AIAA JOURNAL
Vol. 32, No. 11, November 1994

Technical Notes

TECHNICAL NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Two-Equation Turbulence Model for Unsteady Separated Flows Around Airfoils

G. Jin* and M. Braza†
Institut de Mécanique des Fluides de Toulouse, Toulouse 31400, France

Introduction

THE prediction of turbulent unsteady separated flows around airfoils is currently a priority in the domain of aeronautics. There are numerous studies in this domain, but the accurate prediction of the flow structure and aerodynamic parameters, especially near stall conditions, remain open questions, and considerable efforts have to be made to suggest an efficient way to resolve this problem. The majority of the studies devoted to the problem use different classes of turbulence models employing the steady Reynolds-averaged equations. A widely used turbulence model in the context of the eddy-viscosity concept is the two-equation model (Launder and Spalding[1]) and its various versions (Jones and Launder[2] and Chien[3] among others). However, this kind of model was originally conceived for free shear flows and then adapted for boundary-layer flows. It is not evident whether this model may be directly applicable in the elliptic flows around airfoils, as often discrepencies in the flow parameters appear when comparing with the physical experiments (see numerous papers reported in Ref. 4). One of the well-known reasons is that the various applications of the k-ε model provide generally too high a level of the eddy viscosity and of the turbulent kinetic energy production.

In this study, we simulate the turbulent incompressible flow around an airfoil NACA 0012 at a Reynolds number range of 10^6, by an unsteady approach, using the phase-averaging decomposition (Hussain and Reynolds[5]), in the form suggested by the experimental work of Cantwell and Coles.[6] This decomposition provides the phase-averaged Navier-Stokes equations, as reported in the works of Ha Minh et al.,[7] Chassaing,[8] Braza and Noguès,[9] and Franke and Rodi,[10] among others. Furthermore, we introduce the unsteady vorticity into the production term of turbulent energy to improve the behavior of the two-equation model with respect to the physics of the flow.

Turbulence Model

The phase-averaged continuity and momentum equations for an incompressible fluid in a fully elliptic form are

$$\frac{\partial\langle U_j\rangle}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial\langle U_i\rangle}{\partial t} + \langle U_j\rangle\frac{\partial\langle U_i\rangle}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(\nu+\nu_t)\frac{\partial\langle U_i\rangle}{\partial x_j}\right] - \frac{\partial}{\partial x_i}(\langle P\rangle + 2/3\langle k\rangle) + \frac{\partial}{\partial x_j}\left(\nu_t\frac{\partial\langle U_j\rangle}{\partial x_i}\right) \tag{2}$$

The turbulence energy k and its dissipation rate ε are solved by the following equations:

$$\frac{\partial\langle k\rangle}{\partial t} + \langle U_j\rangle\frac{\partial\langle k\rangle}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_k}\right)\frac{\partial\langle k\rangle}{\partial x_j}\right] + P_k - \langle\varepsilon\rangle + W_k \tag{3}$$

$$\frac{\partial\tilde{\varepsilon}}{\partial t} + \langle U_j\rangle\frac{\partial\tilde{\varepsilon}}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial\tilde{\varepsilon}}{\partial x_j}\right] + P_kC_1f_1\frac{\tilde{\varepsilon}}{\langle k\rangle} - C_2f_2\frac{\tilde{\varepsilon}^2}{\langle k\rangle} + W_\varepsilon \tag{4}$$

$$\langle\varepsilon\rangle = \tilde{\varepsilon} - D \tag{5}$$

$$\nu_t = C_\mu f_\mu\frac{\langle k\rangle^2}{\tilde{\varepsilon}} \tag{6}$$

where ν_t is the turbulent viscosity and f_μ, f_1, and f_2 are the functions of a low-Reynolds-number model (Patel et al.[11]). The terms D, W_k, and W_ε represent corrective terms according various near-wall treatments.

