N75 11180

The Solution of Three-Variable Duct-Flow Equations

A. R. STUART

University of Aston
Birmingham, England

R. HETHERINGTON

Rolls-Royce Limited and University of Aston

Past effort in computational techniques in internal flow systems has been concentrated on two-variable problems. This paper establishes a numerical method for the solution of three-variable problems and is applied here to rotational flows through ducts of various cross sections.

An iterative scheme is developed, the main feature of which is the addition of a duplicate variable to the forward component of velocity. Two forward components of velocity result from integrating two sets of first-order ordinary differential equations for the streamline curvatures, in intersecting directions across the duct. Two pseudo-continuity equations are introduced with source/sink terms, whose strengths are dependent on the difference between the forward components of velocity. When convergence is obtained, the two forward components of velocity are identical, the source/sink terms are zero, and the original equations are satisfied.

A computer program solves the exact equations and boundary conditions numerically. The method is economical and compares successfully with experiments on bent ducts of circular and rectangular cross section where secondary flows are caused by gradients of total pressure upstream.

The presence of secondary-flow losses is well known. When a shear flow passes through a bend with a vorticity component directed toward the center of curvature, a secondary flow exists, transverse to the mean flow. The vorticity is produced by a velocity gradient in the flow approaching the bend. This velocity gradient may be produced by viscous losses upstream and by nonuniform work being done on the fluid. The losses in a

secondary flow are due to the energy contained in the transverse flow, much of which is not recovered. (The presence of secondary flow may also cause subsequent parts entered by that fluid to run partly "off design.")

This attack on the secondary-flow problem solves the fully three-dimensional flow equations. The equations of three-dimensional fluid flow are intractable to analytic solution, even with the inviscid and steady flow assumptions. Until now, they have defied numerical solution due to the insufficient core size and speed of the past-generation computer and the lack of a numerical technique. The development of this three-dimensional method of solution was stimulated by the success of various two-dimensional numerical methods. The method is an extension of the two-variable streamline curvature method (refs. 1 and 2).

Although the method as presented is restricted to enclosed ducts, it is also possible to include repeat boundary conditions, thus enabling solutions of the turbomachinery blade passage flow to be obtained.

DEVELOPMENT OF THE THREE-VARIABLE METHOD

To economize on time and effort during the initial development of a three-variable method and to facilitate a clear understanding of the mechanisms involved, attention was restricted to incompressible flows and a simple geometry, for which experimental data was available (ref. 3). The geometry is shown in figure 1. It consists of a rectangular duct which turns through any number of degrees on constant mean radius R_m. Coordinates x and z are fixed in each plane of cross section and y is measured along the centerline.

An Eulerian approach to the equations is used, since the Lagrangian method, which is used in two-variable streamline curvature methods, is excessively complicated in three variables. It requires the storage and manipulation of expressions for two interacting families of stream surfaces and their interaction with the boundaries.

Basic Equations

Continuity

$$\frac{1}{1+x/R_m}\frac{\partial}{\partial x}\left[\left(1+\frac{x}{R_m}\right)u\right]+\frac{1}{1+x/R_m}\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

Momentum

$$\frac{u\partial u}{\partial x}+\frac{v}{1+x/R_m}\frac{\partial u}{\partial y}+\frac{w\partial u}{\partial z}-\frac{v^2}{R_m+x}=-\frac{\partial p}{\partial x} \tag{2}$$
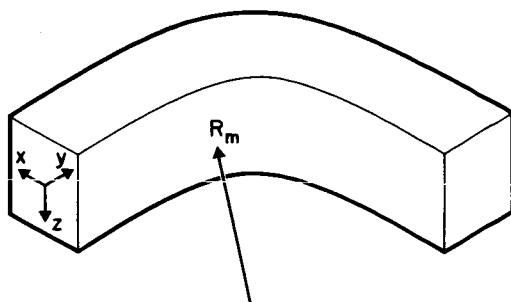
FIGURE 1.—*Geometry of the duct.*

$$\frac{u\partial v}{\partial x}+\frac{v}{1+x/R_m}\frac{\partial v}{\partial y}+\frac{w\partial v}{\partial z}-\frac{vu}{R_m+x}=-\frac{\partial p}{\partial y}\frac{1}{1+x/R_m} \qquad (3)$$

$$\frac{u\partial w}{\partial x}+\frac{v}{1+x/R_m}\frac{\partial w}{\partial y}+\frac{w\partial w}{\partial z}=-\frac{\partial p}{\partial z} \qquad (4)$$

These equations may be verified by considering a simple change of independent variables, $r=R_m+x$ and $\phi=y/R_m$, which yields the well-known inviscid fluid flow equations in cylindrical polar coordinates. The components u, v, and w are physical velocity components normal to the local coordinate surfaces, and the static pressure p is understood to include the specific volume $1/\rho$, which is constant. The term $1/R_m$ is zero outside the bend region, corresponding to infinite radius of curvature.

