C.P. No. 1384

C.P. No. 1384

PROCUREMENT EXECUTIVE, MINISTRY OF DEFENCE

AERONAUTICAL RESEARCH COUNCIL

CURRENT PAPERS

On Three-Dimensional Flow in Centrifugal Impellers

by J. Moore, J. G. Moore and M. W. Johnson

University Engineering Department, Cambridge

LONDON: HER MAJESTY'S STATIONERY OFFICE

1977

£2.50 net

C.P. No.1384*
February 1977

ON THREE-DIMENSIONAL FLOW IN CENTRIFUGAL IMPELLERS

- by -
J. Moore, J.G. Moore and M.W. Johnson

SUMMARY

Evidence of non-uniform flow at the exit of centrifugal impeller passages is discussed and a Rossby number $W/\omega R_n$ which governs the stable location of wake flow in the exit plane of an impeller is presented. In impeller6 with large Rossby numbers the stable location of the wake is on the shroud wall; wakes on the suction side wall are stable in impellers with low Rossby numbers.

The ability of a marching-integration procedure to compute a three-dimensional rotating flow with large secondary velocities leading to the formation of a wake is demonstrated. A possible approach to the calculation of three-dimensional impeller flow is suggested.

*Replaces A.R.C.37 194.

contents

Introduction

The flow in a centrifugal pump or compressor is a complex viscous three-dimensional flow in which the effects of rotation and curvature promote the development of non-unlform flow. Thus, while the Inlet flow may have uniform properties and the outlet flow may also be approximately unlform, the work Input and the efficiency of the machine are governed by the non-uniform flow which la delivered by the Impeller to the diffuser.

It Is therefore necessary to have a detailed understanding of the development of the three-dimensional flow In the rotating Impeller and Its subsequent development in the stationary diffuser, if one is to calculate and predict the performance of centrifugal machines.

Non-uniform flow at impeller exit

Non-uniform velocity profiles at the exit of centrifugal impeller passages have been observed for over 50 years. In fact, even before Prandtl'a famous paper on the boundary layer, there was a suggestion In 1902 by J.A. Smith (1) (reported by Gibson (2)) that under certain conditions the flow tends to leave the leading face (suction side) of a centrifugal pump Impeller passage. The flow then passes along the trailing face (pressure side) and there Is a region of "dead" fluid on the leading face. This work and the classic work of Carrard (3,4) In 1923 was concerned with flow in two-dimensional radial-flow impellers. Carrard modelled the impeller flow as "le Jet et la zone neutre" in what appears to be the first use of the "jet-wake" flow model. However, the concept of a neutral zone or wake was not new in 1923 and Carrard states that "the hypothesis of the existence of a neutral zone is not new in Itself: it has long been supposed that the channels of (centrifugal) wheels must not work with a uniformly filled cross-section."

Carrard made pressure measurements in centrifugal impellers which showed wake flows associated with separation at the leading edge of the Impeller blades on both the suction side at low flow rates and the pressure side at high flow rates. But he also showed that at certain intermediate flow rates the neutral zone started along the suction surface away from the Impeller inlet. He calculated the velocity distribution in the jet using Flugel's (5,6) streamline curvature method and he assumed a zero velocity for the fluid in the wake. Thus, he was able to obtain remarkably good agreement between the measured and calculated velocity distributions and pressure-rise characteristics of his impellers over their whole flow range.

This two-dimensional picture of a jet-wake flow was used to explain observed impeller flow for many years (7-10). Indeed, in the slow speed rig used by Cheshire (11) to study impeller flow during the development of the early jet engines, it was found that there was "an intense region of high radial velocity at the driving face, the velocity diminishing rapidly towards the trailing face,* the total head loss Increasing rapidly in this direction also." Cheshire concluded that "this state can only be caused by total head loss at the trailing face before entering the radial portion, and this is of such magnitude as to represent a complete breakaway of the flow," and his analysis of flow incidence angles at the impeller inlet supported this conclusion.