In the standard eddy-viscosity model, the production of the turbulent kinetic energy P_k is related to the mean strain rate. However, in the flow around an airfoil, it can be observed that the strain is strong in the vicinity of the leading edge, where the flow is almost laminar. According to the standard eddy-viscosity model, an excessive turbulent kinetic energy, and in turn too high a turbulent viscosity, will be produced in this region. Consequently, the separation occurs too late, and it is even completely inhibited. This enhances a steady-state solution of flow. On the other hand, it is well known that in a turbulent flow the vorticity is a key quantity on the motion of different eddies classes. A rotational mass of fluid will give energy to the external environment, and a part of this energy will sustain the turbulent motions. In Ref. 9 it is indicated that introducing a vorticity scale in the eddy viscosity leads to an improved scaling of this quantity in wake flows with coherent structures. Furthermore, the vorticity quantity is very weak in the region near the leading edge, except for a very thin layer close to the wall. In the other regions of the flow, characterized by a significant turbulence level, the vorticity and strain are of the same order of magnitude. These features are also confirmed by our numerical studies.[12] Therefore we consider introducing the local phase-averaged vorticity in the generation term of the turbulent kinetic energy P_k to improve the problem:

$$P_k = 2\nu_tR_{ij}R_{ij} \tag{7}$$

where

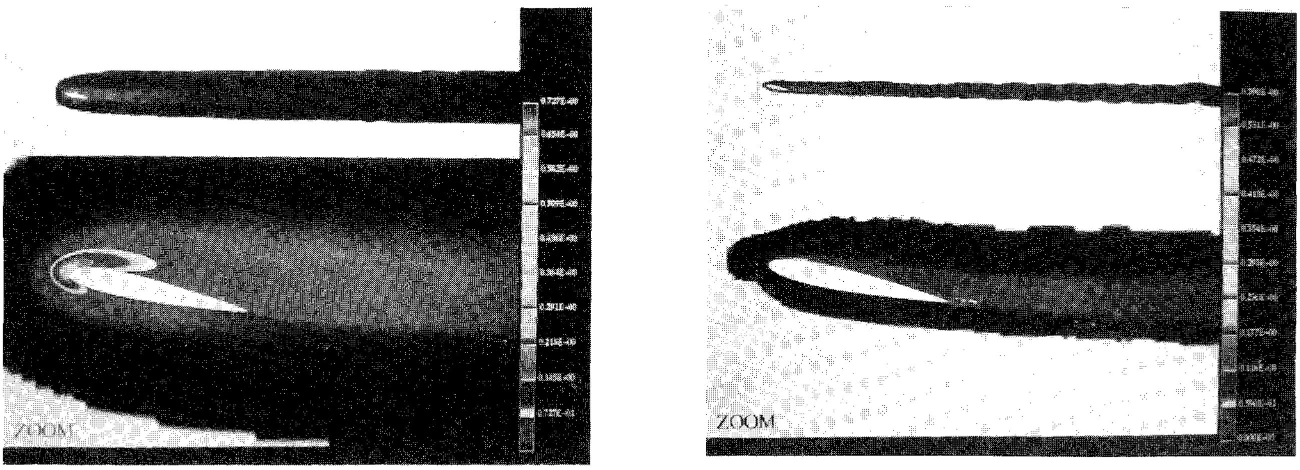
$$R_{ij} = 1/2\left(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i}\right) \tag{8}$$

In this study the preceding equations are used (model II), where the low-Reynolds-number treatment is based on Chien's[3] model.

Received March 7, 1994; revision received May 20, 1994; accepted for publication May 20, 1994.

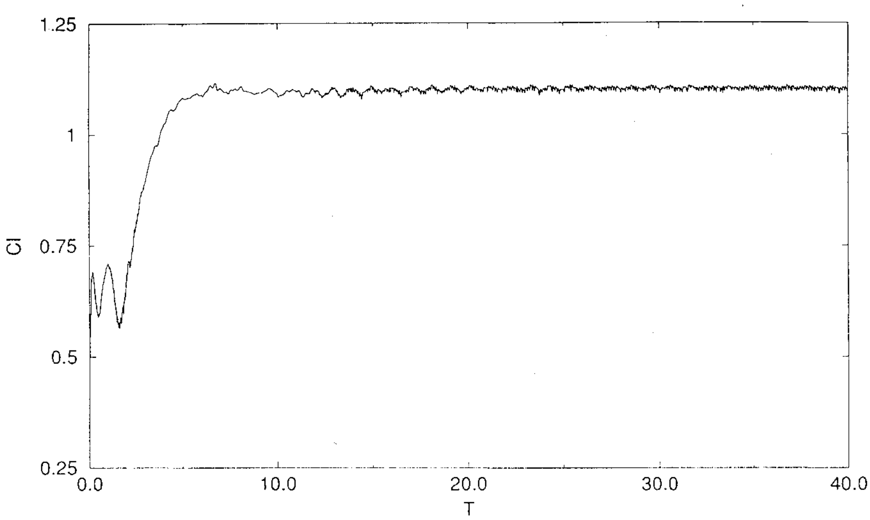
*Research Fellow, Institut National Polytechnique, Avenue du Professeur Camille Soula.

