$u/r)^2]$. More complicated relations may be obtained for three-dimensional flow.

Solutions obtained using Eqs. (5-7) may be matched to the law of the wall or a viscous correction may be invented to modify Eq. (5). One way is to multiply the right hand side of Eq. (5) by $(g\ell/\nu)^2[(g\ell/\nu)^2+28^2]^{-1}$ and add ν to ν_t in Eq. (6). This is a completely empirical prescription, of course, but data is reproduced quite well in the viscous sublayer as well as in the outer flow region.

Acknowledgment

Research supported by the Air Force Scientific Research under Grant AFOSR7502756.

References

[1]Clauser, F. H., "The Turbulent Boundary Layer," *Advances in Applied Mechanics*, Vol. 4, 1954, pp. 1-51.

[2]Mellor, G. L., "Equilibrium Turbulent Boundary Layers," *AIAA Journal*, Vol. 2, Sept. 1964, pp. 1650-1651; see also Mellor, G. L. and Gibson, D. M., *Journal of Fluid Mechanics*, Vol. 24, Feb. 1966, pp. 225-253.

[3]Mellor, G. L. and Herring, H. J., "Two Methods of Calculating Turbulent Boundary Layer Behavior Based on Numerical Solutions of the Equations of Motion," *Proceedings: Computation of Turbulent Boundary Layers*, 1968, AFOSR-IFP-Stanford Conference, Stanford University, Stanford, Calif.

[4]Cebeci, T., Kaups, K., and Moser, A., "Calculation of Three-Dimensional Boundary Layers: III," *AIAA Journal*, Vol. 4, Aug. 1976, pp. 1090-1094.

[5]Blackadar, A. K., "The Vertical Distribution of Wind and Turbulent Exchange in Neutral Atmosphere," *Journal of Geophysical Research*, Vol. 67, July 1962, pp. 3095-3102.

[6]Mellor, G. L. and Herring, H. J., "A Survey of the Mean Turbulent Field Closure Models," *AIAA Journal*, Vol. 11, May 1973, pp. 590-599.

[7]Mellor, G. L. and Yamada, T., "A Hierarchy of Turbulence Closure Models for Planetary Boundary Layers," *Journal of Atmospheric Sciences*, Vol. 31, Oct. 1974, pp. 1791-1806.

[8]Briggs, M., Mellor, G. L. and Yamada, T., "A Second Moment Turbulence Model Applied to Fully Separated Flow," to be published, *Proceedings; Squid Workshop on Turbulence in Internal Flows*, June 1976.

[9]Hanjalic, K. and Launder, B. E., "Fully Developed Asymmetric Flow in a Plane Channel," *Journal of Fluid Mechanics*, Vol. 52, April 1972, p. 689.

[10]So, R. M. C., "A Turbulence Velocity Scale for Curved Shear Flow," *Journal of Fluid Mechanics*, Vol. 70, pp. 37-57.

[11]Mellor, G. L., "A Comparative Study of Curved Flow and Density-Stratified Flow," Vol. 32, July 1975, pp. 1278-1282.

Finite Element Approach to the Viscous Incompressible Flow around a Circular Cylinder

Ciro W. Lucchi*
Dornier GmBH, Friedrichshafen, West Germany

Introduction

THE steady flow of a viscous incompressible fluid around a circular cylinder is examined for Reynolds numbers up to approximately 40. Thus it is a thoroughly laminar flow, having a bubble of recirculation behind the cylinder for Reynolds numbers higher than about 7. Cases at Reynolds numbers above 40 are usually not computed using the steady equations because unsteady vortex shedding starts to develop.

Received Feb. 14, 1977.
Index categories: Computational Methods; Viscous Nonboundary-Layer Flows.
*Research Scientist. Member AIAA.

Numerical results for the steady flow—apparently almost every new method or variant thereof is tested against this problem; two reasons are prominent for this fact: a wealth of data exists from both numerical and experimental investigations; and the problem, by no means simple, is still of such dimensions that a good accuracy can be achieved. The investigators used at first two methods: the finite difference method[1] and the method of truncated series.[2-3] A third method has come into prominence, the finite element method, and it was also applied to the solution of this problem.[4-6] Unfortunately, however, its applications to fluid dynamics were often an extension of its development in solid mechanics and available results show only crude solutions, thus shedding little or no light at all on its comparison with respect to other methods.

