AIAA JOURNAL
Vol. 35, No. 7, July 1997

International Aerospace Literature

During 1997 the *AIAA Journal* will carry selected aerospace literature abstracts on leading research topics from Russia, Japan, France, Germany, Italy, and the United Kingdom. The topics will be chosen and the abstracts reviewed for pertinency by *AIAA Journal* editors. This month feature Transonic Flows from Russia, Japan, France, Germany, Italy, and the United Kingdom.

Support for assembling and publishing the selected abstracts has been provided by the Innovative Science and Technology Directorate of the Strategic Defense Initiative Organization (SDIO), with the sponsorship and technical management of the abstract service by the Office of Naval Research (ONR) under ONR Grant N00014-93-I-1074.

Abstracts in this listing have been taken from the monthly abstract journal *International Aerospace Abstracts* (IAA), published by the American Institute of Aeronautics and Astronautics (AIAA). Additional materials can be obtained through searching the Aerospace Database—available online via DIALOG or STN.

Where noted, paper copies of the original documents cited are available from AIAA Dispatch, 5109 Cherry St., Kansas City, MO 64110; (800) 662-1545. Use the "A" number to identify material you want. Please be advised that most of the original documents are in the original language. Direct questions concerning this abstract section of the *AIAA Journal* to Norma Brennan at AIAA Headquarters.

Russian Aerospace Literature
This month: *Transonic Flows*

A97-13283 Wave drag of bodies of revolution in unsteady transonic flow (Volnovoe soprotivlenie tel vrashcheniya v nestatsionarnom transzvukovom potoke). M. A. NAJDA and A. S. FONAREV (TsAGI, Zhukovsky, Russia). *PMTF—Prikladnaya Mekhanika i Tekhnicheskaya Fizika* (ISSN 0869-5032), Vol. 37, No. 3, 1996, pp. 35–44. 11 Refs. In Russian. Documents available from AIAA Dispatch.

In a previous study (Fonarev, 1993), the integral impulse theorem was used to determine the unsteady wave drag of bodies in plane transonic flow and to obtain the time dependence of the aerodynamic characteristics of an airfoil interacting with a wind gust or a moving shock wave. Here, the results obtained in the previous study are extended to the case of unsteady transonic flow past axisymmetric bodies. The problem is solved using assumptions of the nonlinear transonic theory of small perturbations. An unsteady 'high-frequency' equation for the velocity potential is investigated which makes it possible to consider aperiodic (including discontinuous) changes in flow parameters, such as sudden wind gusts and arriving shock waves.

A97-13205 Numerical simulation of spontaneously condensing steam flows in transonic nozzles. L. I. ZAICHIK, A. S. BOLDAREV, V. A. GASILOV, and O. G. OLKHOVSKAYA (Russian Academy of Sciences, Inst. for Mathematical Modeling, Moscow, Russia) and F. BARON, B. COLLIGNAN, and A. R. LAALI (Electricite de France, Chatou). *Aerothermodynamics of internal flows III; Proceedings of the 3rd International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows,* Beijing, China, 1996 (A97-13101 02-34), World Publishing Corp., Beijing, 1996, pp. 875–880. 6 Refs.

A Eulerian numerical method of modeling wet steam flows is proposed. The governing equations for the condensed matter are derived from the kinetic equation for the droplet distribution function by an averaging procedure. The developed method is applied to the computer simulation of stationary as well as nonstationary transonic flows of overcooled steam under the conditions of spontaneous (homogeneous) condensation. The main purpose of the work is a comparative analysis of various approaches to the simulation of spectrum evolution of droplets' radii. The results of quasi-one-dimensional and two-dimensional modeling studies are also discussed. (Author)

A96-41894 Calculation of vortex flows of a gas in channels of complex configurations (K raschetu vikhrevykh techenij gaza v kanalakh slozhnykh konfiguratsij). O. B. KHAJRULLINA (RAN, Inst. Matematiki i Mekhaniki, Yekaterinburg, Russia). *PMTF—Prikladnaya Mekhanika i Tekhnicheskaya Fizika* (ISSN 0869-5032), Vol. 37, No. 2, 1996, pp. 103–108. 12 Refs. In Russian. Documents available from AIAA Dispatch.

A combined approach to the calculation of vortex flows of a gas in channels of complex configurations is proposed whereby the entire flow region is divided into three overlapping subregions: subsonic, transonic, and supersonic. The method has been implemented in a software package, SOKOL, running on a personal computer. In the subsonic region the flow is calculated by using a finite difference iterative procedure in current function-vortex variables; in the transonic region the flow is calculated by using a McCormack finite difference scheme, included as the TRANS module into the SOKOL code. An illustrative example is presented.

A96-40802 Aerodynamic design of swept infinite wing with flap for subsonic and transonic flight regimes. K. V. KOVALEV (Moscow Inst. of Physics and Technology, Russia). *Proceedings of the ICAS 20th Congress,* Naples, Italy, 1996, Vol. 2 (A96-40679 11-01), AIAA, Inc., Reston, VA, 1996, pp. 2538–2548. 6 Refs.

The methodology is based on the solution of the full potential equation in conservative form with the combined boundary conditions, in which the no-flux condition is set on one part of the wing surface, and the specified pressure distribution is set on another part. The solution of this problem results in determining the surface distribution of the normal velocity. Then the wing shape is changed according to this distribution. This process may be repeated until convergence. The trailing edge closure is provided by specifying the additional normal velocity distribution on the surface. Such an approach makes it possible to control various shape parameters, thickness or camber distribution, etc. The full potential equation is solved using the finite-difference method in the curvilinear airfoil and flap surfaces-fitted coordinate system. The spatial potential derivatives are approximated by a central-difference scheme in subsonic regions of the flow and by an upwind-difference scheme in supersonic regions; both schemes are of the second order of accuracy. The system of finite-difference equations is solved by Newton's procedure; the linear system is solved by the incomplete LU factorization method. The results include different test design cases, demonstrating the abilities of the current design method. (Author)

A96-40767 Application of viscous-inviscid interaction methods for a separated flow calculation about airfoils and high-lift systems. S. V. LYAPUNOV and A. V. WOLKOV (TsAGI, Zhukovsky, Russia). *Proceedings of the ICAS 20th Congress,* Naples, Italy, 1996, Vol. 2 (A96-40679 11-01), AIAA, Inc., Reston, VA, 1996, pp. 2194–2202. 22 Refs.

An approach is considered for calculation of steady attached and separated two-dimensional flows. The approach is based on a viscous-inviscid interaction model. A quasi-simultaneous method of viscous-inviscid coupling is used with a specific boundary condition for an inviscid flow when separation occurs. This approach rapidly converges for both attached and separated flows. Two computer codes based on this approach are developed, one for calculating a transonic flow about a single airfoil and the other for calculating a subsonic flow about multi-element high-lift systems. Calculation of outer (inviscid) flow is carried out by the panel method for low speeds or by solving a modified transonic potential equation for high subsonic speeds. Inner (viscous) flow is described by laminar and turbulent boundary layer equations in integral form. In the regions of pre-separated and separated flow, these equations are solved in the inverse mode in order to avoid the Goldstein singularity at the separation and reattachment points. Calculated results show good agreement with experimental results for both single and multi-element cases. (Author)

A96-39068 Some aspects of helicopter airfoil design. A. A. NIKOLSKY (TsAGI, Moscow, Russia). *Proceedings of the 21st European Rotorcraft Forum,* St. Petersburg, Russia, 1995, Vol. 1 (A96-39051 10-01), NLR, Amsterdam, Netherlands, 1995, pp. II.18.1–II.18.10. 6 Refs.

Two methods of improving helicopter airfoil performance are presented. The first combines the design by an optimization method based on the complex method of nonlinear programming with a new approach to the choice of design variables, providing an effective tool for satisfying several aerodynamic and geometric constraints simultaneously. The second method involves designing an airfoil with prescribed chord or arc length pressure distribution at a given Mach number by solving the nonlinear transonic inverse problem. (Author)

A96-37257 Numerical and experimental investigation of two-dimensional asymmetric nozzles. V. N. ZUDOV, A. V. LOKOTKO, and A. I. RYLOV (Russian Academy of Sciences, Inst. of Theoretical and Applied Mechanics, Novosibirsk, Russia). *AIAA, ASME, SAE, and ASEE 32nd Joint Propulsion*

Conference and Exhibit, Lake Buena Vista, FL, 1996, p. 12 (AIAA Paper 96-3141). 9 Refs.

A series of works on the construction of optimal supersonic two-dimensional asymmetric nozzles and on the investigation of their off-design regimes was carried out. The variation problem of the design of the supersonic two-dimensional asymmetric nozzles with maximum thrust at given lift power and given size limitations was solved. The method of characteristics was used. Structures of flow and power characteristics in nozzles are numerically investigated at transonic and supersonic speeds of external flow. It was experimentally and numericaly studied power characteristics and three-dimensional phenomena of two-dimensional asymmetric nozzle. The problem of the subsonic efflux of the viscous perfect gas jet into the cocurent flow of the same gas was considered. As a result of calculations the major elements of the subsonic turbulent flow structure were defined. A satisfactory agreement between the experimental data and data calculated on the basis of *q–e* (Jones–Launder) turbulence model was observed the universal profile of the mixing layer. (Author)

A96-36532 Analysis of transonic flow by method of viscous-inviscid interaction—Correction for wind tunnel wall interference. Y. B. LIFSHITZ, I. A. SOLNSTEV, and S. A. VELICHKO (TsAGI, Moscow, Russia). *AIAA Fluid Dynamics Conference,* New Orleans, LA, 1996, p. 16 (AIAA Paper 96-1981). 28 Refs.

An application of the viscous-inviscid numerical method to a problem of wind-tunnel-wall interference needs additional boundary conditions on the walls and on the entry and the exit boundaries of test section. The first ones are the same as in a theory of ideal porous or slotted wall; correct entry and exit conditions are taken to obtain well-posed boundary-value problems. The boundary conditions on a body are the same as in the case of free air flow. Numerical results serve for obtaining new properties of wall influence, particularly at high transonic velocities; they show that dependence of aerodynamic coefficients on the height *H* of the tunnel is different for flows with and without shock-induced separation. This result is explained by an asymptotic analysis. Other asymptotic results correlate well with numerical data. If pressure coefficient distribution on the tunnel walls is measured during the tests, our method can be used for correction of the wall interference. (Author)

A96-24297 Numerical simulation of transonic flow around a wing airfoil in a wind tunnel (Chislennoe modelirovanie transzvukovogo obtekaniya profilya kryla v aehrodinamicheskoj trube). S. A. VELICHKO, Y. B. LIFSHITS, V. M. NEJLAND, I. A. SOLNTSEV, and A. M. SOROKIN. *Zhurnal Vychislitel'noj Matematiki i Matematicheskoj Fiziki* (ISSN 0044-4669), Vol. 35, No. 10, 1995, pp. 1518–1537. 23 Refs. In Russian. Documents available from AIAA Dispatch.

A method is developed for the numerical simulation of transonic flow of a viscous gas around an airfoil in the test section of a wind tunnel. The effect of viscosity is taken into account only in the boundary layer on the airfoil and in its wake, with potential flow assumed elsewhere. The solutions in the two regions are matched in the context of viscous-inviscid interaction. The inlet and outlet conditions for the test section, which would ensure the existence and uniqueness of a solution to the boundary value problem are discussed. The boundary value problem is solved numerically using the WINTUN program, specially developed for this purpose. At each iteration, the viscous-inviscid interaction problem in the internal region is solved by using the AIRFOIL code, developed previously for the numerical simulation of infinite flow of a viscous gas around an airfoil.

A96-23744 Distribution of pressure over the nozzle wall of hypersonic ramjet engines under transonic conditions. S. P. PETROV, V. B. ROUTOVSKY, and S. K. SITNIKOW (Moscow Aviation Inst., Russia). *La Recherche Aerospatiale* (ISSN 0034-1223), No. 1, 1996, pp. 71–74. 5 Refs. Documents available from AIAA Dispatch.

In the design of high-speed aircraft for atmospheric and near-space flight, it has been proposed to use a liquid-fuelled rocket engine (LFRE) and hypersonic ramjet engine (HSRE) as the main power plant. Under different conditions it is possible to use additional LFREs or other REs (combined as a rule). But in transonic flight at encircling flow Mach numbers (Me) of 1–2, while the main engine is not working yet, the HSRE nozzle is operating under far-from-design conditions. Break-off zones are possible, leading to the formation of low pressure zones, and, therefore, to considerable thrust loss, since the longitudinal dimension of these zones may be quite large. The purpose of this investigation is to define the general tendency of the static pressure distribution over the open-type nozzle surfaces, under speed variations in the transonic range Me 1–2, and to estimate the decrease in the extent of the negative pressure region and rarefaction level in the forward portion of the nozzle due to a LFRE booster in that region.

A96-20199 Nonclassical boundary value problem for a three-dimensional viscous transonic equation (Neklassicheskaya kraevaya zadacha dlya prostranstvennogo vyazkogo transzvukovogo uravneniya). Y. V. ZASORIN. *Zhurnal Vychislitel'noj Matematiki i Matematicheskoj Fiziki* (ISSN 0044-4669), Vol. 35, No. 9, 1995, pp. 1401–1419. 13 Refs. In Russian. Documents available from AIAA Dispatch.

For a three-dimensional viscous transonic equation, a nonclassical boundary value problem is analyzed which has a system of boundary conditions of a special kind on a one-dimensional manifold. The solution is obtained in the form of a functional series of a special kind. The convergence of the series is investigated, and the solvability of the problem in a space of analytical functionals is proved.

A96-17938 An efficient method for calculating the wave drag of bodies of revolution in the transonic velocity range (Ehffektivnyj metod rascheta volnovogo soprotivleniya tel vrashcheniya v transzvukovom diapazone skorostej). M. A. NAJDA and A. S. FONAREV (TsAGI, Zhukovsky, Russia). *PMTF—Prikladnaya Mekhanika i Tekhnicheskaya Fizika* (ISSN 0869-5032), Vol. 36, No. 3, 1995, pp. 60–68. 13 Refs. In Russian. Documents available from AIAA Dispatch.

An expression is obtained for the wave drag of bodies of revolution in stationary transonic flow, and an algorithm for computing transonic flow past bodies of revolution is developed, which represents an axisymmetric analog of the variable direction method with a monotone algorithm. Results of numerical calculations of the wave drag of several bodies of revolution are compared with experimental data. The drag of a complex three-dimensional flight vehicle configuration in the transonic velocity range is determined numerically using the transonic equivalence rule.

A96-17857 Initial section of a gas combustion jet with nonsymmetric boundary conditions (Nachal'nyj uchastok gazovogo fakela goreniya s nesimmetrichnymi granichnymi usloviyami). G. N. ABRAMOVICH (Moskovskij Gosudarstvennyj Aviatsionnyj Inst., Moscow, Russia). *Aviatsionnaya Tekhnika* (ISSN 0579-2975), No. 3, 1995, pp. 39–43. 4 Refs. In Russian. Documents available from AIAA Dispatch.

An approximate theory is developed for the initial section of a diffusive gas combustion jet with nonsymmetric boundary conditions (with different velocities, temperatures, and thermodynamic properties at the opposite sides of the gas flow) in the turbulent regime. The theory is valid for plane and circular jets. Full combustion in the thin layer (stoichiometric flame front) is assumed. Results of calculations for hydrogen, methane, and propane flames in air are presented for the same temperature in the initial section for subsonic or transonic flow velocities.

A96-17851 Numerical modeling of the nonlinear interaction of an elastic shell with gas flow (Chislennoe modelirovanie nelinejnogo vzaimodejstviya uprugoj obolochki s potokom gaza). M. A. IL'GAMOV and A. L. TUKMAKOV (Russian Academy of Sciences, Inst. of Mechanics and Engineering, Kazan, Russia). *Aviatsionnaya Tekhnika* (ISSN 0579-2975), No. 3, 1995, pp. 3–9. 11 Refs. In Russian. Documents available from AIAA Dispatch.

Results of a numerical modeling of supersonic and transonic transverse flow past an elastic cylindrical shell are presented. Nonstationary shell deformations, flow field around the shell, and its drag coefficient are determined using a two-dimensional formulation. The pressure inside the shell, freestream Mach, and other parameters are analyzed, with allowance made for the relative wall thickness.

A96-16095 Dynamic test rig and test technique for the aircraft models unsteady aerodynamic characteristics measurements in high subsonic and transonic wind tunnels. S. V. KABIN, K. A. KOLINKO, A. N. KHRABROV, and P. D. NUSHTAEV (TsAGI, Zhukovsky, Russia). *ICIASF '95—16th International Congress on Instrumentation in Aerospace Simulation Facilities,* Dayton, OH, 1995, Record (A96-16069 03-35), Inst. of Electrical and Electronics Engineers, New York, 1995, pp. 26.1–26.7. 5 Refs.

The new dynamic test rig is designed in TsAGI to investigate the aerodynamic damping derivatives of the aircraft models in wind tunnels at high subsonic and transonic speeds. The action of the rig is based on the free oscillations method with the use of elastic elements. The dynamic test rig is equipped with the five component strain gage balance. Both direct and cross aerodynamic damping derivatives can be measured. The comparison of the results for the Standard Dynamic Model (SDM) is presented. (Author)

A96-12078 Steady state interaction of elastic turbomachinery blades with three-dimensional flow of an ideal gas (Statsionarnoe vzaimodejstvie uprugikh lopatok turbomashin s prostranstvennym potokom ideal'nogo gaza). Y. S. BULCHINSKIJ and A. S. LIBERZON. *Rossijskaya Akademiya Nauk, Izvestiya, Mekhanika Tverdogo Tela* (ISSN 0572-3299), No. 2, 1995, pp. 168–174. 11 Refs. In Russian. Documents available from AIAA Dispatch.

The problem of the steady state interaction of turbine blades with three-dimensional flow of an ideal gas is analyzed by using a mathematical model based on the linear theory of shallow anisotropic elastic shells of variable thickness in three-dimensional transonic flow of an ideal gas. The loads transmitted to the shell contain components due to flow past the rigid carrier surface, in addition to components due to deformation of the elastic elements. Results of a numerical parametric analysis and deformability estimates for a composite fan blade of an aircraft gas turbine engine are presented.

A96-10244 Characteristics of the interference of impulse shock waves with turbulent shear layers (Osobennosti interferentsii impul'snykh udarnykh voln s turbulentnymi sdvigovymi sloyami). G. F. GLOTOV. *Transactions of Theory and design of flight-vehicle engines; 17th Scientific Lectures on Astronautics Dedicated to the Memory of Outstanding Scientist Pioneers of Space Exploration,* Moscow, Russia, 1993 (A96-10228 01-07), Moscow, Fazis, 1993, pp. 67–71. 7 Refs. In Russian. Documents available from AIAA Dispatch.

Experiments with shock waves of the impulse type are reported in which the characteristics of shock wave interaction with various types of shear layers (turbulent boundary layers, vortex filaments, and wakes) were investigated for Mach 2.5–3.5 and *Re* = 10 exp 6. The interaction characteristics investigated are related to the separation phenomena and depend primarily on the wave propagation velocity relative to the shear layer (supersonic/transonic). The flow patterns were recorded by schlieren photography. The results of the study are used in the analysis of detonation combustion in supersonic flow.

A95-45141 Shock wave formation in transonic flow with a local supersonic region (Formirovanie udarnykh voln v transzvukovom techenii s mestnoj sverkhzvukovoj oblast'yu). A. G. KUZ'MIN. *Sankt-Peterburgskij Universitet, Vestnik, Seriya 1—Matematika, Mekhanika, Astronomiya* (ISSN 0024-0850), No. 3, 1994, pp. 75–80. 9 Refs. In Russian. Documents available from AIAA Dispatch.

The paper is concerned with plane nonviscous flow of a gas in a channel with parallel walls and a small projection on the lower wall simulating an airfoil. The formation of shock waves is investigated numerically in the case of uniform deviations of the airfoil from the shock-free configuration over the entire supersonic region of its surface; the qualitative flow patterns are determined as a function of the deviation. The direct problem of flow over an airfoil of specified shape is solved via the decomposition of the region considered into subsonic and supersonic subregions using a fine grid, which makes it possible to identify the fine flow structure.

A95-44693 Investigation of hypersonic plane exhausted nozzle performance at transonic speeds. N. A. DUBOV, A. P. MAZUROV, and E. V. PAVLYUKOV (TsAGI, Zhukovski, Russia). *ISABE—Proceedings of the 12th International Symposium on Air Breathing Engines*, Melbourne, Australia, 1995, Vol. 1 (A95-44654 12-07), AIAA, Washington, DC, 1995, pp. 427–437 (ISABE 95-7038). 7 Refs.

Results of an experimental investigation of models of exhaust systems of a combined propulsion system consisting of parallel turbojet and scramjet engines for a hypersonic flight vehicle are presented. Tests were conducted at free-stream Mach numbers of 0.6–1 and at turbojet nozzle pressure ratios of 2–5.5. The influence of main geometric parameters on the nozzle thrust performance is discussed. It is shown, in particular, that the boattail cross-section area distribution and the turbojet nozzle exit position with respect to the scramjet nozzle significantly affect the nozzle thrust losses. (Author)

A95-39038 Investigation of transonic flow past thin bodies of 3-D configuration by a bounded gas flow (Issledovanie okolozvukovogo obtekaniya tonkikh tel prostranstvennoj konfiguratsii ogranichennym potokom gaza). K. G. SAYADYAN. *Rossijskaya Akademiya Nauk, Izvestiya, Mekhanika Zhidkosti i Gaza* (ISSN 0568-5281), No. 2, 1995, pp. 149–158. 5 Refs. In Russian. Documents available from AIAA Dispatch.

A simultaneous solution of boundary value problems for the near and far flow fields is used to obtain the pressure coefficient distributions for the flow past a 'Khotol' type aircraft by a bounded gas stream for various transonic regimes in a pipe with a perforated throat. These distributions are compared with the pressure coefficient distribution in the case of unbounded flow past a body. An additional pressure coefficient induced by the walls is calculated for the subsonic flow past the aircraft in a large-radius cylindral pipe. Good agreement is found between the calculations and Malmuth's asymptotic theory for subsonic velocities.

N95-19274 Optical surface pressure measurements: Accuracy and application field evaluation. A. BUKOV, V. MOSHAROV, A. ORLOV, V. PESETSKY, V. RADCHENKO, S. PHONOV, S. MATYASH, M. KUZMIN, and N. SADOVSKII (Moscow M. V. Lomonosov Inst. of the Technology of Fine Chemicals, USSR). *AGARD, Wall Interference, Support Interference and Flow Field Measurements*, p. 9 (SEE N95-19251 05-34).