Secondary Flow

Evidence of a jet-wake flow that was not uniform from hub to shroud was presented by Kearton (8) In 1933. But the study of the three-dimensional character of impeller flows did not begin until the 1950's when several papers (12-15) were published on the generation of a streamwise component of vortlcity In flows in curved and rotating passages. These studies considered the inviscid development of flow containing vortlcity initially oriented perpendicular to the flow direction. They gave a quantitative understanding of the development of secondary flow in regions where viscosity

*N.B. Cheshire (11) refers to the trailing face (auction side) of the impeller blade while Gibson (2) refers to the trailing face (pressure side) of the impeller passage.

wae unimportant and a qualitative picture of **secondary** flow elsewhere. Thus, based on the **results** of **his** study of **incompressible** flow **A.G. Smith** (15) preeented a **qualitative** dlscueslon of the secondary **flows** In the Impeller of a centrifugal **compressor.**

The equations governing the development of the **streamwise** vortlclty along a relative streamline which was derived by Smith (15) has recently been preeented **in** the following simple form by Hawthorne (16);

$$\frac{\partial}{\partial s}\left(\frac{\Omega_s}{W}\right) = \frac{2}{\varrho W^2}\left(\frac{1}{R_n}\frac{\partial p^*}{\partial b} + \frac{\omega}{W}\frac{\partial p^*}{\partial z}\right) \qquad (1)$$

Here $p^* = p + \frac{1}{2}\varrho(W^2 - \omega^2 r^2)$ **is** the rotary stagnation **pressure,** W Is the fluid velocity relative **to the** rotor and ωr Is the rotor tangential velocity; and the equation **is** expressed In atreamllne coordinates where **s** Is the streamwlae direction, n the normal dlrectlon and b the blnormal direction. The two **terms** contributing to the generation of the **streamwise** vortlclty Ω_s are due to curvature of the streamlines **with** radlue R_n and to rotation with angular velocity ω about the axle of rotation **z.** They are associated **with** gradienta of rotary stagnation pressure In the blnormal and axial directions, respectively.

Smith (15) considered the generation of etreamwlee vortlcity In the axial inducer of the Impeller where the relative tangential velocity of the Inlet flow la reduced and in the radial section near the impeller outlet. In the inducer he noted that boundary layers on the hub and shroud walls of the inlet duct would produce radial gradients of p^* which would combine with the tangentially oriented radius of curvature to produce streamwise vorticity. This vorticity would convect fluid with low p^* towards the suction side of the passage along both the hub and shroud walls. In the radial section of the Impeller boundary layers on the hub and shroud walls produce gradients of p^* in the axial direction which would combine with the rotation to produce more streamwise vorticity. Again this vorticlty would convect fluid with low p^* towards the suction side.

Smith noted that Cheshire (11) had found a region of fluid with low p^* on the suction side of his impeller passage and he made two interesting observations on this fact. Firstly, he observed that if all the fluid with low p^* were located on the auction side in the radial part of the impeller then there could be no further generation of streamwiee vorticity. In fact, the condition that p^* becomes smaller as the suction side of the radial channel is approached is a stable one. Secondly, the two processes for generating streamwise vortlcity which he described will result in a tendency for the frictional boundary layer generated on the channel walls to be swept to the suction side of the channel.

Actually, there is a third and possibly more important process for generating streamwise vorticlty and this is associated with the curvature from the axial to the radial directions and with tangential gradients of p^*. Thus, this third process will Influence boundary layer fluid on the suction and pressure sides of the impeller passage and It will generate streamwise vortlcity such that this fluid will tend to migrate towards the shroud wall. In two dimensional Impellers this effect will be negligible and fluid wlth low p^* will tend to settle uniformly over the suction wall as observed by Carrard (3). In three dimensional Impellers, however, It can be a large effect and the fluid with low p^* Is often found either in the shroud- suction-side corner or completely on the shroud wall.

Observations of Three-Dimensional Flow in Impellers

Substantial evidence for the existence of three dimensional flow in centrifugal impellers has been obtained since 1950, and the flows observed are in general agreement with the secondary flow picture outlined above.

A major programme of research to investigate the flow In the Impellers of centrifugal compressors was conducted by NACA and the results of these Investigations were summarised by Hamrick (17) In 1955. Hamrick presents the results of detailed measurements (18) of static pressure, stagnation pressure and velocity In an essentially two-dimensional radial-flow Impeller. This was tested over a range of flow rates at a constant speed and the results show the effects of off-design flow angles at the impeller Inlet, secondary flows and flow through the tip clearance between the blades and the stationary shroud wall.

