Japanese Aerospace Literature
This month: *Computational Fluid Dynamics*

A94-37056 Study on low boom configuration (for supersonic transport aircraft). K. YOSHIDA, T. YAMAKAGE, and H. OGOSHI (Kawasaki Heavy Industries, Ltd., Kobe, Japan), In *Aircraft Symposium*, 31st, Gifu, Japan, Nov. 10–12, 1993, Proceedings (A94-36967 12-01), Tokyo, Japan Society for Aeronautical and Space Sciences, 1993, pp. 588–591. In Japanese. 8 Refs. Documents available from Aeroplus Dispatch.

In the development of a second generation SST, it is improtant to reduce sonic boom. Typical results are summarized of a recent wind tunnel tests and CFD analysis of the characteristics of low boom and low drag configuration.

A94-37052 Effects of vibrational equilibrium on unsteady aerodynamic heating due to shock wave reflection. S. ASO, K.-I. OHYAMA (Kyushu Univ., Fukuoka, Japan), T. FUJIWARA (Nagoya Univ., Japan), and M. HAYASHI (Nishinippon Inst. of Technology, Fukuoka, Japan), In *Aircraft Symposium*, 31st, Gifu, Japan, Nov. 10–12, 1993, Proceedings (A94-36967 12-01), Tokyo, Japan Society for Aeronautical and Space Sciences, 1993, pp. 564–567. 7 Refs. Documents available from Aeroplus Dispatch.

Thin-layer Navier–Stokes equations were solved in order to investigate unsteady aerodynamic heating phenomena induced by shock impingement on a ramp surface, emphasizing high-temperature effects. Particular attention is given to the effect of energy transfer between translational and vibrational energies. The major objectives are identifying the role of the vibrational energy on the unsteady aerodynamic heating phenomena. Calculations were conducted under the conditions of the incident Mach number M (s) = 3.0 and the ramp angle = 35 deg, by changing initial temperatures from 300 K to 2000 K. Quite significant changes of aerodynamic heating phenomena with initial temperature are observed. The results are compared with those without vibrational energy, revealing quite different flowfields between both cases. In particular, at T-infinity = 2000 K, the maximum heat flux with vibrational equilibrium is 2.4 times of that without vibrational equilibrium, suggesting that the vibrational energy plays quite an important role in unsteady aerodynamic heating phenomena at high temperatures. (Author)

A94-36986 BEM analysis of unsteady aerodynamics on an elastically vibrating lifting body. T. UMEJIMA, G. BEPPU (Tokai Univ., Hiratsuka, Japan), H. MATSUSHITA (National Aerospace Lab., Tokyo, Japan), and M. YANAGIZAWA (Tokyo, Science Univ., Japan), In *Aircraft Symposium*, 31st, Gifu, Japan, Nov. 10–12, 1993, Proceedings (A94-36967 12-01), Tokyo, Japan Society for Aeronautical and Space Sciences, 1993, pp. 158–161. In Japanese. 7 Refs. Documents available from Aeroplus Dispatch.

The BEM based on Green's function is extended functionally to calculate unsteady aerodynamic forces acting on an elastically vibrating lifting body. This paper treats an aeroelastically scaled wind tunnel model as a lifting body which executes harmonic oscillation in a compressible flow. The key variables which describe the boundary condition are panel center velocity vector and normal vector increment at a panel center. The present paper shows how to determine these two variables using FEM vibration analysis data. The model has rigid airfoil segments attached to a flexible spar at a partial spanwise position and leading and trailing edge control surfaces; variable determination is, therefore, treated separately according to these structurally different parts. Several typical cases of elastic vibration and control surface oscillation are calculated by the present method and compared with doublet point method analysis. The correspondence between these two methods is fairly good. (Author)

A94-36985 Numerical simulation of viscous flow around two-dimensional airfoil in unsteady motions. S. ASO and Y. KUMAMOTO (Kyushu Univ., Fukuoka, Japan), In *Aircraft Symposium*, 31st, Gifu, Japan, Nov. 10–12, 1993, Proceedings (A94-36967 12-01), Tokyo, Japan Society for Aeronautical and Space Sciences, 1993, pp. 154–157. In Japanese. 5 Refs. Documents available from Aeroplus Dispatch.