†Chargé de Recherche, C.N.R.S., Avenue de Professeur Camille Soula.

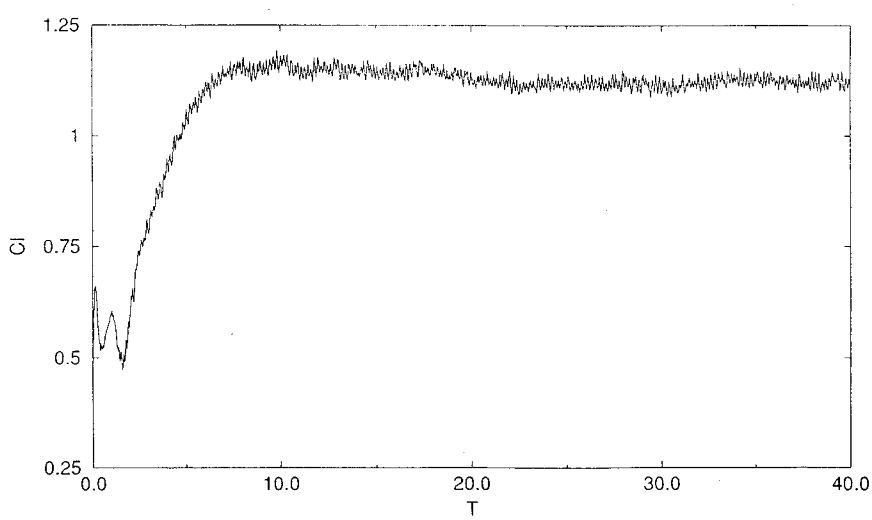


a) **Model I** b) **Model II**

Fig. 1 Isocontours of turbulent kinetic energy.



a) **Model I**



b) **Model II**

Fig. 2 Time-dependent evolution of the lift coefficient.

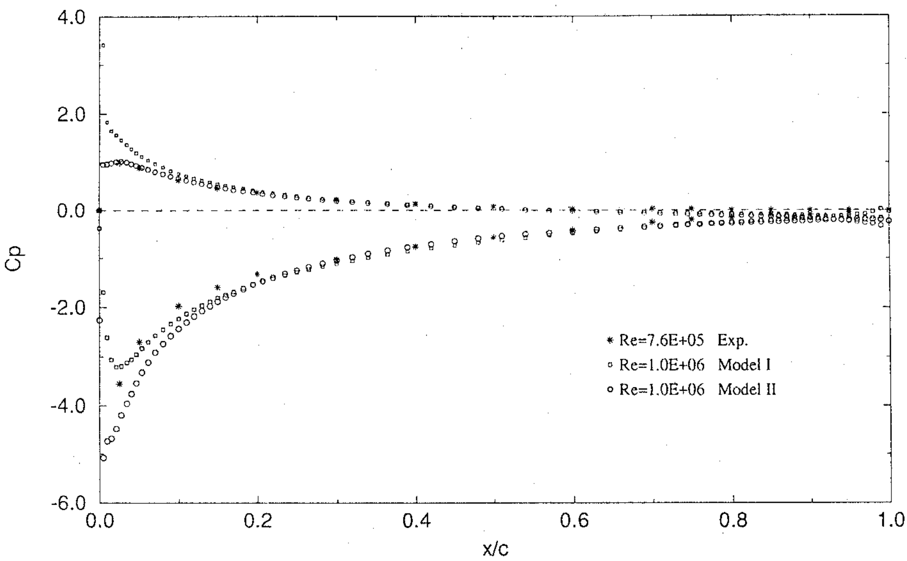


Fig. 3a Mean pressure coefficient along the airfoil.