At approximately one-half the design flow rate there was separation at the inlet on the suction side of the blades near the shroud. The fluid with low p^* associated with this separated flow passed down the channel and flowed from the shroud-suction-side corner until it was distributed approximately uniformly over the suction side wall - the stable location in the two-dlmensional Impeller. Towards the exit of the impeller the hub-nhroud height of the impeller passages became small compared with the blade-to-blade wldth of the passage (1:5.5). Then the flow through the tip-clearance (-3.3 %) had a significant effect on the flow in the Impeller passages; It caused secondary flows which tended to extend the region of low p^* fluid onto the shroud wall.

At approximately 1.5 times the design flow rate there was separation at the Inlet on the pressure side of the blades again near the shroud. The fluid with low p^* was convected by secondary flow across the shroud wall to a position near the suction side where It remained, showing no tendency to move onto the suction slde wall. This suggests that the Influence of the curvature from the axial to the radial direction on the generation of streamwise vorticity is relatively larger than the influence of rotatlon. The relative importance of curvature and rotation as influences on secondary flow may be estimated using the Rossby number $\frac{W}{\omega R_n}$ *

---

* Hawthorne (16) and Lakshminarayana and Horlock (19) consider the relative Importance of rotatfon and curvature in the plane of rotation, such as may be found with forward or backward curved Impeller blades. They use a Rossby number with the corresponding radius of curvature and they comment that in centrifugal compressors their Rossby number "is unlikely to be greater than 1/4, so that rotation induced secondary vorticity dominates."

(see equation (1)). Since the rotation rate ω and the radius of curvature R_n of the flow from the axial to radial directions were fixed for the constant speed teats on the NACA impeller, the influence of curvature will become more important at high flow rates. For the teat results presented by Hamrlck, the values of the Roeeby number may be estimated to be 1/4 at the design flow rate, 3/8 at 1.5 times design flow and 1/8 at 0.5 times design flow. These figures suggest that rotation was important at all flow rates, that rotation was dominant at low flow rates and that curvature and rotation were of approximately equal importance at the high flow rates In the NACA tests.

At the design flow rate the loss of stagnation pressure at the inlet was relatively small as the flow did not separate from the blade surfaces. Thus the secondary flows were associated with boundary layer growth. However, as the outlet cross-sectional area of the impeller passage was approximately equal to the Inlet throat area, the changes in relative velocity were small and the boundary layers did not thlcken appreciably. Flufd with low p^* accumulated on the suction side wall due to the action of secondary flows, but the stagnation pressure losses associated with this fluid were small compared wlth the losses caused near the shroud wall by flow through the tip-clearance.

It may be concluded from Hamrick's results that the flow distribution at the exit of a centrifugal impeller can be strongly influenced by secondary flow. The secondary flow is caused by the curvature and rotation of the impeller passage. The final location of low stagnation pressure fluid in the cross-sectional plane at the exit of the Impeller depends on the relative magnitude of the two effects and on how far the generation of streamwise vorticity has progressed. Even in the nearly two-dimensional Impeller used by Hamrick the effects of curvature were significant at high flow rates and the fluid with law stagnation pressure moved towards the shroud wall.

An impeller In which the effects of curvature are even more important has been tested recently by Eckardt (20). At the test condition of 14000 RPM and a mass flow rate of 5.31 kg/s the value of the Rossby number $W/\omega R_n$ for Eckardt's impeller is approximately 1.0. An extrapolation of' Hamrick's results suggests that fluid with low stagnation pressure will accumulate an the shroud wall and this is exactly what Eckardt observed. Eckardt comments In answer to discussion of his paper that "the circumferential position of it8 (the wake's) cure, In the presented case at $y/t = 0.65$, shifts between $y/t = 0.5 - 0.8$, depending on mass flow and speed." Although he offers no further comment, It is clear from the present argument that a stable wake location closer to the suction side ($y/t = 0.8$) is expected at high speed and low flow, and a stable wake location towards the middle of the shroud wall is expected at low speed and high flow.

Influence of Impeller Exit Flow on Diffuser Performance

A question which naturally arises at this point is where in the impeller exit plane would the designer wish to have the low stagnation preeaure fluid if he had the choice ? Indeed, how is the performance of the diffuser influenced by the location ?

Much attention has been concentrated on the subsequent development of jet-wake flow In vaneless dlffueers. Several workers (21-23) have considered the case of jet and wake flows which are uniform between the hub and shroud walls. But as we have seen such flow is associated with a small Rossby number, and even the nearly two-dimensional impeller of Hamrick (18) only produced axially uniform flow at low flow rates. Thus, It is a special case and the work of Ellis (24) and Rebernik(25) has shown that this is only one of many different exit flow distributions and it leads to only one of many flows in the vaneless diffuser. Furthermore, as the Rossby number changes with flow rate, the Impeller exit flow distribution will also change unless the Rossby number is very large or very small. The designer needs to know what flow to produce at the impeller exit and how to produce It.