Optical pressure measurement (OPM) is a new pressure measurement method rapidly developed in several aerodynamic research centers: TsAGI (Russia), Boeing, NASA, McDonnell Douglas (all USA), and DLR (Germany). Present level of OPM-method provides its practice as standard experimental method of aerodynamic investigations in definite application fields. Applications of OPM-method are determined mainly by its accuracy. The accuracy of OPM-method is determined by the errors of three following groups: (1) errors of the luminescent pressure sensor (LPS) itself, such as uncompensated temperature influence, photo degradation, temperature and pressure hysteresis, variation of the LPS parameters from point to point on the model surface, etc.; (2) errors of the measurement system, such as noise of the photodetector, nonlinearity and nonuniformity of the photodetector, time and temperature offsets, etc.; and (3) methodological errors, owing to displacement and deformation of the model in an airflow, a contamination of the model surface, scattering of the excitation and luminescent light from the model surface and test section walls, etc. OPM-method allows getting total error of measured pressure not less than 1%. This accuracy is enough to visualize the pressure field and allows determining total and distributed aerodynamic loads and solving some problems of local aerodynamic investigations at transonic and supersonic velocities. OPM is less effective at low subsonic velocities (M less than 0.4), and for precise measurements, for example, an airfoil optimization. Current limitations of the OPM-method are discussed on an example of the surface pressure measurements and calculations of the integral loads on the wings of canard-aircraft model. The pressure measurement system and data reduction methods used on these tests are also described. (Author)

A95-28384 Experimental studies of fluctuations structure in compressible flows. V. A. LEBIGA and V. N. ZINOV'EV (Russian Academy of Sciences, Inst. of Theoretical and Applied Mechanics, Novosibirsk, Russia). *PICAST'1 1993—Proceedings of the Pacific International Conference on Aerospace Science and Technology*, National Cheng Kung Univ., Tainan, Taiwan, 1993, Vol. 2 (A95-28244 07-99), pp. 988–995. 21 Refs.

Some results of investigations of nonstationary processes in compressible flows (both subsonic and supersonic) carried out in the Institute of Theoretical and Applied Mechanics of the Siberian Division of the Russian Academy of Sciences are presented. Measurements of turbulent fluctuations behind nozzles of different types (Laval, screen, ventilated, etc.) in subsonic and supersonic flows are discussed. Examples of fluctuation measurements in boundary layers at supersonic velocities are given. A new approach to the interpretation of acoustic fluctuation measurements based on a mode diagram method has been used to study hot-wire data on the intensity, spectra, and mode composition of fluctuations in transonic flows in channels with different types of walls (rigid, perforated, slotted, with jets along the walls, etc.) for different typical sources of fluctuations. New information about the structure and sources of disturbances in test sections of wind tunnels is adduced. (Author)

A95-28363 Perforated wall interference corrections for model testing at high subsonic Mach numbers. O. K. SEMENOVA and A. V. SEMENOV (TsAGI, Zhukovski, Russia). *PICAST'1 1993—Proceedings of the Pacific International Conference on Aerospace Science and Technology*, National Cheng Kung Univ., Tainan, Taiwan, 1993, Vol. 2 (A95-28244 07-99), pp. 859–863. 5 Refs.

A subsonic problem of perforated wall interference corrections for lifting models is considered. Different approaches for determining and decreasing the effect of wind tunnel walls are presented for tests at high subsonic Mach numbers. Wall pressure measurements with known boundary conditions permit one to check lifting model schematic representations and to take into account all flow features essential for the problem. Some aspects of model representation for high incidence tests are illustrated. An appropriate boundary equation porosity parameter corresponds to real perforated wall characteristics of the wind tunnel, and the interference flowfield is predicted for arbitrary porosity distribution on test section walls. Additional wall-induced flowfield data are interpreted as wall interference corrections. The predicted wall interference corrections are compared with the experimental results obtained for a large transonic wind tunnel with sectionally variable permeability. Some results are presented on a test condition optimization procedure which decreases interference velocity and velocity gradients in the model location. (Author)

A95-26229 AEROFOIL—A computer codes library for numerical analysis of viscous transonic flow around a wing section. Y. B. LIFSHITS and S. A. VELICHKO (TsAGI, Zhukovski, Russia). *La Recherche Aerospatiale* (ISSN 0034-1223), No. 2, 1995, pp. 73–83. 25 Refs.

The AEROFOIL codes library is intended for computational study of subsonic and transonic viscous gas flow around an airfoil at high Reynolds number. It includes several sections which allow the preparation of input information, computation of the flow, analysis of the results, and comparison of different flows. The library enables the numerical data to be saved in archives and supports rapid extraction of different results. The main functional part of the library contains the subroutines for flow simulation based on the interaction of the inviscid flow with viscous shear layers. A modified transonic potential theory is used in the outer domain, making it possible to obtain solutions including shock waves which are very close to those of the Euler equations. The outer solution is coupled with the viscous layer in such a way that the same number of iterations is required for attached or separated flow. Some examples presented for code validation show that the AEROFOIL results agree well with the experimental data and with solutions of the Navier–Stokes equations. Moreover, they can be obtained using much less computer time. (Author)

A95-22654 Transonic vortical gas flows. E. G. SHIFRIN and O. M. BELOTSERKOVSKII (Russian Academy of Sciences, Inst. of Computer Aided Design, Moscow, Russia). Chichester, UK and New York, John Wiley and Sons, 1994, p. 371 (ISBN 0-471-95066-1). 160 Refs. Documents available from AIAA Dispatch.

The present volume discusses general properties of the transonic flow of an ideal gas. The flow in the vicinity of the point of orthogonality in the sonic line to the velocity vector is discussed. Problems and theoretical investigation, and numerical methods for solving profiling and direct problems of the Laval nozzle are presented. Attention is also given to the subcritical flow over a profile, supercritical subsonic flow over a profile, transonic flow over a corner point, transonic flow behind a detached shock wave in supersonic gas flow about a body, and secondary shocks.

A95-17524 Numerical solution of Navier–Stokes equations using iterative methods of the variational kind (Chislennoe reshenie uravnenij Nav'e-Stoksa s ispol'zovaniem iteratsionnykh metodov variatsionnogo tipa). I. Y. BABAEV, V. A. BASHKIN, and I. V. EGOROV. *Zhurnal Vychislitel'noj Matematiki i Matematicheskoj Fiziki* (ISSN 0044-4669), Vol. 34, No. 11, 1994, pp. 1693–1703. 8 Refs. In Russian. Documents available from AIAA Dispatch.

An efficient numerical procedure is proposed for solving grid equations arising in the finite difference approximation of steady state Euler and Navier–Stokes equations by monotonic schemes. The procedure employs the Newton method, LU decomposition, and the generalized minimum-residual iterative method. Results of calculations of transonic flow of an ideal gas past a NACA0012 airfoil are presented.

A95-17522 Exact solutions for some external problems described by unsteady viscous transonic equations (Tochnye resheniya nekotorykh vneshnikh zadach, opisyvaemykh nestatsionarnymi vyazkimi transzvukovymi uravneniyami). Y. V. ZASORIN. *Zhurnal Vychislitel'noj Matematiki i Matematicheskoj Fiziki* (ISSN 0044-4669), Vol. 34, No. 10, 1994, pp. 1476–1488. 11 Refs. In Russian. Documents available from AIAA Dispatch.

Green's functions are obtained in explicit form for a series of initial-boundary value problems associated with unsteady viscous transonic equations. Some aspects of the correct solvability of the problems in Kipriyanov–Schwartz spaces are examined.

Japanese Aerospace Literature
This month: *Transonic Flows*

A97-15837 Unsteady aerodynamics measurements on an elastic wing model of SST. M. TAMAYAMA, H. MIWA, and J. NAKAMICHI (National Aerospace Lab., Chofu, Japan). *AIAA 35th Aerospace Sciences Meeting and Exhibit*, Reno, NV, 1997, p. 16 (AIAA Paper 97-0836). 7 Refs.

Unsteady aerodynamics measurements on a half model of SST arrow wing were conducted in the Transonic Wind Tunnel at National Aerospace Laboratory. The model is an elastic one designed for flutter tests. It has a partial flap along the trailing edge. The unsteadiness of the flow was generated by the forced oscillation of the flap. The model is equipped with 36 unsteady pressure sensors and four accelerometers. The data were acquired in the transonic regime ($M = 0.8$–0.9) at angles of attack from 0 to 4 deg. The flap is oscillated around a mean deflection angle in a frequency range up to 30 Hz. Pressure distributions at two span stations and the accelerations at tip region were measured. The data were processed in time domain as well as in frequency domain. The Cp distributions and power spectrum distributions are presented. The coherence and the phase differences between the pressure signals and the flap deflections are also shown. The influences of flow conditions upon the unsteady pressure distributions and the dynamic displacements of the wing are discussed. (Author)

A97-13122 Experimental study of centrifugal impeller/diffuser interaction for transonic discharge rates. K. YAMAGUCHI and T. NAGASHIMA (Tokyo Univ., Japan). *Aerothermodynamics of internal flows III; Proceedings of the 3rd International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows*, Beijing, China, 1996 (A97-13101 02-34), World Publishing Corp., Beijing, 1996, pp. 175–185. 11 Refs.

To fill the gap between practice and theoretical predictions of centrifugal compressor performance, a series of experiments has been carried out to provide some cross-check results against numerical simulation of the unsteady impeller/diffuser interaction, in particular with high rotational speed and at large mass flow rates. The experimental data of pressure measurements revealed interesting flow features in the diffuser passage associated with the flow choking and the shock wave occurrence, which were also confirmed by schlieren flow visualization. (Author)

A96-41403 Orbital re-entry experiment vehicle ground and flight dynamic test results comparison. T. YOSHINAGA, A. TATE, and M. WATANABE (National Aerospace Lab., Chofu, Japan) and T. SHIMODA (NASDA, Tokyo, Japan). *Journal of Spacecraft and Rockets* (ISSN 0022-4650), Vol. 33, No. 5, 1996, pp. 635–642. 12 Refs.

The angular motion of an orbital reentry experiment (OREX) vehicle model was tested in a transonic wind tunnel using a single-degree-of-freedom test method to investigate the angular motion of the OREX vehicle. The purpose of the OREX is to test heat shield materials and to obtain measurements of an aerodynamically heated flowfield. Approximate static and dynamic derivatives of the model were measured by a local linear curve-fitting method. The angular motion of real OREX flight data was compared with the wind-tunnel test results. The maximum amplitude of the OREX vehicle in the transonic region is almost the same as that obtained in the wind-tunnel test. (Author)

A96-40775 Effect of lateral blowing on aerodynamic characteristics of low aspect-ratio wings at high angles of attack. K. KARASHIMA and K. SATO (Inst. of Space and Astronautical Science, Sagamihara, Japan). *Proceedings of the ICAS 20th Congress*, Naples, Italy, 1996, Vol. 2 (A96-40679 11-01), AIAA, Inc., Reston, VA, 1996, pp. 2271–2277. 3 Refs.

Measurements of aerodynamic characteristics of clipped delta wings with a low aspect ratio are made in a wind tunnel at subsonic/transonic Mach numbers to investigate the feasibility of the enhancement of wing performance by lateral blowing. It is shown that blowing brings about a considerable lift increase in wide ranges of angles of attack and freestream Mach numbers without serious degradation of lift to drag ratio. The aerodynamic mechanism of the lift augmentation is attributed to two processes. One is a circulation enhancement that results from a favorable modification of the pressure on the upper surface of the wings and takes place at low angles of attack. The other is due to the fact that the blowing enhances the leading-edge vortex so as to delay its breakdown and occurs at high angles of attack. Moreover, it is emphasized that lateral blowing is more efficient for increasing the lift than the so-called spanwise one in the sense of blowing momentum. (Author)

A96-40754 Robust stabilization of a transonic flutter based on a linearized transonic math model. H. MATSUSHITA, K. SAITOH, and M. HASHIDATE (National Aerospace Lab., Tokyo, Japan). *Proceedings of the ICAS 20th Congress*, Naples, Italy, 1996, Vol. 2 (A96-40679 11-01), AIAA, Inc., Reston, VA, 1996, pp. 2090–2098. 14 Refs.

To the benefit of simplicity from linear control, transonic aerodynamics is linearized to obtain a linear aeroelastic math model, and the robust control design method is applied to yield linear control laws. An aeroelastic model of high aspect ratio wing with a leading and a trailing edge control surface was designed and fabricated in order to investigate the applicability of active control to manage transonic flutter. Flutter tests were carried out in the transonic wind tunnel at the National Aerospace Laboratory, and the model exhibited typical transonic nonlinear phenomena, such as a transonic dip and limit cycle oscillation. A transonic full potential code, USTF3, is extended to calculate the generalized aerodynamic forces due to eigenmode oscillation, and linear approximation is carried out by getting a describing function and fitting a linear finite state aerodynamic model to the calculated data. Since a finite state linearized model has many sources of modeling errors, the H-α loop-shaping method with the normalized left coprime factors approach is applied to provide a stability robustness against model uncertainty. (Author)

A96-40692 An inverse-direct hybrid Navier–Stokes solver using pseudo-analytic functions. TERUOMIYAZAKI (Kokushikan Univ., Tokyo, Japan). *Proceedings of the ICAS 20th Congress*, Naples, Italy, 1996, Vol. 2 (A96-40679 11-01), AIAA, Inc., Reston, VA, 1996, pp. 1468–1478. 6 Refs.

An inverse-direct hybrid Navier–Stokes solver using pseudo-analytic function theories, which can deal with numerous parameters and variables appearing in the unsteady three-dimensional compressible viscous solvers, is presented. An interchange of the necessary information between the direct and inverse methods can be smoothly and sufficiently executed without any artificial technique. The application of the present method to the conventional blade design is straightforward. Using calculated results for the unsteardy inlet transonic rotor flow and the linear cascade flows with and without air injection, the reconstruction of blade suction surfaces is illustrated by numerical examples. (Author)

A96-40618 Numerical simulation of flowfields of the NAL Two-dimensional Transonic Wind Tunnel. K. MATSUNO and N. SATOFUKA (Kyoto Inst. of Technology, Japan) and N. SUDANI and M. FUKUDA (National Aerospace Lab., Chofu, Japan). *Proceedings of the ICAS 20th Congress*, Naples, Italy, 1996, Vol. 1 (A96-40526 11-01), AIAA, Inc., Reston, VA, 1996, pp. 809–817. 12 Refs.

Numerical computations of the flow fields of the two-dimensional transonic wind tunnel of the National Aerospace Laboratory, Japan, are presented. The full Navier–Stokes equations are solved numerically by a cell-centered finite-volume method. In the present numerical method the inviscid flux is estimated by the approximate Riemann solver by Roe with MUSCL higher-order interpolation, while the viscous flux is approximated by a Galerkin-like discretization formula with the Baldwin–Lomax turbulence model. The present numerical study consists of two parts: the simulation of the whole flow field of the wind tunnel and the simulation of the flow field near the airfoil mounted in the wind tunnel. The computations are compared with the experimental data. The flow mechanism at the near region of an airfoil model mounted in the tunnel is discussed. (Author)

A96-40533 Numerical investigation of strongly non-linear transonic flow problems around the airfoil. S. YAMAMOTO (Tohoku Univ., Sendai, Japan). *Proceedings of the ICAS 20th Congress*, Naples, Italy, 1996, Vol. 1 (A96-40526 11-01), AIAA, Inc., Reston, VA, 1996, pp. 12–18. 21 Refs.

The purpose of the present paper is to propose a higher-resolution numerical method for simulating unsteady transonic viscous flows around the airfoil considering the gas-liquid phase change which must be one of typical nonlinear flow problems. The complex flow phenomena involving shocks, vortices, and their interactions are investigated. The fundamental equations comprising the compressible Navier–Stokes equations and the model equations for the phase change based on the classical condensation theory are solved using the fourth-order accurate compact MUSCL TVD scheme and the maximum second-order implicit scheme proposed by the author. As numerical examples, steady and unsteady transonic viscous flows around the NACA0012 airfoil in the moist air at low and high angles of attack are calculated, and the results are compared to that in dry air. The phase change and the induced condensation shock are obtained. The influence of humidity on airfoil performance is numerically estimated. (Author)

A96-37455 Wind-tunnel tests on aerodynamic characteristics of express reentry capsule. K. SUZUKI (Inst. of Space and Astronautical Science, Sagamihara, Japan); K. ODA (Tokai Univ., Japan); and M. HONGOH and T. ABE (Inst. of Space and Astronautical Science, Sagamihara, Japan). *Proceedings of the 19th International Symposium on Space Technology and Science*, Yokohama, Japan, 1994 (A96-37401 10-12), Agne Shofu Publishing, Inc., Tokyo, Japan, 1994, pp. 441–447. 6 Refs.

The aerodynamic characteristics of the reentry capsule for the EXPRESS mission are experimentally investigated at Mach numbers from 0.4 to 4.2 using transonic and supersonic wind tunnels. The capsule configuration is a blunted cylinder with a conical skirt. For the sake of the conical skirt, the capsule has sufficient static stability with respect to the planned center-of-gravity location both in the supersonic regime and in the transonic regime. Consequently, the capsule configuration with a conical skirt has favorable aerodynamic characteristics in the transonic regime from the viewpoint of the static stability of the capsule. However, schlieren pictures show that an abrupt change in the aerodynamic characteristics with the angle of attack may occur in the transonic regime due to the appearance and disappearance of shock waves on the lower and upper sides of the conical skirt. Surface flow visualizations reveal that the turbulent transition is induced by the interaction between the shock wave and the boundary layer separation in front of the junction of the conical skirt. (Author)

A96-36676 Detection of boundary-layer transition in a cryogenic wind tunnel by using luminescent paint. K. ASAI, H. KANDA, and T. KUNIMASU (National Aerospace Lab., Tokyo, Japan) and T. LIU and J. P. SULLIVAN

(Purdue Univ., West Lafayette, IN). *AIAA 19th Advanced Measurement and Ground Testing Technology Conference*, New Orleans, LA, 1996, p. 12 (AIAA Paper 96-2185). 16 Refs.

A technique for boundary-layer transition detection in a cryogenic wind tunnel has been developed. This technique is based on the thermal quenching of luminescent molecules. Calibration results show that luminescence of the paint composed of a ruthenium complex and a silicone polymer is strongly sensitive to temperature over the 90–220 K range. This capability allows one to visualize thermal signatures across the boundary-layer transition. A thin coating of paint was applied on three airfoil models with different thermal insulation properties. The paint was excited by a xenon light, and the luminescence image was acquired using a high-resolution digital camera. To enhance the surface temperature signatures between the laminar and turbulent regions, either the flow or the model substrate was cooled or heated in an active manner. Transition patterns have been successfully visualized by processing the luminescent images. Boundary-layer transition has been detected by using this technique over a cryogenic temperature range from 90 to 150 K in subsonic and transonic flows. (Author)

A96-33003 Numerical flutter simulation of a binary system in transonic region. H. KHEIRANDISH and G. BEPPU (Tokai Univ., Hiratsuka, Japan) and J. NAKAMICHI (National Aerospace Lab., Tokyo, Japan). *Proceedings of the 33rd Aircraft Symposium*, Hiroshima, Japan, 1995 (A96-32875 08-31), Japan Society for Aeronautical and Space Sciences, Tokyo, 1995, pp. 627–630. 5 Refs.

The need to properly compute unsteady viscous flows surrounding airfoils at transonic speeds remains an outstanding problem in fluid dynamics. In a transonic flow where shock-boundary interactions and separation are dominant, methods based on the Navier–Stokes equations are needed. An N–S code based on the Harten TVD method was developed to calculate unsteady transonic flows about airfoils. This code was applied to the unsteady flow about a NACA0012 oscillating in pitch and was also employed to simulate two-dimensional flutter. (Author)

A96-32943 Performance test of flutter emergency stopper for transonic wind tunnel effect of third cart device. K. SUZUKI, N. HOSOE, Y. IIJIMA, Y. KOMATU, T. KARASAWA, Y. OGUNI, S. NAKAMURA, and S. SUZUKI (National Aerospace Lab., Tokyo, Japan). *Proceedings of the 33rd Aircraft Symposium*, Hiroshima, Japan, 1995 (A96-32875 08-31), Japan Society for Aeronautical and Space Sciences, Tokyo, 1995, pp. 333–336. 3 Refs. In Japanese.

A description is given of experimental results on the effectiveness of a flutter emergency stopper for the NAL 2×2 m Transonic Wind Tunnel. The stopper consists of a part of a side wall plate of the third cart and an actuator using an oil cylinder. (Author)

A96-32942 Wind tunnel tests of arrow-wing flutter model. M. HASHIDATE, K. SAITOH, Y. ANDO, K. SUZUKI, H. MIWA, M. TAMAYAMA, H. MATSUSHITA, and J. NAKAMICHI (National Aerospace Lab., Tokyo, Japan). *Proceedings of the 33rd Aircraft Symposium*, Hiroshima, Japan, 1995 (A96-32875 08-31), Tokyo, Japan Society for Aeronautical and Space Sciences, 1995, pp. 329–332. 4 Refs. In Japanese.

An SST-type arrow wing flutter model for unsteady pressure measurement and active flutter control tests in the NAL 2×2 m Transonic Wind Tunnel has been designed and fabricated. This paper describes the main features of an elastic wing model, the development of a deformation measurement system, and the main results obtained with the wind tunnel test. (Author)

A96-32920 Effect of lateral blowing on aerodynamic characteristics of a wing-body combination at transonic speeds. K. KARASHIMA, K. SATO, and T. TANIKATSU. *Proceedings of the 33rd Aircraft Symposium*, Hiroshima, Japan, 1995 (A96-32875 08-31), Japan Society for Aeronautical and Space Sciences, Tokyo, 1995, pp. 193–196. 3 Refs. In Japanese.

A feasibility test is made for clarifying the effect of lateral blowing on the aerodynamic characteristics of a wing-body combination at subsonic and transonic speeds, where a couple of small sonic jets are applied in parallel to both trailing edges of the wings. It is shown that the blowing brings about a considerable amount of lift-augmentation as well as a certain small increase of drag over wide ranges of angles of attack and freestream Mach numbers. Thus, the blowing increases the associated L/D to an appreciable extent for small angles of attack, although the effect may decrease to nil as the angle of attack increases beyond 15 deg. (Author)

A96-32919 Unsteady pressure measurement on a supercritical airfoil. H. MIWA, M. SATO, H. KANDA, Y. OGUNI, and M. SIGEMI (National Aerospace Lab., Tokyo, Japan). *Proceedings of the 33rd Aircraft Symposium*, Hiroshima, Japan, 1995 (A96-32875 08-31), Japan Society for Aeronautical and Space Sciences, Tokyo, 1995, pp. 189–192. 4 Refs. In Japanese.

Unsteady pressures were measured on a supercritical airfoil in the High Reynolds Number Two-dimensional Transonic Wind Tunnel of the National Aerospace Laboratory of Japan. To study the self-sustained oscillation due to shock-induced separation, the pressure signals were measured up to VHF of 10 kHz. We adopted a space-time correlation analysis to measure the convection velocities in the separated flow and attempted to determine the time period of the periodic shock motion at the heavy buffet condition. (Author)

A96-31215 An inverse design method for transonic multiple wing systems on integral equations. K. MATSUSHIMA (Fujitsu Ltd., Chiba, Japan) and S. TAKANASHI (National Aerospace Lab., Tokyo, Japan). *AIAA 14th Applied Aerodynamics Conference*, New Orleans, LA, 1996, TP. Pt. 2 (A96-31147 08-02), AIAA, Reston, VA, 1996, pp. 682–690 (AIAA Paper 96-2465). 12 Refs.