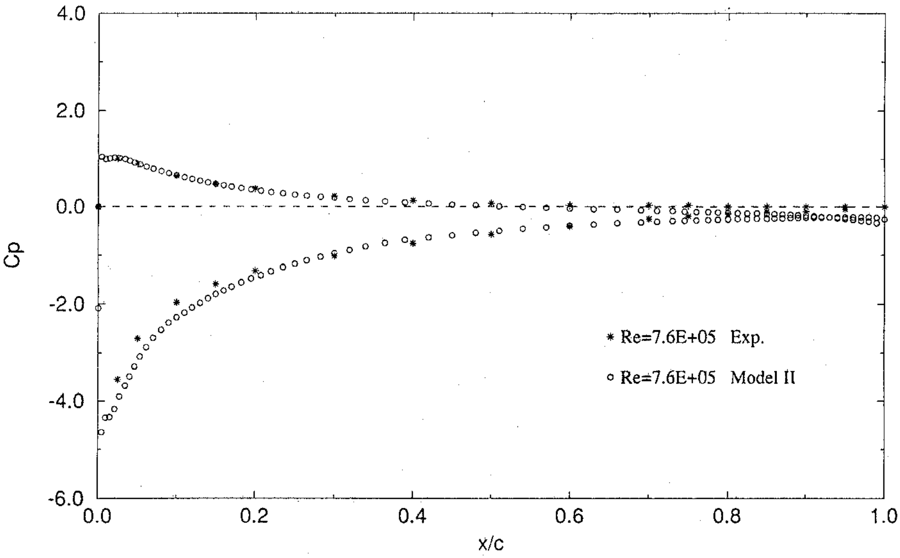


Fig. 3b Mean pressure coefficient along the airfoil.

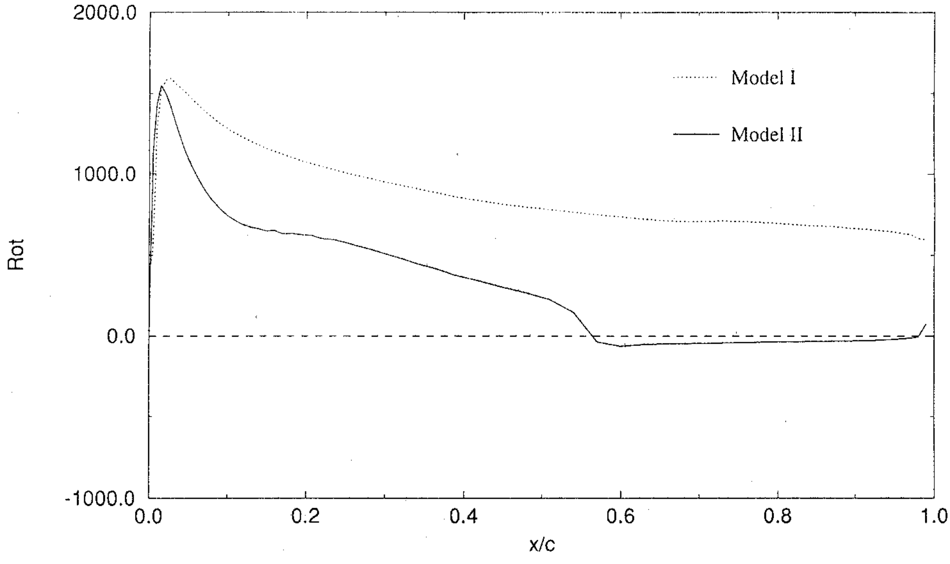
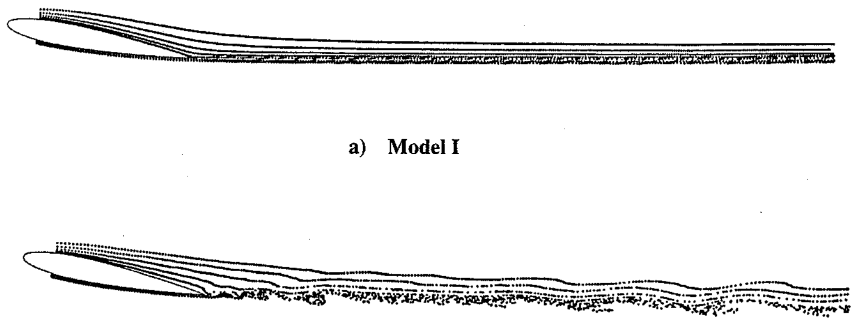


Fig. 4 Mean wall vorticity on the suction side.



a) Model I

b) Model II

Fig. 5 Streamlines.