Within the spectrum of all the work done till now, other aspects vary too. As for example the form in which the Navier-Stokes equations were solved. Some are typical of the two-dimensional flow solutions: having as variables the streamfunction and the vorticity or the streamfunction alone. Other forms use the physical variables. Another point of importance is whether the equations are solved in the physical plane or in a transformed plane.

A last characteristic for the fast convergence of results is the representation of the far field. The usual techniques involve either a transformation bringing infinity to a finite distance or imposing the uniform flow at a finite distance or matching on the boundary the numerical solution with an asymptotic solution.

Experimental results—reliable results are available for this range of Reynolds number.[7] The drag coefficient was measured and the flow patterns were photographed. A more detailed description of the present work may be found in Ref. 8.

The Method

Equation—because of the hypotheses that characterize the flow, the Navier-Stokes equations are reduced to the continuity and two momentum equations:

$$\begin{aligned} &\partial_j u_j = 0 \\ &\rho u_i \partial_i u_j = -\partial_j p + \mu \partial_i \partial_i u_j \end{aligned} \qquad (i,j=1,2)$$

Since μ is considered constant, cross-differentiating and substracting the momentum equations and defining the streamfunction ψ from the continuity equation will yield a biharmonic-type of equation:

$$\frac{1}{Re}\nabla^4\psi + \psi_x\nabla^2\psi_y - \psi_y\nabla^2\psi_x = 0 \tag{1}$$

where Re is the Reynolds number (based on the diameter of the cylinder) and the subscripts indicate differentiation. No transformation of the physical space is performed.

Galerkin method—assuming for the streamfunction a trial function of the form $\psi' = \psi_j N_j(x,y)$ where the ψ_j are the generalized variables and the N_j the shape functions, this procedure[9] together with the use of Green's theorem yields the following set of nonlinear algebraic equations:

$$\frac{1}{Re}\int_S \nabla^2 N_i \nabla^2 N_j \psi_j \mathrm{d}S + \int_S [N_{i,x}N_{j,y} - N_{i,y}N_{j,x}]\nabla^2 N_k \psi_j \psi_k \mathrm{d}S + \frac{1}{Re}\int_C N_{i,n}\omega \mathrm{d}s - \frac{1}{Re}\int_C N_i \omega_n \mathrm{d}s + \int_C N_i \psi_s \omega \mathrm{d}s = 0 \tag{2}$$

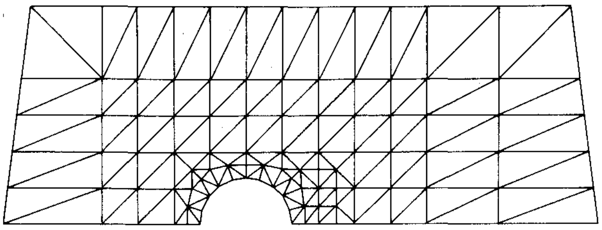

Fig. 1 Domain of computation.

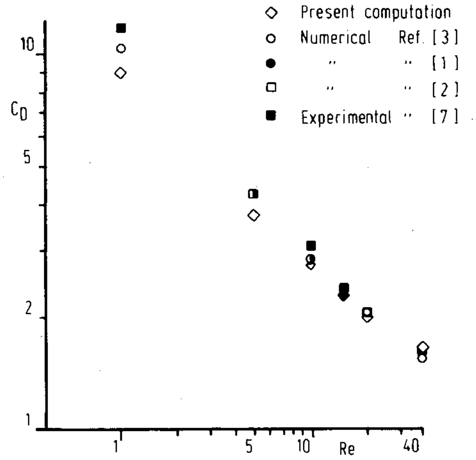

Fig. 2 Drag coefficient vs Reynolds number.

where ω is the vorticity, C is the contour and the subscripts x, y, s, and n indicate differentiation with respect to those variables.