An inverse design method that treats multiple wings (or multicomponents of a wing) is proposed and examined. The method takes into consideration the mutual interaction among wings to provide section shapes of wings which realize a specified surface pressure distribution. It can be applied to two-dimensional and three-dimensional aerodynamic design problems in various degrees of a flow field, potential flow, inviscid and viscous flow, subsonic and transonic flow. The primary idea is the extension of Takanashi's integral equation method. An inverse problem is reformulated to be integral equations which express the relation of pressure differences to geometrical changes of wing sections for a multiple wing system. Most of the integral calculation is done analytically, and finally the equations are solved numerically by introducing piecewise function approximation. This method works well on several preliminary design problems. From the practical point ot view, it is promising for complex aerodynamic designs. (Author)

A96-27253 High resolution schemes for the compressible Navier–Stokes equations and their applications. H. DAIGUJI, X. YUAN, S. YAMAMOTO, and K. ISHIZAKA (Tohoku Univ., Sendai, Japan). *The recent developments in turbulence research; Proceedings of the Sino-Japan Workshop on Turbulent Flows*, Tsinghua Univ., Beijing, China, 1994 (A96-27252 06-34), Beijing, International Academic Publishers, 1995, pp. 3–21. 15 Refs.

High resolution schemes of the compressible Navier–Stokes equations suitable for the direct numerical simulation of turbulent flows are presented. The delta-form implicit equations using the fourth-order compact MUSCL-type TVD scheme are efficiently solved by the LU SGS algorithm. Some calculated results of three-dimensional compressible mixing layer flows and three-dimensional transonic turbine nozzle flows are shown. The present schemes are very effective for the transonic flows with weak discontinuities. (Author)

A96-26176 Behavior of shock waves in transonic cascades during high-frequency blade pitching oscillation. I. FUJIMOTO and T. HIRANO (Takushoku Univ., Japan); S. ISHII (Nihon Univ., Japan); and H. TANAKA (Tokai Univ., Japan). *Proceedings of the Yokohama International Gas Turbine Congress*, Yokohama, Japan, 1995, Vol. 2 (A96-26107 06-37), Gas Turbine Society of Japan, Tokyo, Japan, 1995, pp. II.201–II.208. 6 Refs.

An accurate photographing and image processing system for capturing the high-speed periodic motion of shock waves in unsteady transonic flow cascades is developed using a video camera with a moderate frame rate, instead of a high-speed video camera. The feasibility of generating successive pictures in one period whose frequency is up to 500 Hz is confirmed. The behavior of the shock waves is clarified in the range up to 200 Hz. Although the amplitude of the shock wave displacement does not change much within the range of the present experiment, the phase lag relative to the blade oscillation reaches around $\pi/2$ as the blade oscillation frequency increases to 200 Hz. (Author)

A96-26162 Effects of inlet passage width contraction of low solidity cascade diffusers on performance of transonic centrifugal compressor. H. HAYAMI (Kyushu Univ., Kasuga, Japan) and A. UMEMOTO (Toto Co., Technology Div., Kitakyushu, Japan). *Proceedings of the Yokohama International Gas Turbine Congress*, Yokohama, Japan, 1995, Vol. 2 (A96-26107 06-37), Gas Turbine Society of Japan, Tokyo, Japan, 1995, pp. II.99–II.102. 5 Refs. Grant MOESC-4650153.

Low solidity circular cascades have been applied successfully as part of the diffuser system of a transonic centrifugal compressor. Three types of diffusers with and without an inlet passage width contraction or a pinch upstream of the cascade were tested. An appropriately contracted diffuser realized a high compressor efficiency at high rotor speed by the decrement in input power. The effects of a passage width contraction on surge and choke are also discussed. The choke occurred at the throat of the cascades of a pinched-type diffuser at less flow than the inducer choke flow, in spite of a low solidity cascade. The pinched diffuser had a stabilizing effect on the diffuser characteristics, but only a small amount of gain in the surge margin was obtained, owing to the change in impeller characteristics. (Author)

A96-26151 Navier–Stokes analysis of unsteady transonic flows through gas turbine cascades with coolant ejection. T. TANUMA (Toshiba Corp., Yokohama, Japan) and S. YAMAMOTO and H. DAIGUJI (Tohoku Univ., Sendai, Japan). *Proceedings of the Yokohama International Gas Turbine Congress*, Yokohama, Japan, 1995, Vol. 2 (A96-26107 06-37), Gas Turbine Society of Japan, Tokyo, Japan, 1995, pp. II.25–II.32. 16 Refs.

An implicit time-marching higher-order accurate finite-difference method for solving the two-dimensional compressible Navier–Stokes equations is applied to the numerical analyses of unsteady transonic viscous flows through gas turbine cascades with trailing edge coolant ejection. The unsteady aerodynamic mechanisms associated with the interaction between the trailing edge vortices and shock waves, and the effect of coolant ejection, are evaluated. (Author)

A96-26125 Effect of tip clearance in three-dimensional cascade flow analysis. O. NOZAKI, K. KIKUCHI, and A. TAMURA (National Aerospace Lab., Chofu, Japan); K. SHIMIZU and Y. MIYAKE (Mitsubishi Heavy Industries, Ltd., Aichi, Japan); and T. WATANABE (Tokyo Univ., Japan). *Proceedings of the Yokohama International Gas Turbine Congress*, Yokohama, Japan, 1995, Vol. 1 (A96-26107 06-37), Gas Turbine Society of Japan, Tokyo, Japan, 1995, pp. I.135–I.142. 7 Refs.

The three-dimensional transonic flow fields of a turbine stage have been analyzed by computational solutions of Navier–Stokes equations employing an implicit finite deference method with a high-accuracy upwind scheme. Numerical solutions were compared for different approaches to deal with tip clearance flow. Experimental data were also introduced for verifying the computation. The turbine stage model trader consideration consists of inlet guide vanes and rotor blades. Two approaches were examined for dealing with the flow through tip clearance. These approaches have been applied to the turbine for a certain flow condition to evaluate their advantages. In this paper the numerical approaches are illustrated and discussed, with a comparison of the resultant flow fields as well as over-all characteristics with experimental data. (Author)

A96-26123 Numerical simulation of tip leakage flows in a transonic compressor. A. MATSUOKA, K. HASHIMOTO, and H. KATO (Kawasaki Heavy Industries Ltd., Kobe, Japan) and O. NOZAKI and A. TAMURA (National Aerospace Lab., Chofu, Japan). *Proceedings of the Yokohama International Gas Turbine Congress*, Yokohama, Japan, 1995, Vol. 1 (A96-26107 06-37), Gas Turbine Society of Japan, Tokyo, Japan, 1995, pp. I.121–I.126. 7 Refs.

A numerical and experimental investigation of three-dimensional flows in a transonic axial-flow compressor was conducted. Three-dimensional viscous flow fields through the rotor blades, both with and without tip clearance, of a transonic compressor were numerically studied by solving Navier–Stokes equations with a high-accuracy upwind scheme. For validating the capability of the analysis, detailed experimental data such as the dynamic static pressure contours on the rotor casing and the radial distributions of flow parameters at the rotor exit were measured. The numerical results and the available experimental data yielded a good level of consistency under careful comparisons. (Author)

A96-26111 A higher-order CFD approach to shock/vortex interaction and its application to turbomachinery. S. YAMAMOTO (Tohoku Univ., Sendai, Japan); X. YUAN (Tsinghua Univ., Beijing, China); and H. DAIGUJI (Tohoku Univ., Sendai, Japan). *Proceedings of the Yokohama International Gas Turbine Congress*, Yokohama, Japan, 1995, Vol. 1 (A96-26107 06-37), Gas Turbine Society of Japan, Tokyo, Japan, 1995, pp. I.27–I.34 (Grant MOESC-05240102). 20 Refs.

A higher-order CFD approach for simulating the unsteady shock/vortex interaction is presented and applied to the simulation of some practical cascade flows in which shocks are strongly interacting to the boundary layer and the wake. The main concept of the present approach is to employ the fourth-order compact MUSCL TVD scheme proposed by the authors in space difference and a maximum second-order integral scheme in time integration. The set of the higher-order accurate schemes enable us to get shocks, contact discontinuities, and vortices very clearly. Moreover, since both schemes are constructed of quite simple forms, the extension to the existing CFD codes is also very easy. As numerical examples, the unsteady transonic flow around the airfoil with high angle of attack is calculated to show the reliability of the present approach, and the unsteady transonic turbine cascade flows and the unsteady supersonic compressor cascade flows considering the on- and off-design conditions are calculated as the application. The possibility or potential of the present CFD approach is discussed by showing some interesting calculated results. (Author)

A96-22055 Visualization of boundary-layer transition on transonic airfoils using liquid crystal coatings. N. SUDANI, M. NOGUCHI, H. KANDA, M. SATO, and Y. ISHIDA (National Aerospace Lab., Chofu, Japan). *Flow visualization VII; Proceedings of the 7th International Symposium on Flow Visualization*, Seattle, WA, 1995 (A96-22001 05-35), New York, Begell House, Inc., 1995, pp. 538–543. 7 Refs.

The objectives of this study are to establish a new technique for visualization of transition on airfoils using temperature-sensitive liquid crystals and to investigate the applicability of the liquid crystal coatings to suction surfaces for laminar flow control. Visualization results are presented for two-dimensional supercritical airfoil models with a clean surface and a natural laminar-flow airfoil model with a slotted surface around the midchord. (Author)

N95-29712 Experimental studies on boundary-layer transition on a reentry vehicle at transonic and supersonic speeds. K. SUZUKI and T. ABE. ISAS-659.

The boundary-layer transition on the EXPRESS reentry capsule at transonic and supersonic speeds is studied experimentally by the wind tunnel tests. For the diagnostic of the turbulent transition of the boundary layer, the China-clay method is used. The experimental results clarify that when the freestream Mach number increases, the transition point moves downstream on the body surface and the distance between the beginning of the transition and its completion to the fully turbulent flow becomes larger. The effects of the freestream Mach number on the location of the boundary-layer transition are described successfully in terms of two nondimensional quantities, that is, the transition Reynolds number and the local Mach number at the boundary-layer edge. The oil-flow pictures reveal that in the transonic regime, the separation bubble is formed at the junction between the blunt nose and the conical part of the body and therefore the transition begins behind the reattachment point of the separation bubble. The effects of the turbulent transition on the aerodynamic characteristics of the reentry body are investigated by using the technique of the boundary-layer trip and the experimental results show that the aerodynamic characteristics of the EXPRESS reentry vehicle are not sensitive to the boundary-layer transition. (Author)

N95-29640 Transonic, supersonic and hypersonic wind-tunnel tests on aerodynamic characteristics of reentry body with blunted cone configuration. K. SUZUKI and T. ABE. ISAS-658.

The aerodynamic characteristics of the early version of the 'EXPRESS' reentry capsule, having a blunted cone configuration with flat bottom, are investigated experimentally over a wide range of the Mach number from 0.6 to 7.1 by using the transonic, supersonic, and hypersonic wind tunnels. The continuity of the experimental data with respect to the Mach number is examined especially at the Mach numbers where the experimental facility is changed from the transonic wind tunnel to the supersonic wind tunnel and from the supersonic wind tunnel to the hypersonic wind tunnel. The experimental data in the supersonic and hypersonic regime show the tendency that when the Mach number becomes large, the aerodynamic characteristics become independent of the Mach number. It is clarified that in the transonic regime the aerodynamic center of the capsule moves forward and the static stability of the vehicle on the pitch plane is significantly reduced. To overcome such reduction in the static stability of the vehicle, it is necessary to deploy the parachute behind the capsule by the time when the flight Mach number decreases to the transonic regime and the static stability around the center-of-gravity vanishes. The critical Mach number for the parachute deployment is determined for various center-of-gravity locations. When the center-of-gravity of the capsule is located around the center of the body, the parachute must be deployed at supersonic speed. The effects of the nose shape and the bottom configuration on the aerodynamic characteristics of the capsule are experimentally investigated. The validity of the Newtonian flow analysis in prediction of the aerodynamic characteristics of the capsule is assessed by comparisons with the experimental data. (Author)

A95-44714 Aerodynamic instability of transonic cascade with shock movement. T. SHIRATORI (Tokyo Metropolitan Inst. of Technology, Hino, Japan) and Y. NOGUCHI (Salford Univ., UK). *Proceedings of the ISABE—12th International Symposium on Air Breathing Engines*, Melbourne, Australia, 1995, Vol. 1 (A95-44654 12-07), AIAA, Washington, DC, 1995, pp. 644–654 (ISABE 95-7059). 9 Refs.

A numerical study to investigate the aerodynamic instability of the transonic cascade with shocks and the role of shock behavior has been carried out. The pitch oscillating two-dimensional transonic cascade which consists of double-circular arc airfoils with 5% thickness is evaluated. The shock behavior and the aerodynamic instability are numerically analyzed at various exit static pressures and frequencies. The numerical results indicate that the shock movement on the suction surface acts as an aerodynamically unstable factor at the relatively high reduced frequencies, and the shock movement on the pressure surface acts as an unstable factor at the low reduced frequency. The results also indicate that the pitching transonic cascade with shocks becomes aerodynamically unstable due to the passage shock movement when the pitching axis is moved forward at relatively high frequencies and the pitching axis is moved backward at low frequencies with an interblade phase angle of 180 deg. (Author)

A95-42795 Elastic deflection effects on transonic aerodynamics of a flutter wing model with control surfaces. K. SAITOH, M. HASHIDATE, H. MATSUSHITA, and T. KIKUCHI (National Aerospace Lab., Tokyo, Japan). *AIAA 1st Aircraft Engineering, Technology, and Operations Congress*, Los Angeles, CA, 1995, p. 7 (AIAA Paper 95-3926). 14 Refs.

Normal direction deflections of an elastic wing are measured by detecting an image on a wind tunnel wall reflected from a laser light target embedded on a wing and processing the video signal obtained. This measurement system allows for the easy and precise measurement of the elastic deformation of the wing. Computing the pressure distribution on the wing surface by a three-dimensional full potential USTF3 code, taking into account the mean elastic deflection of the wing measured above, cleared the elastic deformation effects on the transonic aerodynamics. Dynamic elastic deformation of the wing during flutter is also acquired by processing the high-speed video data which shows that the phase difference between the bending and torsion components change, spanwise. At the outboard wing station, the torsion component leads to bending, which promotes flutter, while at the middle part, the torsion and the bending component go in phase, which leads to flutter suppression. (Author)

A95-40615 Temperature measurement in transonic duct with swept-back bump by laser-induced fluorescence method. M. INOUE, M. MASUDA, M. FURUKAWA, T. MURAISHI, and Y. TAKAHASHI (Kyushu Univ., Fukuoka, Japan). *Modern techniques and measurements in fluid flows; Proceedings of the 2nd International Conference on Fluid Dynamic Measurement and Its Applications (IFCDMA)*, Tsinghua Univ., Beijing, China, 1994 (A95-40601 11-35), International Academic Publishers, Beijing, China, 1994, pp. 153–157. 19 Refs.

The laser-induced fluorescence method is developed to investigate the three-dimensional transonic flow with complicated shock wave-boundary layer interaction. This diagnostic system uses an argon-ion laser with seeded iodine as fluorescence material, and is applied to a rectangular duct with a swept-back bump. The temperature distributions in the duct are obtained, and the structure of the flow field is clarified including the curved shock wave and the boundary-layer separation. (Author)

A95-40262 An experimental study of transonic flow behavior over a deformable wing section. M. MAKINO and A. MINEO (Nihon Univ., Funabashi, Japan). *Proceedings of the 1st Asian-Pacific Conference on Aerospace Technology and Science*, Hangzhou, China, 1994 (A95-40201 11-31), Beijing, International Academic Publishers, 1994, pp. 463–467.

In transonic flow the flow field around a wing section is so sensitive to the shape of its outline that the flow condition changes greatly by slight deformation of the wing surface. In the present experiment the wind tunnel model is a hollow wing, the outer plate of which is made of an aluminum alloy plate suitable in thickness, and the curvature of the wing surface can be changed smoothly and continuously by regulation of the air pressure inside the wing in the flow at a fixed Mach number. On the upper surface of the wing model, 13 static pressure holes line up in the stream direction. The pressure distribution on the surface was measured in Mach numbers 0.60–0.98. At the same time the color video was taken by schlieren method in order to observe the shock wave patterns. (Author)

A95-40260 Effect of lateral blowing on aerodynamic characteristics of low aspect-ratio wings at high angles of attack. K. KARASHIMA and K. SATO (Inst. of Space and Astronautical Science, Sagamihara, Japan). *Proceedings of the 1st Asian-Pacific Conference on Aerospace Technology and Science*, Hangzhou, China, 1994 (A95-40201 11-31), International Academic Publishers, Beijing, 1994, pp. 452–457. 3 Refs.

Measurements of the aerodynamic characteristics of clipped delta wings with low aspect ratio are performed in a wind tunnel at subsonic and transonic Mach numbers to clarify the aerodynamic effectiveness of the lateral blowing. It is shown that the blowing gives rise to a considerable increase in the lift of the wings in the wide ranges of angle of attack and freestream Mach number without serious degradation of L/D performance, and it is further deduced that the aerodynamic mechanism of the lift-augmentation may be attributed to two fluid dynamic processes. The first is the circulation enhancement that results from improvement of potential pressure on the upper surface of the wing and takes place at relatively low angles of attack. The other is the strengthening of the separation vortex at the leading-edge, and it occurs at high angles of attack. A comparison between the lateral blowing and the so-called spanwise one clearly reveals that the former is more effective for lift-augmentation than the latter in terms of efficiency of the blowing momentum. (Author)

A95-39282 Numerical simulation of unsteady viscous flow around an airfoil with oscillating spoiler. K. ISOGAI (Kyushu Univ., Fukuoka, Japan) and M. YOSIDA (National Aerospace Lab., Tokyo, Japan). *AIAA Atmospheric Flight Mechanics Conference*, Baltimore, MD, 1995, TP (A95-39269 10-08), AIAA, Washington, DC, 1995, pp. 147–150 (AIAA Paper 95-3439). 8 Refs.

Numerical simulations of unsteady viscous flows around an airfoil with oscillating spoiler have been performed using the compressible Navier–Stokes equations. The Yee-Harten Total Variation Diminishing (TVD) scheme and the Baldwin–Lomax algebraic turbulence model are employed. The unsteady pressure distributions on the 16% thick supercritical airfoil with oscillating spoiler in low subsonic and transonic Mach numbers have been calculated, being compared with the existing experimental data. (Author)

A95-38474 Transonic wind-tunnel flows about a fully configured model of aircraft. Y. TAKAKURA (Tokyo Noko Univ., Japan) and S. OGAWA and Y. WADA (National Aerospace Lab., Tokyo, Japan). *AIAA Journal* (ISSN 0001-1452), Vol. 33, No. 3, 1995, pp. 557–559. Previously cited in issue 20, Accession A93-48207. 10 Refs. Documents available from AIAA Dispatch.

The transonic flows about a fully configured model of an ONBEA-M5 aircraft within the Japanese National Aerospace Laboratory (NAL) transonic wind tunnel are numerically solved to investigate the reliability of numerical solutions. The multidomain technique is used to solve the flow field around the complicated configuration. A simple model is presented to estimate the outflow/inflow effects at the perforated wall of the transonic wind tunnel. The results show good coincidence with the experimental data of NAL wind-tunnel tests with regard to local C(p) distributions on the main wing and total forces C(L) and C(D).

A95-36731 Dynamic test of the Orbital Reentry Vehicle (OREX) in a transonic wind tunnel with comparison to flight data. T. YOSHINAGA, A. TATE, and M. WATANABE (National Aerospace Lab., Chofu, Japan) and T. SHIMODA (NASDA, Tokyo, Japan). *AIAA 13th Applied Aerodynamics Conference*, San Diego, CA, 1995, TP. Pt. 2 (A95-36627 09-02), AIAA, Washington, DC, 1995, pp. 1129–1138 (AIAA Paper 95-1901). 12 Refs.

The angular motion of an OREX model was tested in a transonic wind tunnel using a single DOF test method to investigate the angular motion of OREX reentering the atmosphere for the test of heat-shield materials and the measurements of the aerodynamic heated flow field. Approximate static and dynamic derivatives of the model were measured by local linear curve fitting. The angular motion of real OREX flight data was compared with the wind tunnel test results. (Author)

A95-36506 Applications of genetic algorithm to aerodynamic shape optimization. K. YAMAMOTO and O. INOUE (Tohoku Univ., Sendai, Japan). *AIAA 12th Computational Fluid Dynamics Conference*, San Diego, CA, 1995, Collection of Technical Papers. Pt. 1 (A95-36501 09-34), AIAA, Washington, DC, 1995, pp. 43–51 (AIAA Paper 95-1650). 15 Refs.

This paper presents an aerodynamic shape optimization method based on a genetic algorithm with the compressible Euler or Navier–Stokes equations. A new algorithm to represent aerodynamic configurations is proposed to find optimum aerodynamic configurations efficiently. We solve three aerodynamic shape optimization problems. First, an inverse problem is solved to understated the basic character and to examine the possibility of the genetic algorithm. Second, a drag minimization problem is solved to find a streamlined body. Finally, the lift/drag ratio maximization problem in a transonic flow is solved, and the optimized configuration is compared with a supercritical airfoil. The results show that the genetic algorithm is powerful and robust in solving aerodynamic shape optimization problems. (Author)

A95-35758 Research and development of two-stage fan component in HYPR project. M. SUZUKI and N. KUNO (Mitsubishi Heavy Industries, Ltd., Komaki, Japan). *AIAA, ASME, SAE, and ASEE 31st Joint Propulsion Conference and Exhibit*, San Diego, CA, 1995, p. 8 (AIAA Paper 95-2344).

The design and experimental rig test of a two-stage transonic axial-flow fan component for turbofan engine in a super/hypersonic transport propulsion system are described. Engine tests incorporating this fan component were performed, enabling a comparison of rig test results with engine test data. The overall performance is found to be acceptable for demonstrator engine use. (Author)

A95-25946 Design of new generation rotor blade airfoils using Navier–Stokes. M. NAKADATE and M. OBUKATA (Fuji Heavy Industries, Ltd., Utsunomiya, Japan). *Proceedings of the 20th European Rotorcraft Forum*, Amsterdam, Netherlands, 1994, Vol. 2 (A95-25916 06-01), National Aerospace Lab., Amsterdam, Netherlands, 1994, pp. 33.1–33.15. 11 Refs.

A study on the design of new generation rotor blade airfoils has been carried out to develop new rotors for helicopters. Beginning by discussing the design criteria for rotor blade airfoils, aerodynamic analysis methods used in the design are outlined and design procedures in terms of analysis methods are described. State-of-the-art Navier–Stokes code was extensively used in the design procedures to satisfy the design criteria and to achieve the design goals. In the practical design, a parametric analysis on the leading edge shape is followed by an optimization of the shape and then the shape is refined to obtain the final airfoil. A two-dimensional transonic wind tunnel test was conducted on the new airfoil to confirm the design. The analysis results agreed well with the wind tunnel results and it was confirmed that the airfoil achieved or exceeded the design goals. (Author)

A95-22434 Unsteady analysis of transonic helicopter rotor noise. T. AOYAMA (National Aerospace Lab., Chofu, Japan); K. KAWACHI (Tokyo Univ., Japan); S. SAITO (National Aerospace Lab., Mitaka, Japan); and J. KAMIO (Tokyo, Univ., Japan). *Proceedings of the 19th European Rotorcraft Forum*, Como, Italy, 1993, Vol. 1 (A95-22426 05-01), National Aerospace Lab., Amsterdam, Netherlands, 1993, pp. B2.1–B2.12. 17 Refs.