Manipulation of Equations

Following usual streamline curvature procedure, the u and w velocity components are replaced by new dependent variables λ and μ

where

$$u=\lambda v \qquad (5)$$

and

$$w=\mu v \qquad (6)$$

(Most authors use tan λ and tan μ, but this is not necessary for the present analysis.) The "in-plane" components u and w, which are expected to be relatively small in an enclosed duct, are expressed as fractions of the dominating velocity component v, normal to the planes of cross section. No approximation is here implied, but the transformations (5) and (6) are singular when v is zero, a condition which must be avoided. Using (5) and (6), equations (1) through (4) become

$$\frac{1}{1+x/R_m}\frac{\partial}{\partial x}\left[\left(1+\frac{x}{R_m}\right)\lambda v\right]+\frac{1}{1+x/R_m}\frac{\partial v}{\partial y}+\frac{\partial}{\partial z}\,(\mu v)=0 \tag{7}$$

$$\lambda v\,\frac{\partial}{\partial x}\,(\lambda v)+\frac{v}{1+x/R_m}\frac{\partial}{\partial y}\,(\lambda v)+\mu v\,\frac{\partial}{\partial z}\,(\lambda v)-\frac{v^2}{R_m+x}=-\frac{\partial p}{\partial x} \tag{8}$$

$$\lambda v\,\frac{\partial v}{\partial x}+\frac{v}{1+x/R_m}\frac{\partial v}{\partial y}+\mu v\,\frac{\partial v}{\partial z}+\frac{\lambda v^2}{R_m+x}=\frac{-1}{1+x/R_m}\frac{\partial p}{\partial y} \tag{9}$$

$$\lambda v\,\frac{\partial}{\partial x}\,(\mu v)+\frac{v}{1+x/R_m}\frac{\partial}{\partial y}\,(\mu v)+\mu v\,\frac{\partial}{\partial z}\,(\mu v)=-\frac{\partial p}{\partial z} \tag{10}$$

Bernoulli's equation is also derived from equations (8) through (10).

$$\lambda\,\frac{\partial P}{\partial x}+\frac{1}{1+x/R_m}\frac{\partial P}{\partial y}+\mu\,\frac{\partial P}{\partial z}=0 \tag{11}$$

where the total pressure

$$P=p+\tfrac{1}{2}v^2(1+\lambda^2+\mu^2) \tag{12}$$

The five equations (8) through (12) are not independent, as Bernoulli's equation is linearly dependent on the three momentum equations. One equation must be omitted, and (9) is selected since it is identical with (11) in the trivial case $\lambda=\mu=0$.

Still adhering to the two-variable streamline curvature method, the $\partial v/\partial y$ terms are eliminated from equations (8) and (10) with the aid of (7) to obtain

$$v^2\left[\frac{1}{1+x/R_m}\frac{\partial\lambda}{\partial y}+\mu\,\frac{\partial\lambda}{\partial z}-\frac{1+\lambda^2}{R_m+x}-\lambda\,\frac{\partial\mu}{\partial z}\right]=-\frac{\partial p}{\partial x} \tag{13}$$

and

$$v^2\left[\frac{1}{1+x/R_m}\frac{\partial\mu}{\partial y}+\lambda\,\frac{\partial\mu}{\partial x}-\frac{\lambda\mu}{R_m+x}-\mu\,\frac{\partial\lambda}{\partial x}\right]=-\frac{\partial p}{\partial z} \tag{14}$$

However, p is related to v^2 and P by (12). Using this equation in (13) and (14)

$$\tfrac{1}{2}(1+\lambda^2+\mu^2)\,\frac{\partial}{\partial x}\,(v^2)=\frac{\partial P}{\partial x}+v^2\left[\frac{1}{1+x/R_m}\frac{\partial\lambda}{\partial y}-\frac{1+\lambda^2}{R_m+x}\right.$$

$$\left.-\lambda\,\frac{\partial\lambda}{\partial x}-\mu\,\frac{\partial\mu}{\partial x}+\mu\,\frac{\partial\lambda}{\partial z}-\lambda\,\frac{\partial\mu}{\partial z}\right] \tag{15}$$

and

$$\tfrac{1}{2}(1+\lambda^2+\mu^2)\,\frac{\partial}{\partial z}\,(v^2)=\frac{\partial P}{\partial z}+v^2\left[\frac{1}{1+x/R_m}\,\frac{\partial \mu}{\partial y}-\frac{\lambda\mu}{R_m+x}\right.$$

$$\left.-\lambda\,\frac{\partial\lambda}{\partial z}-\mu\,\frac{\partial\mu}{\partial z}+\lambda\,\frac{\partial\mu}{\partial x}-\mu\,\frac{\partial\lambda}{\partial x}\right] \qquad (16)$$

There are now four equations, (7), (11), (15) and (16), for the four dependent variables λ, μ, v, and P. Boundary conditions are required to close the system.

Boundary Conditions

It is necessary to appeal to the physics of the problem to obtain the correct boundary conditions. Some of these are obvious: $\lambda=0$ at the walls given by $x=$ constant and $\mu=0$ at the walls given by $z=$ constant (the no-flow conditions). Upstream conditions are easily come by: straight shear flow where the static pressure is constant and either v or P is specified at the inlet cross section. However, the conditions downstream are not so evident, being complicated by the presence of secondary flow. Two different downstream boundary conditions have been tried, both of which are sufficient to close the system of equations and boundary conditions from a numerical or computational point of view. The first condition is $\partial\lambda/\partial y=\partial\mu/\partial y=0$; the second is $\partial v/\partial y=0$. The latter is a little more symmetric and converges faster, but both produce near-identical flow fields except over the last few computing planes. If, far downstream, there is a uniform swirling flow pattern, repeated at all subsequent planes of cross section, both boundary conditions are correct.