In order to reveal the flow structure and mechanism of dynamic stall of airfoils, dynamic stall phenomena have been investigated numerically by solving full Navier–Stokes equations with a third-order upwind scheme. In this paper, we have calculated separated flows around oscillating airfoils in pitch by a moving mesh system. Quite significant flow patterns due to dynamic stall are revealed at various conditions with two different airfoils. (Author)

A94-36984 Transonic and supersonic flow characteristics around a hemisphere-cone-cylinder. I. NAKOMORI and Y. NAKAMURA (Nagoya Univ., Japan), In *Aircraft Symposium*, 31st, Gifu, Japan, Nov. 10–12, 1993, Proceedings (A94-36967 12-01), Tokyo, Japan Society for Aeronautical and Space Sciences, 1993, pp. 150–153. In Japanese. 4 Refs. Documents available from Aeroplus Dispatch.

Numerical solutions and experimental results are reported for transonic and supersonic flows past a cone-cylinder, and a hemisphere-cone-cylinder. First, we applied an axisymmetric code to the flow around a cone-cylinder, so that it was confirmed to be accurate enough for practical use. Then, to obtain a highly accurate flow field around a hemisphere-cone-cylinder with the finite volume method, an improved extrapolation technique was employed. As a result, we were able to sharply capture the weak re-compression wave on the cone surface after the overexpansion produced on the hemisphere surface. (Author)

A94-36983 The flow around a rotating spherical object. S. WAKABAYASHI and E. MORISHITA (Tokyo, Univ., Japan), In *Aircraft Symposium*, 31st, Gifu, Japan, Nov. 10–12, 1993, Proceedings (A94-36967 12-01), Tokyo, Japan Society for Aeronautical and Space Sciences, 1993, pp. 146–149. In Japanese. 2 Refs. Documents available from Aeroplus Dispatch.

The Laminar boundary layer on a rotating object in an otherwise undisturbed fluid of infinite extent is investigated numerically by solving N-S equations. First, the flow around a rotating sphere is solved numerically. This flow has been investigated both theoretically and experimentally, but the results are not sufficient in some regions and inconsistent with each other. Compared with those results, the present numerical results are considered sufficient and more consistent with the experimental ones. Second, the flow around a rotating partial spherical shell is solved numerically. This flow induces some lift near the edge and increases with the Reynolds number. The effects of curvature and edge angle are also investigated. (Author)

A94-33298 Direct Monte Carlo simulation of axisymmetric free jet. T. SOGA, H. KAWATO, and B. G. KIM (Nagoya Univ., Japan), Nagoya University, School of Engineering, Memoirs (ISSN 0919-0805), Vol. 45, No. 2, 1993, pp. 223–232. 13 Refs. Documents available from Aeroplus Dispatch.

Axisymmetric expansion flow from a circular orifice into a vacuum was studied based upon the DSMC method. The numerical simulation was carried out for a gas composed of hard sphere molecules. It is found that the radial temperature is slightly higher than the axial temperature in the vicinity of the orifice. The degree of such a nonequilibrium decreased in accordance with the decrease of the Knudsen number Kn. For the case of Kn = 0.02 the difference between two temperatures was less than 2 percent. Present simulation revealed that in the vicinity of the axis of symmetry orifice flow expands like a spherical source flow, but the off axis flow exhibits the peculiar features of axisymmetric expansion flow predicted by Grundy (1969). The simulated expansion flow along the axis of symmetry includes continuum to free molecular flows. Present results, the terminal velocity, the terminal temperature, and the terminal Mach number, showed good agreement with the previous results of spherical source flow expansion. (Author)

A94-33297 Numerical simulation of the plume impingement on the lunar surface. T. SOGA, B. G. KIM, and S. KOGURE (Nagoya Univ., Japan), Nagoya University, School of Engineering, Memoirs (ISSN 0919-0805), Vol. 45, No. 2, 1993, pp. 214–222. 13 Refs. Documents available from Aeroplus Dispatch.