A combined CFD/extended Kirchhoff's equation method has been developed to analyze the high-speed impulsive (HSI) noise of a helicopter rotor. The method solves Euler equations to yield the pressure distributions around a rotor blade. To predict the HSI noise, the behavior of shock wave should be evaluated precisely; the present CFD code used predicts the shock wave by using a higher-order upwind scheme. In case of a forward flight condition, the Newton iterative method is used to get unsteady solutions. The Kirchhoff's equation, extended for moving surfaces, is then used to find the acoustic pressures by using the Euler solutions on the Kirchhoff surface in which all the acoustic sources are enclosed. The HSI noise of a nonlifting hovering rotor is calculated, and good agreement is obtained between calculated and experimental results. The comparison between HSI noise levels of two types of advanced tip shape and a conventional rectangular tip shape are also presented for nonlifting hovering conditions. (Author)

A95-19130 Similarity rule for jet-temperature effects on transonic base pressure. K. ASAI (National Aerospace Lab., Tokyo, Japan). *AIAA Journal* (ISSN 0001-1452), Vol. 33, No. 2, 1995, pp. 276–281. Previously cited in issue 10, Accession A94-30265. 11 Refs.

On the basis of the similarity rule for jet interaction, a hot jet can be simulated in a cryogenic wind tunnel with a test gas at ambient or moderately elevated temperatures. Using this approach, jet-temperature effects on the base pressure of a cylindrical afterbody model at transonic speeds were investigated in the 0.1-m transonic cryogenic wind tunnel at the National Aerospace Laboratory. Mixtures of N_2 and either CH_4, Ar, or He at varying temperatures were used as a jet gas to determine separate effects of jet temperature, specific heat ratio, and gas constant. It was found that data obtained for various jet conditions can be correlated very well with two similarity parameters: the plume maximum diameter for the plume shape effect and the jet to freestream mass flux ratio for the jet entrainment effect. The same model was tested in an ambient wind tunnel. It was found that an ambient temperature gas having low molecular weight could simulate the jet temperature effects on the transonic base pressure. (Author)

N95-10135 Aeroelastic tailoring research. J. NAKAMICHI, H. EJIRI, T. KIKUCHI, M. MINEGISHI, and T. SOTOZAKI. In Japanese. PB94-180031; NAL/TR-1208.

Research on aeroelastic tailoring at NAL is presented. Flutter boundaries for an optimized forward swept wing have been experimentally verified at angles of attack of 0 and 2 deg (which is the design point) in the transonic range. It was found that two different mode flutters and a divergence occur at almost the same dynamic pressure for the optimized forward swept wing. The aeroelastic behavior of the optimized forward swept wing is investigated and is compared with that of parametric studies performed previously. (NTIS)

A95-11790 Computation of unsteady transonic cascade flow using the Euler and Navier–Stokes equations of contravariant velocities. S. YAMAMOTO and H. DAIGUJI (Tohoku Univ., Sendai, Japan). *JSME International Journal, Series B: Fluids and Thermal Engineering* (ISSN 0914-8817), Vol. 37, No. 3, 1994, pp. 522–530. 15 Refs.

The purpose of the present paper is to investigate the steady and unsteady flows through subsonic and transonic turbine cascades using the Euler and Navier–Stokes solvers developed by the authors. Use of the fundamental equations of contravariant velocity components also proposed by the authors is very convenient for treating several kinds of boundary conditions. Some efficient numerical schemes for the unsteady calculation, shock capturing, and turbulent quantities are also developed. As numerical examples, we show the numerical results of some turbine cascade flows by assuming the flows to be inviscid or viscous (turbulent) and steady or unsteady. The results are compared with each other and with the experimental data. Finally, the reliability and the limitation of the present numerical solvers for the turbine cascade cases are discussed. (Author)

French Aerospace Literature
This month: *Transonic Flows*

A97-15998 Use of improved numerical optimization methods for the definition of advanced profiles (Mise en oeuvre de methodes ameliorees d'optimisation numerique pour la definition de profils avances). B. MIALON (ONERA, Chatillon, France). ONERA, TP 1996-64, 1996. In French. 9 Refs. Documents available from AIAA Dispatch.

This paper presents the latest improvements introduced into the two-dimensional numerical optimization method utilized at the Direction de l'Aerodynamique at ONERA. They concern, on one hand, the minimization code, and, on the other hand, the shape modification module. An approximation technique for the aerodynamic functions (objective and/or constrained) was introduced in the code for minimization under constraints. An application to the case of the optimization of a transonic profile is presented; it shows that the utilization of the approximation concept gives satisfactory results and allows for an important economy of calculation time, provided the number of decision variables is not too important. The shape modification techniques utilizing the Bezier (1977) curves are also integrated by the optimization method. This parameterization technique presents interest in considerably expanding the research space for an optimal solution, while in rapport with more classical techniques in the literature of profiles or shape functions. Two types of two-dimensional numerical optimization applications are presented. One deals with the definition of wings of heavy stationary aircraft; the other deals with the study of laminar profiles for the blades of a nonstationary helicopter.

A97-15736 Improvements of a zonal method for the computation of viscous separated flows. V. SAINT-MARTIN and F. THIVET (ONERA, Centre d'Etudes et de Recherches de Toulouse, France). *AIAA 35th Aerospace Sciences Meeting and Exhibit,* Reno, NV, 1997, p. 12 (AIAA Paper 97-0721). 16 Refs.

A zonal method consisting of a strong coupling between Euler, Navier–Stokes and boundary-layer equations is presented. Coupling procedures assuring the continuity of the solution across the zonal boundaries are studied and improved. Results are given for subsonic and transonic two-dimensional applications. They demonstrate that the zonal method is more efficient and accurate than a global Navier–Stokes approach. (Author)

A97-15294 Study of passive control in a transonic shock wave/boundary layer interaction. R. BUR, B. CORBEL, and J. DELERY (ONERA, Chatillon, France). *AIAA 35th Aerospace Sciences Meeting and Exhibit,* Reno, NV, 1997, p. 12 (AIAA Paper 97-0217). 14 Refs.

Passive control applied to a turbulent shock wave/boundary layer interaction has been investigated by considering a two-dimensional channel flow. The field resulting from application of passive control has been probed in great detail by using a two-component LDV system to execute mean velocity and turbulence measurements. Four different perforated plates have been considered, as also the solid wall reference case. The performed measurements have shown that passive control deeply modifies the inviscid flowfield structure, the unique strong shock being replaced by a lambda shock system. This fractionation of the compression induces a substantial reduction of the wave drag associated with the interaction. On the other hand, the combined injection-suction effect taking place in the control region provokes an important thickening of the boundary layer. There results an increase of the viscous drag which nearly outbalances the gain in wave drag. A determination of the total drag in the control region was made. It was found that passive control induced a modest decrease of this drag compared to the solid wall case. Moreover, the experimental wall transpiration velocity distribution in the control region is well represented by usual laws. (Author)

A97-15187 Optimal shape design, reverse mode of automatic differentiation and turbulence. B. MOHAMMADI (INRIA, Le Chesnay, France). *AIAA 35th Aerospace Sciences Meeting and Exhibit,* Reno, NV, 1997, p. 12 (AIAA Paper 97-0099). 19 Refs.

A new approach for optimal shape design is introduced. The main ingredients are an unstructured CAD free framework for geometry deformation and automatic differentiation in reverse mode. Transonic inviscid and viscous turbulent flows are investigated. Both two and three dimensional configurations are considered. These cases involve up to several thousand control parameters. (Author)

A97-13943 Near-wall Reynolds-stress three-dimensional transonic flow computation. G. A. GEROLYMOS and I. VALLET (Paris VI Univ., Orsay, France). *AIAA Journal* (ISSN 0001-1452), Vol. 35, No. 2, 1997, pp. 228–236. 49 Refs.

A computational method for the Favre-Reynolds-averaged three-dimensional compressible Navier–Stokes equations using near-wall Reynolds-stress closure is described. The mean-flow and turbulence-transport equations are discretized using a finite volume method based on MUSCL Van Leer flux-vector-splitting with Van Albada limiters. The mean-flow and turbulence equations are integrated in time using a fully coupled approximately factored implicit backward Euler method. The resulting scheme is robust and achieves optimal convergence with local-time-step Courant–Friedrichs–Lewy = 50. The turbulence closure is validated by comparison with classic flat-plate boundary-layer data. Results are presented for the three-dimensional Delery transonic channel test case and compared with *k*-epsilon computations. An analysis of the limitations of the closure is attempted, and possible improvements are suggested. (Author)

A97-10718 Prediction of HSI noise using a coupled Euler/Kirchhoff method for a helicopter in hoverflight. J. ZIBI, O. ROUZAUD, and C. POLACSEK (ONERA, Chatillon, France). *Proceedings of the 22nd European Rotorcraft Forum,* Brighton, UK, 1996, Vol. 1 (A97-10676 01-05), Royal Aeronautical Society, London, 1996, pp. 49.1–49.11. 21 Refs.

A coupled multibladed Euler/Kirchhoff method has been developed at ONERA to predict the high speed impulsive (HSI) noise from helicopter rotors in hover flight with transonic flow. This method has been validated with UH-1H rotor experimental tests. CFD output, provided by the WAVES Euler solver, are transferred to the acoustic Kirchhoff code, KARMA, through an interface program which computes the pressure and its normal gradients on the Kirchhoff surface needed to predict the acoustic pressure signature. Aerodynamic computations are in very good agreement with experimental results for the perturbation pressure on the sonic circle. A parametric study of the location of the Kirchhoff surface is performed from the sonic circle up to the external boundary of the aerodynamic mesh. HSI noise predictions are very stable and accurate for control surfaces located from 1.35 rotor radius up to the last section of the aerodynamic mesh. These satisfactory results are mainly due to the fact that no artificial viscosity is needed in the numerical scheme of the WAVES code, and it is concluded that the coupling method WAVES + KARMA is efficient with respect to HSI noise predictions in hover. (Author)

A96-40706 Navier–Stokes analysis for engine airframe integration. E. CHAPUT, L. BARRERA, C. GACHERIEU, and L. TOURRETTE (Aerospatiale, Toulouse, France). *Proceedings of the ICAS 20th Congress,* Naples, Italy, 1996, Vol. 2 (A96-40679 11-01), AIAA, Inc., Reston, VA, 1996, pp. 1606–1618. 20 Refs.

The experience gained at Aerospatiale over the last two decades to overcome advanced experimental techniques and numerical methods for the design of engine-airframe integration, is now being strengthened by the industrial use of turbulent Navier–Stokes methods. The efficiency of this approach is demonstrated for the study of viscous effects around the pylon of a transonic aircraft. The separation of the boundary layers arises and disappears for the same flight conditions that are observed in wind-tunnel simulations. The problem of simulating the behavior of the bleed system for the supersonic transport aircraft air intakes is presented. The numerical difficulty in diminishing the influence of artificial dissipation with respect to the physical one is pointed out in a comparison between Euler and Navier–Stokes analysis. (Author)

A96-40578 Rear fuselage flow characteristics for a complete wing-body configuration at transonic conditions. E. COUSTOLS, A. SERAUDIE, and A. MIGNOSI (ONERA, Centre d'Etudes et de Recherches de Toulouse, France). *Proceedings of the ICAS 20th Congress,* Naples, Italy, 1996, Vol. 1 (A96-40526 11-01), AIAA, Inc., Reston, VA, 1996, pp. 423–433. 10 Refs.

This study deals with experiments conducted in the transonic, pressurized T2 wind tunnel of CERT. The $\frac{1}{80}$th scale model of a modern subsonic transport aircraft was handled in the test section using a fin-sting. Tests were carried out at a freestream Mach number of 0.82 and a Reynolds number based upon a model chord length of close to 2.5 million. Various measurements were performed, mainly in the vicinity of the fuselage's downstream rear part: oil-flow visualizations; pressure distributions; boundary layer surveys along the fuselage symmetry lines and some lateral lines using an LDA system; and near-wake surveys with both pressure and velocity measurements. Those experiments were aimed at scrutinizing the flows developing along the rear part of a fuselage in a configuration very close to flight applications. (Author)

A96-40538 Experimental study on transonic shock wave/turbulent boundary layer interactions and separation instabilities—Suction and Reynolds number effects. D. CARUANA (ONERA, Centre d'Etudes et de Recherches de Toulouse, France), C. BULGUBURE (Dassault Aviation, Merignac, France), and A. MIGNOSI (ONERA, Centre d'Etudes et de Recherches de Toulouse, France). *Proceedings of the ICAS 20th Congress,* Naples, Italy, 1996. Vol. 1 (A96-40526 11-01), AIAA, Inc., Reston, VA, 1996, pp. 66–75. 5 Refs.

In transonic flow, the shock wave/boundary layer interaction and flow separation may have an important influence on the aerodynamic behavior of the aircraft. Experimental studies in this field have been conducted by CERT-ONERA in cooperation with Dassault Aviation. Aerodynamic wind tunnel studies on a stiff two-dimensional airfoil have been performed to analyze the effects of the Reynolds number, the boundary layer suction upstream of the shock, and the downstream displacement of the transition tripping on the occurrence of unsteadiness resulting from shock wave/boundary layer interaction and flow separation. The OALT25 laminar airfoil was selected for these tests. The experiments were performed in a transonic, cryogenic, pressurized T2 wind tunnel equipped with self-adaptive top and bottom walls. High Reynolds NUMBERS were obtained by using high pressure and low temperature flow. This paper describes the measurements carried out and illustrates the most interesting results. (Author)

A96-39195 Three-dimensional computation of the VEGA2 turbine stage (Calcul tridimensionnel de l'etage de turbine VEGA2). A. FOURMAUX, G. BILLONNET, and C. TOUSSAINT (ONERA, Chatillon, France). Revue Francaise de Mecanique (ISSN 0373-6601), No. 2, 1996, pp. 131–133. In French.

This paper presents computations carried out on the VEGA2 transonic turbine stage test rig. Two approaches are implemented: The first approach is fully unsteady and provides data which should be compared with data provided by unsteady sensors. The second approach uses an averaging technique between two blade rows and only provides steady results; but it is less costly. (Author)

A96-39194 Three-dimensional flows in transonic and supersonic compressors (Ecoulements tridimensionnels dans les turbomachines transsoniques et supersoniques). G. BOIS, F. LEBOEUF, I. TREBINJAC, and A. VOUILLARMET (Lyon I Univ.; Societe Metraflu, Ecully, France). *Revue Francaise de Mecanique* (ISSN 0373-6601), No. 2, 1996, pp. 117–129. 11 Refs. In French. Documents available from AIAA Dispatch.

Aspects of research on three-dimensional flow in axial and centrifugal compressors at LMFA are presented. Numerical simulation developments are considered, with emphasis on scheme improvement and software development. Simulations of mean field, turbulent field, instantaneous effects, and thermal characteristics are also discussed. Finally, consideration is given to experimental issues, including the development of high-velocity measurement techniques and data processing methods.

A96-39193 Measurement of unsteady stagnation pressure (Mesure de pression d'arret instationnaire). C. FRADIN (ONERA, Chatillon, France). *Revue Francaise de Mecanique* (ISSN 0373-6601), No. 2, 1996, pp. 111–115. In French.

A pitot probe whose natural frequency is 330 kHz has been used to make instantaneous measurements at the exit of a TURBOMECA transonic axial compressor. The results allow us to derive the total pressure ratio contours of the fluid just downstream of the rotor, corresponding to four operating points of the compressor. Measurements made at the design point are then compared with the three-dimensional Navier–Stokes computational results. To improve the accuracy of these measurements, fast response pressure transducers with a constant transfer function in a large bandwidth are required. (Author)

A96-39057 Pressure measurements around a rotor model for high-speed noise study. P. GNEMMI, C. JOHES, and J. HAERTIG (Saint-Louis, French–German Research Inst., France). *Proceedings of the 21st European Rotorcraft Forum*, St. Petersburg, Russia, 1995, Vol. 1 (A96-39051 10-01), NLR, Amsterdam, Netherlands, 1995, pp. I.8.1–I.8.11. 8 Refs.

The Institute of Saint-Louis (ISL) has built a rotor model operating in the open air, which simulates hovering flight, to study helicopter rotor noise. This rotor model can reach transonic speed at the blade tip with a Reynolds number not too far from that of full-scale rotors. The aim of the experiment presented in this paper consists in measuring, in the very near field around the rotor model, the pressure generated by the rotor blades. Two parameters are studied: the distance between the measurement point and the rotor center and the rotation velocity (or the hover Mach number). Pressure measurements carried out very close to the blade tip are of high quality, and they are obtained without great difficulty except when the hover Mach number is close to the delocalization Mach number. Results are presented for hover Mach numbers ranging from 0.73 to 0.93 and for distances ranging from 1.025 to 1.115 rotor radius. They provide interesting information for analyzing the high-speed noise generation, and this first signatures collection may be used to validate aerodynamic computations. (Author)

A96-36493 Assessment of turbulence models for the transonic flow around the DLR-F4 wing/body configuration. L. TOURRETTE (Aerospatiale Aircraft Business, Toulouse, France). *AIAA 27th Fluid Dynamics Conference*, New Orleans, LA, 1996, p. 14 (AIAA Paper 96-2034). 13 Refs.

The transonic flow around the DLR-F4 wing/body configuration is simulated using the Baldwin–Lomax and Granville algebraic turbulence models, the Wolfshtein one-equation model, and the Chen and Patel (1987) two-layer model, combining the Wolfshtein model near walls with the standard k–ϵ model away from walls. Regarding the shock location, the best results are obtained with the Granville model, the models with transport equations being intermediate between the two algebraic models. (Author)

A96-31985 Computation of unsteady three-dimensional transonic nozzle flows using k–ϵ turbulence closure. G. A. GEROLYMOS and I. VALLET (Paris VI Univ., Orsay, France) and A. BOLCS and P. OTT (Lausanne, Ecole Polytechnique Federale, Switzerland). *AIAA Journal* (ISSN 0001-1452), Vol. 34, No. 7, 1996, pp. 1331–1340. 32 Refs.

A three-dimensional Navier–Stokes solver is applied to the computation of unsteady nozzle flows resulting from fluctuating backpressure and validated by comparison with experimental measurements. The flow is modeled by the three-dimensional compressible Navier–Stokes equations using the Launder–Sharma near-wall k–ϵ turbulence model. The mean-flow and turbulence- transport equations are integrated in time using a first-order implicit scheme with third-order MUSCL upwind-biased Van Leer flux-vector-splitting. The configuration studied is a three-dimensional nozzle, with thick sidewall boundary layers. The three-dimensional effect is produced by the shock wave-boundary layer interaction at the corners of the nozzle, where an important three-dimensional recirculating zone appears. The flow is unsteady because of backpressure fluctuation, produced experimentally by a rotating rod of elliptical cross section. The unsteady computations were run for a backpressure fluctuation frequency of 180 Hz. The computed and measured results compare satisfactorily. An analysis of steady and unsteady losses is undertaken. (Author)

A96-31984 Implicit computation of three-dimensional compressible Navier–Stokes equations using k–ϵ closure. G. A. GEROLYMOS and I. VALLET (Paris VI Univ., Orsay, France). *AIAA Journal* (ISSN 0001-1452), Vol. 34, No. 7, 1996, pp. 1321–1330. 99 Refs.

A computational method for the numerical integration of the Favre–Reynolds-averaged three-dimensional compressible Navier–Stokes equations using the Launder–Sharma near-wall k-epsilon turbulence closure is developed. The mean flow and turbulence transport equations are discretized using a finite volume method based on MUSCL Van Leer flux-vector splitting with Van Albada limiters. The mean flow and turbulence equations are integrated in time using a fully coupled approximately factored implicit backward Euler method. The resulting scheme is robust and was found stable for local-time steps with Courant–Friedrichs–Lewy number equal to 50. Higher time steps are possible but not optimal for convergence. Results are presented for the three Delery transonic channel test-cases; although these are nominally two-dimensional, three-dimensional computations presented quantify the important three-dimensional effects induced by the sidewall boundary layers. Finally computational results are compared with the experiment for a geometrically three-dimensional transonic nozzle. (Author)

A96-27387 Aerodynamic and acoustic calculations of transonic nonlifting hovering rotors. J. PRIEUR and M. COSTES (ONERA, Chatillon, France) and J. D. BAEDER (Maryland Univ., College Park). *American Helicopter Society, Journal* (ISSN 0002-8711), Vol. 41, No. 2, 1996, pp. 17–26. 17 Refs.

Results from an Euler and a full potential solver are compared for a nonlifting transonic hovering rotor. The correlation is very good on the blade, but the aerodynamic disturbances are damped by the full potential code off the blade tip. The ability of both codes to provide near-field input data for acoustic predictions using Lighthill's acoustic analogy is checked. Acoustic results using Euler input data show a fairly good correlation with experiment and with direct Euler far-field calculations, even in the case of delocalization. On the contrary, the precision of the full potential results is insufficient for that purpose, and further work is needed to improve the code's stability and accuracy off the blade. (Author)

A96-24890 European research cooperation on propellers noise and aerodynamics. C. CASTAN, A. DUMAS, and P. CLOAREC (Aerospatiale Aeronautique, Toulouse, France). *Aspects of engine airframe integration for transport aircraft; Proceedings of the DLR Workshop*, Braunschweig, Germany, 1996 (A96-24878 05-05), DLR, Cologne, Germany (DLR-Mitteilung 96-01), 1996, pp. 16.1–16.18. 10 Refs.

The knowledge gained in Europe on transonic propellers remained limited to uncoordinated national efforts all along the 1970s and the 1980s. With the 1990s, the early Brite/Euram Workprogram took into account the European lack of knowledge in the field of advanced propellers and identified two critical areas, one in the field of aerodynamics, the other in the field of acoustics. European organizations like the CEC, the GARTEUR, and Airbus Industrie, sponsored a number of programs that led to a consistent approach to high speed propellers. Those described here are SNAAP, GEMINI, GEMINI II, APIAN and the Tractor PropFan, to which Aerospatiale contributed in cooperation with a large number of partners from industry, universities, and research centers. (Author)

A96-24884 3D boundary layer computations on wing-pylon-nacelle configuration. R. HOUDEVILLE (ONERA, Centre d'Etudes et de Recherches de Toulouse, France); P. BARDOUX (ONERA, Chatillon, France); and V. MOREUX (Aerospatiale Aeronautique, Toulouse, France). *Aspects of engine airframe integration for transport aircraft; Proceedings of the DLR Workshop*, Braunschweig, Germany, 1996 (A96-24878 05-05), DLR, Cologne, Germany (DLR-Mitteilung 96-01), 1996, pp. 9.1–9.16. 26 Refs.

To calculate the flow over complex civil aircraft geometries, the Euler version of the CANARI solver is used to provide boundary conditions to the boundary layer solver 3C3D. This code was developed to handle complex flow configurations on realistic geometries; it solves the Prandtl equations along the local streamlines without the help of the generalized coordinates, as it is usually done in boundary layer calculations. Emphasis is put on the description of the boundary layer solver and two examples of application are presented: the first one is a twin-engine transonic civil aircraft, and the second one a wing-pylon-nacelle configuration. (Author)

A96-24882 Application of Navier–Stokes methods for engine/airframe integration. E. CHAPUT, C. GACHERIEU, and L. TOURRETTE (Aerospatiale, Toulouse, France). *Aspects of engine airframe integration for transport aircraft; Proceedings of the DLR Workshop*, Braunschweig, Germany, 1996 (A96-24878 05-05), DLR, Cologne, Germany (DLR-Mitteilung 96-01), 1996, pp. 7.1–7.12. 17 Refs.