Method of Solution of the Equations

The extent to which the two-variable procedure may be followed has now been reached. Examination of the equations indicates the following:

(1) Bernoulli's equation (eq. (11)). Given values for λ and μ throughout the flow field, P may be calculated from the starting values at the inlet cross section.

(2) The momentum equations (eqs. (15) and (16)). Either of these may be integrated for v when λ, μ, and their derivatives are known.

(3) The continuity equation (eq. (7)). Assuming that v is given throughout the flow field, this equation may be integrated for λ if μ is known or for μ if λ is known.

These integrations will be for *linear, first-order, ordinary differential equations* with *nonconstant coefficients*. Bernoulli's equation is written

$DP/Dt = 0$ along a streamline to fit this classification. It is inconvenient, though, to have two equations, either of which may be solved explicitly for v, and only one equation to solve for both λ and μ. Two methods of solution have been tried.

First Method of Solution

A pattern similar to that proposed by Wu (ref. 4) was followed. Separate sets of two-dimensional solutions were sought, with an iterative procedure connecting them as shown in figure 2. This did not work. Alternating between one set of two-dimensional solutions and the other is not sufficient to produce convergence since neither solution "realizes" that it is not the same as the other. The information conveyed between the solutions is not sufficient to produce convergence.

Although Wu's proposals differ in that his two-dimensional solutions are calculated on S_1 and S_2 stream surfaces using two stream functions, the method follows the pattern suggested by Wu. A few variations on this method have also been tried, but without success. This suggests that for a method to have any chance of success it must "know" about the "error" or difference between separate two-dimensional solutions, $v^x - v^z$ for example, and act on this information until the error is reduced to zero.

Second Method of Solution

Let the result of integrating equation (15) in the x-direction be v^x, and the result of integrating equation (16) in the z-direction be v^z. The error $v^x - v^z$ is related to a static pressure difference by equation (12). Physically, this pressure difference will change the curvature of the streamlines, and thus λ and μ must be influenced by $v^x - v^z$. The best choice seems to be the replacement of the continuity equation (eq. (7)) by the two equations,

$$\frac{\partial}{\partial x}\left[\left(1+\frac{x}{R_m}\right)\lambda v^x\right] + \frac{\partial v^x}{\partial y} + \left(1+\frac{x}{R_m}\right)\frac{\partial}{\partial z}(\mu v^z) = \beta(v^x - v^z) \qquad (17)$$

and

$$\frac{\partial}{\partial x}\left[\left(1+\frac{x}{R_m}\right)\lambda v^x\right] + \frac{\partial v^z}{\partial y} + \left(1+\frac{x}{R_m}\right)\frac{\partial}{\partial z}(\mu v^z) = \beta(v^z - v^x) \qquad (18)$$

where β is a constant. In these equations, the right-hand sides represent source/sink terms and each reduces to the continuity equation (eq. (7)) when $v^x = v^z$. One additional equation and one additional unknown have been introduced and now (17) is integrated directly for λ, (18) for μ, (15) for v^x, (16) for v^z, and (11) for P. In (17) and (18), v^x and v^z are selected appropriately to make the boundary conditions for the velocity

integrations explicit. The essential feature of these equations is that when $v^x = v^z$ they reduce to the physically correct equations.
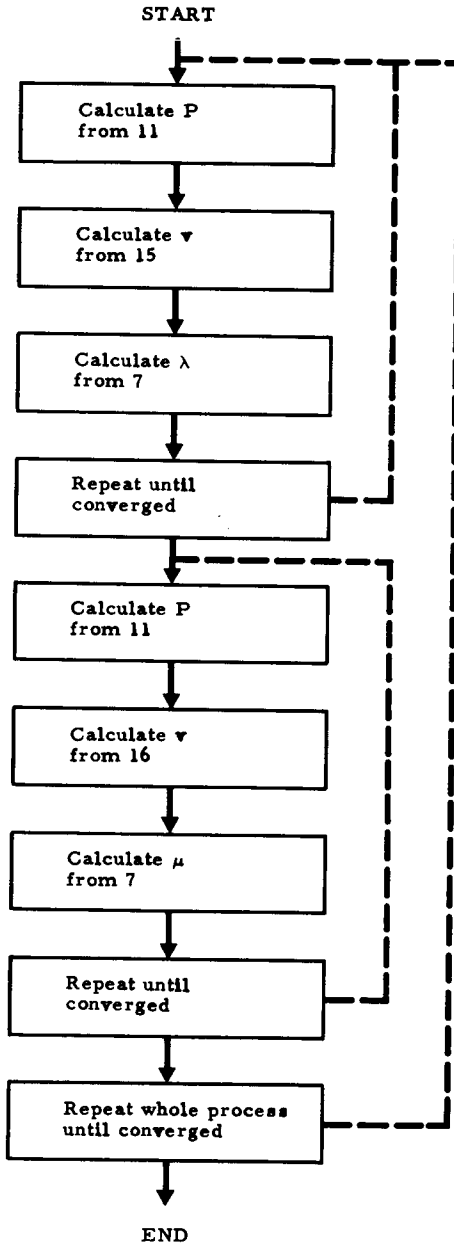
START

```
Calculate P
from 11

Calculate v
from 15

Calculate λ
from 7

Repeat until
converged

Calculate P
from 11

Calculate v
from 16

Calculate μ
from 7

Repeat until
converged

Repeat whole process
until converged
```

