estimate and how it was arrived at. Furthermore, how different from the initial estimate was the final flow field?

Additional References

1 Ivanov, Yu. V., *Effective Combustion of Overfire Fuel Gases in Furnaces*, Estrosizdat, Tallin, 1959.
2 Tatom, F. B., and Kezios, S. P., "Interaction of a Turbulent Compressible Underexpanded Two-Dimensional Jet with a Crossflowing Subsonic Free Stream," *Proceedings, AIAA 4th Fluid and Plasma Dynamics Conference*, Palo Alto, Calif., June 21–23, 1971.
3 Waldrop, W. R., and Farmer, R. C., "Three Dimensional Computation of Buoyant Plumes," *Journal of Geophysical Research*, Vol. 79, No. 9, Mar. 20, 1974.

DAVID K. GARTLING.[4] The authors have presented an interesting and useful analysis of an important problem in fluids engineering. Their work has again demonstrated the power of a numerical approach for the treatment of complex, multidimensional fluid dynamics problems.

As shown by the figures, the author's finite difference procedure does very well in predicting the qualitative behavior of the jet and in most cases also provides a good quantitative description However, in Fig. 2 it can be observed that there is a significant departure between the experimental data and the numerical predictions for R values of 4 and 6. While this writer is aware of the difficulties in using experimental data to verify a numerical model, some comment by the author's on this lack of agreement for particular R values would seem appropriate.

Though the author's analysis has shown generally good agreement between experimental and numerical velocity data, a discussion of the computed pressure field was not presented. Very often numerical models predict very accurate velocity fields and relatively poor pressure fields, especially in low speed flows. Have the authors attempted to compare their pressure predictions with experimental data, if such data exists?

A final question involves the location of the upstream computational boundary. The location of the boundary at $4r_0$ upstream of the jet seems to be a very short distance in which to assume the velocity field varies from a uniform free stream value to a highly three dimensional field near the jet. Was this boundary location tested by numerical experimentation?

Authors' Closure

We would like to thank Dr. Gartling and Dr Tatom for their interesting comments.

Regarding Dr. Gartling's first point, the cases of $R = 4$ and 6 in Fig. 2 should be viewed along with the results for similar values of R in Fig. 3. When the data from different sources show this amount of spread, no comment on the departure between predictions and individual experimental data seems necessary. Further, it should be remembered that, for numerical solutions and for experimental measurements, the location of maximum velocity is difficult to determine with great precision.

No pressure fields were presented in the paper, because corresponding experimental data were not available. However, there is no reason to believe that the predicted pressure fields are not satisfactory.

Dr. Gartling's final point is also raised by, and to some extent answered by, Dr. Tatom. This matter and other boundary-condition practices should be judged by reference to the aim and emphasis of the work. We focussed attention on the main properties of the jet, such as the location of its center line and the value of the maximum velocity. We did not attempt to predict accurately other fine details such as the nonuniformity of the velocity profile at the jet orifice, the boundary layer on the plate, the recirculation zone downstream of the jet. (It would have required a much finer grid to do justice to these details.) The locations of the boundaries and the specifications of the boundary conditions were satisfactory in this context. They may have, however, influenced some finer details of the flow field.

We are grateful to Dr. Tatom for the additional references. Regarding his comments about the locations of the boundaries, it is sufficient to state that we did ensure that the *presented* results remained unaffected by any further enlargement of the calculation domain and thus could be regarded as the results for an unconfined situation.

The flow field in Fig. 1 is a qualitative representation of what happens. It reflects the results of our numerical predictions as well as published experimental studies. The grid spacing employed was quite nonuniform, especially in the y direction. The finite-difference scheme used a control-volume approach around each grid point. The wall function method has its limitations, but it often seems to work well even where it is not expected to.

Lastly, our initial estimate of the flow field was most simple-minded: within the calculation domain, we set the velocity equal and parallel to the mainstream velocity and assumed the pressure to be uniform everywhere.