The parallel Navier–Stokes code NSMB has been used to accurately compute the flow field around a twin-engine aircraft configuration. A cost-effective time-integration method is used in conjunction with a genuine multidomain implementation of both algebraic and $\frac{1}{2}$-equation turbulence models. The validation of the approach is presented, dealing with two-dimensional and three-dimensional test cases relevant to the general application of engine/airframe integration. An accurate prediction of the separation of the boundary layer induced by a pressure gradient is obtained by an enhanced Johnson–King type turbulence model. The computed flow about a transonic wing-body configuration is quite sensitive to the turbulence model in use. The successful application of the code to the twin-engine aircraft AS28G configuration is presented with an emphasis on sensitivity to mesh sizes ranging from 0.4 to 3.5 million points. (Author)

A96-20925 Stable conservative multidomain treatments for implicit Euler solvers. A. LERAT and Z. N. WU (Ecole Nationale Superieure d'Arts et Metiers, Paris, France). *Journal of Computational Physics* (ISSN 0021-9991), Vol. 123, No. 1, 1996, pp. 45–64. 53 Refs. Grant DRET-90-162; DRET-92-169. Document available from AIAA Dispatch.

Implicit time-dependent volume methods on block-structured grids are here used to solve the steady compressible Euler equations in multidomain treatments. Both continuous and discontinuous one-dimensional matchings are given for unconditionally Gustafsson, Kreiss, Sundstrom (1972) stable and conservative treatments; these are extended to two-dimensional patched grids. Transonic flow calculations over one- and two-element airfoils are used to illustrate the efficiency of the interface conditions in question in this method.

A96-17078 Steady transonic flow computations using overlapping grids. Z.-N. WU (Ecole Nationale Superieure d'Arts et Metiers, Paris, France). *Proceedings of the 14th International Conference on Numerical Methods in Fluid Dynamics*, Bangalore, India, 1994 (A96-17053 03-34), Berlin, Springer-Verlag (Lecture Notes in Physics., Vol. 453), 1995, pp. 183–187. 5 Refs. Grant DRET-92-169.

An implicit multidomain technique is considered for steady state transonic flow computations using overlapping grids. The convergence to steady state and uniqueness of steady state solution of this technique are studied. (Author)

A96-15492 New mixed Van Leer flux splitting for transonic viscous flow. R. FERRAND and S. AUBERT (Lyon, Ecole Centrale, Ecully, France). *AIAA Journal* (ISSN 0001-1452), Vol. 34, No. 1, 1996, pp. 190–193. 7 Refs. Document available from AIAA Dispatch.

A new mixed Van Leer method that introduces a central scheme contribution in the low Mach number region is introduced. The expression of the Van Leer flux is preserved, and only conservative variable coefficients are modified in a MUSCL formulation. The results show conservation of robustness in shock capture and a dramatic improvement in the treatment of viscosity in the boundary layers.

A95-38520 Validation of code using turbulence model applied to three-dimensional transonic channel. J. CAHEN (SNECMA, Moissy-Cramayel, France) and V. COUAILLIER, J. DELERY, and T. POT (ONERA, Chatillon, France). *AIAA Journal* (ISSN 0001-1452), Vol. 33, No. 4, April 1995, pp. 671–679. Previously cited in issue 07, Accession A93-22693. 18 Refs.

The flow induced by a swept bump mounted on the lower wall of a transonic channel has been thoroughly analyzed on the basis of surface pressure measurements, surface flow visualizations, and field measurements by a three-component LDV system. The same flow has been computed by a code solving the full time-averaged Navier–Stokes equations by a method using an explicit centered finite difference scheme applied to a finite volume approach. An algebraic model using the mixing length concept and the Jones–Launder (*k*, epsilon) transport equation model have been considered. Both models give a faithful prediction of the essential features of the flow; in particular, the shock pattern forming in the channel is well predicted. However, the (*k*, epsilon) model shows a clear superiority in the prediction of the behavior of the interacting dissipative layers. (Author)

A95-36626 Numerical vs. non-numerical robust optimisers for aerodynamic design using transonic finite-element solvers. A. DERVIEUX, J.-M. MALE, and N. MARCO (INRIA, Sophia-Antipolis, France); J. PERIAUX and B. STOUFFLET (Dassault Aviation, Saint-Cloud, France); H. Q. CHEN (NAI, China; Dassault Aviation, Saint-Cloud, France); and M. CEFROUI (Paris VI Univ., France). *AIAA 12th Computational Fluid Dynamics Conference and Open Forum*, San Diego, CA, 1995, Collection of Technical Papers. Pt. 2 (A95-36501 09-34), AIAA, Washington, DC, 1995, pp. 1339–1347 (AIAA Paper 95-1761). 12 Refs.

How to solve shape optimization with unstructured meshes? We consider exact gradients, descent or one-shot algorithms and hierarchical parametrization. Applications are transonic flows governed by Euler equations around an airfoil or in a three-dimensional nozzle. (Author)

A95-35348 Solution of unsteady Euler equations for flow around a supersonic wing in the transonic region (Resolution des equations d'Euler instationnaires autour d'une voilure de type supersonique dans le domaine transsonique). F. REY (Aerospatiale, Div. Avions, Toulouse, France). *EAC '94—The supersonic transport of second generation; Proceedings of the 7th European Aerospace Conference*, Toulouse, France, 1994 (A95-35310 09-01), Association Aeronautique et Astronautique de France, Paris, 1995, pp. 505–519. 7 Refs. In French.

Aerospatiale-Avions has an Euler code available to simulate unsteady flows around rigid or flexible bodies. To perform aeroelastic applications, we solve the Euler equations in a small displacement hypothesis to keep a fixed mesh. First, we present an application on a supercritical wing of Airbus type and compare some results with experimental data. Then, an application to a supersonic wing of the ATSF type is discussed. (Author)

A95-34520 Feasibility of arbitrary pitching motion controlled by piezoceramic actuators to reduce blade-vortex interaction noise. M. P. AMERIGO (Eurocopter France, Marignane) and J. D. BAEDER (Maryland Univ., College Park). *Proceedings of the AHS 51st Annual Forum, Fort Worth, TX, 1995*, Vol. 2 (A95-34426 09-01), American Helicopter Society, Alexandria, VA, 1995, pp. 1238–1253. 20 Refs.

Previous work utilizing piezoceramic actuators to dynamically modify the twist of rotor blades has concentrated on vibration reduction. This study investigates the feasibility of using the same approach for noise control by reducing the fluctuating loads caused by blade-vortex interactions (BVIs). A two-dimensional airfoil version of the Transonic Unsteady Rotor Navier–Stokes code is modified to model the interaction of a pitching airfoil with a vortex. The airfoil motion is prescribed using quasi-steady thin airfoil theory and a knowledge of the time history of the blade loading during a previous vortex passage. The influence of the vortex on the loading of the airfoil is found to be alleviated considerably by pitching the airfoil. Also, the time rate of change of the loading, which influences the amplitude of the accompanying acoustic signature, is reduced by a factor of 2 after only one control loop. A strong transonic case indicates that such an approach remains effective even when supersonic pockets are present. An analytical study of the response of the blade with embedded piezoceramic actuators indicates that the frequencies of the twist required to perform this control are within the capacity of the state-of-the-art 'smart rotors'. (Author)

A95-32985 Assessment of a low storage technique for multi-stage turbomachinery Navier–Stokes computations. A. FOURMAUX (ONERA, Chatillon, France). ONERA, TP 1994-157, 1994. 6 Refs.

For industrial applications, CPU memory and time needed for numerical simulations of the unsteady flow through turbomachine stages have to be reasonable. In this paper, a technique is presented that is specially intended to reduce the computational domain. The actual blade-to-blade passage geometry can be computed owing to an approximate treatment of the flow continuity condition on periodic boundaries. This treatment induces some approximations in the description of the unsteady flow phenomena. These are analyzed in the case of a transonic turbine stage in which the viscous flow is computed. The main conclusion is that this approach, which has been in use for several years now at ONERA for Euler flows, can also be used fruitfully for Navier–Stokes flow simulations. (Author)

N95-19258 Experimental techniques for measuring transonic flow with a three dimensional laser velocimetry system. Application to determining the drag of a fuselage (Technique experimentale de mesure en ecoulement transsonique avec un systeme de velocimetrie laser tridimensionnel. Application a la determination de la trainee d'un fuselage). A. SERAUDIE, A. MIGNOSI, J. B. DOR, and S. PRUDHOMME. *AGARD, Wall Interference, Support Interference, and Flow Field Measurements*. p. 6 (SEE N95-19251 05-34). In French.

Recent developments in laser anemometry have been used to design a three dimensional laser system which has been in operation at the CERT ONERA's T2 wind tunnel since December 1989: fiber optics (to lead the light between the source and the emitting optics), Fast FOURIER Transfer Doppler processors (to analyze the Doppler signals), high power transmission system (to provide color separation), digital control of displacement motors and real time operation (to move the measuring point during the run). This device works well for the short run times of the T2 wind tunnel, providing a good accuracy which allows 30–50 measurement points during 60–120 s of the test. After a complete description of the three-dimensional laser velocimetry system, the present paper will develop some typical measurements which have been performed. For each case we will present some test results obtained under transonic conditions: shock wave probing (shape and location on the upper side of a two-dimensional transonic model); and three-dimensional velocity measurements in forward and backward scatter configurations with the wall approach for areas without good accessibility. To obtain the drag of a fuselage, a vertical plane located downstream of the model was measured with two devices: laser velocimetry in order to obtain the three components of velocity; and a pressure rake providing the static and total pressures. The combination of these measurements (pressure and velocity) allowed the calculation of the total drag of the three-dimensional model. (Author)

N95-17871 Test data on a non-circular body for subsonic, transonic and supersonic Mach numbers. P. CHAMPIGNY. *AGARD, A Selection of Experimental Test Cases for the Validation of CFD Codes*, Vol. 2, p. 11 (SEE N95-17846 04-02).

Measurements on a noncircular body were made in ONERA wind tunnels. This body, representative of nonconventional missile shapes, was studied for Mach numbers from 0.4 to 3.0 (S2MA wind-tunnel) and 4.5 (S3MA wind-tunnel), angles of attach up to 20 deg and sideslip angles up to 10 deg, with a free transition. The data base consists of static pressure measurements. The intent of the experiment was to provide data for evaluation of three-dimensional flow computation methods, as part of a research program sponsored by the 'Direction des Recherches, Etudes et Techniques' of the French Ministry of Defense. The flow exhibits large separation regions and strong vortices, even at low angles of attack, due to the particular shape of the body (lenticular cross-section). (Author)

A95-30023 Computation of unsteady nozzle flow due to fluctuating back-pressure using Euler equations. G. A. GEROLYMOS and J.-P. BREUS (Paris VI Univ., France). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, 1994, p. 14 (ASME Paper 94-GT-91). 50 Refs.

A numerical method is developed for the integration of the unsteady two-dimensional Euler equations using a third-order upwind-biased scheme with Van Leer flux-vector-splitting and Van Albada limiters, with MUSCL space discretization, and an explicit two-stage Runge–Kutta time integration procedure. The method is applied to the numerical computation of flows in a transonic nozzle with fluctuating back-pressure and compared with available experimental data. The effects of frequency and amplitude on the shock-wave response are studied in detail. Despite the use of an inviscid flow model the unsteady pressures are quite satisfactorily predicted over the range of frequencies studied. The numerical method is then used to study the effect of the back-pressure fluctuation amplitude on the shock-wave oscillation. At large amplitudes the flowfield response to back-pressure fluctuation is essentially nonlinear. (Author)

German Aerospace Literature
This month: *Transonic Flows*

A97-13155 A finite element method for the compressible Navier–Stokes equations and its application to transonic nozzle flows. J. STILLER (Dresden Univ. of Technology, Germany). *Aerothermodynamics of internal flows III; Proceedings of the 3rd International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows*, Beijing, China, 1996 (A97-13101 02-34), World Publishing Corp., Beijing, 1996, pp. 437–444. 7 Refs.

A streamline upwind Petrov–Galerkin FEM for solving the compressible Navier–Stokes equations is presented. It is based on conservative variables and does not require a special equation of state. The method was successfully applied to simulate the transonic flow in a double throat nozzle at Reynolds numbers ranging from 1×10^2 to 16×10^2. (Author)

A96-40585 Transition control on a transonic laminar airfoil with suction panel. L. JORGNESEN and W. NITSCHE (Berlin, Technische Univ., Germany). *Proceedings of the ICAS 20th Congress*, Naples, Italy, 1996, Vol. 1 (A96-40526 11-01), AIAA, Inc., Reston, VA, 1996, pp. 494–503. 11 Refs.

The efficiency of future airliners can be improved significantly by applying laminar wing technology. Hybrid laminar flow-control designs employ suction panels at the wing leading edge to stabilize the laminar boundary layer. The present investigation was aimed at a control system for such suction panels. For this purpose, a transonic laminar airfoil with a segmented suction panel was used to develop a sensor-based monitoring system. The system measures online the transition position on the active suction panel without disturbing the laminar flow. Tests were performed at different flow and boundary layer suction conditions and checked against reference techniques. Furthermore, the sensor system was employed to control the suction distribution of the airfoil in order to minimize the total drag. (Author)

A96-38462 Experiments on transonic flutter of a two-dimensional supercritical wing with emphasis on the non-linear effects. G. SCHEWE and H. DEYHLE (DLR, Inst. fuer Aeroelastik, Goettingen, Germany). *Unsteady aerodynamics; Proceedings of the Conference*, London, UK, 1996 (A96-38453 10-02), Royal Aeronautical Society, London, 1996, pp. 9.1–9.10. 8 Refs. Documents available from AIAA Dispatch.

Results of experiments in which flutter measurements were performed with a supercritical two-dimensional wing, primarily in the transonic flow regime, are reported. It is found, in particular, that the critical dynamic pressure at flutter onset in the transonic regime can be half of the corresponding subsonic value. The results provide experimental evidence of the limit cycle oscillation. The possiblity of controlling transonic flutter by relatively small forces is demonstrated. Finally, areas of further research are outlined.

A96-36575 Theoretical knowledge base for accelerated transonic design. H. SOBIECZKY (DLR, Inst. fuer Stroemungsmechanik, Goettingen, Germany). *AIAA 1st Theoretical Fluid Mechanics Meeting*, New Orleans, LA, 1996, p. 12 (AIAA Paper 96-2115). 31 Refs.

Based on the classical gasdynamic models for transonic flow, airfoil and three-dimensional configuration design tools are developed for fast predesign studies. Singularities, sonic and shock surfaces are seen as part of the initial geometries in inverse design studies. For the direct approach, geometry preprocessors are developed and visualization postprocessors are adapted to control the designed and expected mechanisms. Creating a knowledge base for future aerodynamic design expert systems is seen as a goal of this effort combining theoretical aerodynamics and modern software technology. (Author)

A96-30275 Numerical investigation of oscillating nacelles in transonic flow (Numerische Untersuchung der transsonischen Stroemung an schwingenden Triebwerksgondeln). A. KNIPFER (DLR, Inst. fuer Aeroelastik, Goettingen, Germany). DLR, Cologne, Germany (DLR-Forschungsbericht 95-39), 1995, p. 131. 27 Refs. In German. DLR-FB 95-39; ISSN 0939-2963.

The nacelles of modern aeroengines are constantly increasing in size. Thus, engine airloads are becoming stronger and their importance for aeroelastic stability is becoming more significant. In this report an annular wing serves as a model for the nacelle. The principal goal is the computation of the loads acting on an oscillating annular wing in transonic inviscid flow. A Finite Volume Method based on a solution of the Euler equations is utilized. The influence of the computed airloads on aeroelastic stability is investigated with reference to the example of an isolated, elastically mounted annular wing. In addition, the fan jet is simulated by extending a steady actuator disk method in such a way that unsteady flows can be treated. (Author)

A96-24888 Application of an inverse design method to the design of transonic nacelles. M. HEPPERLE and W. BARTELHEIMER (DLR, Inst. fuer Entwurfsaerodynamik, Braunschweig, Germany) and O. BOUSQUET (SNECMA, Paris, France). *Aspects of engine airframe integration for transport aircraft; Proceedings of the DLR Workshop*, Braunschweig, Germany, 1996 (A96-24878 05-05), DLR, Cologne, Germany (DLR-Mitteilung 96-01), 1996, pp. 13.1–13.18. 16 Refs.

We describe the extension and application of a design method to nacelles for turbofan engines under transonic flow conditions. Starting from a generic nacelle shape and a prescribed pressure or Mach number distribution, the inverse design problem for the unknown nacelle geometry is solved. The method uses an existing Euler solver to obtain solutions of the transonic, compressible flow field around the configuration. Examples show that the method is capable of handling axisymmetric and three-dimensional flow conditions on isolated and installed nacelles, and that it can be used to design the external as well as the internal surfaces of a nacelle. (Author)

A96-24887 Interference phenomena of upper-wing-mounted engines. A. RONZHEIMER, R. RUDNIK, O. BRODERSEN, and C.-C. ROSSOW (DLR, Inst. fuer Entwurfsaerodynamik, Braunschweig, Germany). *Aspects of engine airframe integration for transport aircraft; Proceedings of the DLR Workshop*, Braunschweig, Germany, 1996 (A96-24878 05-05), DLR, Cologne, Germany (DLR-Mitteilung 96-01), 1996, pp. 12.1–12.14. 6 Refs.

The influence of upper wing mounted engines on the aerodynamic interference of a transport aircraft is investigatcd numerically, focussing on cruise flight conditions. The geometry of the configuration is identical to the existing VFW 614 aircraft, but without tail and fin. It is shown that the main effects are dominated by the flow above the wing in front of the engine and pylon. Further effects result from the fact that engine, pylon, wing, and fuselage form a channel inboard of the pylon. Therefore, the development of exceptional transonic regions at moderate freestream Mach numbers is established. (Author)

N96-13598 Cost efficient calculation of compressible potential flow around a helicopter rotor including free vortex sheet by a field panel method. A. ROETTGERMANN and S. WAGNER. *AGARD, Aerodynamics and Aeroacoustics of Rotorcraft*, p. 10 (SEE N96-13582 02-01). Grant DFG-WA424/8.

To consider compressible transonic effects a vortex lattice method for the computation of the rotor flow is coupled with a field panel method. For this purpose Cartesian grids are used which are not adapted to the contour and only discretize the domain of the nonlinear flow. The basis for this procedure is the separation of the full potential equation into the Laplacian operator and the nonlinear terms. The developed program ROFPM is validated at several test cases of the CARADONNA rotor. (Author)

A96-18191 The European Transonic Wind Tunnel—Testing at flight Reynolds numbers. T. SAUNDERS (European Transonic Windtunnel GmbH, Cologne, Germany). *AIAA 34th Aerospace Sciences Meeting and Exhibit*, Aerodynamics of a centrifugal compressor, Reno, NV, 1996, p. 12 (AIAA Paper 96-0227).

The history of the establishment of ETW by France, Germany, the Netherlands, and the United Kingdom is described briefly. The engineering concept is for a transonic cryogenic wind tunnel of high quality with a level of productivity appropriate for industrial use in the design and development of new civil and military aircraft projects. Commissioning followed erection of the tunnel on schedule and revealed only relatively minor problems requiring adjustment of the original design. Calibration tests started during commissioning and showed flow quality of the very high standard required by the specification. Early model tests, mostly carried out in collaboration with clients, to validate the wind tunnel have shown good repeatability and have clearly indicated the predicted strong effect of Reynolds number on drag. (Author)

A96-17189 Shock boundary-layer interaction on transonic airfoils for laminar and turbulent flow. M. SWOBODA and W. NITSCHE (Berlin, Technical Univ., Germany). *Journal of Aircraft* (ISSN 0021-8669), Vol. 33, No. 1, 1996, pp. 100–108. 12 Refs.

This article deals with the interaction between shocks and the boundary layer in a transonic airfoil flow. Here, examining comparatively the shock boundary-layer interaction in a laminar boundary layer as well as in a fully turbulent one was the main focus of interest. The experimental investigations were carried out in the transonic wind tunnel of the Technical University of Berlin. Apart from conventional time-averaging measuring techniques, such as schlieren photography, Laser-2-Focus anemometry, pressure measurement, and visualization, two unsteady measuring techniques were used for investigating the dynamic effects associated with the shock boundary-layer interference. The first unsteady measuring technique concerns a new type of highly sensitive piezoreceivers, and in the second technique a piezofoil array was mounted on the airfoil model. The unsteady measuring techniques were used to detect details in the separation regions and to determine the position of transition in the case of laminar flow. (Author)

A96-17083 Far field boundary conditions based on characteristic and bicharacteristic theory applied to transonic flows. D. SCHULZE (Berlin, Technical Univ., Germany). *Proceedings of the 14th International Conference on Numerical Methods in Fluid Dynamics*, Bangalore, India, 1994 (A96-17053 03-34), Berlin, Springer–Verlag (Lecture Notes in Physics. Vol. 453), 1995, pp. 211–215. 6 Refs.

The influence of two different far boundary conditions on the solution of a transonic profile flow is studied when the distance to the outer grid boundary is reduced. One of the boundary conditions makes use of characteristics in one-dimensional flows. The other results from an analysis of bicharacteristics in two-dimensional flows. These boundary conditions were combined with a cell-vertex method for solving the Euler equations in two-dimensional. The method was used to simulate the transonic flow past a NACA0012 profile set at an angle of attack of 1 deg and a freestream Mach number of 0.85. (Author)

A96-12473 Blunt body flow—The transonic region. S. MOELDER (DLR, Goettingen, Germany). *Shock waves at Marseille I: Hypersonics, shock tube and shock tunnel flow; Proceedings of the 19th International Symposium on Shock Waves*, Marseille, France, 1993 (A96-12460 01-02), Berlin and New York, Springer–Verlag, 1995, pp. 101–104. 4 Refs.

For supersonic flow of an ideal gas over planar and axisymmetric bodies we consider the transonic region bounded by the sonic line, the body surface, the downstream surface of the shock wave and the limiting characteristics. Using curved shock theory, we establish the conditions under which the shock surface is (type I) or is not (type II) a boundary of the transonic region. It is found that the existence of type I or type II flow is determined solely by the values of free stream Mach number, specific heat ratio of the gas, and transverse-to-longitudinal curvature of the shock surface. When the curvature ratio is greater than 1, only type II flow is possible irrespective of the specific heat ratio. When the curvature ratio is less than $-2/(\gamma - 1)$ then only type I is realized. When the curvature ratio is between 1 and $-2/(\gamma - 1)$ the appearance of type I or type II is determined by the free stream Mach number. With proper choice of curvatures the results apply to general three-dimensional shock waves as well. (Author)

A96-11042 Viscous flow field computations for turbine cascades using different turbulence models. L. J. LENKE and H. SIMON (Duisburg Univ., Germany). *Numerical methods in laminar and turbulent flow.* Vol. 9, Pt. 1; *Proceedings of the 9th International Conference*, Atlanta, GA, 1995 (A96-11001 01-34), Pineridge, Swansea, Wales, UK, 1995, pp. 758–772. 16 Refs.

