiative heat transfer problems under a reentry condition. The radiative structure of the hypersonic air shock layer generated around a body is closely related to the thermally nonequilibrium structure of the shock layer. A radiative heat transfer which is comparable with the convective one is calculated at such a high Mach number as 35 at an altitude 70 km. This result suggests the importance of radiative heat transfer in thermal design of a reentry vehicle.

A92-48255 Transition between weak and strong turbulence observed in complex Ginzburg-Landau equation with a quintic nonlinearity. SADAYOSHI TOH and HIROSHI IWASAKI, *Physical Society of Japan Journal* (ISSN 0031-9015), Vol. 61, No. 5, May 1992, pp. 1495–1504. 11 Refs.

The 1D complex Ginzburg-Landau equation with a quintic nonlinearity is studied here in order to confirm the possibility of the transition between weak and strong turbulence and between soft and hard turbulence. The characteristics of asymptotic states are also examined. Three types of turbulent states are observed as dissipative effects decrease. The first state is a weak chaos driven by modulational instability, the second is a turbulent state characterized by the occurrence of relatively small bursts, and the third is a strongly intermittent turbulence dominated by bursts which blow up and get close to a singular solution of the 1D nonlinear Schroedinger equation until dissipation starts to take effect.

A92-47098 Numerical studies on a reacting mixing layer with chemical heat release. XIAO WANG and TOSHI FUJIWARA, *Japan Society for Aeronautical and Space Sciences, Transactions* (ISSN 0549-3811), Vol. 35, No. 107, May 1992, pp. 1–13. 17 Refs.

A numerical simulation of a nonpremixed, temporally-developing reacting mixing layer is performed by solving the time-dependent, compressible Navier-Stokes and species conservation equations. The chemical reaction is based on a binary, one-step, irreversible model. Diffusive properties are considered to be temperature-dependent. The conservation equations are integrated by means of finite difference method and the utilized numerical algorithm is MackCormack second-order explicit predictor-corrector method. The results are obtained at different heat release parameters, showing, in qualitative agreement with the previous studies, that heat release has effects of suppressing the growth rate of hydrodynamic instability, smearing and diffusing both the vortical and reactive-zone large-scale structures, and reducing product formation. In general, with the increase of diffusivities, hydrodynamic instabilities are reduced as indicated by quick saturation of rollup and pairing processes and reduction of vorticity strength; reactive-zone structures become diffusive and product formation is enhanced by a small amount, relative to the corresponding case of constant diffusivities.

A92-29521 Error characteristics of a vortex panel method in two-dimensional flow. AKIO ICHIKAWA, *Japan Society for Aeronautical and Space Sciences Journal* (ISSN 0021-4663), Vol. 40, No. 457, 1992, pp. 119–126. 7 Refs.

A numerical investigation is performed on the error analysis of a vortex panel method in the 2D flow. The method uses step or linear vortex distributions on straight line elements. The airfoil is divided into a set of elements by three ways: semicircular method, equispaced method, and quartercircular method. The following results are obtained: (1) the location of the optimum control point is the center of the elements for the step vortex distributions and the edge for the linear vortex distributions; (2) for dividing the way of the airfoil, the semicircular-method is the best which is followed by the quartercircular method and then by the equispaced method; (3) when the airfoil is divided by the best way, the accuracy of the solution is first order in the element size for the step vortex distributions and is second order for the linear vortex distributions.

A92-54499 Numerical simulation of reactive flows by a time-split algorithm. S. G. R. KUMAUR and TOSHI FUJIWARA, *Nagoya University, Faculty of Engineering, Memoirs* (ISSN 0027-7657), Vol. 43, No. 2, 1991, pp. 258–274. 11 Refs.

The propagation of shock-initiated reaction in a 2D channel is studied by a time-split algorithm. The convection and reaction processes are solved in a decoupled manner. The Euler equation describing the 2D inviscid flow is solved by an explicit TVD scheme, while the equations describing the reaction kinetics are solved implicitly at each nodal point. The initiation of combustion by a shock wave and its development into a detonation in a 2H2 + O2 + 7 Ar system are studied using the present algorithm. The time history of the process is presented.