Boundary conditions—the only flow treated within the present work is the symmetric unbounded flow around a circular cylinder. Thus three types of boundary conditions exist: the no-slip condition on the cylinder, the line of symmetry and the outer boundary. The numerical solution in the inner domain is matched on the outer boundary with the Oseen asymptotic solution valid in the outer domain. [10]

Drag coefficient—it is obtained by a momentum balance using a line contour around the body. [11]

Finite element—the surface integrals of Eq. (2) involve first-and second-order derivatives of the streamfunction. Compatibility criteria[12] would require continuity of first derivatives across the elements' interfaces. It was however shown that convergence and good accuracy can be achieved by an element that provides function conformity but no slope conformity. [13] Dealing with a similar equation as treated in Ref. 13 and because of its simplicity (as compared with elements that satisfy slope conformity) and its good characteristics, the BCIZ element was chosen for this approach.

Nonlinear algebraic equations—they are solved by the Newton iterative technique, each step being solved by direct elimination.

Results

The investigation has been restricted to the flow around a circular cylinder in an unbounded region at Reynolds numbers 1, 5, 10, 15, 20, and 40. Extensive studies in the case of Reynolds number 10 were first carried out and the domain of computation thus obtained (Fig. 1) was then used in the other cases.

Considering that only 188 variables were used in the present work, the results for the drag coefficient compare well with other numerical methods (that use from 600 to 6000 variables) and with the experiment (Fig. 2).

In the case of low Reynolds numbers (1 and, in less measure, 5) the perturbations generated by the solid body propagate upstream and sidewise to larger distances than at large Reynolds numbers, thus requiring larger dimensions in those directions than the domain of computation provides. At high Reynolds numbers the opposite situation appears: the upstream and sidewise perturbations decay quite fast; the wake however becomes rather long and, for an accurate representation, the downstream dimension should be increased. Figure 3 finally presents the vorticity distribution around the cylinder. Comparison with Ref. 3 has to be careful; while in Ref. 3 the vorticity is a variable, in the present work it has to be obtained by a double differentiation of the streamfunction. Thus the vorticity is linear over the elements and discontinuous across the interfaces. Furthermore, while in Ref. 3 thirty points are placed around the half cylinder, only ten elements cover the same arc in the present computation.

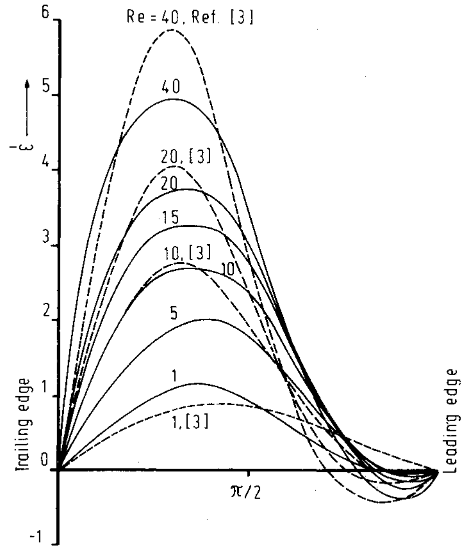

Fig. 3 Vorticity distribution on the cylinder.

Conclusions

Although the results cannot be said to be very accurate, the method has been shown to yield reasonably good results within the contraints of poor meshes. Reasons for this are the following: the BCIZ element is well adapted to the given equation; an asymptotic solution represents the far field instead of the uniform flow condition; and the mesh was optimized.

The choice of the biharmonic-type of equation led to a solution method having distinct advantages: a similar equation has been well investigated in the problem of plate bending; only one variable is involved; and streamlines are a natural outcome of this approach.

The main disadvantage was the rather complex form of the contour integrals generated by the use of Green's theorem to transform the integrand in the Galerkin technique (it must be borne in mind that contour integrals may arise whenever the Galerkin technique is used).

The present work thus pointed out once more the excellence of the BCIZ element and the application of the finite element method to this flow problem. Furthermore it showed that simpler approaches are necessary to make this method practical in fluid dynamics problems.