END

FIGURE 2.—*Iterative procedure for Method 1.*

Revised Boundary Conditions

Although the boundary conditions described are sufficient to determine a solution, they do not lend themselves to an easy integration of the equations in their present form. First, (17) and (18) are first-order equations; each requires only one boundary condition for λ and μ and there are two for each. Second, (15) and (16) also require one boundary condition each for v^x and v^z and there are none. This matter is rectified by the requirements that two boundary conditions may be satisfied by each of the first-order pseudo-continuity equations (eqs. (17) and (18)). These requirements are found as follows. Equations (17) and (18) are integrated, first with respect to x from $x = -\frac{1}{2}X$ across the duct to $x = \frac{1}{2}X$ and then with respect to y from $y = 0$ at the inlet to some station y', to yield

$$\int_{-1/2X}^{1/2X} v^x \, dx + \int_0^{y'} \int_{-1/2X}^{1/2X} \left(1 + \frac{x}{R_m}\right) \frac{\partial}{\partial z} (\mu v^z) \, dx \, dy = 0 \qquad (19)$$

and

$$\int_{-1/2X}^{1/2X} v^z \, dx + \int_0^{y'} \int_{-1/2X}^{1/2X} \left(1 + \frac{x}{R_m}\right) \frac{\partial}{\partial z} (\mu v^z) \, dx \, dy = 0 \qquad (20)$$

where the $\beta(v^x - v^z)$ terms have been omitted, and the boundary conditions $\lambda = 0$ at $x = \pm\frac{1}{2}X$ have been incorporated. Alternatively, a repeat condition, $\lambda(-\frac{1}{2}X,y,z) = \lambda(\frac{1}{2}X,y,z)$ and $v^x(-\frac{1}{2}X,y,z) = v^x(\frac{1}{2}X,y,z)$, yields the same results. The requirement (19) is used as a boundary condition for equation (15). The procedure is repeated with the roles of x and z interchanged to obtain a similar requirement for the other pseudo-continuity equation and a boundary condition for (16).

In general, sets of coupled partial differential equations cannot be put into explicit form, so it is necessary to select one variable in an equation and guess or assume values for all others. Each of the variables must take its turn as the unknown in one of the equations. When all variables have been found, the equations are solved again and this iterative procedure is continued until convergence is obtained.

Iterative Scheme

Figure 3 shows the iterative scheme. Each block represents the integration of the appropriate equation for the unknown variable throughout the entire flow field. In each integration, the most up-to-date values are used for all other variables. This scheme is chosen for its simplicity and because it also simplifies the boundary conditions of (19) and (20). Between the calculation of μ and the next calculation of λ, v^z remains unchanged; hence, on subtracting equations (19) and (20)

$$\int_{-1/2X}^{1/2X} v^x\, dx = \int_{-1/2X}^{1/2X} v^z\, dx \tag{21}$$

This is the boundary condition used for v^x. Similarly, the boundary condition for v^z is

$$\int_{-1/2Z}^{1/2Z} v^z\, dz = \int_{-1/2Z}^{1/2Z} v^x\, dz \tag{22}$$

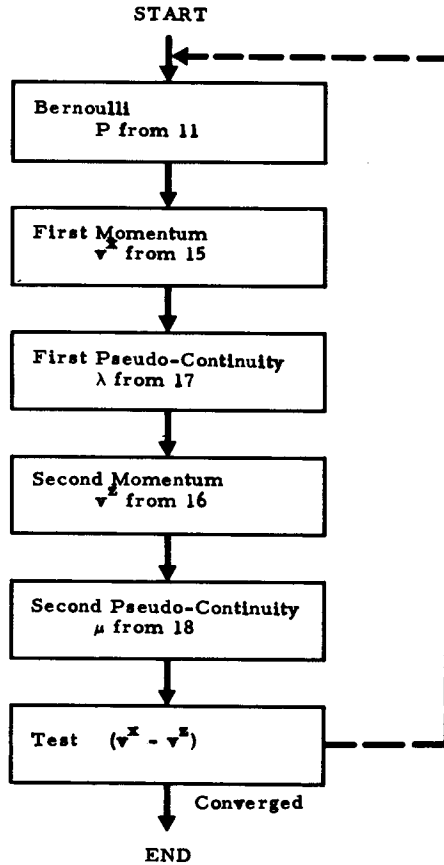The right-hand sides of these equations are known from previous calculations.