A92-17503 Numerical simulation for various flowfields of aero-engine components. KOJI MATSUNAGA, YASUNORI ANDO, ATSUSHIGE TANAKA, and HIDEMI TOH, *Ishikawajima-Harima Engineering Review* (ISSN 0578-7904), Vol. 31, July 1991, pp. 230–237. 12 Refs.

Advances in aero-engine performance and economy are achieved by a fusion of many individual efforts in technology. Significant advances in the evolution and rationalization of aerodynamic technology appear in the development and utilization of CFD. CFD demonstrates valuable applications and is an essential complement to testing and experimentation. A brief review is presented of the current status and the future of CFD on the aero-engine development including: (1) 3D compressible Navier-Stokes computation for turbine-vane configurations; (2) incompressible Navier-Stokes computation for gas turbine combustor; and (3) Euler and Navier-Stokes computations for some ducts, stator/downstream strut interaction and swan-neck duct.

A92-17502 Supersonic inlet flow computation. SHIN-ICHI KURODA, *Ishikawajima-Harima Engineering Review* (ISSN 0578-7904), Vol. 31, July 1991, pp. 227–229. 10 Refs.

Supersonic inlet plays the role of partial or whole air-compression process in supersonic/hypersonic air-breathing engines and is a key factor of the engine performance. The ultimate purpose of the present study is to clarify the complex supersonic inlet flow structure by using the computational fluid dynamics (CFD) and thereby contribute to the development of the inlet. In the present paper, a preliminary computation is performed for the flowfield about the experimental inlet model which is designed at Mach number 3.0 and has a bleed chamber. To handle the complex body configuration, a zonal method with slightly overlapped grid is adopted. The Fortified Navier-Stokes approach is used as the interface scheme, which connects each zone with high accuracy and permits the movement of discontinuities across the zonal boundary.

A92-17501 CFD application to 2D/3D flow fields in Scramjet engine. TOSHIRO FUJIMORI, MASAFUMI KAWAI, TAKAKO SUZUKI, YASUNORI ANDO, and YASUNORI OHMORI, *Ishikawajima-Harima Engineering Review* (ISSN 0578-7904), Vol. 31, July 1991, pp. 221–226. 12 Refs.

CFD plays a major role in the research and development of hypersonic flight vehicles on the premise that numerical approaches provide simulations of various conditions, including supersonic combustion for which no ground test capability exists. The 2D/3D CFD codes have been developed to simulate the supersonic/hypersonic turbulent reacting flow in supersonic combustion ramjet (Scramjet) engines. The TVD scheme is used to capture shocks, and a finite reaction-rate mode of hydrogen-air combustion is utilized. The current results for the components of Scramjet engines are presented; i.e., inlet, combustor and nozzle by using the CFD codes. Validation of these results are compared with existing experimental and computational results.

A92-12423 A numerical simulation of separated flows around bodies. SHIGERU ASO and ATSUHIRO SAKAMOTO, *Kyushu University, Technology Reports* (ISSN 0023-2718), Vol. 64, Aug. 1991, pp. 249–255. 12 Refs.

Dynamic stall phenomena have been investigated numerically by solving incompressible Navier-Stokes equations by a third-order upwind scheme in order to reveal the flow structure and mechanism of dynamic stall. At first, in order to examine the validity of the calculations separated flows around circular cylinder are calculated. The results show excellent agreements with the experiments. Also, separated flows around a wing section at fixed attack angle are calculated and the results show excellent agreements with experiments which are conducted by the present authors. Finally, separated flows around oscillating airfoil in pitch are calculated by using moving mesh system. The flow conditions are selected from the experiments. The calculated separated region is small in pitching-up process and it becomes large in a pitching-down process. Quite different characteristics of flow patterns between a pitching-up and pitching-down processes are obtained.

A91-46822 Numerical simulation of outer flow around an ACV by discrete vortex methods. RYUICHI HAYASHI, TERUHIKO KIDA, and ZENSABURO YASUTOMI, *Japan Society for Aeronautical and Space Sciences Journal* (ISSN 0021-4663), Vol. 39, No. 448, 1991, pp. 258–267. 23 Refs.

Documents available from AIAA Technical Library Journal Announcement: IAA9120 Two discrete vortex methods are used to compute the outer flow around an air cushion vehicle (ACV). The flow patterns for a two-dimensional ACV and the flow model are discussed. A numerical computational model of the three-dimensional ACV is also demonstrated.