A Flow Study in Radial Inflow Turbine Scroll-Nozzle Assembly[1]

G. F. CAREY.[2] The paper presents some numerical results for flow in a radial turbine. Although the analysis begins with the compressible potential formulation, (equation (2)), both the finite element analysis and subsequent numerical experiments consider simple incompressible flow (the Laplacian). Consequently, the resulting finite element formulation (equations (4-8)) is very standard. Perhaps one should note in addition that, while the statements of equations (5) and (6) are common even in the textbook literature, it is strictly more correct to state that these integrals are the element contributions to the finite element system or, alternatively, replace the element basis function N_i by the nodal patch function p_j at node j. Here p_j is composed from the element basis functions N_i adjacent to node j. As a final point on the finite element formulation, the branch cut contributions are very sparingly described and not brought out clearly in the presentation. The treatment of the Kutta condition should be elaborated upon and the branch cut contributions included in the finite element equations, to complete the description of the finite element problem.

From a turbo-machine design standpoint, the principal detractions of the method are the following: the actual three-dimensional flow problem has been replaced by a two-dimensional flow problem with the same flow area; viscous effects, particularly

[4]Fluid Mechanics and Heat Transfer Division 1261, Sandia Laboratories, Albuquerque, N. Mex.

[1]By A. Hamed, E. Baskharone, and W. Tabakoff, published in the March, 1978, issue of the Journal of Fluids Engineering, Vol. 100, No. 1, p. 31.

[2]Texas Institute for Computational Mechanics (TICOM), the University of Texas at Austin, Austin, Texas.

in the blade-to-blade portion of the flow domain are not included; compressibility is not included, despite the attempt to include the density in the formulation. Although these factors prohibit general flow studies using this analysis, the authors have in-investigated two different blade configurations with linearly varying area scroll and obtained an indication of the general performance of different vane shapes for the flows considered.

As mentioned above, the density ρ is included in the formulation of the governing equations. This treatment of ρ together with the claim in the final remarks that the analysis can be extended to compressible flows necessitates some qualifying remarks: a major point not treated in the paper is the fact that the density field depends implicitly on the unknown potential solution ($\rho = \rho(\mathbf{v}) = \rho\nabla\phi$)) so that the finite element system arising from equation (7) would be quite nonlinear. An alternative procedure for treating the compressibility effects is to "uncouple" the density equation and solve the problem iteratively. Thus, beginning with, say, the incompressible solution we introduce the density relation to estimate the actual density field $\rho(x, y)$ and then using this estimate in equation (7) the dependence of ρ on ϕ is removed from the finite element solution for the next potential field iterate. Thus ϕ is recomputed and the procedure continues, iteratively improving the potential field and density fields successively. This strategy is adequate for very low speed flows but at higher speeds difficulties have been experienced in computing converged solutions for similar compressible flows arising elsewhere in engineering, in for example, compressible throughflow calculations in turbo-machines [22][3] and in computational transonics [23, 24]. In the latter instance the mathematical aspects of these difficulties have been more substantially investigated.

Additional References

22 Oates, G. C., Knight, C. J., and Carey, G. F., "A Variational Formulation of the Compressible Throughflow Problem," ASME *Journal of Engineering for Power*, Vol. 98, No. 1, Jan. 1976, pp. 1–8.

23 Shen, S. F., "An Aerodynamicist Looks at the Finite Element Method," *Finite Elements in Fluids*, Vol. 2, R. H. Gallagher, et al. (eds.), Wiley, 1975.

24 Carey, G. F., "Computational Transonics," *Finite Elements in Fluids*, Vol. 3, R. H. Gallagher, et al. (eds.), Wiley (in press).

D. ADLER.[4] The authors solved the complicated problem of the flow in a multiple connected domain of inward flowing gas. The assumptions and simplifications used in the formulation of the problem and the development of the finite element solution procedure are well justified in a first attempt, considering the complexity of the real three-dimensional compressible flow.

The governing partial differential equation is elliptic and requires boundary conditions to be known all around the flow region. The difficulty lies in the inner exit boundary on which V_{n2} has to be specified. The V_{n2} distribution is actually a result of the upstream flow and the upstream passage geometry. The specification of exit boundary conditions in elliptic flow problems is a classical problem solved in many cases by locating the exit boundary far downstream at a place where a uniform flow distribution can be assumed to be correct. In the present case, this is not possible. Uniform V_{n2} distribution cannot be assumed because the exit boundary cannot be moved far enough.

The authors do not explain how V_{n2} was specified. This information surely will interest many readers. Further, it would be interesting to know how does the V_{n2} distribution effect the mass flow distribution between the nozzle channels (Fig. 8).