Wake Flow or Back-Flow ?

The accumulated low stagnation pressure fluid observed by Hamrick and Eckardt had a significant radial velocity even at the exit of the impeller. In the case of Hamrick's impeller this was due to the relatively constant area of the passages and even when the blade unloaded on the suction side the fluid had sufficient momentum to continue down the channel. In Eckcrdt's impeller the low momentum fluid was away from the suction side and so It did not experience the unloading of the blades. In both cases this fluid is a significant fraction of the total mass-flow rate and Eckardt states that It is about 15 % of his Impeller flow. In both cases also this wake flow exhibits no reverse flow. It is not a separated flow In the sense of two-dimensional boundary layer separation; It is instead an accumulation of low momentum fluid which has flowed towards the low pressure regions of the channel by the action of secondary flows. These observations are of great Importance to the designer for they Indicate that he may be able to calculate the wake flow as well as the jet flow by marching-integration methods, starting at the impeller Inlet and marching to the exit.

Requirements for a Calculation Procedure

It is apparent from the discussion above that the calculation of the performance of centrifugal machines will be complex in all but the simplest cases. Carrard was able to calculate the pressure rise in his two-dimensional Impellers using a simple two-dlmenelonal Jet-wake model and considering only the flow In the rotor. But two-dimensional flow is a special case and practical machlnes wlth an axial Inlet and a radial outlet exhibit three-dlmsnelonal flow with secondary flow velocities that can be locally large compared with the through flow velocity. An extension of Carrard's ideas is required and this must Include a calculation of the three-dimensional flow including the Jet and the wake flow and allowing the locatlon of the wake to be governed by secondary flow. It must also include an analysis of the subsequent development of the non-uniform Impeller exit flow as It passes through a vaneless diffuser and then poseibly through a vaned diffuser as well.

The source8 of fluid with low etagnation pressure which accumulate as a wake flow are

1/ boundary layers on the walls of the impeller channels,

2/ separated flow at the Impeller inlet,

3/ boundary layer flow and regions of low stagnation pressure in the inlet flow, 4/ flow through the tip-clearance between the impeller blades and the stationary shroud wall.

A calculation procedure for impeller flow should Include each of these sources and a method of calculating their convection towards the wake. The convection is influenced by curvature In the inducer and in the axial to the radial bend and by rotation. It is clear that only a general calculation scheme for three-dimensional flow will handle all these effects simultaneously.

Moore (26) has shown that such calculations can be made for a relatively simple flow in the rotating radial-flow passage shown in Figure 1. In Moore's flow the sources of low stagnation pressure fluid were the boundary layers and the inlet flow. But his simple analysis included only the effects of the boundary layers and neglected secondary flow in the potential flow which was assumed to have uniform stagnation pressure. Nevertheless, he was able to calculate the development of the boundary layers on all four walls of the rectangular passage and he showed that secondary flows developing on the hub and shroud walls were of sufficient magnitude to convect low-momentum boundary layer fluid to a wake on the suction aide of the passage. Moore's calculation procedure was specifically intended for his simple geometry. The potential flow analysis applied to simple two-dimensional radial flow and the boundary layer analysis was specific to the walls of his rectangular passage. The procedure can not be easily extended to more complex geometries. However, It contains a calculation of the whole flow and it allows the migration of boundary layer fluid towards a wake. A method with more generality which includes these features is necessary for Impeller flow calculations.

A Marching-Integration Calculation Procedure for Impeller Flows

A marching integration procedure for the calculation of three-dimensional parabolic flows has been suggested by Patankar and Spaldlng (27). This may be extended and combined with the results of an inviscid streamline-curvature calculation to give an economical procedure for calculating three-dlmenelonal viscous flow throughout an Impeller passage.

The procedure hinges on the simultaneous solution of the continuity and momentum equations In finite difference form. These equations for steady flow may be written In vector form as

$$\nabla \cdot (\rho \underline{V}) = 0$$

and

$$(\underline{V} \cdot \nabla)\, \underline{V} = \underline{X} - \frac{1}{\rho}\, \nabla \cdot \Pi \,,$$

where X Is the body force vector and Π Is the stress tensor (28).