FIGURE 3.—*Iterative procedure for Method 2.*

Uniqueness

These boundary conditions for the velocity are not of a familiar type, and the existence of a unique solution must be explored. Equation (15) is like the equation

$$\frac{d}{dx} v^2 = v^2 g(x) + f(x) \tag{23}$$

A solution is sought, subject to the condition

$$\int_0^1 v \, dx = Q \tag{24}$$

If two solutions, v_1 and v_2, exist then

$$\frac{d}{dx} (v_1{}^2 - v_2{}^2) = (v_1{}^2 - v_2{}^2) \, g(x) \tag{25}$$

subject to

$$\int_0^1 (v_1 - v_2) = 0 \tag{26}$$

The solution of (25) is

$$v_1{}^2 - v_2{}^2 = E \exp \left(\int_0^x g(s) \, ds \right) \tag{27}$$

where E is determined by (26) as follows

$$E \int_0^1 \frac{\exp \left(\displaystyle\int_0^x g(s) \, ds \right)}{v_1 + v_2} \, dx = 0 \tag{28}$$

The further restriction that $v \geq 0$ is necessary for uniqueness. Now, the integrand of (28) is always positive; whence $E = 0$ and $v_1 = v_2$.

Stability

Theoretically, it is only possible to perform a stability analysis for trivial flows where the total pressure is constant, but the resulting criterion is found to have general application. A straight flow without shear is considered, where $v^x = v^z = V$ and $\lambda = \mu = 0$ is the required solution. Small perturbations from this trivial solution are examined and the following stability criterion is obtained

$$\left\{ 1 - r \left[1 + 2 \left(0.63 \frac{X}{\Delta y} \right)^2 \right] \right\}^2 + \left[\frac{3}{\pi} \left(0.63 \frac{X}{\Delta y} \right)^2 r\beta \right]^2 < 1 \tag{29}$$

where X is the duct width, Δy is the grid spacing in the y direction, and r is the relaxation factor on the velocity.

A computer program has been written in FORTRAN IV to solve the problem as outlined. An experiment with a 5-inch by 5-inch 90° bend on a 15-inch mean radius, where the velocity varied across the duct from about 30 feet/second to 80 feet/second at inlet, was simulated. The results, in the form of P_0 contours, are presented for comparison with experiment. The small discrepancies can be accounted for by the presence and separation of the boundary layer. Otherwise, an adequate prediction of the flow is obtained.

Comparison With Experiment

Numerical computations have been carried out for the experiments of Joy (ref. 3) for ducts of rectangular cross section bending through 90° and subject to substantial inlet total head variations across the duct. A comparison has also been made with ducts of circular cross section (Eichenburger reported in ref. 5). The theory presented in this paper is directly applicable to the rectangular duct but requires modification to the circular geometry although the equations are of a similar form.

In figure 4 the total pressure contours at inlet to the duct are presented. In figures 5, 6, and 7 the computed contours are compared with experiment at three stations down the duct.

Station 1	6 inches upstream of the bend
Station 2	30° of turning
Station 3	60° of turning
Station 4	90° of turning

The duct is 5 inches by 10 inches in cross section with a mean radius of 15 inches. For consistency with Joy, the total pressure contours are labelled as velocity contours computed on the assumption of constant static pressure. Similar comparisons are shown for the circular-cross-sectioned duct in figures 8, 9, and 10. The duct is of 6 inches diameter and 30 inches mean radius.

In general, the experimental contours are predicted by the theory. For the circular duct, the agreement is particularly good except in the immediate vicinity of the wall where the viscous forces in the boundary layer are dominant, causing reductions in total pressure. The discrepancies in predictions for the rectangular duct near the inside of the bend are probably due to the occurrence of separation of the boundary layer near Station 3.

A measure of the convergence of the numerical procedure for the rectangular bend is presented in figure 11 showing good convergence after 58 cycles. This procedure took 14 minutes on an IBM 360/65.

LIST OF SYMBOLS

p Static pressure

P_0 Total or stagnation pressure

u, v, w Velocity components

x, y, z Coordinates

β Strength of source/sink distribution

λ, μ Flow directions as defined

ρ Density
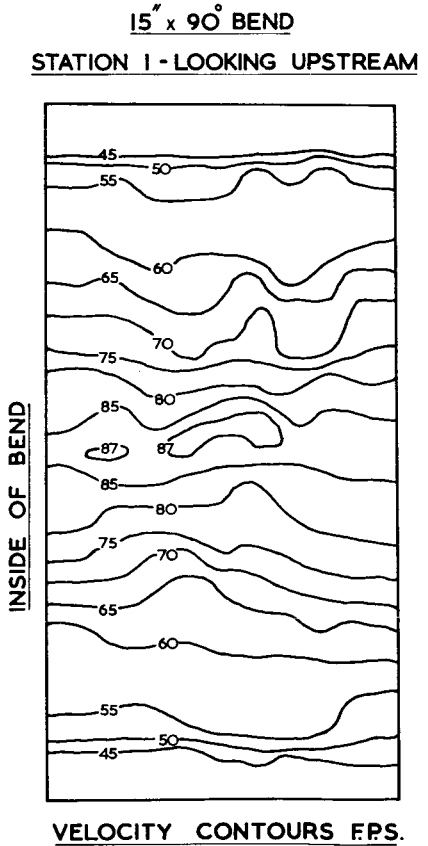
FIGURE 4.—*Velocity contours in rectangular duct.*