[3]Numbers 22–24 in brackets designate Additional References at end of discussion.

[4]Professor, Faculty of Mechanical Engineering, Technion Israel Institute of Technology, Haifa, Israel.

T. C. PRINCE.[5] The authors have demonstrated an effective method for solving an unusually complex potential flow field. They point out correctly that it would be more difficult to solve this problem by a stream function formulation, but the only true practical difference (for subsonic flow) is in the density of the "stiffness" matrix. Thompson's straight sided elements (14) were an erroneous application of cubic elements without the necessary isoparametric transformation. He went on to demonstrate equal accuracy with stream and potential formulations, using cubic isoparametric elements.

The discussion of the Kutta-Zhukovski condition is not clear. The requirement of no force on the cuts implies the same jump in potential everywhere on the cut, giving a continuous solution regardless of whether the Kutta condition is satisfied. If the value of the jump (circulation around the airfoil) were known, the matrix would be symmetric. In the present problem, the value of the jump must be adjusted to satisfy some condition on smoothness of the solution. The classical requirement is that the stagnation streamline be attached to the trailing edge. No reliable direct statement of this condition is available in the finite element method. It is my understanding that the authors have chosen a condition on uniformity of tangential velocity on the downstream boundary, where the normal velocity is prescribed uniform by the boundary condition.

The authors employ a mixture of notations in equation (8). The only simplification which should be made from equation (8) is to neglect the derivatives with respect to the axial coordinate z. A quasi-three-dimensional solution could be obtained by retaining variable thickness of the flow field in the z direction.

It is unfortunate that the authors did not determine the accuracy given by their choice of elements. A better case for quadratic elements might be made in this problem than in most two-dimensional flows. On the other hand, the accuracy appears to be sufficient to give a useful solution to a problem which has not been analyzed previously. While the assumption of potential flow is not realistic in the illustrated examples, analysis by the authors' method should permit design for attached flow. This should lead to more predictable and possibly better aerodynamic performance.

CH. HIRSCH.[6] The authors present an interesting application of the Finite Element Method to a complicated geometrical configuration. However, several points require some clarification.

1 It is well known that even for potential function formulation, linear elements give less accuracy than higher order elements, as shown e.q. in reference [14] of the paper. Acceptable accuracies can nevertheless be obtained with linear elements at the cost of using a refined mesh. It is not clear from the paper, Fig. 3, how fine the mesh is in the blade passages, from point of view of the precision of the geometrical representation of the blades. Could the authors specify the number of elements used along the blades as well as the influence of the mesh size on the accuracy of the numerical results.

2 The introduction of the Kutta condition through the potential discontinuities along the cuts is not very clearly described. If uniform conditions (uniform normal and tangential velocities) are imposed at exit then the potential jumps should have the same value along each cut and only one unknown would remain. Also the numerical process leading to the determination

[5]General Electric Co., Cincinnati, Ohio.

Note: Reference [15] is available as a dissertation under University Microfilms No. 76-28343.

[6]Professor, Vrije Universiteit Brussel, Faculteit Van De Toegepaste Wetenschappen, Brussels, Belgium.

of these discontinuities from the prescription of continuity of the unknown value of f in equation (3) requires a more extensive description.

The influence of these conditions on the nonsymmetric and nonbanded structure of the stiffness matrix is not obvious and appears as an important source of increase of computer time which could perhaps be avoided. Also typical values of computer running times would be interesting.

3 The interest of this paper would be enhanced by presentation of comparisons with experimental data, like the velocity distribution along the blades reported in reference [11] of the paper.

Authors' Closure

The authors wish to thank Drs. Hirsch, Prince, Carey, and Adler for their comments and discussions. Since some of the questions were shared by the first two reviewers, we will give a combined closure to their discussions first.

Drs. Prince and Hirsch raised questions related to the details of the boundary conditions at the cuts extending from the nozzle blades, and to the Kutta condition. The requirement of equal flow velocities at the two sides of each cut were imposed as boundary conditions. Since the potential function formulation involves the normal derivative at the boundaries, it was the circumferential velocity component that was set equal on the two sides of each radial cut. The equality in the radial velocity component which is in the direction tangent to the cut is inherent, since the difference in the values of the potential function on the two sides of a given cut is the same due to the absence of any exerted forces. Our analysis is based on the use of the potential function formulation with only the normal velocity component involved at all the boundaries including the radially inward downstream station. Unlike in the stream function formulations generally [13, 14, and 16], and some potential function formulations [15], our solution did not involve any specifications of the tangential velocity component or of the flow angle at the downstream station. There was no need therefore for any iterative procedures involving adjustment of the downstream tangential velocity component to satisfy the Kutta condition as those used in such analyses. It is important however to notice that the symmetry of the stiffness matrix is sacrificed in our case.