For a marching Integration to be possible we muet choose the direction of marching such that the velocity always has a positive component in this direction. Since impeller geometries vary wldely we consider the flow In general orthogonal coordinates s_1, s_2, s_3, with scaling factors h_1, h_2, h_3 and velocity components u_1, u_2 and u_3. The marching Integration proceeds In the s_1 direction, which closely follows the direction of bulk flow.

Linearization of the momentum equation.

To Integrate the momentum equation over a forward step we first divide the equation Into two parts; the flrot part containing unknown quantities and the second containing quantities which may be calculated from upstream values and are thus known. Also we linearize the equations with respect to the unknown velocity components. We then combine the continuity and momentum equations to give momentum equations which can be used to calculate the three unknown velocity components u_n. These equatlons may be written In the following form

$$\frac{1}{h_1h_2h_3}\left\{\frac{\partial}{\partial s_1}(\rho h_2h_3u_1u_n)+\frac{\partial}{\partial s_2}(\rho h_1h_3u_2u_n)+\frac{\partial}{\partial s_3}(\rho h_1h_2u_3u_n)\right\}$$

$$=\frac{1}{h_1h_2h_3}\left\{\frac{\partial}{\partial s_2}\left(\frac{h_3h_1}{h_2}\mu\frac{\partial u_n}{\partial s_2}\right)+\frac{\partial}{\partial s_3}\left(\frac{h_1h_2}{h_3}\mu\frac{\partial u_n}{\partial s_3}\right)\right\}-\frac{1}{h_n}\frac{\partial p}{\partial s_n}+F_n$$

for $n=1,2,3$

In these equations, the left hand side represents part of the convection term $(\underline{V}\cdot\nabla)\underline{V}$. The other part of the convection term is due to curvature of the coordinate system and together with the body force it is included as part of the term, F_n, which la considered known. The term Involving the stress tensor π la separated into a pressure term, vlecous terms which contain the unknown velocity u_n and additional vlscoua terms which are also included In the known F_n. The choice of which viscous terms to take as known and which as unknown may seem arbitrary. However the above choice has the advantage that the equations are of the same form. Thus-when they are expressed In finite difference form the coefficients of the unknown velocity components are the same for all three equations. Since the marching integration proceeds in the s_1 direction the viscous terms due to velocity gradients In this direction are neglected.

Uncoupling the s_1 momentum equation

The linearized momentum equations are coupled together by the pressure p so that the velocity components also satisfy eontlnuity. The momentum equation for u_1 depends only Indirectly on the transverse pressure gradients $\partial p/\partial s_2$, $\partial p/\partial s_3$ and is therefore not sensitive to small error8 in these gradients.

These transverse pressure gradients can be estimated using inviscid streamline curvature calculations. Therefore, In the calculation procedure the pressure $\bar{p}$ In the u_1 momentum equation Is distinguished from the pressure p in the momentum equations for the secondary flow velocities u_2 and u_3; and It Is assumed that the transverse gradients of $\bar{p}$ are known throughout the flow from streamline curvature calculations made prior to the marching integration. The pressure $\bar{p}$ is corrected uniformly at each step so that, over the cross-section of the passage, the correct mass flow is obtained.

The secondary flow momentum equation8

The momentum equations for the secondary flow velocities u_2 and u_3 may be solved In finite difference form once a pressure distribution is chosen. The problem is to choose the pressure distribution so that the velocity components also satisfy continuity.

In the procedure, an estimated pressure distribution p^e Is used to obtain estimates of u_2 and u_3. We then assume that corrections p^c In the pressure are related to corrections, u_2^c and u_3^c, of the velocity components by the abbreviated momentum equation,

$$\frac{\rho u_1}{h_1} \frac{\partial u_n^c}{\partial s_1} \simeq - \frac{1}{h_n} \frac{\partial p^c}{\partial s_n}$$

This expression is substituted into the continuity equation and the resulting equation for the pressure correction p^c is solved. The improved pressure estimate $p^e + p^c$ is now used in the momentum equations to obtain improved values of u_2 and u_3. This correction method can be Improved but It was found satisfactory for the present calculations.

The finite difference grid

The number of grid points required are kept to a mininum by using a non-uniform grid spacing with linear profiles of the flow properties between the grid points. This allows grid points to be concentrated in the near-wall regions where they are required for the accurate calculation of the secondary flows which convect fluid with low p^*.