15″ x 90° BEND

STATION 1 - LOOKING UPSTREAM
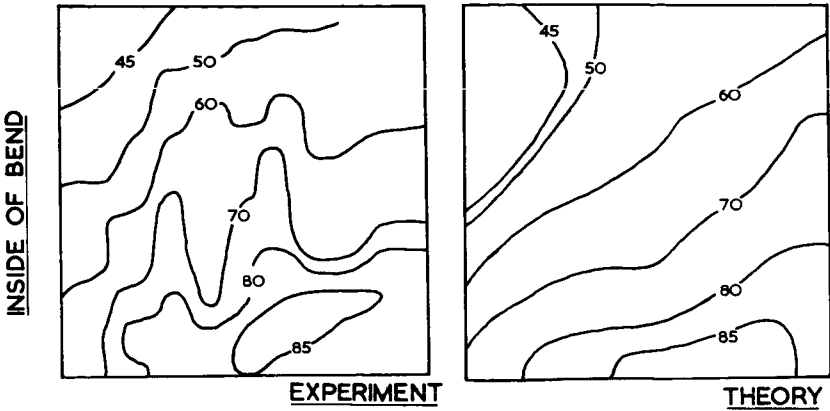
VELOCITY CONTOURS F.P.S.

15″ x 90° BEND

STATION 2 - LOOKING UPSTREAM

FIGURE 5.—*Velocity contours in rectangular duct; comparison between theory and experiment.*
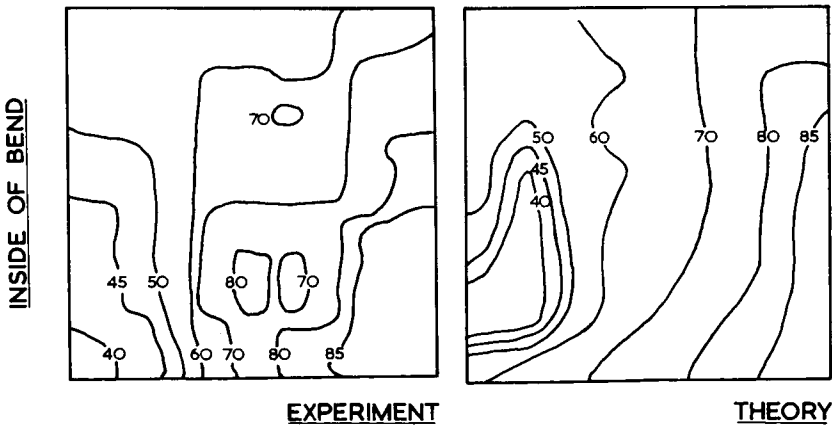
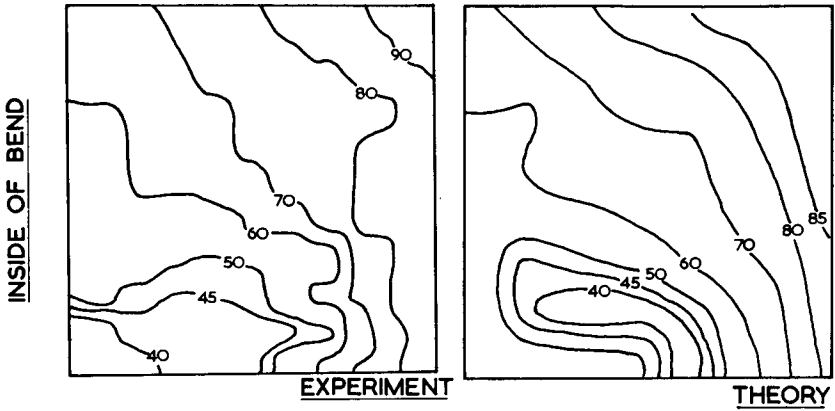15′ x 90° BEND

STATION 3 - LOOKING UPSTREAM

FIGURE 6.—*Velocity contours in rectangular duct; comparison between theory and experiment.*

15″ x 90° BEND
STATION 4 - LOOKING UPSTREAM

EXPERIMENT   THEORY

FIGURE 7.—*Velocity contours in rectangular duct; comparison between theory and experiment.*

Total pressure Contours, due to EICHENBURGER, in a 6-inch–diameter circular duct, bent on a mean radius of 30 inches.
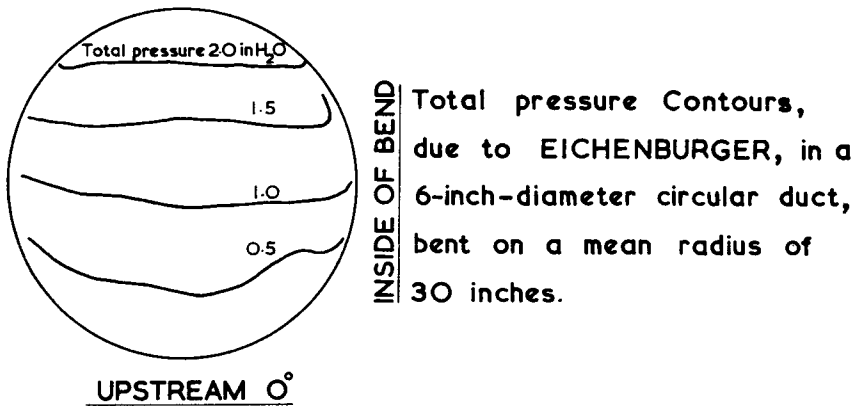
UPSTREAM 0°