The results are compared with the standard k-ε model (model I), with the same near-wall treatment. As is discussed in the results, the new treatment of the turbulent energy production term leads to a more physical distribution of the turbulent kinetic energy and especially in the region near the leading edge. The weakness of the standard k-ε model in the leading-edge region is also reported in the work of Kato and Launder,[13] where in the P_k term a product of the strain by the vorticity was suggested.

Results

The numerical tests are carried out for an incompressible flow around a NACA 0012 airfoil at an attack angle of 12 deg and Reynolds number of 10^6. The numerical method resolves the time-dependent equations in primitive variables, in a non-orthogonal curvilinear coordinate system, using an H-type grid of 276×162 nodes; 71 points are distributed on the suction side and 71 points on the pressure side of the airfoil. A non-reflecting type boundary condition[14] is employed for all transported variables at the outlet boundary. The computations are carried out on the IBM 3090-600VF computer of the Center National Universitaire Sud de Calcul (CNUSC), and the computational time is provided by the C3NI program (Centre des Compétences en Calcul Numérique Intensif) of CNUSC-IBM.

Figure 1 illustrates the distribution of the turbulent kinetic energy at $t = 35$. As expected, model I shows a very high level of k, especially in the front region, although the flow is almost laminar there. Owing to this, the turbulent kinetic energy is remarkably diffused over an extended width in the wake. Model II provides a much lower level of the turbulent kinetic energy in the stagnation region and a wavy variation trail in the wake region.

Let us examine now the global aerodynamic parameters, according to both models. Figures 2a and 2b show the time-dependent evolution of the lift coefficient for models I and II. After a transient phase, the lift coefficient reaches an established state. The mean value is found equal to 1.11 (model I) and 1.12 (model II). The experimental value is 1.09 (Ref. 15). Both models give satisfactory results compared with the experiments. The mean drag coefficient is found to be 0.203 in the case of model I and 0.090 for model II. For this Reynolds number, the experimental drag coefficient was not available. We compare with the experimental result for Reynolds number of $Re = 0.76 \times 10^6$, which gives a value of 0.045 (Ref. 16). Model I has the tendency to overpredict the drag coefficient because of a too high level of the turbulent viscosity. Model II reduces the drag coefficient and gives a value closer to the physical experiment.

Figure 3a shows the mean pressure coefficient along the wall. It is noticeable that in the case of the model I the pressure rises unphysically in the vicinity of the leading edge. Model II gives a correct behavior and a good tendency in this area compared with the experiment ($Re = 0.76 \times 10^6$). Figure 3b shows the mean pressure coefficient of a more recent calculation at a Reynolds number 0.76×10^6, with model II. It can be seen that the numerical results in this case become closer to the experimental ones and that a good agreement is obtained.

Figure 4 shows the mean wall vorticity on the suction side. Model I does not indicate any separation because of a high level of the turbulent viscosity near the leading edge. However, with model II, it is clearly shown that separation occurs at $x/c = 0.552$. Hence, this model allows the separation to be predicted. The behavior of these models with respect to the separation is shown by means of the streamlines (Fig. 5). Five streamlines are used for the suction side and two for the pressure side. With model I, the first streamline on the suction side is coincident with the wall; no separation appears. With model II, the first streamline leaves from the wall at the separation point, and the formation of trailing-edge vortices is shown.

Discussion

In the present study a new energy production term is suggested in the two-equation turbulence model for predicting time-dependent, separated flows around airfoils. With this model, the separation can be predicted, especially in a flow where the separation point is not a priori imposed. The computational results provide satisfactory aerodynamic parameters. Other tests are in progress, for various configurations and Reynolds numbers.

The authors would like to note that the new scaling of the turbulent energy production term is limited to the considered flows for the moment. This modification may suggest that the turbulent stresses could be modeled as

$$-\langle u_i u_j \rangle = f(R_{ij}) \tag{9}$$

or more generally

$$-\langle u_i u_j \rangle = f(R_{ij}, S_{ij}) \tag{10}$$

with

$$S_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \tag{11}$$

Acknowlegments

The authors thank P. Chassaing and H. Ha Minh for their helpful discussions.

References

[1]Launder, B. E., and Spalding, D. B., "The Numerical Computation of Turbulent Flow," *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, March 1974, pp. 269–289.