The influence of the turbulence modelling on viscous flow field calculation results has often been discussed in the past. For a meaningful comparison of different turbulence models the access to reliable measurement data is necessary. Therefore different turbine profiles are chosen for transonic two-dimensional flow field calculations using different turbulence models. The algebraic model of Baldwin and Lomax, the standard k–ϵ model with wall functions, a low-Reynolds number and the k–ω model are considered in this investigation. The main differences between the models become apparent in the trailing edge region. The turbulence modeling influences the boundary layer thickness and the shape of the shear layers and the separation region in the wake flow. For the high Mach numbers appearing in this region, a strong influence on the flow field due to small shear layer changes has been found. (Author)

A95-44872 Design and flow field calculations for transonic and supersonic radial inflow turbine guide vanes. A. W. REICHERT (Siemens AG, Muelheim, Germany) and H. SIMON (Duisburg Univ., Germany). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, Houston, TX, 1995, p. 14 (ASME Paper 95-GT-097). 8 Refs.

The design of radial inflow turbine guide vanes depends very much on the discharge conditions desired, especially if the choking mass flow is reached. Because of the choking mass flow condition and supersonic discharge Mach numbers, an inverse design procedure based on the method of characteristics is presented. Various designs which correspond to different discharge Mach numbers are shown. Viscous and inviscid flow field calculations for varying discharge conditions show the properties of the guide vanes at design and off-design conditions. In a previous paper (Reichert and Simon, 1994), an optimized design for transonic discharge conditions was published. In the present paper, additional results concerning the optimum design are presented. For this optimum design, an advantageous adjusting mechanism for a variable-geometry guide vane was developed. The effect of guide vane adjustment on the discharge conditions was investigated using viscous flow field calculations. (Author)

A95-44866 Aerodynamics of a centrifugal compressor impeller with transonic inlet conditions. H. KRAIN, B. HOFFMANN, and H. PAK (DLR, Inst. fuer Antriebstechnik, Cologne, Germany). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, Houston, TX, 1995, p. 9 (ASME Paper 95-GT-079). 28 Refs.

Performance and laser measurement results are presented for a transonic centrifugal compressor stage, equipped with a backswept rotor designed for 586 m/s tip speed and a mean relative inlet tip Mach number of 1.30. The flow-field features of the rotor are analyzed in detail with the help of laser measurements and the results obtained from a three-dimensional viscous calculation. Both laser measurements and calculations are carried out for the impeller's design point, and a comparison between measured and calculated data is presented for four measurement planes representing the inlet-, exit-, and channel-flow conditions. The maximum relative Mach number is found to be 1.45, and a jet/wake type flow exists in the rear part of the flow channel with the wake concentrating in the shroud region. For the operating point investigated, the wake development is due more to the strong shroud curvature in the meridional plane than to a shock-induced boundary layer separation. Laser measurements and calculations point to a significant flow displacement in the impeller exit region caused by the wake flow. The theoretical results indicate that a flow separation occurs also at the front side of the vaneless diffuser. (Author)

A95-44865 Computation of three-dimensional viscous transonic turbine stage flow including tip clearance effects. R. MERZ, J. KRUECKELS, J. F. MAYER, and H. STETTER (Stuttgart Univ., Germany). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, Houston, TX, 1995, p. 8 (ASME Paper 95-GT-076). 17 Refs.

A numerical method to solve the three-dimensional Navier–Stokes equations for the flow in transonic turbine stages with tip gap leakage is presented. Viscous flow in a transonic turbine stage was simulated. The high pressure difference at the rotor blade tip results in a supersonic jet. The relative motion of the casing wall is oriented against the tip leakage flow and tends to reduce it. Very large velocity gradients in the tip region pose a challenge for the numerical simulation. Computational results are compared with experimental data obtained in operation. Measurements include data for the tip leakage jet. The numerical method is based on a conservative finite-volume cell-vertex scheme in cylindrical coordinates with central difference approximation and Runge–Kutta time stepping. Convergence is accelerated by use of a multigrid method and implicit residual smoothing with variable coefficients. The Baldwin–Lomax turbulence model is used for closure. The boundary condition treatment at inlet and outlet as well as the coupling of stator and rotor flow is achieved by use of non-reflective boundary conditions. The tip region is discretized by an additional grid within a multiblock approach. (Author)

A95-44727 The performance of a new axial single stage transonic compressor. G. SCHULZE, C. BLAHA, and D. K. HENNECKE (Darmstadt, Technical Univ., Germany) and J. M. HENNE (MTU Motoren- und Turbinen-Union Muenchen GmbH, Munich, Germany). *ISABE—Proceedings of the 12th International Symposium on Air Breathing Engines*, Melbourne, Australia, 1995, Vol. 2 (A95-44654 12-07), AIAA, Washington, DC, 1995, pp. 783–792 (ISABE 95-7072). 11 Refs.

This paper presents the first test results of a new axial single-stage transonic compressor. The goal of the research activities with this test rig is the evaluation of the computational methods used for design and detailed flow analysis. The compressor design was carried out by using quasi-three-dimensional and fully three-dimensional methods. In addition to a design-point optimization, the three-dimensional Euler solver, modified by inclusion of viscous effects, was used to predict the entire compressor map. This map and some calculation results are presented, followed by a brief description of the test rig and the measurement technique. The measured compressor map and downstream flow distributions for different operating points are shown and discussed in comparison with the predictions. (Author)

A95-42628 Influence of 3D boundary-layer displacement on unsteady transonic potential flow computations. U. R. MUELLER and H. HENKE (Daimler-Benz Aerospace Airbus GmbH, Bremen, Germany). *Proceedings of the International Forum on Aeroelasticity and Structural Dynamics 1995*, Manchester Business School, UK, 1995, Vol. 1 (A95-42613 11-39), Royal Aeronautical Society, London, 1995, pp. 16.1–16.8. 10 Refs.

A fully three-dimensional viscous-inviscid interaction scheme was developed that furnishes the simultaneous solution of an inviscid three-dimensional solver and a three-dimensional boundary-layer method. Here, a transonic small perturbation potential method was enhanced by a three-dimensional integral boundary-layer technique. Example calculations for the LANN wing at $M = 0.82$ proved the scheme to work well in both the direct mode of boundary-layer calculation and the simultaneous mode. (Author)

A95-42623 Prediction of unsteady aerodynamic forces for elastically oscillating wings using CFD methods. W. WEGNER (DLR, Inst. fuer Aeroelastik, Goettingen, Germany). *Proceedings of the International Forum on Aeroelasticity and Structural Dynamics 1995*, Manchester Business School, UK, 1995, Vol. 1 (A95-42613 11-39), Royal Aeronautical Society, London, 1995, pp. 11.1–11.14. 22 Refs.

Theoretical flow procedures that occur in the transonic speed range in modern commercial aircraft are researched. One main point is the calculation of periodic motion-induced air forces in order to identify unwanted aeroelastic instablities for an entire configuration as well as for individual components. A useful aerodynamic stability criterion can be derived from the power coefficient. The influence of the friction of viscous flow is ignored. The theoretical results of selected standard wings are compared with available measurements. The starting point of a calculation is the spatial three-dimensional Euler equations which are solved by an efficient upwind procedure in which time-dependent boundary conditions are observed. The upwind procedure is based on the complete Riemann solution in a closed form. In contrast to presently employed methods based on approximate Riemann solutions or on flux splitting, the numerical fault is dependent only on the local discretization and not on the local gradient of the solution. (Author)

A95-42621 The importance of geometric modeling of elastically oscillating wings in unsteady transonic CFD calculations. D. NEISIUS (DLR, Inst. fuer Aeroelastik, Goettingen, Germany). *Proceedings of the International Forum on Aeroelasticity and Structural Dynamics 1995*, Manchester Business School, UK, 1995, Vol. 1 (A95-42613 11-39), Royal Aeronautical Society, London, 1995, pp. 9.1–9.6. 6 Refs.

Aeroelastic investigations increasingly apply modern CFD methods in unsteady aerodynamics. To solve three-dimensional problems, the wing is still often overly simplified as was done in formerly used linear methods (thin geometrics, no geometric deformation). The neglected details of the geometric model of a wing are, for example, static bending and torsion as they appear on the big wings of modern passenger aircraft at travel speed, as well as the modeling of the exact form of the wingtip. All of these details have an influence on the unsteady phenomena of Euler solutions. This influence is shown, e.g., on the AMP wing for various types of geometric models. The main objective here is the comparison of the unsteady Euler solutions of different models of the same wing. In some cases there is a significant difference in the solutions—in pressure distribution as well as in integral coefficients. (Author)

A95-40575 A comparison of nucleation theories in transonic nozzle flows with homogeneous condensation. C. F. DELALE (DLR Inst. fuer Stroemungsmechanik, Goettingen, Germany). DLR, Cologne, Germany (DLR-Forschungsbericht 94-37), 1994, p. 38. DLR-FB 94-37; ISSN 0939-2963. 10 Refs.

The classical and Dillmann-Meier theories are compared in transonic nozzle flows of moist air under atmospheric supply conditions using the asymptotic solution of Delale, Schnerr, and Zierep. The comparison is made in slender nozzles using two surface tension formulas, one fitted to the experiments of Peters and Paikert by the classical theory, the other extrapolated from the well known room temperature value in the range of temperatures investigated. The droplet growth law is fixed by the Hertz–Knudsen formula. It is shown that the Dillmann–Meier theory predicts higher nucleation rates than the classical theory together with a delay for the onset of condensation when either of the surface tension formulas are employed; however, the pressure distribution downstream of the condensation zones is precisely achieved in comparison with the static pressure measurements. (Author)

A95-38649 Experiments on the influence of angle of attack and miss distance on 2D-BVI. F. OBERMEIER and O. SCHUERMANN (Max-Planck-Inst. fuer Stroemungsforschung, Goettingen, Germany). *Proceedings of the CEAS/AIAA 1st Joint Aeroacoustics Conference*, Munich, Germany, 1995, Vol. 1 (A95-38611 10-71), DLR, Bonn, Germany, 1995, pp. 357–364 (CEAS/AIAA 95-050). 17 Refs.

Experimental results of a parametric study on two-dimensional blade vortex interaction noise are presented. Experiments were performed in a transonic wind-tunnel; the airfoil was a NACA 23012 profile. The angle of attack of the airfoil was varied systematically between 0 and 10 deg and the miss distance of the vortex between 0 and 46 mm. The Mach number was always $M = 0.57$. The vortices were produced by a vortex generator ahead of the airfoil; only vortices with a well defined structure were used in the experiments. Results are presented. The absolute density distribution just ahead of the airfoil was measured by means of a Mach–Zehnder interferometer. A sufficiently high time solution was made possible by the use of a Cranz–Schardin camera. The instantaneous pressure distribution at the airfoil's surface was recorded by a set of KULITE-transducers mounted in the leading edge region. These simultaneous measurements make it possible to check the validity whether the pressure can be determined from the density by means of an adiabatic equation of state. The angle of attack turns out to be a very important parameter. An increasing angle leads to a stronger compressibility wave while the transonic wave becomes weaker. This also affects the directional pattern of the emitted sound. For very small miss distance both the compressibility and the transonic waves seem to get weaker. From the surface pressure measurements the sound radiation was calculated by an integral method. It indicates, like in the interferometric flow visualization, that the transonic effects are much stronger than the compressible effects. (Author)

A95-36722 Unsteady flow control on rotor airfoils. W. GEISSLER and H. SOBIECZKY (DLR, Goettingen, Germany). *AIAA 13th Applied Aerodynamics Conference*, San Diego, CA, 1995, TP. Pt. 2 (A95-36627 09-02), AIAA, Washington, DC, 1995, pp. 1024–1032 (AIAA Paper 95-1890). 12 Refs.

Airfoil designs for helicopter rotors are based on compromises. On the advancing side the blade encounters high speed transonic flows including moving shock waves; on the retreating side, however, the high incidence of the blade may lead to dynamic stall with strong hysteresis effects in force and moment loops. It is the aim of the present numerical investigation to systematically develop airfoils which are changing their shape during operation. With specially designed dynamically deforming airfoil sections the dynamic stall characteristics can successfully be influenced. Rather small airfoil modifications show already considerable benefits with respect to a reduction of the negative (nose-down) pitching moment peak, as well as with the avoidance of negative aerodynamic damping. (Author)

A95-36672 Transonic airfoil design with expert systems. R. E. ZORES (DLR, Goettingen, Germany). *AIAA 13th Applied Aerodynamics Conference*, San Diego, CA, 1995, TP. Pt. 1 (A95-36627 09-02), AIAA, Washington, DC, 1995, pp. 466–473 (AIAA Paper 95-1818). 13 Refs.

The combination of new technologies with classical methods for designing transonic airfoils is described. While designing supercritical airfoils the skilled aerodynamicist is usually completely aware of the laws and criteria that guide the designing process. An expert system can help inexperienced engineers reduce the time of learning and exploring new generated airfoils. It speeds up the process of generating, calculating, and analyzing the airfoil. Proven programs, such as inverse CFD solvers, beside new technologies from computer science, can be used to build new tools for aerodynamic design on workstations. A selection of improved workstation tools to design transonic airfoils is described. The capability of the expert system is shown by two examples. The first example raises a more academic question on airfoils which can be concave on their upper side. The second example shows how an expert system can be used to support wind tunnel tests. (Author)

A95-36540 An improved integral equation method for the design of transonic airfoils and wings. W. BARTELHEIMER (DLR, Inst. fuer Entwurfsaerodynamik, Braunschweig, Germany). *AIAA 12th Computational Fluid Dynamics Conference*, San Diego, CA, 1995, Collection of Technical Papers. Pt. 1 (A95-36501 09-34), AIAA, Washington, DC, 1995, pp. 463–473 (AIAA Paper 95-1688). 11 Refs.

A design method for transonic airfoils and wings based on the solution of the Euler/Navier–Stokes equations is described. The applied design strategy is an inverse design method based on the work of Takanashi. The difference between the computed pressure distribution of a given geometry and the prescribed target pressure distribution is iteratively reduced by the solution of an inverse formulated transonic small perturbation equation. In this design method an analysis code is required, such as the Euler/Navier–Stokes code CEVCATS developed at DLR. It is shown that Takanashi's method must be modified in order to ensure the convergence of the design in transonic flow. In addition a smoothing algorithm based on Bezier curves is used to obtain a smooth surface. To estimate the accuracy and convergence of the design method, a redesign of a known transonic airfoil and wing is conducted. In all designed cases, very accurate results are obtained within a small number of design cycles. (Author)

A95-35165 The Cryogenic Ludwieg-Tube of DLR and its new adaptive wall test section. H. ROSEMANN and E. STANEWSKY (DLR, Goettingen, Germany) and G. HEFER (European Transonic Wind Tunnel, Cologne, Germany). *AIAA 26th Fluid Dynamics Conference*, San Diego, CA, 1995, p. 10 (AIAA Paper 95-2198). 9 Refs.

The Cryogenic Ludwieg-Tube of DLR (KRG) is a blow-down wind tunnel especially designed for high Reynolds number research in transonic flow. Reynolds numbers of up to 60×10 exp 6 (two-dimensional) are accomplished by applying stagnation pressures of up to 1 MPa together with low temperatures down to 100 K utilizing cryogenic technology. After verifying the concept and evaluating the performance of the KRG with a slotted wall test section, a second test section with two-dimensional adaptive walls was built to eliminate interferences from the top and bottom wall. A single step algorithm based on a Cauchy's integral method is used for two-dimensional adaptation to compute the interference-free wall contours without needing a model presentation. The walls are adjusted by 19 stepper motors each. A Reynolds number study carried out with a laminar type airfoil in the slotted wall test section shows a strong loss of lift in the Reynolds number range between 8×10 exp 6 and 20×10 exp 6 due to the forward movement of the transition point. First results from the adaptive wall test section demonstrate the applicability of the adaptation procedure. (Author)

N95-17847 2-D airfoil tests including side wall boundary layer measurements. W. BARTELHEIMER, K. H. HORSTMANN, and W. PUFFERT-MEISSNER. *AGARD, A Selection of Experimental Test Cases for the Validation of CFD Codes*, Vol. 2, p. 11 (SEE N95-17846 04-02).

The data presented in this contribution were obtained in the DLR Transonic Wind Tunnel Braunschweig. The intent of the experiment was to provide data giving information on the development of the TWB-side wall boundary layer in the presence of a typical transonic airfoil model for further investigation of the influence of the side wall boundary layer on two-dimensional airfoil measurements. For this purpose boundary layer pitot pressure were measured in 13 different side wall positions around the airfoil. Airfoil pressure distributions were obtained in several spanwise positions by sliding the airfoil model in a spanwise direction. The test cases investigated correspond to the design conditions of the airfoil ($Ma = 0.73$, $\alpha = 1.5$ deg) and to a slow ($\alpha = 0$ deg) and a high ($\alpha = 3.0$ deg) lift value at the same Mach number. For these cases wall pressure distributions were measured on the center slat of the top and bottom walls. Additionally to the pressure measurements some oil flow pictures were made on the upper airfoil surface and the adjacent wind tunnel side wall to get more insight in the structure of the flow. To have well defined wind tunnel boundary conditions for the evaluation by computational methods, the slotted top and bottom walls of the test section were closed for these specific tests. This means, of course, that the presented airfoil pressure distributions do not correspond to free flight conditions and are not comparable to wind tunnel results obtained in slotted or perforated transonic test sections. (Author)

A95-30129 Improved blade profile loss and deviation angle models for advanced transonic compressor bladings. II—A model for supersonic flow. W. M. KOENIG and D. K. HENNECKE (Darmstadt Technical Univ., Germany) and L. FOTTNER (Munich Univ. of the German Armed Forces, Neubiberg, Germany). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, 1994, p. 8 (ASME Paper 94-GT-336). 12 Refs.

New blading concepts, as used in modern transonic axial-flow compressors, require improved loss and deviation angle correlations. The new model presented in this paper incorporates several elements and separately treats blade-row flows having subsonic and supersonic inlet conditions. The second part of the present report focuses on the extension of a well-known correlation for cascade losses at supersonic inlet-flows. It was originally established for DCA-bladings and is now modified to reflect the flow situation in bladerows having low-cambered, arbitrarily designed blades including precompression blades. Finally, the steady loss increase from subsonic to supersonic inlet-flow velocities demonstrates the matched performance of the different correlations of the new model. (Author)

A95-30128 Improved blade profile loss and deviation angle models for advanced transonic compressor bladings. I—A model for subsonic flow. W. M. KOENIG and D. K. HENNECKE (Darmstadt Technical Univ., Germany) and L. FOTTNER (Munich Univ. of the German Armed Forces, Neubiberg, Germany). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, 1994, p. 9 (ASME Paper 94-GT-335). 19 Refs.

New blading concepts as used in modern transonic axial flow compressors require improved loss and deviation angle correlations. The new model presented in this paper incorporates several elements and treats separately blade-row flows having subsonic and supersonic inlet conditions. In the first part of this paper, two well-established profile loss correlations for subsonic flows are extended to quasi-two-dimensional conditions and to custom-tailored blade designs. Instead of a deviation angle correlation, a simple method based on singularities is utilized. The comparison between the new model and a recently published model demonstrates the improved accuracy in prediction of cascade performance achieved by the new model. (Author)

Italian Aerospace Literature
This month: *Transonic Flows*

A97-15172 Multipoint transonic airfoil design by means of a multiobjective genetic algorithm. A. VICINI and D. QUAGLIARELLA (Centro Italiano Ricerche Aerospaziali, Capua, Italy). *AIAA 35th Aerospace Sciences Meeting and Exhibit*, Reno, NV, 1997, p. 10 (AIAA Paper 97-0082). 12 Refs.

A numerical procedure for the aerodynamic design of transonic airfoils by means of genetic algorithms, with single point, multipoint, and multiobjective optimization capabilities, is presented. Both direct and inverse design problems can be addressed. Results of an investigation on the relative efficiency of different genetic operator combinations are reported; the test problems used to this end consist in the search for the maximum of a mathematical function, and in an aerodynamic inverse design problem. It is shown how different optimization problems are better addressed by different genetic algorithms, depending on design space size and topology, and cross correlation of variables. The multiobjective approach to design is introduced with the aid of two mathematical tests. The problem of the optimization of the drag rise characteristics of a transonic airfoil is addressed using a single point, a multipoint, and a multiobjective approach, and a comparison is established between the results obtained using the three different strategies. (Author)

A97-10749 A new Kirchhoff formulation for transonic rotor noise. P. D. FRANCESCANTONIO (Agusta S.p.A., Samarate, Italy). *Proceedings of the 22nd European Rotorcraft Forum*, Brighton, UK, 1996, Vol. 2 (A97-10676 01-05), Royal Aeronautical Society, London, 1996, pp. 83.1–83.8. 17 Refs.

A new boundary integral formulation is presented for the evaluation of the noise radiated in a uniform medium by generic sources. The method requires a knowledge of pressure, velocity, and density disturbances on a smooth closed surface surrounding the source, and assumes that the propagation is linear outside the surface itself. When applied to the prediction of transonic rotor noise, the method can be used in the same manner as the Kirchhoff approach, but the new integral equations are derived, releasing the nonpenetration condition in the Ffowcs–Williams Hawkings equation. The main advantage of the proposed formulation in respect of the Kirchhoff method is that it does not require a knowledge of the surface pressure normal derivative. Two different formulations are presented that differ in the way in which a time derivative is handled. Comparisons with experiment and with the Kirchhoff method are presented for a hovering rotor in transonic conditions at various tip Mach numbers. (Author)

A96-40725 Unsteady airloads prediction for oscillating airfoils at separated flow conditions. D. P. COIRO (Napoli Federico II Univ., Naples, Italy) and A. PAGANO (CIRA—Centro Italiano Ricerche Aerospaziali, Capua, Italy). *Proceedings of the ICAS 20th Congress*, Naples, Italy, 1996, Vol. 2 (A96-40679 11-01), AIAA, Inc., Reston, VA, 1996, pp. 1800–1809. 29 Refs.

A viscous-inviscid interaction procedure for unsteady transonic flow using full potential and integral boundary layer equations is presented. Unsteady full potential equations have been used in both conservative and nonconservative formulations. Unsteady compressible boundary layer equations are used both in direct and inverse forms, and many different formulations of these are presented. Coupling between inviscid and viscous flow is performed using a semiinverse approach. Steady coupling is performed using both conservative and nonconservative full potential formulations. Unsteady coupling is performed using only nonconservative full potential equations. The numerical results obtained for NLF and NACA0012 airfoils undergoing steady and unsteady motion are discussed and compared with experiments. (Author)

A96-40562 Application of an unsteady two-equation turbulence model to the numerical prediction of the transonic buffet of an airfoil. R. ARINA (Torino Politecnico, Turin, Italy), N. CERESOLA (Alenia S.p.A., Turin, Italy), and P. G. PIANTA (CNR, CSDF, Turin, Italy). *Proceedings of the ICAS 20th Congress*, Naples, Italy, 1996, Vol. 1 (A96-40526 11-01), AIAA, Inc., Reston, VA, 1996, pp. 290–296. 13 Refs.

A two-equation *k–R* turbulence model, with pointwise formulation, is applied to the solution of the phase averaged Navier–Stokes equations for the simulation of shock-induced flow oscillations over an 18% thick circular-arc airfoil. The governing equations are solved with an implicit time discretization, with pseudotime subiterations. The numerical computations show that the unsteady separated flow is properly predicted and better calculated than previous simulations obtained using algebraic turbulence models. The reduced frequency of oscillations is predicted to be 10% lower than the experimental one. The surface pressure distribution is in agreement with previous calculations performed with a one-equation turbulence model. (Author)

A96-39060 Recent developments on a BEM for aerodynamics and aeroacoustics of rotors. M. GENNARETTI, U. IEMMA, and L. MORINO (Roma III Univ., Rome, Italy). *Proceedings of the 21st European Rotorcraft Forum*, St. Petersburg, Russia, 1995, Vol. 1 (A96-39051 10-01), NLR, Amsterdam, Netherlands, 1995, pp. II.4.1–II.4.10. 16 Refs.