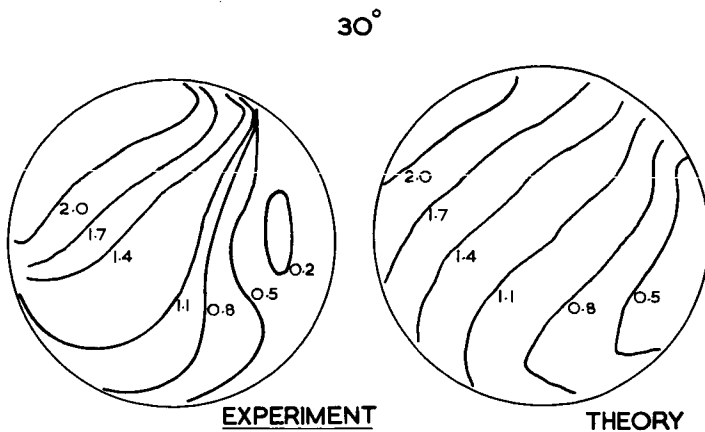FIGURE 8.—*Total pressure contours in circular duct.*

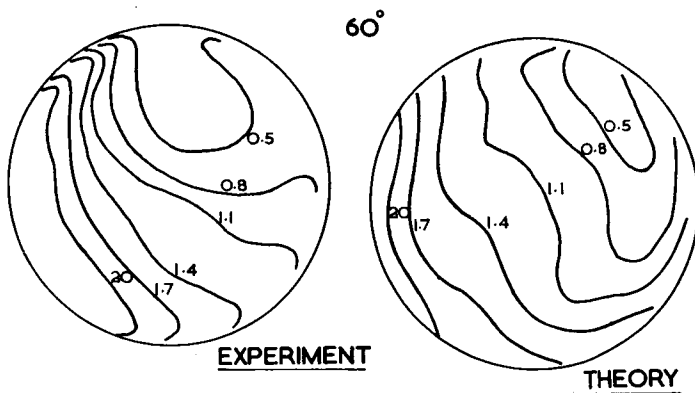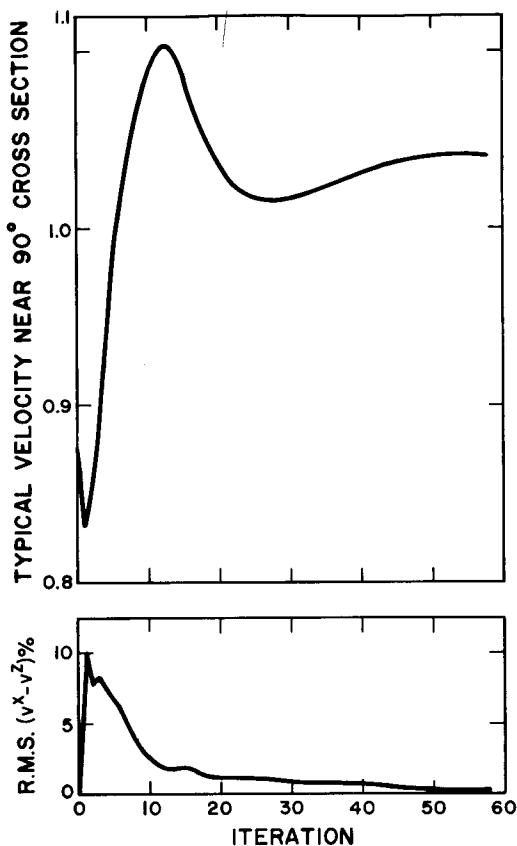FIGURE 9.—*Total pressure contours in circular duct; comparison between theory and experiment.*

FIGURE 10.—*Total pressure contours in circular duct; comparison between theory and experiment.*

FIGURE 11.—*Convergence of computing scheme.*

REFERENCES

1. SMITH, L. H., The Radial-Equilibrium Equation of Turbomachinery. *Trans. ASME, J. Eng. Power*, January 1966.
2. SILVESTER, M. E., AND R. HETHERINGTON, Three-Dimensional Compressible Flow Through Axial Flow Turbomachines. *Numerical Analysis: An Introduction*, J. Walsh, ed., Academic Press, 1966.
3. JOY, W., *Experimental Investigation of Shear Flow in Rectangular Bends*. M.S. thesis, M.I.T., 1950.
4. WU, C. H., *A General Theory of Three-Dimensional Flow in Subsonic Turbomachines of Axial, Radial and Mixed Flow Types*. NACA TN 2604, 1952.
5 HAWTHORNE, W. R., Secondary Circulation in Fluid Flow. *Proc. Roy. Soc. London*, Vol. 206, 1951, pp. 374–387.

DISCUSSION

H. MARSH (Cambridge University): The authors have successfully solved a major problem in the calculation of internal flows, namely the three-dimensional flow in a duct. Until recently, it has not been possible to solve this problem due to the lack of adequate computing facilities. It would be interesting to know the computer storage requirements for this program.