The finite difference equations for continuity and momentum conservation are formed by integrating over control volumes with boundaries midway between the grid points. These equations are solved using the tri-diagonal matrix-algorithm double-sweep method of solution. This and other aspects of the procedure are similar to the Patankar and Spalding method,

Equation9 for the conservation of energy and other properties can be written In a form similar to the momentum equation and thus can be included In the marching integration procedure.

Calculation of Flow in a Rotating Radial-Flow Passage

The applicability of this procedure to the calculation of impeller flowe will be demonstrated by calculating the medium-flow measured by Moore (26).

Moore measured the flow development in the rotating radial-flow passage shown In Figure 1. The figure shows a schematic and a sectional view of the test-section which was mounted on a rotating turntable. The teat-section had a constant height of 3 In. and the side walls were radii with an Included angle of 15°. The length of the test-section waa 24 in. and the square Inlet was at a radius of 12 In. The rotational speed was 206 RPM.

At the medium flow rate the mean velocity at the Inlet to the teat section was 54.9 ft/s. At this flow rate, Moore measured the three-dimensional flow distribution, 0.5 In. downatream of the Inlet, and the results of the measurements are shown in Figure 2. In the present calculations two flow distributions have been used to start the marching Integration. One baeed on the measured asymmetric three-dimensional velocity distribution at x =0.5 inches, and, for comparison, a second distribution assuming a purely radial Inlet flow symmetrical about the mid-height of the passage.

Moore's flow was Incompressible and we have adopted a simple viscosity model which uses the Prandtl mixing length to calculate the turbulent contribution to the viscosity. The mixing length was taken a8 the smaller of 0.08δ and 0.41 y, where δ is the boundary layer thickness and y is the distance to the nearest wall. Negative values of radial velocity were not allowed and all the velocity components were set to zero at the walls. The transverse pressure gradient8 through the flow were taken as $\frac{\partial \bar{p}}{\partial s_2} = 2\rho\omega\bar{u}_1 W = 66\ \mathrm{kN/m^2}$ and $\frac{\partial \bar{p}}{\partial s_3} = 0$, where $s_2 = -Y/W$ and $s_3 = Z/h$ and the symbols are defined in Figure 1. This value of $\partial\bar{p}/\partial s_2$ is consistent with Moore'8 measurements which indicated a constant static pressure difference between the pressure and suction side walls.

Symmetrical inlet flow

The velocity distribution at the inlet was chosen to be symmetrical about the mid-height of the rotating passage and it was assumed that at the Initial station the flow was radial. The marching integration was performed for one half of the channel using a 13 by 8 grid and a symmetry condition was applied at the mid-height plane. The Inlet flow was assumed to consist of a potential flow core with uniform stagnatlon pressure and boundary layers of uniform thickness on each of the three walls. Initially 5 points represented each boundary layer profile, grid points being located at 0, 1, 5, 20 and 100 per cent of the Inlet boundary layer thickness. The results of the flow calculations are presented In Figures 3-7.

Figures 3 and 4 show contours of radial velocity, as a fraction of the local mean velocity, over six cross-sectional planes In the channel. A pictorial representation of the corresponding secondary velocities is shown In Figures 5 and 6, and the development of the radial weloclty at the mid-height plane of the channel is shown with the measured development in Figure 7.

Secondary flows transport low momentum fluid from the pressure side boundary layer into the suction side layer and these, together with the corresponding movement of the potential flow towards the pressure side, are shown in Figures 5 and 6. In these calculations the tangential velocity remained approximately constant across the top and bottom walls and the initial large cross flows from the pressure side corners result from the deceleration of the fluid In these corners due to the radial adverse pressure gradient. Between x = 14.5 Inches and x = 18.5 inches fluid with radial velocity less than one half of the mean velocity flows from the pressure side wall. This fluid and most of the fluid with low p^* accumulates on the suction side and shows an Initial tendency to move towards the mld-height of the passage as shown In Figures 3 and 4. Streamwise vorticity is then generated as seen in the secondary velocities at x = 20.5 inches In Figure 6 and this vortlclty tends to return the fluid with low p^* to its stable location near the suction side wall In this flow which has a Rossby number $W/\omega R_n = 0$.