The tangential forces exerted by the different nozzle blades are generally not equal due to the differences in the flow fields in the different channels. Even when the downstream conditions are uniform, these differences are caused by the variation in the conditions at inlet to every channel which are affected by the scroll geometry. This provides an explanation to Dr. Hirsch's question as to why generally the potential jumps do not have the same value across all the cuts.

We agree with Dr. Prince's comment on the similarity between the stream function and the potential function formulations of subsonic flows in cascades, only when the downstream tangential velocity is specified and modified iteratively according to the Kutta condition in the solution. The similarity does not extend however to our formulation where only the normal velocities are involved in all the boundary conditions, and the correct downstream tangential velocities, evolve naturally from the calculation. Furthermore we believe that the stream function formulation is completely unsuitable for handling the present problem in which the mass flow distribution between the different channels is not initially known, but is under investigation.

Our computed results were found to compare favorably with the available experimental data for the pressure distribution on the nozzle blades in spite of the two dimensional flow field approximations, and the incompressible flow analysis. A total of 817 nodes and 1292 elements were used in our solution with a computation time of 8 minutes on AMDAHL 470 computer.

In reference to Dr. Carey's comments, we do not think our notations, which have been commonly used by other investigators, need further discussions since they can be found in standard textbooks [18 and 20]. Furthermore, the complexity of the problem under investigation needs no explanation to anybody familiar with turbomachine flows. We feel that the reasoning for any assumptions and simplifications made, were well described in the text, aside from those already well known to be justified in turbomachine flow calculations. The simplest extension to the compressible flow was referred to in our paper, in which the same solution procedure could be used iteratively after uncoupling the density. Other techniques whose applicability extend to higher Mach numbers are better discussed in other references [15].

In answer to Dr. Adler's question concerning the specification of the downstream boundary conditions, the same classical approach of placing that boundary far enough with a uniform v_{n2} has been used. The value of v_{n2} is adjusted according to the radial location of the downstream boundary to satisfy the global continuity requirements. We found that, placing the exit station at a radial distance from the nozzle blade trailing edges, more than three quarter the radial chord projection, did not produce any appreciable difference in the flow conditions between the nozzle blades or in the scroll, nor did it alter the mass flow distribution between the different channels.

On Vortex Wind Power[1]

J. P. JOHNSTON[2] and J. EATON.[2] Dr. So offers an interesting theoretical model for the "vortex wind power" turbine proposed by Yen. Like the incomplete model offered by Yen [1], it allows one to estimate the maximum theoretical power output of the turbine. Although it has the appearance of greater rigor than the analysis by Yen, it also rests on many assumptions and approximations. Nevertheless, it does appear to provide a basis for estimation of the limits of performance for this class of wind turbine.

We found two errors in So's analysis and one point which should be questioned. There is a sign error in the right-hand term of equation (2d); it should be negative. The second error is more serious; the maximum wind power given by equation (17) is of the correct form but, as shown below, its numerical coefficient is 6.44 times too large.

The point in question concerns the assumption made in the development of equation (13). We don't feel it is necessary that the static pressure $p(0)$ and the axial velocity w_0 in the center of the vortex at $z = 0+$ (above the turbine) be exactly equal to the central static pressure and axial velocity at $z = 0-$ (below the turbine). Nevertheless, this is a possible condition for elimination of the parameter A. If the condition is not imposed, then the results become a function of one more free parameter and consequently would be of little practical value. Other means for elimination of the parameter A should be examined.

Return now to the question of the maximum turbine power, the power that may be extracted if all losses inside the turbine

[1]By Ronald M. C. So, published in the March, 1978, issue of the ASME JOURNAL OF FLUIDS ENGINEERS, Vol. 100, No. 1, pp. 79–82.

[2]Professor, and Research Assistant, respectively, Department of Mechanical Engineering, Stanford University, Stanford, Calif.