[2]Jones, W. P., and Launder, B. E., "The Prediction of Laminarization with a Two-Equation Model of Turbulence," *International Journal of Heat and Mass Transfer*, Vol. 15, No. 2, 1972, pp. 301–314.

[3]Chien, K.-Y., "Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Number Turbulence Model," *AIAA Journal*, Vol. 20, No. 1, 1982, pp. 33–38.

[4]Haase, W., Brandsma, F., Elsholz, E., Leschziner, M., and Schwamborn, D., "EUROVAL–An European Initiative on Validation of CFD Codes," *Notes on Numerical Fluid Mechanics*, Vol. 42, Vieweg Verlag, 1993, Chap. 5.

[5]Hussain, A. K. M. F., and Reynolds, W. C., "Measurements in Fully Developed Turbulent Channel Flow," *Journal of Fluids Engineering*, Vol. 97, No. 4, 1975, pp. 568–580.

[6]Cantwell, B., and Coles, D., "An Experimental Study of Entrainment and Transport in the Turbulent Near Wake of a Circular Cylinder," *Journal of Fluid Mechanics*, Vol. 136, Nov. 1983, pp. 321–374.

[7]Ha Minh, H., Viegas, J. R., Rubesin, M. W., Vandromme, D. D., and Spalart, P., "Physical Analysis and Second-Order Modeling of an Unsteady Turbulent Flow: The Oscillating Boundary Layer on a Flat Plate," *Proceedings of the 7th Turbulent Shear Flow Symposium* (Stanford, CA), Paper 11-5, Aug. 1989.

[8]Chassaing, P., "Turbulence en Mécanique des Fluides," Institut National Polytechnique, Toulouse, France, Oct. 1990.

[9]Braza, M., and Nogès, P., "Numerical Simulation and Modeling of the Transition Past a Rectangular Afterbody," *Proceedings of the 8th Turbulent Shear Flow Symposium* (Munich, Germany), Paper I-2, 1991.

[10]Franke, R., and Rodi, W., "Calculation of Vortex Shedding Past a Square Cylinder with Various Turbulence Models," *Proceedings of the 8th Turbulent Shear Flows Symposium* (Munich, Germany), Paper 20-2, 1991.

[11]Patel, V. C., Rodi, W., and Scheuerer, G., "Turbulence Models for Near-Wall and Low Reynolds Number Flows: A Review," *AIAA Journal*, Vol. 23, No. 9, 1985, pp. 1308–1319.

[12]Braza, M., Jin, G., and Chassing, P., "Etude Numérique de Certains Phénomènes de Transition dans le Sillage de Profils d'Aile en Conditions Naturelles et Forçées," *Actes du Colloque de Mécanique des Fluides Numérique de Toulouse*, Institut de Méchanique des Fluides de Toulouse, Toulouse, France, 1993, pp. 16-1, 16-4.

[13]Kato, M., and Launder, B. E., "The Modelling of Turbulent Flow Around Stationary and Vibrating Square Cylinders," *Proceedings of the 9th Symposium on Turbulent Shear Flows* (Kyoto Univ., Japan), Paper 10-4, 1993.

[14]Jin, G., and Braza, M., "A Non-Reflecting Outlet Boundary Condition for Incompressible Unsteady Navier-Stokes Calculations," *Journal of Computational Physics*, Vol. 107, No. 2, 1993, pp. 239–253.

[15]Miley, S. J., "A Catalog of Low Reynolds Number Airfoil Data for Wind Turbine Applications," Dept. of Aerospace Engineering, Texas A&M Univ., Rept. DE82-021712, College Station, TX, Feb. 1982.

[16]Michos, A., Bergeles, G., and Athanassiades, A., "Aerodynamic Characteristics of a NACA 0012 Airfoil in Relation to Wind Generators," *Wind Engineering*, Vol. 7, No. 4, 1983, pp. 247–262.