Impingement of axisymmetric free jet from a supersonic nozzle (M(e) = 5) into a vacuum on a plate was studied based upon the Direct Simulation Monte Carlo method for hard sphere molecules. The distance from the nozzle exit to the plate was four times of the nozzle diameter. Plume envelopment due to plume impingement was found on the basis of the continuum flow theory. Results obtained for the nozzle Knudsen number KnD = 0.00625 showed good agreement with the results of PLM. The maximum shear stress on the flat plate is found at R/D about 0.7 for the case of KnD = 0.00625. (Author)

A94-27745 Breakdown and rearrangement of a vortex street in the far wake of a cylinder. T. KARASUDANI and M. FUNAKOSHI (Kyushu Univ., Japan), Kyushu University, Research Institute for Applied Mechanics, Reports, Vol. 39, No. 110, Feb. 1994, pp. 1–27. 31 Refs. Documents available from Aeroplus Dispatch.

The breakdown and rearrangement of a primary vortex street shed from a circular cylinder in the far wake are experimentally examined for R (the Reynolds number based on the diameter of the cylinder) in the range 70–154. According to the vorticity fields obtained using digital image processing for visualized flow fields, the primary vortex street breaks down into a nearly parallel shear flow of Gaussian profile at a certain downstream distance, before a secondary vortex street of large scale appears further downstream. The process leading to the nearly parallel flow can be explained as the evolution of the vortex regions of an inviscid fluid if we invoke the observation that the distance between the two rows in the primary vortex street increases with the downstream distance. Numerical computations with the discrete vortex method also support this explanation. (Author)

A94-19973 Numerical investigation of perpendicular staged blowing into a subsonic cross flow. Y. SUGIYAMA, M. YOSHIDA, and J.-S. CHOI (Nagoya Univ., Japan) Nagoya University, Faculty of Engineering, Memoirs (ISSN 0919-0805), Vol. 45, No. 1, Oct. 1993, pp. 118–136. 7 Refs. Documents available from Aeroplus Dispatch.

Numerical solutions have been obtained for a steady viscous flow induced by a small boundary-layer blowing through three slots. The analysis is assumed to be 2D, incompressible, and also laminar. The results show that a decrease in slot diameter and pitch and velocity ratio results in a reduction of the recirculating regions, but the present temperature field strongly is not affected by a recirculating flow near the blowing slots. A blowing fluid is strongly accelerated and heated by the cross flow at the leading edge of a plume. However, the stream-wise velocity component of the blowing fluid is almost constant along the plume. Moreover an increase in slot diameter or a decrease in slot pitch increases local Nusselt number and film cooling effectiveness on the slotted wall. It is also indicated that the calculated recirculating zones vary with numerical schemes used. (Author (revised))

A94-19940 Numerical analysis of separated flows through stalled cascade. T. NISHIZAWA and H. TAKATA, Japan Society of Mechanical Engineers, Transactions B (ISSN 0387-5016), Vol. 59, No. 567, Nov. 1993, pp. 3399–3406. In Japanese. 9 Refs. Documents available from Aeroplus Dispatch.

Flow behavior of a rotating stall is studied numerically by means of the vortex model which was developed to solve the flows through stalled cascades. The flow features and pressure distributions, as well as the aerodynamic forces and moments exerted on the blades are examined. It is clarified that the strong suction due to the stall and unstall vortices has a considerable influence on the unsteady aerodynamic characteristics of the blades. The development of a rotating stall of large amplitudes from various initial disturbances is also shown. Some important aspects of the rotating stall, such as inception point, the hysteresis, the number of stall cells, and its timewise changes are discussed and compared with the results deduced from a conventional linearized actuator disk theory. (Author)

A94-13507 3D displayed multifractal structure of turbulent dissipation field. I. HOSOKAWA (Univ. of Electro-Communications, Chofu, Japan), Physical Society of Japan, Journal (ISSN 0031-9015), Vol. 62, No. 7, July 1993, pp. 2204–2207. 9 Refs. Documents available from Aeroplus Dispatch.

A 3D generalized binomial Cantor set, which is in itself a multifractal, is used to display the 3D domain with strong energy dissipation (above a certain threshold). This artificially made display efficiently simulates the corresponding domain of a realization of isotropic turbulence by direct numerical simulation. It is concluded that all dissipative regions in real turbulence form the multifractal designated by this model, and that the domain surface has a fractal dimension corresponding to the threshold value.