Fluid with low p^* lo continually generated along the pressure side wall and after $x \approx 18.5$ inches In the calculations It is convected first towards the mid-height, then towards the suction side and finally It passes close to the pressure wall as It moves onto the top or bottom wall. This calculated pressure side flow is similar to Taylor-Goertler vortex flow with two complete cells near the mid-height and two 'open" cells in the corners. It shows the complexity which can be caused by small scale effects in impeller flow calculations. This thickening of the pressure side boundary layer in the calculation with the symmetrical inlet flow Is shown In Figure 7 and it was not observed in the experiments. However, the general agreement between the calculated and measured radial velocities at the mid-height of the channel is good. The wake develops on the suction side with similar shape and size to the measured wake and It thickens markedly after $x = 10.5$ Inches as observed.

Asymmetric inlet flow

The measured asymmetric Inlet velocity distribution shown in Figure 2 was used with interpolation and extrapolation to obtain the initial secondary velocities shown in Figure 10. The distribution of radial velocity was obtained by using linear Interpolation between the nine central measurements, extrapolation of the potential flow to the boundary layer edges and boundary layers of unlform thickness on each of the four walls. The resulting initial condition is necessarily approximate but it can be used to obtain an estimate of the Influence of non-uniform Inlet flow on the flow development, and ideally It should result In better agreement with the measurements. The reeults of these flow calculations using a 13 by 13 grid are presented in Figures 7-11.

Figures 8 and 9 show contours of radial velocity which may be compared with the contours obtained for the symmetrical flow shown in Figures 3 and 4. Similarly, the calculated secondary velocities shown In Figures 10 and 11 may be compared with the velocities shown in Figures 5 and 6, and the radial velocity at the mid-height plane is shown in Figure 7.

The influence of the asymmetric inlet flow dominates the flow initially. Figure 10 shows strong secondary flow Into the top corner on the pressure side, across the top wall and onto the suction side. The low momentum fluid builds up In two pockets in the corners at the top of the channel and these are evident at $x = 10.5$ inches in Figure 8. Figure 9 then shows the migration of the fluid from the pressure side pocket across the top wall to join the bulk of the low momentum fluid near the centreline in the suction-side wake. Here, as In the symmetric flow calculation, wake flow extends along the centrellne towards the pressure side, especially at x = 18.5 lnchea. The secondary velocities shown In Flgure 11 Indicate that streamwise vortlclty then develops which tends to return this fluid to the auction slde and tends to distribute the wake more uniformly over the suction side wall.

Figures 8 and 10 also show a comparleon of the calculated velocity dlatrlbutlon at x = 10.5 Inches with the measured velocity distribution at x = 11 Inches. The radial and the secondary velocities are In extremely good agreement except near the bottom wall where the calculated boundary layer thlcknesa and the resulting tangential velocities are larger than observed. The relatively low velocities meaaured In the top pressure-side corner support the calculated accumulation of a pocket of low momentum fluid, Indeed In taklng the measurements Moore noted that at the data polnt closest to the top wall in this corner the pressure was "more unsteady".

The asymmetric flow calculation gives a thinner pressure-side boundary layer at the channel mid-height than calculated for the symmetric flow. This Is shown In Figure 7, and the difference between the two calculations may be seen quite clearly by comparing

Figures 4 and 9. In the asymmetric flow there is a relatively complete transport of boundary layer fluid from the pressure side wall by secondary flow. The asymmetric flow calculation also results In an apparently thicker wake on the suction side at x = 14.5 Inches and x = 18.5 Inches. However, Figure 9 shows that this is mostly due to the local accuaulatlon of fluid with low p^* on the centreline In this calculation, and the streamwise vorticity subsequently reorienta the wake fluid to give similar wake thicknesses at x = 20.5 inches.

Moore's channel was sufficiently long that In both calculations most of the fluid with low p^* had accumulated on the suction side wall by x = 20.5 Inches. Thus, at this radial location the velocity distributions are very similar In the two calculations. However, the development of the flow In the two case8 Is quite different. The details of the flow development may well have a significant effect on the performance of centrifugal machines especially If wake fluid has not reached Its stable location at the Impeller exit.

Concluding Remarks

It is clear that centrifugal machines often have non-uniform flow at the Impeller exit and that this flow can have a large effect on performance. Fluid with low stagnation pressure is found to accumulate in "wake" flows which are often a significant fraction of the mass flow through the machine. The calculation of the size and location of these wake flows involves the calculation of the development of complex viscous three-dimensional flow in Impeller geometries. A possible approach to the solution of this problem is offered by the combination of a marching-integration procedure for the calculation of parabolic flows with a streamline curvature calculation of the approximate characteristics of the primary flow through the machine. The ability of a marching integration procedure to compute a complex three-dimensional flow with large secondary velocities leading to the formation of a wake, has been demonstrated. It appears that an extension of this procedure to impeller flow calculations may be facilitated by general. orthogonal coordinates which have been used in this work.