Two methods of solution have been examined, but only the successful second method is described in detail. The first method is similar to that proposed by Wu (ref. 4). Until now, this has been considered a viable method for calculating the full three-dimensional flow field. The authors have investigated this technique and have found that they could not obtain convergence. This negative result is extremely important and it deserves a more detailed discussion. Smith (ref. D–1) has described the methods which are available for calculating the two separate two-dimensional flow fields. Until now, it has been assumed that by alternating between the two solutions, the full three-dimensional flow field might be calculated. It would be helpful if the authors would give more details of the basis for their conclusion that the first method of solution does not work.

In the second method of solution, the error $v^x - v^z$ is related to a static pressure difference, but it is not clear why this term should have any physical significance. The replacement of the continuity equation by two equations with source/sink terms is a numerical technique which is used in order to obtain a convergent solution. It is therefore unlikely that the intermediate values of the error $v^x - v^z$ have any physical meaning.

In the derivation of the boundary conditions, the authors have omitted the source/sink term but have not discussed this point. Perhaps they would outline their argument for neglecting these terms. It is possible to argue that any convenient boundary condition can be used, provided that it approaches the true boundary condition as the solution converges.

This is a major contribution to methods of flow calculation and the authors must be congratulated on their presentation in this paper. If this work can be extended to include compressibility, then it would provide a single comprehensive technique for calculating inviscid three-dimensional duct flows.

W. R. HAWTHORNE (Cambridge University): I agree with what Mr. Stuart says. I think the work of Rowe (ref. D–2) should be referred

to in this excellent paper which seems to me to be a substantial break-through on three-dimensional and secondary flow calculations. But I also want to raise the question of getting the right downstream boundary conditions. It isn't clear from what the author was saying how and where the boundary conditions were established. Were they established at 30°, 40°, or down the bend? In one case, he showed a section 1 foot downstream from the bend. How far downstream could you go before you get the right downstream conditions?

STUART AND HETHERINGTON (authors): In reply to H. Marsh, we would like to state the following:

(1)   The computations were performed on an IBM 360/65 and re-quired between 120 000 and 180 000 bytes (depending on whether the program was overlayed or not). Typical execution times were 10 to 14 minutes CPU.

(2)   The conclusion that simple alternation between two separate two-dimensional solutions does not produce the required three-dimen-sional flow field is based on our failure to make such methods produce identical fields for the axial velocity from both two-dimensional solutions, in the absence of the $\beta(v^x - v^z)$ terms in equations (17) and (18). For test computations with $\beta = 0$, convergence has not been obtained, and over a considerable portion of the flow field (about half), near the start of the bend, the secondary flow turned in the wrong direction. This even propagated upstream where no secondary flow is to be expected.

(3)   We agree that physically no terms $\beta(v^x - v^z)$ exist. The argument for the use of such a term as a numerical device is as follows: Physically, fluid will tend to flow from *high* pressure toward *low* pressure regions until the pressure gradient is balanced by acceleration. Now, an imbalance may exist between the pressure gradient of one two-dimensional solution and the acceleration or curvature terms of the other (since these balance their own pressure gradient, which is not necessarily identical to that of the first solution). This imbalance between respective pressure gradients is related to $v^x - v^z$, which term or "error" is used to change λ and μ ac-cordingly in the psendo-continuity equations (17) and (18).

(4)   The neglect of some terms $\beta(v^x - v^z)$ in the derivation of the revised boundary conditions is justified as follows: In practice, the pseudo-continuity equations (17) and (18) are solved with additional source and sink terms $S^x(y,z)$ and $S^z(x,y)$ to allow for the effect of the terms omitted. Now, it must be shown that these terms vanish when convergence is obtained. If the two solutions v^x and v^z converge (remember that these are both the axial velocity, and *not* components in the x and z directions), then subtracting equation (17) from equation (18) yields, at most

$$S^x(y,z) = S^z(x,y) = f(y)$$

Integrating equation (17) or (18) over a plane of cross section yields

$$f(y) \int dx\, dz = \frac{dQ}{dy}$$

where Q is the mass flow. Since Q is constant, $f(y) = 0 = S^x = S^z$.

(5)   In its present form, the method has been extended to compressible flows (Mach number less than 0.98) and the computer program can handle the following problems:

(a)   Bent rectangular ducts
(b)   Bent circular ducts, including S-bends
(c)   Three-dimensional flow through a turbomachinery blade passage specified by random points, including rotors.

Arbitrary values for total temperature, total pressure, and static pressure, varying across the inlet section may be specified as input data to the program. Future work in the Department of Mathematics at the University of Aston will attempt to extend the method further, to include viscous and turbulent flows.

As regards the right boundary conditions mentioned by Sir William Hawthorne, we would like to offer the following reply.

The computing mesh is usually extended two or three planes further downstream of the region of interest (there being expense involved in using too many), but from the calculations we have done, the condition downstream does not have much effect two or three planes upstream (i.e., about three pipe diameters) of where the downstream boundary condition is applied. The flow fields for the two different downstream boundary conditions described were within 0.1 percent of being identical two planes upstream of where the conditions were established.

REFERENCES

D-1. Smith, D. J. L., Computer Solutions of Wu's Equations for the Compressible Flow Through Turbomachines. In *Fluid Mechanics and Design of Turbomachinery*, Int. Symp., Penn. State U., September 1970. NASA SP-304, 1973.
D-2. Rowe, M., Measurements and Computation of Flow in Pipe Bends. *J. Fluid Mech.*, Vol. 43, Part 4, 1970.