A general boundary integral formulation for the transonic aerodynamic and aeroacoustic analyses of lifting bodies in arbitrary motion is presented. Emphasis is placed on the analysis of the contribution of the wake and of the nonlinear field terms appearing in the boundary integral representation for the velocity potential. Numerical results obtained from the formulation outlined include both aerodynamics and aeroacoustics of helicopter rotors in hover and forward flight in subsonic flows. They are compared with both existing numerical results and experimental data. Transonic results are presented for steady flows around both fixed-wing and hovering rotors. (Author)

A96-30814 BEM for aerodynamics and aeroacoustics of rotors in subsonic/transonic forward flight. M. GENNARETTI, U. IEMMA, and L. MORINO (Rome III Univ., Italy). *AIAA and CEAS 2nd Aeroacoustics Conference*, State College, PA, 1996, p. 9 (AIAA Paper 96-1699). 12 Refs.

This paper deals with some recent developments of a boundary element methodology for the unified aerodynamic/aeroacoustic analysis of subsonic and transonic potential flows past helicopter rotors in hover and forward flight. In this methodology, first the potential solution is derived on the rotor-blade surfaces. Then, the boundary integral representation gives the potential everywhere in the field, and the Bernoulli theorem yields the acoustic pressure. Numerical results are presented for helicopter rotors in hover and forward flight at both subsonic and transonic rotor speeds. Comparisons with existing numerical results and experimental data are included. (Author)

A96-26152 Numerical investigation of the wake decay downstream of a transonic turbine guide vane with trailing edge coolant flow ejection. S. COLANTUONI, A. COLELLA, and G. SANTORIELLO (Alfa Romeo Avia, Italy) and C. KAPTEIN and J. AMECKE (DLR, Goettingen, Germany). *Proceedings of the Yokohama International Gas Turbine Congress*, Yokohama, Japan, 1995, Vol. 2 (A96-26107 06-37), Gas Turbine Society of Japan, Tokyo, Japan, 1995, pp. II.33–II.40. 6 Refs.

Results are described of a numerical investigation on the wake decay downstream of a transonic inlet guide vane (isentropic exit Mach number = 1.05) cooled at the trailing edge. The blade geometry, which is well representative of that of advanced technology turbine aeroengines, has been subjected to a research program in the framework of a BRITE/Euram project. The code is based on a conservative formulation of the Reynolds-averaged two-dimensional compressible Navier–Stokes equations. The solver uses a finite volume approach and a five-stage explicit Runge–Kutta algorithm for time integration. Turbulence effects are evaluated by means of the 'eddy' viscosity concept, and the Baldwin–Lomax algebraic model is employed. Numerical results show a good capability of the code to predict blade isentropic Mach number and flow traversing profiles at the cascade exit, in the cases of both no-coolant and coolant ejection. The investigation allowed one to see the effect of coolant ejection on the shock structure, thereby helping towards a better understanding of the experimental flow pattern. Moreover, the flow details of the wake-mixing process in the case of coolant ejection are compared with the experimental data. The effect of the coolant ejection on the wake evolution and decay is discussed. (Author)

A96-25587 A comparison of time maching and pressure-correction algorithms for transonic turbine blades. V. MICHELASSI (Florence Univ., Italy) and E. L. PAPANICOLAOU and G. S. THEODORIDIS (Karlsruhe Univ., Germany). *Computational fluid dynamics in aeropropulsion; Proceedings of the ASME International Mechanical Engineering Congress and Exposition*, San Francisco, CA, 1995 (A96-25576 06-02), American Society of Mechanical Engineers, New York, 1995, pp. 115–126. 25 Refs.

The paper describes the comparison of a classical time-marching algorithm with a pressure-correction method reformulated for compressible flows. The comparison is carried out for inlet guide vanes in the presence or not of shocks. The time marching code is based on the scalar approximate factorization algorithm. The pressure-correction solver uses a modified SIMPLE algorithm to enforce mass conservation for compressible fluids. The equations are discretized by means of centered finite volumes. The two solvers use different artificial damping schemes which yield very similar results for the inviscid flow over a 4% bump. The comparison for subsonic, transonic, and supersonic turbine blades yielded some differences in the shock resolution, although losses and exit flow angles predictions are very similar. (Author)

N96-13610 A boundary integral method for unified transonic aerodynamic and aeroacoustic analysis of hovering rotors. M. GENNAARETTI, U. IEMMA, and L. MORINO. *AGARD, Aerodynamics and Aeroacoustics of Rotorcraft*. p. 10 (SEE N96-13582 02-01).

The subject of this paper is the unified aerodynamic and aeroacoustic analysis of transonic hovering rotors. The aerodynamic/aeroacoustic problem is stated in terms of the velocity potential, whereas the solution is determined by applying a shock-capturing boundary integral formulation. Particular emphasis is given to the analysis of the nonlinear terms in the equation for the velocity potential, whose contribution cannot be neglected for the transonic flow case analysis. Their contribution is expressed in a conservative form. Starting from the solution for the potential, the Bernoulli theorem is used to determine both the pressure distribution on the body surface (aerodynamic solution) and the acoustic pressure in the field (aeroacoustic solution). Numerical results are presented in order to show the capability of the methodology in determining both the aerodynamic and aeroacoustic solutions for transonic rotor configurations. (Author)

A96-18252 Appraisal of numerical methods in predicting the aerodynamics of forward-swept wings. G. LOMBARDI and M. V. SALVETTI (Pisa Univ., Italy) and M. MORELLI (CSIR, Medium Speed Wind Tunnel, South Africa). *AIAA 34th Aerospace Sciences Meeting and Exhibit*, Reno, NV, 1996, p. 12 (AIAA Paper 96-0289). 22 Refs.

The capabilities of different numerical methods in evaluating the aerodynamic characteristics of a forward-swept wing in subsonic and transonic flow are analyzed. The numerical results, obtained by means of potential, Euler, and Navier–Stokes solvers, are compared with experimental data. In particular, attention is focused on the aerodynamic quantities related to pressure distributions. Moreover, an attempt is made to investigate the capability of a laminar Navier–Stokes solver to predict separation lines. This analysis is carried out by comparing the numerical velocity field to the experimental visualizations, obtained through liquid crystals. All the numerical solvers give results consistent with the physical limits of the equations that they discretized. (Author)

A96-11969 Numerical approximation of the transonic flow on a pitching airfoil. D. AMBROSI and L. VIGEVANO (Milano, Politecnico, Milan, Italy). *Aerotecnica Missili e Spazio*, Vol. 74, Nos. 1 and 2, 1995, pp. 47–54. 14 Refs.

We extend a finite volume method for the compressible Euler equations to arbitrarily moving meshes. First, we briefly discuss the physical considerations which characterize the scheme as well as its mathematical background. Then we consider the proper way to account for the mesh movement. The numerical method is preliminarly used for a steady state test case and then applied to the computation of the transonic flow around a pitching NACA0012 airfoil. For this latter case the numerical results are compared with experimental data and discussed. (Author)

A95-40101 Recent experience on the control of flow separation on flight surfaces of defence aircraft. A. FERRETTI (Alenia, Turin, Italy). *High lift and separation control; Proceedings of the Conference*, Univ. of Bath, UK, 1995 (A95-40090 11-02), Royal Aeronautical Society, London, 1995, pp. 11.1–11.13. 10 Refs. Documents available from AIAA Dispatch.

A review is presented of the typical design improvements that have to be pursued during the development of contemporary combat aircraft. Improvement of the aerodynamic characteristics of a given aircraft configuration may be sought through a delay in the occurrence of flow separation phenomena (in terms of angle of attack or freestream Mach number). This paper presents results on the improvement in usable lift achieved with the use of vortex generators and controls deflection on the wing of a conventional combat aircraft flying at transonic speed. The effects of different leading edge devices on the maximum lift achievable by a contemporary delta wing aircraft are illustrated, together with the results of an optimization study of the wing tip region, aimed at alleviating the strong interaction between vortical flow and shock wave. Comparisons with experimental results (wind tunnel and/or flight) are presented showing the efficacy of the criteria adopted in the aircraft design.

A95-38664 An integral method for transonic aerodynamics and aeroacoustics of rotors in forward flight. M. GENNARETTI, U. IEMMA, L. LUCERI, and L. MORINO (Roma III Univ., Rome, Italy). *Proceedings of the CEAS/AIAA 1st Joint Aeroacoustics Conference*, Munich, Germany, 1995, Vol. 1 (A95-38611 10-71), Deutsche Gesellschaft fuer Luft- und Raumfahrt, Bonn, Germany, 1995, pp. 501–511 (CEAS/AIAA 95-066). 39 Refs.

This work presents a general boundary integral formulation for the unified aerodynamic and aeroacoustic analyses of lifting bodies in arbitrary motion, for transonic potential flows. Once the boundary integral representation for the velocity potential is derived, contributions from wakes and nonlinear field terms are analyzed. Then, from the knowledge of the potential solution, the acoustic field is determined by applying the Bernoulli theorem. Numerical results obtained from the formulation outlined concern both aerodynamics and aeroacoustics of helicopter rotors in hover and forward flight in subsonic flows. They are compared with both existing numerical results and experimental data. Transonic results are presented only for nonlifting hovering rotors. (Author)

A95-30587 Analysis of some interference effects in a transonic wind tunnel. G. LOMBARDI (Pisa Univ., Italy) and M. MORELLI (Council of Scientific and Industrial Research, Pretoria, South Africa). *Journal of Aircraft* (ISSN 0021-8669), Vol. 32, No. 3, 1995, pp. 501–509. 17 Refs.

The effects of the walls of a test section on a model in transonic flow were investigated by using the AGARD Calibration Model B. Tests were carried out in a closed-circuit pressurized tunnel, with a confined square test section of 1.5 m width, with tapered slots giving a 5% porosity. Two models with different dimensions were used, with 0.85 and 0.056% blockage ratios. Longitudinal aerodynamic characteristics were analyzed by means of measurements performed at varying angles of attack (up to 24 deg) and Mach numbers from 0.3 to 1.2. In some flow conditions wall interference effects were probably present. However, the forces and moments dependent on the pressure distribution were likely to be related to the same factors, and therefore, the above effects tended to disappear when longitudinal stability and lift-dependent drag were analyzed as a function of lift characteristics. The drag rise Mach number evaluation seems be fully free from blockage effects. The dimensions of the tested larger model can be considered to be the largest reasonable ones for industrial applications, but, probably, not sufficiently small when high accuracy is required. (Author)

A95-30090 Aerodynamic performance of a transonic turbine guide vane with trailing edge coolant ejection. II—Numerical approach. V. MICHELASSI and F. MARTELLI (Florence Univ., Italy) and J. AMECKE (DLR, Inst. fuer Experimentelle Stroemungsmechanik, Goettingen, Germany). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, 1994, p. 12 (ASME Paper 94-GT-248). 14 Refs.

The effect of coolant ejection from the trailing edge of a turbine blade is studied by means of numerical simulation codes. Two solvers based on explicit and implicit algorithms, with and without multigridding, are used in connection with H, HOH, and HC grid types. A simple procedure to model the coolant jet at the trailing edge of the blade is presented which does not require excessive grid refinement. The features of the different approaches are discussed in terms of isentropic Mach number distribution on the blade, base pressures and wake profiles at various exit Mach numbers and coolant flow rates which match the experimental conditions. The coolant ejection effect on the shock pattern is discussed. The trailing edge ejection is found to have a marginal effect on the wake shape and on the measured and computed base pressures. (Author)

A95-28256 Transitional boundary layer analysis on turbine blade cascades by means of high-frequency-response hot-film probes. F. PITTALUGA, G. BENVENUTO, and U. CAMPORA (Genova Univ., Genoa, Italy). *Proceedings of the PICAST'1 1993—Pacific International Conference on Aerospace Science and Technology*, National Cheng Kung Univ., Tainan, Taiwan, 1993, Vol. 1 (A95-28244 07-99), National Cheng Kung Univ., Tainan, Taiwan, 1993, pp. 104–110. 14 Refs.

An experimental strategy is discussed, together with its detailed results, aimed at detecting transonic boundary layer transition behavior on the suction surfaces of turbine blade cascades. The technique relies on multiple-sensor (8 filaments) hot-film surface probes of the glue-on type, mounted flush on the blade surfaces, and suitable for detecting fluctuations in heat transfer up to a frequency of about 50 kHz. Transition from the laminar to the turbulent regime is observed by inspection of the oscilloscope traces coming from the probe signals, and, with a certain level of quantitative meaning, through the analysis of their effective rms voltage values. Interestingly enough, even the mean-time voltage levels turn out to be useful. The mean-time surface heat-transfer rates, directly related to them, correspond to a separating flow region by an evident minimum in their distribution. Finally, a rather extensive series of experimental results coming from an instrumented high-deflection blade cascade is shown and discussed, corresponding to three different inlet relative flow angles. Responses are interpreted and compared with respective surface pressure distributions and schlieren visualizations. Transition clearly appears as a spread-out phenomenon, covering more than 30% of the length of the blade surface. (Author)

A95-25937 Irrotational and rotational transonic flows using a boundary integral equation method. M. GENNARETTI, U. IEMMA, F. SALVATORE, and L. MORINO (Roma III Univ., Rome, Italy). *Proceedings of the 20th European Rotorcraft Forum*, Amsterdam, Netherlands, 1994, Vol. 1 (A95-25916 06-01), National Aerospace Lab., Amsterdam, Netherlands, 1994, pp. 23.1–23.12. 23 Refs. Documents available from AIAA Dispatch.

A boundary integral equations methodology for the analysis of irrotational and rotational transonic flows is discussed. Numerical resulting concerning two-dimensional and three-dimensional configurations are presented and compared with experimental data and existing CFD results. The capability of the full-potential formulation to predict rotational flow solutions is demonstrated. Subsonic viscous flow results show the capability of the technique in capturing viscous effects.

A95-22451 Boundary element method for unified transonic aerodynamic and aeroacoustic analyses of rotors. U. IEMMA and M. GENNARETTI (Roma I Univ., Rome, Italy) and L. MORINO (Roma III Univ., Rome, Italy). *Proceedings of the 19th European Rotorcraft Forum*, Como, Italy, 1993, Vol. 1 (A95-22426 05-01), National Aerospace Lab., Amsterdam, Netherlands, 1993, pp. C9.1–C9.11. 22 Refs. Documents available from AIAA Dispatch.

An account is given of recent developments of boundary methods capable of conducting unified analyses of aerodynamics and aeroacoustics for potential transonic flows. In CFD approaches to such problems, this technique allows a more computationally efficient analysis of flows associated with such complex configurations as that of a helicopter rotor/fuselage. A potential integral representation is derived in which the linear contributions are expressed by body and wake surface integrals, while a field terms takes nonlinear effects into account.

A95-21400 Design method for 2-D transonic rotational flows. F. LAROCCA and L. ZANNETTI (Torin, Politecnico, Turin, Italy). *AIAA 33rd Aerospace Sciences Meeting and Exhibit*, Reno, NV, 1995, p. 8 (AIAA Paper 95-0648). 10 Refs.

A numerical technique for solving design problems for a two-dimensional duct is presented. The unsteady Euler equations are integrated numerically by using a time-dependent procedure and by adopting an upwind finite volume approximation that belongs to the class of the second order ENO schemes. The method is checked with an analytical solution. The design of diffusers that perform transonic compression through a shock wave in the flow core, but are shockless at the walls, is shown to be feasible. (Author)

A94-28608 Full potential and Euler solutions for transonic unsteady flow. D. AMBROSI, L. GASPARINI, and L. VIGEVANO (Milano Politecnico, Milan, Italy). *Proceedings of the 12th Associazione Italiana di Aeronautica e Astronautica, National Congress*, Como, Italy, 1993, Vol. 1 (A94-28567 09-01), Associazione Italiana di Aeronautica e Astronautica, Rome, 1994, pp. 489–500. 13 Refs.

A comparison between full potential and Euler solutions for transonic unsteady flow is presented. The limitations of the potential model are discussed together with its advantages, and the main features of both computational methods are summarized. The comparison is carried out for a NACA0012 airfoil oscillating at a freestream Mach number ranging from 0.755 to 0.825 in order to investigate the limit of application of the potential approximation in an unsteady flow. (Author)

United Kingdom Aerospace Literature
This month: *Transonic Flows*

A97-10745 Numerical simulation of the transonic blade-vortex interaction. N.-L. NG and R. HILLIER (Imperial College of Science, Technology and Medicine, London, UK). *Proceedings of the 22nd European Rotorcraft Forum*, Brighton, UK, 1996, Vol. 2 (A97-10676 01-05), Royal Aeronautical Society, London, 1996, pp. 79.1–79.13. 24 Refs.

The transonic, parallel blade-vortex interaction is investigated numerically. Simulations of the unsteady interaction are made using a time-accurate, high resolution Euler code based on a second-order Godunov-type method. The incident vortex has an assumed structure based on Sculley's velocity profile. A local mesh refinement scheme along the lines of that developed by Berger (1984–1989) is described. The method is based on embedded meshes and allows the resolution of flow/geometric features to be increased without prohibitive expense. The behavior of the mesh interfaces for this scheme is also examined. Numerical test cases are presented which demonstrate the ability of the code to simulate steady and unsteady flows about a NACA0012 airfoil. The nature of the interaction for a freestream Mach number of 0.8 is considered in detail. The flow for this case is transonic, and three sources of noise generation are identified. These involve flow mechanisms at the leading edge, at the trailing edge, and at the shocks. The nature of the noise generation is found to be highly directional. For small vortex-blade offsets, the blade interacts with the 'near-field' of the vortex, where the associated induced velocities and flow gradients result in a highly impulsive interaction. (Author)

A96-44269 The RAE–Vickers rocket powered transonic aircraft model—1945 to 1948. J. BECKLAKE (Herstmonceux Science Center, UK). *IAF 47th International Astronautical Congress*, Beijing, China, 1996, p. 14 (IAA Paper 96-2204). 17 Refs. Documents available from AIAA Dispatch.

An overview of the REI-Vickers Rocket Propelled Transonic Model Aircraft, also known as the Transonic Model Aircraft (TMA), is presented. The TMA was to be carried aloft beneath a Mosquito aircraft to about 36,000 ft. Following release the liquid fuel motor would ignite, powering the research vehicle through the sound barrier. The TMA was stabilized by a modified version of the V1 control platform and the data recorded via a combination of optical and photographic observations from the mother plane, radar tracking using an American SCR 584 radar, and by a six-channel telemetry system. The craft's Alpha motor was based on German technology and the Walter 509-109 Messerschmitt 163 powerplant in particular. The successful flight of the TAM A3 vehicle proved that the design of the Miles M52 supersonic aircraft was satisfactory and could have safely flown through the sound barrier.

A96-42741 Calibration of a four-hole pyramid probe and area traverse measurements in a short-duration transonic turbine cascade tunnel. A. J. MAIN, C. R. B. DAY, G. D. LOCK, and M. L. G. OLDFIELD (Oxford Univ., UK). *Experiments in Fluids* (ISSN 0723-4864), Vol. 21, No. 4, 1996, pp. 302–311. 13 Refs.

A four-hole pyramid probe has been calibrated for use in a short-duration transonic turbine cascade tunnel. The probe is used to create area traverse maps of total and static pressure, and pitch and yaw angles of the flow downstream of a transonic annular cascade. A four-hole pyramid probe was used which has a 2.5-mm section head and has the side faces inclined at 60 deg to the flow to improve transonic performance. The probe was calibrated in an ejector-driven perforated-wall transonic tunnel over the Mach number range 0.5–1.2, with pitch angles from −20 to +20 deg and yaw angles from −23 to +23 deg. A computer-driven automatic traversing mechanism and data collection system was used to acquire a large probe calibration matrix (approximately 10,000 readings) of nondimensional pitch, yaw, Mach number, and total pressure calibration coefficients. A novel method was used to transform the probe calibration matrix of the raw coefficients into a probe application matrix of the physical flow variables. The probe application matrix is then used as a fast look-up table to process probe results. With negligible loss of accuracy, this method is faster by two orders of magnitude than the alternative of global interpolation on the raw probe calibration matrix. The blowdown tunnel (with a mean nozzle guide vane blade ring diameter of 1.1 m) creates engine-representative Reynolds numbers, transonic Mach numbers, and high levels (approximately 13%) of inlet turbulence intensity. Contours of experimental measurements at three different engine-relevant conditions and two axial positions have been obtained. An analysis of the data is presented. (Author)

A96-39460 Measurements of transonic shock structures using shearography. M. BURNETT and P. J. BRYANSTON-CROSS (Warwick Univ., Coventry, UK). *Laser interferometry VIII—Applications; Proceedings of the Meeting*, Denver, CO, 1996 (A96-34949 10-35), Society of Photo-Optical Instrumentation Engineers, Bellingham, WA, (SPIE Proceedings. Vol. 2861), 1996, pp. 124–135. 9 Refs.

In this paper a shearography approach is used to encode flow field density measurements. The fringes are projected through the flow field and automatically analyzed using the FFT method. The subsequent 'wrapped' phase map is 'unwrapped' using the largely noise immune Minimum Spanning Tree (MST) technique. This allows the flow field to be solved despite the presence of discontinuities such as shocks. This technique was applied to the results made on a two-dimensional transonic wind tunnel where whole field measurements were made. The subsequent fringe patterns were each solved by the automatic fringe analysis technique. The shock structures were observed to be the same as those revealed by earlier flow visualizations. The density measurements correspond well to previous holographic measurements. (Author)

A96-38461 Numerical simulation of the transonic blade-vortex interaction. N.-L. NG and R. HILLIER (Imperial College of Science, Technology and Medicine, London, UK). *Unsteady aerodynamics; Proceedings of the Conference*, London, UK, 1996 (A96-38453 10-02), Royal Aeronautical Society, London, 1996, pp. 8.1–8.11. 20 Refs.

The transonic, parallel blade-vortex interaction (BVI) has been investigated numerically. Simulations of the unsteady interaction are made using a time accurate, high resolution Euler code based on a second-order Godunov-type method. The incident vortex has an assumed structure based on Sculley's velocity profile. Numerical test cases are presented which demonstrate the ability of the code to simulate steady and unsteady flows about a NACA0012 aerofoil. The nature of the interaction for a free stream Mach number of 0.8 is considered in detail. The flow for this case is transonic, and three sources of noise generation have been identified. These involve flow mechanisms at the leading edge, at the trailing edge, and at the shocks. The nature of the noise generation is found to be highly directional. For small vortex-blade offsets, the blade interacts with the near-field of the vortex, where the associated induced velocities and flow gradients result in a highly impulsive interaction. For close interactions, the dependence of the interaction on the assumed vortex structure is evident. (Author)

A96-36495 Aerodynamic instability of pitch oscillating transonic cascade and the effects of turbulence models. Y. NOOGUCHI (Salford Univ., UK) and T. SHIRATORI (Tokyo Metropolitan Inst. of Technology, Japan). *AIAA 27th Fluid Dynamics Conference*, New Orleans, LA, 1996, p. 11 (AIAA Paper 96-2036). 9 Refs.