A94-13433 Numerical study on unstart phenomena due to compound choking of scramjet engine inlets. I—Compound choking of inlets in a shear flow. II—Relation between thermal choking and compound choking. T. SATO and S. KAJI, Japan Society for Aeronautical and Space Sciences, Journal (ISSN 0021-4663), Vol. 41, No. 476, 1993, pp. 515–530. In Japanese. 5 Refs. Documents available from Aeroplus Dispatch.

Unstart phenomena due to compound choking of airframe-integrated scramjet engine inlets were investigated numerically. It is shown that the boundary layer flow spreads and pushes out the main flow in engine inlets and that engines designed assuming a uniform inlet flow can be made unstart easily because of the existence of a low flow speed region like a boundary layer. When sidewalls of engines sweep backward, the transition from start to unstart is rather continuous and the inlet performance of unstart conditions is not so badly deteriorated as that of inlets without sweep. The effect of the amount and position of heating in a combustion chamber on thermal choking is studied and the relation of compound choking on thermal choking is investigated. It is found that the inlet can be thermally choked quite easily in the presence of boundary layers and that the choked flow condition satisfies the same characteristics of compound choking as the case of a shear flow without heating. It is also found that the transition to unstart conditions can be delayed by expanding the region of heat addition towards the rear portion of the engine.

A94-10797 A modified version of the Reynolds-stress model applicable right up to a wall and its application to turbulent Couette flow. M. KANO (Gumma College of Technology, Maebashi, Japan), Japan Society for Aeronautical and Space Sciences, Transactions (ISSN 0549-3811), Vol. 36, No. 112, Aug. 1993, pp. 59–71. 22 Refs. Documents available from Aeroplus Dispatch.

The Reynolds-stress model, proposed by Shima (1988) as applicable right up to a wall, has been partly modified, and the performance of this model has been investigated in a turbulent channel flow. Investigated in all respects in this paper, the modified version of the Reynolds-stress model is found superior or at least equal to the existing Shima model and anisotropic k-epsilon model. The proposed model is applied to a turbulent Couette flow to confirm the universal validity of this model. In this flow, the turbulence has two strong characters. Near the wall the three components of the turbulence intensities differ significantly and are similar to those of the wall turbulence. But the core region exhibits strong indications of homogeneity in turbulence structure. The proposed model describes these properties quantitatively. (Author (revised))

A94-10359 A fundamental study on second-moment closure of turbulent scalar transport. N. SHIKAZONO and N. KASAGI, Japan Society of Mechanical Engineers, Transactions B (ISSN 0387-5016), Vol. 59, No. 563, July 1993, pp. 2279–2286. In Japanese. 21 Refs. Documents available from Aeroplus Dispatch.

This study focuses on the second-moment closure of turbulent scalar transport. Special attention was given to the characteristic time scales of scalar-pressure gradient correlation and dissipation terms in the turbulent scalar flux transport equation. The correlation coefficients of those terms and the turbulent scalar flux were introduced in order to represent influences of the Prandtl number nu/alpha, time scale ratio k(theta)epsilon/epsilon(theta)k, and turbulent Reynolds number k-squared/nu(epsilon). An effort was also made to take into account effects of the turbulence anisotropy produced by a strong mean shear rate. The proposed model provides excellent predictions of scalar flux budgets in isotropic turbulence, homogeneous shear flows, and fully developed channel flows. (Author (revised))

A94-10354 Comparison of several numerical methods for solving two-dimensional incompressible viscous flows. A. KIMURA, H. YANAGI, M. KUROKAWA, N. SATOFUKA, K. MORINISHI, and H. NISHIDA, Japan Society of Mechanical Engineers, Transactions B (ISSN 0387-5016), Vol. 59, No. 563, July 1993, pp. 2181–2187. In Japanese. 7 Refs. Documents available from Aeroplus Dispatch.

This paper presents a comparative study of several numerical methods which have been proposed for solving 2D incompressible viscous flows. These methods are the vorticity stream function method, the MAC method on a staggered grid, Chorin's (1992) artificial compressibility method with/without convergence acceleration, and Abdallah's (1991) method on a nonstaggered grid. The steady laminar flow in a square driven cavity and a 2D channel with 90 deg bend are chosen as test problems. It is found that all of the methods give practically the same computed results except for the pressure distributions in the 2D channel problem obtained by the vorticity stream function method. The numerical experiments show that the artificial compressibility method with convergence acceleration has the best convergence rate. (Author (revised))

A93-54316 Assessment of epsilon-equations for wall shear flows with DNS database. I—The case of k-epsilon two-equation modeling. Y. NAGANO, M. S. YOUSSEF, and M. SHIMADA, Japan Society of Mechanical Engineers, Transactions B (ISSN 0387-5016), Vol. 59, No. 562, June 1993, pp. 1972–1979. 18 Refs. Documents available from Aeroplus Dispatch.