Acknowledgment

This Note is based partially on the research sponsored by the U.S. Office of Naval Research.

References

[1]Keller, H. B. and Takami, H., "Numerical Studies of Steady Viscous Flow about Cylinders," *Numerical Solution of Non-Linear*

Differential Equations, edited by D. Greenspan, Wiley, N.Y., 1967, pp. 115-140.

[2]Dennis, S. C. R. and Chang, G. Z., "Numerical Solutions for Steady Flow Past a Circular Cylinder at Reynolds Numbers up to 100," *Journal of Fluid Mechanics*, Vol. 42, 1970, pp. 471-489.

[3]Nieuwstadt, F. and Keller, H. B., "Viscous Flow past Circular Cylinders," *Computers & Fluids*, Vol. 1, 1973, pp. 59-71.

[4]Olson, M. D., "A Variational Finite Element Method for Two-Dimensional Steady Viscous Flows," *Proceedings of the Speciality Conference on Finite Element Methods in Civil Engineering*, Engineering Institute of Canada, Montreal, 1972, pp. 585-616.

[5]Taylor, C. and Hood, P., "A Numerical Solution of the Navier-Stokes Equations using the Finite Element Technique," *Computers & Fluids*, Vol. 1, 1973, pp. 73-100.

[6]Gartling, D. and Becker, E. B., "Computationally Efficient Finite Element Analysis of Viscous Flow Problems," *Proceedings of the International Conference on Computational Methods in Nonlinear Mechanics*, Austin, Texas, 1974, pp. 603-614.

[7]Tritton, D. J., "Experiments on the Flow past a Circular Cylinder at Low Reynolds Numbers," *Journal of Fluid Mechanics*, Vol. 6, 1959, pp. 547-567.

[8]Lucchi, C. W., "Finite Element Approach to the Viscous Incompressible Flow around a Circular Cylinder," Cornell University, Ithaca, New York, M.S. Thesis, Jan. 1977, also Dornier GmbH, Friedrichshafen, West Germany, Bericht 76/52 B, Nov. 1976.

[9]Finlayson, B. A., *The Method of Weighted Residuals and Variational Principles*, Academic Press, New York, 1972.

[10]Lagerstrom, P. A., "Laminar Flow Theory," *High Speed Aerodynamics and Jet Propulsion*, Vol. 4, edited by F. K. Moore, Princeton University Press, Princeton, N.J., 1964, pp. 20-285.

[11]Imai, I., "On the Asymptotic Behavior of Viscous Fluid Flow at a Great Distance from a Cylindrical Body, with Special Reference to Filon's Paradox," *Proceedings of the Royal Society of London*, Vol. A 208, 1951, pp. 487-516.

[12]Zienkiewicz, O. C., *The Finite Element Method in Engineering Science*, McGraw-Hill, New York, 1971.

[13]Bazeley, G. P., Cheung, Y. K., Irons, B. M., and Zienkiewicz, O. C., "Triangular Elements in Plate Bending—Conforming and Non-Conforming Solutions," *Proceedings of the First Conference on Matrix Methods in Structural Mechanics*, Wright-Patterson AFB, Ohio, 1965, pp. 547-576.

Boundary-Layer Effects on Pressure Variations in Recovery Tube

A. V. Murthy*
National Aeronautical Laboratory, Bangalore, India

FOR generating high Reynolds number transonic flows, the application of the Ludwieg tube principle has been considered to be an attractive proposition because of its simplicity and also its high flow quality. To improve the efficiency of operation of the basic Ludwieg tube, the addition of a long tube called the recovery tube has been suggested.[1] While the calculation procedure for the ideal flow quantities in the recovery tube is simple, deviations from these calculations can be expected because of the real gas effects. The purpose of this paper is to present an analysis to calculate the departure from the ideal flow behavior due to the turbulent boundary-layer growth on the tube wall. The analysis, based on the solution of one-dimensional unsteady flow equations with distributed mass sources, predicts that, at the tube entry, the pressure increases with time due to boundary-layer growth. This can cause early termination of the uniform test section flow by unchoking of the throat. The agreement with the limited available data on pressure measurements[2] indicates that the analysis can be used with confidence to obtain preliminary estimates of the boundary-layer effects in the recovery tube for larger tunnels like that proposed for the European Transonic Ludwieg Tube Tunnel.[1]

Presented at the AIAA 9th Aerodynamic Testing Conference, Arlington, Texas, June 7-9, 1976 (in bound volume of Conference papers, no preprint number); submitted July 16, 1976; revision received March 1, 1977.