A numerical study has been carried out on pitch oscillating two-dimensional transonic cascade flows with three pitching modes. Two relatively simple but popular turbulence models, Baldwin–Lomax model (BLM) and Johnson–King model (JKM), are used. The purpose of this numerical experimentation is to establish the effects of various features on aerodynamic instability in pitch motion. These include shock movements, flow separations and turbulence models. The results show the shock positions predicted by JKM is slightly upstream of BLM. This is the same trend to the earlier study with changes in the pitching frequency. The upstream pitching axis position contributes to the instability in pitch. When the interblade phase angle is 0 deg instead of 180 deg, flow behavior changes. The difference in overall aerodynamic instability is caused by the different shock locations predicted by BLM and JKM. The variation in the predicted separation and wake between the two models contributes to the difference in the aerodynamic instability. The shock location is more significant factor for the difference between the models for aerodynamic instability. (Author)

A96-28261 Application of the FAME method to the simulation of store separation from a combat aircraft at transonic speed. T. A. BLAYLOCK (Defence Research Agency, Farnborough, UK). *Numerical grid generation in computational field simulations; Proceedings of the 5th International Conference*, Mississippi State Univ., Mississippi State, 1996, Vol. 1 (A96-28186 07-64), Mississippi State Univ., Mississippi State, MS, 1996, pp. 805–814. 8 Refs.

A method known as FAME is described for generating surface and field meshes around complex configurations. The extension of the method to deal with configurations with components in relative motion is described. Euler flow results are presented for a store released from a Tornado under-fuselage pylon. (Author)

A96-18068 Rapid design space approximation for two-dimensional transonic aerofoil design. M. E. TOPLISS, C. A. TOOMER, and D. P. HILLS (British Aerospace, Ltd., Sowerby Research Centre, Bristol, UK). *AIAA 34th Aerospace Sciences Meeting and Exhibit*, Reno, NV, 1996, p. 10 (AIAA Paper 96-0095). 13 Refs.

This paper presents a strategy for exploring the design space for use in aerodynamic optimization. This is achieved by perturbing the values of the geometric shape variables, angle of attack, and freestream Mach number for a two-dimensional transonic aerofoil. The approach uses the 'quasi-analytical' method, which arises from the direct differentiation of the nonlinear governing equations. Using only one traditional CFD analysis, the design space can be explored cheaply with little loss in accuracy for small perturbations. This approach is extended further by developing a new technique by applying the 'quasi-analytical' method to provide information at selected positions in the design space and then to the approximation of the function for the aerodynamic quantities by a series of polynomials. This provides a significant cost reduction and simplifies the integration within modern engineering methods. Results are presented for a typical transonic airfoil, giving comparisons with individual CFD analyses. (Author)

A96-16535 The application of PIV (Particle Image Velocimetry) to long range transonic flow measurements. P. J. BRYANSTON-CROSS (Warwick Univ., Coventry, UK). *Optical techniques in fluid, thermal, and combustion flow; Proceedings of the Conference*, San Diego, CA, 1995 (A96-16526 03-35), Society of Photo-Optical Instrumentation Engineers, Bellingham, WA (SPIE Proceedings. Vol. 2546), 1995, pp. 98–111. 9 Refs.

A number of PIV measurements have been made at transonic speeds. The initial objective of the work has been to explore if such measurements could be made remotely and processed in an accurate and automatic fashion. Subsequently PIV measurements have been made remotely at optical standoff distance of up to 1 m. PIV results are presented, made on a 1/12th scale model of an Airbus wing, where a separation induced shock has been measured in three dimensions. The paper also presents results achieved using PIV in mapping the flow within a full size annular turbine cascade with a velocity measurement accuracy of 1%. Measurements are presented showing the wake measured downstream of a annular turbine stator row. The measurements clearly show the presence of a vortex street which has a width of approximately 3 mm. (Author)

A96-16121 The application of PIV (Particle Image Velocimetry) to transonic flow measurements. P. J. BRYANSTON-CROSS (Warwick Univ., Coventry, UK). *ICIASF '95—16th International Congress on Instrumentation in Aerospace Simulation Facilities*, Dayton, OH, 1995, Record (A96-16069 03-35), Inst. of Electrical and Electronics Engineers, New York, 1995, pp. 53.1–53.11. 9 Refs.

A number of PIV measurements have been made at transonic speeds. The initial objective of the work has been to explore if such measurements could be made remotely and processed in an accurate and automatic fashion. Subsequently PIV measurements have been made remotely at optical standoff distance of up to 1 m PIV results are presented, made at ARA Bedford on a 1/12th scale model of an Airbus wing, where a separation-induced shock has been measured in three dimensions. The paper also presents results achieved using PIV at DRA Pyestock in mapping the flow within a full size annular turbine cascade with a velocity measurement accuracy of 1%. Measurements are presented showing the wake measured downstream of a annular turbine stator row. The measurements clearly show the presence of a vortex street which has a width of approximately 3 mm. (Author)

A96-13334 Comparison of computational and experimental results in axi-symmetric transonic turbulent flows. Y. NOGUCHI (Salford Univ., UK) and T. SHIRATORI (Tokyo Metropolitan Inst. of Technology, Hino, Japan). *Computational methods and experimental measurements VII; Proceedings of the 7th International Conference (CMEM '95)*, Capri, Italy, 1995 (A96-13323 02-31), Computational Mechanics Publications, Southampton, UK and Billerica, MA, 1995, pp. 337–344. 9 Refs. Documents available from AIAA Dispatch.

An extension of previous work to test two popular turbulence models for applied compressible flows is carried out. This work covers the transonic region, and two turbulence models are compared with measured data. Significant discrepancies in the shock patterns are observed between the measured and calculated results. This is explained by the differences in the transition position, boundary layer development wind tunnel wall effects, and Reynolds number. The calculated shock locations agree with experiments. It is shown that the shocks are not stable and the shock location has a wide range of variations.

A95-44635 Computed effects of heat transfer on the transonic flow over an aerofoil. S. RAGHUNATHAN and D. MITCHELL (Queen's Univ., Belfast, UK). *AIAA Journal* (ISSN 0001-1452), Vol. 33, No. 11, 1995, pp. 2120–2127. 22 Refs.

The effects of the model surface-to-freestream adiabatic temperature ratio (Tw/Tad) on subsonic flows at zero pressure gradient and transonic flow over a NACA0012 airfoil are evaluated using a computational fluid dynamic approach. The analysis, based on the thin-layer Navier–Stokes equations with a Baldwin–Lomax turbulence model, indicated that surface heat transfer has significant effects on both subsonic and transonic flows, confirming some of the experimental data available. The results have implications in wind-tunnel testing at nonadiabatic surface conditions. (Author)

A95-42685 Conjectures on new transonic aeroelastic phenomena. J. ANDERSON (Glasgow Univ., UK). *Proceedings of the International Forum on Aeroelasticity and Structural Dynamics*, Manchester Business School, UK, 1995, Vol. 2 (A95-42613 11-39), Royal Aeronautical Society, London, 1995, pp. 84.1–84.11. 15 Refs.

It is shown that the apparently disparate transonic aeroelastic behavior observed in computational studies of a symmetric typical section aerofoil in two-dimensional flow, subject to a structural pretwist, can be interpreted in terms of the 'global' response characteristics associated with asymmetric perturbation of the pitchfork/Hopf degeneracy. Evidence based on genericity and continuity arguments suggests the existence of new transonic aeroelastic phenomena not previously encountered in computational studies. (Author)

A95-42643 The effects of turbulence models on unsteady transonic cascade flow computations. Y. NOGUCHI (Salford Univ., UK) and T. SHIRATORI (Tokyo Metropolitan Inst. of Technology, Japan). *Proceedings of the International Forum on Aeroelasticity and Structural Dynamics*, Manchester Business School, UK, 1995, Vol. 1 (A95-42613 11-39), Royal Aeronautical Society, London, 1995, pp. 35.1–35.12. 10 Refs.

A numerical study is being carried out on pitch oscillating two-dimensional turbulent transonic cascade flows. Two relatively simple but popular turbulence models, namely the Baldwin–Lomax (BLM) and Johnson–King (JKM) models, are used for the computations. The purpose of this numerical experimentation is to establish the magnitude of the effects of turbulence models. The results show the JKM predicts the shock locations slightly upstream of the BLM as it did in steady flow cases. However, the resultant differences in the shock driven parameters, such as the pitching moment, can be significant. It is also interesting to note that the JKM has a time delay when compared with the BLM. This time delay is thought to be caused by the extra equation in the JKM. The comparisons showed that the effects of the turbulence models could be significant in terms of aerodynamic pitching instability, and a careful validation is essential for an accurate prediction. (Author)

A95-42630 Aeroelastic studies using transonic flow CFD modelling. K. J. BADCOCK, G. SIM, and B. E. RICHARDS (Glasgow Univ., UK). *Proceedings of the International Forum on Aeroelasticity and Structural Dynamics*, Manchester Business School, UK, 1995, Vol. 1 (A95-42613 11-39), Royal Aeronautical Society, London, 1995, pp. 18.1–18.12. 13 Refs.

An unfactored solution method developed for the Navier–Stokes equations is applied to study the nonlinear aeroelastic response of an aerofoil. The flow solver efficiency has been found to compare favorably with the standard ADI method. A loosely coupled approach is adopted with the structural equations and the flow equations being solved in sequence. An Euler verson of the code is used to explore the limit cycle and flutter-divergence interaction behavior on an NACA64A006 aerofoil constrained in pitch-heave motion. Similar results to that found in an earlier study were obtained for symmetrical flow cases as well as for pre-twist. This study provides a firm foundation for more advanced work ongoing including viscous effects and the extension of the work to three dimensions in order to explore the problem of aeroelasticity of wings. (Author)

A95-40103 The high-speed characteristics of leading-edge slats—Observations from transonic wind tunnel tests. N. J. TAYLOR (Defence Research Agency, Farnborough, UK) and R. JORDAN (Aircraft Research Association, Bedford, UK). *High lift and separation control; Proceedings of the Conference*, Univ. of Bath, UK, 1995 (A95-40090 11-02), Royal Aeronautical Society, London, 1995, pp. 13.1–13.15. 8 Refs.

By referring to wind tunnel test data gathered during an extensive UK research program, this paper describes how leading-edge slats influence the flow development on swept wing panels at moderate and high subsonic freestream Mach numbers and provides evidence of the attendant improvements in high-speed high-lift performance. The roles played by several geometric features, particularly the slat angle, are also discussed. By examining the ways in which the physical principles governing their aerodynamic effectiveness at high Mach numbers differ from those appropriate to low-speed flight, it is shown that i) the optimum slat angle and extension at high Mach number are appreciably smaller than those required to produce maximum benefit at low-speed and ii) while leading-edge slats become progressively less effective in delaying the onset of severe rear separation as the Mach number is raised, they play an increasingly important role in controlling the forward growth of shock-induced separation on the main element whilst simultaneously generating high lift and thrust. (Author)

A95-38534 Computational analysis of buffet alleviation in viscous transonic flow over a porous airfoil. M. A. GILLAN (Short Brothers, Belfast, UK). *AIAA Journal* (ISSN 0001-1452), Vol. 33, No. 4, 1995, pp. 769–772. Previously cited in issue 19, Accession A93-47215. 16 Refs. Documents available from AIAA Dispatch.

A computational investigation into the effects of passive control on periodic transonic viscous flow over an 18%-thick circular-arc airfoil is presented. To perform this analysis, an explicit finite volume cell-vertex-centered Navier–Stokes solver was developed, in conjunction with an orthogonal boundary conforming hyperbolic C-grid generator. The ability of passive control to suppress Tijdeman-type-B SIO, which invariably leads to buffet, is demonstrated using a Navier–Stokes solver.

A95-35236 Behavior of turbulence models in steady and unsteady transonic cascade flows. Y. NOGUCHI (Salford Univ., UK) and T. SHIRATORI (Tokyo Metropolitan Inst. of Technology, Japan). *AIAA 26th Fluid Dynamics Conference*, San Diego, CA, 1995, p. 11 (AIAA Paper 95-2286). 11 Refs.

The behavior and effects of turbulence models in unsteady two-dimensional transonic cascade flows are studied. This qualitative numerical experimentation is performed in order to understand the flow mechanism and the effects of turbulence models in such a complex flow field. Large differences between the turbulence models used are observed in the parameters which are sensitive to the shock location, such as pitching moment. The Johnson–King model (JKM) is known to predict shock locations slightly upstream of the Baldwin–Lomax model (BLM) in some adverse pressure gradient conditions in steady flows. It is confirmed that this tendency is unchanged in the unsteady flows tested. The unsteady cases have a time delay when compared with the steady cases. This time delay is slightly smaller with the JKM than the BLM. This is thought to be caused by the addition of the streamwise convection and diffusion effects in the JKM. The maximum Reynolds shear stresses around the trailing edge vary significantly between the turbulence models. However the wake region seems to be interacting with the separated region, and careful study into handling of the wake seems to be necessary. (Author)

N95-20799 Testing in the ARA Transonic Wind Tunnel. D. JORDAN. ARA-MEMO-395.

This note is an overview of the ARA 2.74 × 2.44 m Transonic Wind Tunnel facility which covers the full Mach number range $M = 0$–1.4. A description of the main features of the tunnel circuit is given together with operational details. The test capabilities encompass a wide variety of model configurations which can be mounted on interchangeable full or half-span model carts. Load measurements are made using strain-gauge balances in conjunction with an appropriate sting or support strut. Several standard model support arrangements are provided but specialized rigs are frequently developed for particular requirements. High pressure air is available for tests involving propulsion simulation. An advanced data acquisition and data reduction system produces on-line corrected results which are presented on inter-active color graphic displays. Emphasis is placed on the consistently good flow quality in the working section, the high accuracy and repeatability of the aerodynamic measurements, the routine corrections incorporated in the data reduction, and the high productivity of testing in this tunnel. (Author)

A95-31190 Computation of multi-element aerofoil high-lift performance at low-speed and transonic flow conditions. L. J. JOHNSTON (Univ. of Manchester Inst. of Science and Technology, UK). *Proceedings of the 34th Israel Annual Conference on Aerospace Sciences*, Tel Aviv, Israel, 1994 (A95-31151 08-99), Ayalon Offset, Ltd., Haifa, Israel, 1994, pp. 412–423. 14 Refs.

A computational method to predict the aerodynamic performance of mechanical high-lift systems is described. The Reynolds-averaged Navier–Stokes equations applicable to compressible, two-dimensional mean flow are solved using a cell-centered, finite-volume spatial discretization and an explicit multistage scheme to time march to steady-state solutions. The governing mean-flow equations are solved in conjunction with a two-equation, high-Reynolds number k–ϵ turbulence model. Unstructured computational grids are used to deal with the geometric complexity associated with practical multielement aerofoil configurations. Results are presented for the low-speed, high-lift NLR 7301 aerofoil/trailing-edge flap configuration and the SKF 1.1 transonic aerofoil/maneuver flap configuration. The agreement between predictions and experiment is reasonable for the cases considered. However, it is concluded that a more sophisticated turbulence model is required in order to improve the quantitative accuracy at maximum lift conditions. (Author)

A95-31188 Transonic flow drag prediction using Navier–Stokes solvers and transport equation turbulence models. L. J. JOHNSTON (Univ. of Manchester Inst. of Science and Technology, UK). *Proceedings of the 34th Israel Annual Conference on Aerospace Sciences*, Tel Aviv, Israel, 1994 (A95-31151 08-99), Ayalon Offset, Ltd., Haifa, Israel, 1994, pp. 387–398. 7 Refs.

A description is given of a numerical method to predict the viscous transonic flow development around airfoil sections. The method solves the Reynolds-averaged Navier–Stokes equations in conjunction with a one-equation turbulence model. A modification of this model is introduced to enable the method to deal with blunt-base airfoils. The governing mean-flow and turbulence transport equations are discretized in space using a cell-centered finite-volume scheme, with a time-marching procedure being adopted to obtain steady-state solutions. Results are presented for several transonic airfoils. Computed drag levels are in good agreement with experiment for subcritical flow conditions. However, wave drag is overpredicted at the higher Mach numbers or incidence angles associated with supercritical flow conditions; this is attributed to an underprediction of the shock wave boundary layer interaction when using an eddy-viscosity based turbulence model. (Author)

N95-19546 An investigation of drag repeatability in half model testing in the ARA Transonic Wind Tunnel. I. F. BURNS, J. E. GREEN, and D. R. STANNILAND. ARA-MEMO-392.

In 1990 the variability of drag measurements on a half model in the ARA Transonic Tunnel increased from the usual one drag count to between two and three drag counts. An investigation showed the primary source of the variability to be a damaged balance, with a possible air leak into the tunnel plenum chamber as an additional contributor. When these sources had been eliminated, drag repeatability was restored to approximately one count. It was found however that this was mainly a systematic, diurnal variation, caused by cyclic variation of the heat exchange between the tunnel airflow and the plenum chamber structure. A procedure was established, using measured sidewall pressures, to correct for this effect and thereby reduce drag variability within a test series to rather less than one count. As a further improvement to the tunnel for half model testing, the test section has been modified to extend the region of uniform flow and improve the aerodynamics of the diffuser. Initial results of the commissioning trials with this modification are reported. (Author)

N95-19260 Transonic and supersonic flowfield measurements about axisymmetric afterbodies for validation of advanced CFD codes. M. BURT and P. MILLER (Miller and Wilson Aerodynamics Research, Bath, England) and J. AGRELL (Aeronautical Research Inst. of Sweden, Bromma). *AGARD, Wall Interference, Support Interference and Flow Field Measurements*, p. 28 (SEE N95-19251 05-34).

Two axisymmetric afterbody experimental programs, aimed at providing necessary and sufficient data for CFD code validation, were conducted in the FFA S5 suckdown wind-tunnel. Flow conditions covered the range of transonic to supersonic. Mean and fluctuating flowfield velocities in a single longitudinal plane were measured using LDA along many traverses, both over the afterbody and in the jet and mixing regions. Flow separated on the boattail of the AGARD 10 and 15 deg geometries at all conditions tested. Separation also occurred on a conical afterbody at supersonic Mach number. Comprehensive sets of boundary condition data were also recorded, through a wide variety of techniques. Extensive error analyses have been undertaken to evaluate the accuracy of all data. Transonic Navier–Stokes computations on the configurations were performed and showed the benefit of having static pressure information along the slotted tunnel roof. An algebraic stress model of turbulence returned predictions of afterbody surface pressures superior to two more simple models, in both attached and separated flow. (Author)

A95-30293 A three-dimensional moving mesh method for the calculation of unsteady transonic flows. A. L. GAITONDE and S. P. FIDDES (Bristol Univ., UK). *Aeronautical Journal* (ISSN 0001-9240), Vol. 99, No. 984, 1995, pp. 150–160. 21 Refs.

A three-dimensional moving mesh method for solving the Euler equations describing the compressible flow about a wing undergoing arbitrary motions and deformations is described. A finite-volume formulation is chosen where the volumes distort as the wing moves or deforms. By using transfinite interpolation, a technique for generating the required sequence of grids has been developed. Furthermore, as the speeds of the grid at the vertices of the finite volumes are required by the flow solver, transfinite interpolation is also used to obtain these by interpolation of the boundary speeds. A two-dimensional version of the method has also been developed and results for both two- and three-dimensional transonic flows are presented and compared with experimental data where available. (Author)

A95-30079 Unsteady 3D flow in a single-stage transonic fan. II—Unsteady stator exit flow field. M. A. CHERRETT, J. D. BRYCE, and R. B. GINDER (Defence Research Agency, Farnborough, UK). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, 1994, p. 9 (ASME Paper 94-GT-224). 15 Refs.

Detailed unsteady aerodynamic measurements have been taken in a single-stage transonic fan with a very high stage-hub loading. Two-dimensional dynamic yawmeter probes, capable of measuring mean levels and fluctuations in stagnation pressure, static pressure and yaw angle have been traversed at rotor exit and downstream of the stator, along with several types of pneumatic three-dimensional probe. Part I described measurements taken at rotor exit. This paper, Part II, describes measurements taken at stator exit when the fan was operating at near peak efficiency, on the design speed characteristic. The measurements indicate the effects of rotorstator interaction on the development of the viscous endwall-corner flows at the hub and casing. In addition, they illustrate that significant changes in stagnation pressure level occur within much of the stator exit flow field during the rotor passing cycle. (Author)

A95-30078 Unsteady 3D flow in a single-stage transonic fan. I—Unsteady rotor exit flow field. M. A. CHERRETT, J. D. BRYCE, and R. B. GINDER (Defence Research Agency, Farnborough, UK). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, 1994, p. 9 (ASME Paper 94-GT-223). 15 Refs.

Detailed unsteady aerodynamic measurements have been taken in a single-stage transonic fan with a very high stator-hub loading. Two-dimensional dynamic yawmeter probes, capable of measuring mean and fluctuating levels of stagnation pressure, static pressure and yaw angle have been traversed at rotor exit, and downstream of the stator along with several types of pneumatic three-dimensional probe. This first part of our investigation describes the dynamic yawmeters and their performance, and presents ensemble-averaged stagnation pressure and random stagnation pressure unsteadiness measurements taken at rotor exit. These are used to illustrate the salient features of the rotor flow field, and the effects of compressor aerodynamic loading. (Author)

A95-30054 Heat transfer measurements in an annular cascade of transonic gas turbine blades using the transient liquid crystal technique. R. F. MARTINEZ-BOTAS, G. D. LOCK, and T. V. JONES (Oxford Univ., UK). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, 1994, p. 8 (ASME Paper 94-GT-172). 16 Refs.

Heat transfer measurements have been made in the Oxford University Cold Heat Transfer Tunnel employing the transient liquid crystal technique. Complete contours of the heat transfer coefficient have been obtained on the aerofoil surfaces of a large annular cascade of high pressure nozzle guide vanes (mean blade diameter of 1.11 m and axial chord of 0.0664 m). The measurements are made at engine representative Mach and Reynolds numbers (exit Mach number 0.96 and Reynolds number 2.0×10 exp 6). A novel mechanism is used to isolate five preheated blades in the annulus before an unheated flow of air passes over the vanes, creating a step change in heat transfer. The surfaces of interest are coated with narrow-band thermochromic liquid crystals, and the color crystal change is recorded during the run with a miniature CCD video camera. The heat transfer coefficient is obtained by solving the one-dimensional heat transfer equation for all the points of interest. This paper describes the experimental technique and presents results of heat transfer and flow visualization. (Author)

A95-30050 A numerical study of the interaction of a transonic compressor rotor overtip leakage vortex with the following stator blade row. W. N. DAWES (Cambridge Univ., UK). *ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, 1994, p. 14 (ASME Paper 94-GT-156). 32 Refs.

This paper presents a numerical simulation of the unsteady flow in a transonic compressor stage. The simulations were performed using an unstructured mesh, three-dimensional Navier–Stokes solver with a sliding interface between the mesh blocks containing the rotor and stator blades to permit a time-resolved calculation. The focus in the simulation is the endwall flow, its contrast with the mid-span flow and, in particular, the interaction of the rotor overtip leakage flow with the downstream stator. The endwall region of both the rotors and the stators was predicted to be proportionally responsible for much more loss production than the mid-span sections of the blading. Unsteady effects were predicted to be responsible for rather more of the total loss production in the rotors than in the stators but the unsteady loss production in the rotors did not seem to be associated with the endwall flow but rather occurred over the bulk of the blade span caused by unsteady shock motion. By contrast, the rotor overtip leakage flow was shown to cause a considerable degradation of the stator performance near the casing. (Author)