Experimental Investigation of High-Speed Twin Jets

Gamal H. Moustafa*
Menoufia University, Shbien El-Kom, Egypt

Introduction

TWIN jets have been used in many engineering applications. Recently, the development of high-performance aircraft, in which high-speed twin jets are used, is a subject that has received increasing attention. Low-speed twin jets have been investigated extensively, and sufficient data on them are available for many practical engineering purposes.[1,2] Most of the work has been performed in incompressible turbulent jets. However, there have been very few studies of high-speed twin jets. The acoustic and screech phenomena of high-speed twin jets are the aim of most earlier investigators. Miller and Comings[3] studied the subsonic twin jet at only one nozzle spacing. They found that the region of subatmospheric static pressure between the converging jets accounted for their convergence and for the reversal of a considerable fraction of the total flow at the centerline against a strong downstream force of turbulent shear. The phenomenon of supersonic twin jet resonance was defined and studied by Seiner et al.[4] The effect of nozzle spacing on the coupled interaction of supersonic twin jets was examined by Wlezien.[5] He found that, for closely spaced nozzles, coupling occurred at low Mach numbers and was suppressed at high Mach numbers.

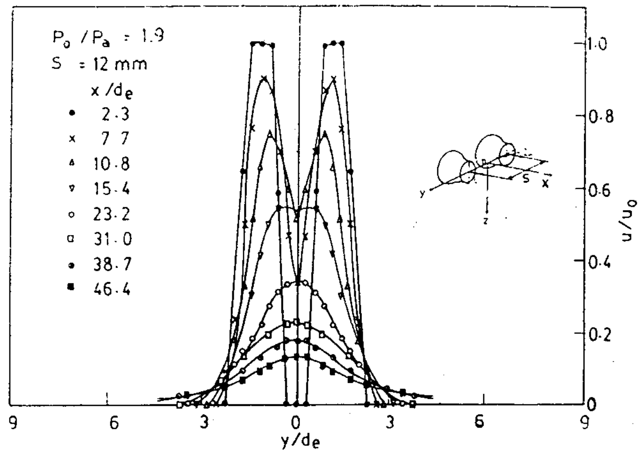


Fig. 1a Mean velocity profile.

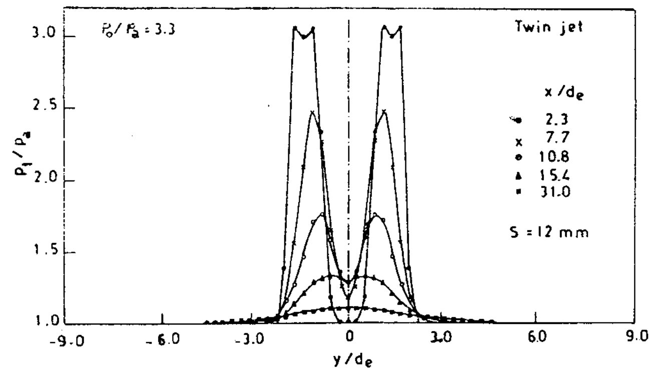


Fig. 1b Total pressure profiles of underexpanded jets.

The main parameters governing the flowfield of a high-speed twin freejet are stagnation/ambient pressure ratio (p_0/p_a) and nozzle spacing S. The interaction between the jet and ambient fluid with varying these parameters just discussed is especially important in many engineering fields. The purpose of the present investigation is to study the effect of the parameters on the twin jet structure, development, and propagation. Only the mean flowfield is highlighted in this investigation.

Experimental Apparatus and Procedure

A blowdown high-pressure supply system was used to provide the airflow to a settling chamber. Before reaching the nozzle, air was passed through three mesh screens set 3 cm apart to reduce disturbances at the nozzle inlet. Two axisymmetric convergent nozzles having an exit diameter d_e of 4.2 mm set in a common wall were used. The spacing between the two nozzles was chosen as 12, 16, 18, and 22 mm. The stagnation/ambient pressure ratio was varied from 1.13 to 4. The Reynolds number based on d_e was varied from 4.2×10^4 to 1.3×10^5. Measurements of total pressure were made across the jet at several downstream locations using a pitot tube with an inside diameter of 0.5 mm. The distance between the nozzle centerline and the nearest wall was about 3 m so that wall effects were negligible in the experiments. The stagnation pressure was maintained at the desired value within an accuracy of ±1.7%. The total pressure (pitot tube reading) measured in the jet (p_t) was uniform within ±0.6%. The stagnation temperature was uniform at 35 ±1°C during the experiments.

Received Oct. 28, 1992; revision received May 20, 1993; accepted for publication Aug. 10, 1993.

*Lecturer, Department of Power Mechanical Engineering, College of Engineering and Technology.