Index category: Boundary Layers and Convective Heat Transfer—Turbulent.

*Scientist, Aerodynamics Division.

Analysis

The flow into the recovery tube is idealized by inflow through a choke into a long tube closed at one end (Fig. 1a). The pressure inside the tube is kept below the ambient pressure by a diaphragm attached to the nozzle end. The rupture of the diaphragm initiates the propagation of a shock wave and a following contact surface into the tube. In this analysis, it is assumed that the nozzle throat is choked and the deceleration of the flow to subsonic Mach numbers in the tube (region 3) occurs through a stationary shock located in the diffuser section. This phenomenon is similar to that happening in the recovery tube of transonic tunnel.[1] Across the contact surface, the pressure and the velocity are continuous but the temperature has a jump since region 2 ahead of it contains the gas processed by the shock wave. The boundary-layer growth in the tube is considered in two parts. Firstly, the unsteady boundary layer in region 2 induced by the passage of the shock wave. Secondly, for region 3 between the tube entry and the contact surface, a steady flat plate type boundary layer starting from the entry section of the tube from the instant the constact surface has passed is assumed. The downstream edge of this boundary layer is assumed to move along with the contact surface and at any station in between, the boundary-layer growth is considered time independent. For region (2) induced by the shock wave, the unsteady boundary-layer growth is calculated by considering the equivalent steady-state problem with shock-fixed coordinates.[3]

For calculating the perturbation in the flow quantities, the boundary-layer growth is assumed to act as distributed mass sources the strength of which is dependent on the vertical velocities induced at the outer edge of the boundary layer. For the one-dimensional flow in a tube of uniform cross section with distributed mass sources, the solution of the governing differential equations can be written as[4]

$$\Delta p^{\pm} = \pm \frac{2\gamma\rho}{ad(1 \pm M)} \int_{\mp\infty}^{\kappa} v(\xi, t - \frac{\kappa - \xi}{a - u})\,d\xi \qquad (1a)$$

$$\Delta p = \Delta p^{+} + \Delta p^{-} \qquad (1b)$$

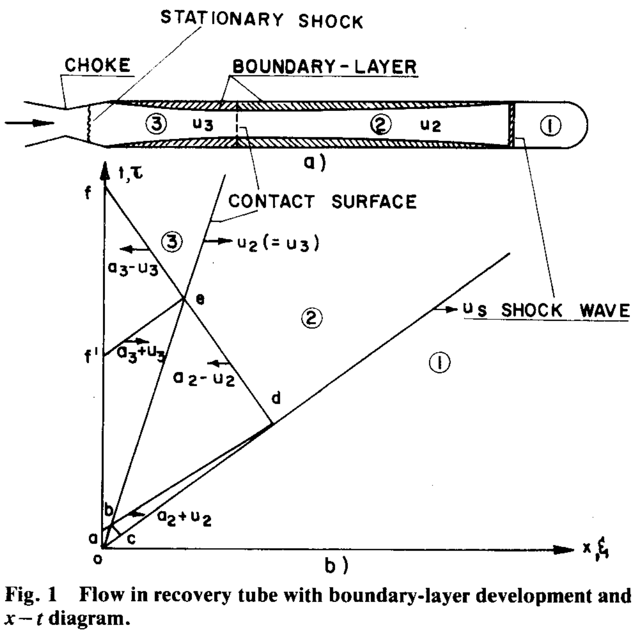

Fig. 1 Flow in recovery tube with boundary-layer development and $x-t$ diagram.