Using direct numerical simulation (DNS) data base for wall turbulence, we performed critical assessment of representative dissipation-rate equations (i.e., epsilon of average-epsilon equations) for k-epsilon turbulence models and proposed a new dissipation-rate equation. The DNS data of mean velocity and turbulent kinetic energy are used as the known exact quantities for solving the epsilon or average-epsilon equation. Thus, the obtained values of epsilon are the true solutions with no contaminants. The assessment indicates that, except for one recent model, the performance in predicting the exact epsilon profiles is poor, irrespective of the model type. The proposed model constructed on the basis of the proposal of Nagano and Hishida (1987) provides a good performance, though the model formulation is very simple. (Author (revised))

A93-54314 Study on flow field around slender diamond cone traveling at hypersonic speed. M. NISHIO and M. KOTAKE, Japan Society of Mechanical Engineers, Transactions B (ISSN 0387-5016), Vol. 59, No. 562, June 1993, pp. 1925–1931. 8 Refs. Documents available from Aeroplus Dispatch.

This paper proposes a method to analyze the flow field around a slender diamond cone traveling at hypersonic speed. We used the equivalence principle to analyze the problem. In order to apply the equivalence principle, we assumed that the flow field around a slender diamond cone traveling at hypersonic speed coincides with the two-dimensional unsteady flow field around a diamond shape body expanding at a constant rate. Using conformal mapping, we solved the flowfield to satisfy the boundary condition around the expanding body. Using this analysis, the velocity distribution and the streamline around the body became clear. Consequently, we obtained the flow field around a slender diamond cone traveling at hypersonic speed. (Author (revised))

A93-53836 Numerical analysis on fully developed turbulent flow in a square duct with a rough wall. H. SUGIYAMA, M. AKIYAMA, M. MATSUMOTO, M. HIRATA, and N. NINOMIYA, Japan Society of Mechanical Engineers, Transactions B (ISSN 0387-5016), Vol. 59, No. 561, May 1993, pp. 1510–1517. 13 Refs. Documents available from Aeroplus Dispatch.

In this study, a numerical analysis is carried out for fully developed turbulent flow in a square duct with a toughened wall, using the Reynolds stress model. Calculated results are compared with the experimental data available. The present method can predict well a large eddy caused by the secondary flow of the second kind in a half cross section, and indicated the existence of a small eddy in the upper wall corner region. However, a small eddy which appeared in the calculated results is not recognized in the experimental data. Calculated results of normal stresses and turbulent energy are in comparatively good agreement with the experimental data. These examinations suggest that the present method utilizing the Reynolds stress model, classified by an anisotropic turbulent model of high Reynolds number, is able to predict the turbulent flow in a square duct with roughened walls. (Author (revised))

A93-53800 Direct numerical simulation of reacting mixing layers—Effect of heat release and density difference of mean flow. M. TANAHASHI and T. MIYAUCHI, Japan Society of Mechanical Engineers, Transactions B (ISSN 0387-5016), Vol. 59, No. 560, April 1993, pp. 1338–1343. 9 Refs. Documents available from Aeroplus Dispatch.

The objective of this paper is to clarify the effect of heat release and density difference in mean flow on the development of mixing layers. To achieve this purpose, we have performed a direct numerical simulation of a two-dimensional chemically reacting mixing layer by means of a spectral method. Low Mach number approximation was used to take into account the effect of density change. From these results, we have obtained the following conclusions. (1) Heat release causes baroclinic torque in the outside region of the coherent structure. Because of this baroclinic torque, the growth of the mixing layer and chemical reaction are suppressed. (2) The density difference of mean flow suppresses the growth of the mixing layer and chemical reaction because of the baroclinic torque and expansion. (3) The growth of the mixing layer and chemical reaction are further suppressed by the coexistence of heat release and density difference. (Author (revised))