The Rossby number $W/\omega R_n$ which governs the stable location of wake flow at the impeller exit is based on the rotation rate ω, the relative velocity W and the radius R_n of the curvature from the axial to the radial direction. Large values of this number Indicate that curvature is dominant and the stable location of the wake is on the shroud wall. Small values signify the dominance of rotation and a stable wake location on the suction side wall. It is Interesting to note that low speclflc-speed Impellers tend to have low Rossby numbers, while high specific speed Impellers tend to have high Rossby numbers. Qualitative understanding of Impeller flow may be obtained by consideration of the generation of streamwise vortlclty.

Acknowledgement

The computer calculations performed in thls study have been sponsored by Rolls-Royce (1971) Ltd. as part of the Rolls-Royce / Whittle Laboratory collaboration at Cambridge University. The partial support of one of the authors (J.G.M.) under this contract is gratefully acknowledged. The authors also wish to thank Mr. 2.H. Timmis and Mr. C.M. Pratt of the Rolls-Royce Hellcopter Engine Group for their encouragement.

References

1. Smith, LA., "Notes on Some Exparlmentsl Researches on Internal Flow in Centrifugal Pumps and Allied Machines", Engineering, Vol. Lxxiv p 763, Dec 5, 1902.

2. Gibson, A.H., Hydraulics and its Applications, Constable 1st Ed., 1908.

3. Carrard, A., "Sur le Calcul des Rows Centrifuges", La Technique Moderne, T. XV No. 3 pp 65-71 and No. 4 pp 100-104, Feb. 1923.

4. Carrard, A., "On Calculations for Centrlfugal Wheels", translation by J. Moore, Unlv. of Cambridge, Dept. of Eng. Report No, CUED/A Turbo/TR 73, 1975.

5. Flugel, G., "Ein neues Verfahren der graphischen Integration", Dissertation Munich, 1914.

5. Stodola, A., Steam and Gas Turbines Vol. II, McGraw-Hill Book Co., Inc, 1927, pp992-997, 1252-1270, (Reprlnted, Peter Smith (New York), 1945)

7. Fischer, K. and Thoma, D., "Investigation of the Flow Conditions in a Centrifugal Pump", A.S.M.E. Trans., HYD-54-8, 1932.

8. Kearton, W.J., "Influence of the Number of Impeller Blades on the Pressure Generated In a Centrifugal Compreesor and on Its General Performance", Proc. Inst. Mech. Engs. Vol. 124, pp 481-568, 1933.

9. Church, A.H., "Centrifugal Pumps and Blowers", 1944, Reprinted, Robert E. Krieger, Huntington, New York, 1972.

10. Dean, R.C., Jr, "On the Unresolved Fluid Dynamics of the Centrifugal Compressor", Advanced Centrifugal Compressors, A.S.M.E., Special Publication, pp l-55, 1971.

11. Cheshire, L.J., "Centrifugal Compressors for Aircraft Gas Turbines", Proc. I. Mech E. Vol. 153, p 440, 1945.

12. Squire, H.B. and Winter, K.G., 'The Secondary Flow in a Cascade of Airfoils In a Non-uniform Stream", J. Aero. SC., Vol 18, pp 271-277, 1951.

13. Hawthorne, W.R., "Secondary Circulation In Fluid Flow", Proc. Roy. Soc. A. Vol. 206, pp 374-387, 1951.

14. Kramer, J.J. and Stanitz, J.D., "A Note on Secondary Flow in Rotating Radial Channels", NACA Report 1179, 1954.

15. Smith, A.G., "On the Generation of a Streamwise Component of Vorticity in a Rotating Passage", Aero. Quart. Vol. 8, pp 369-383, 1957.

16. Hawthorne, W.R., "Secondary Vorticity in Stratified Compressible Fluids In Rotating Systems, " Univ. of Cambridge, Dept. of Eng. Report No. CUED/A-Turbo/TR 63, 1974.

17. Hamrick, J.T., "Some Aerodynamic Investigations In Centrifugal Impellers", Trans. ASME, April 1956, pp 591-602.

1.8. Hamrick, J.T., Mizisin, J. and Michel, D.J., "Study of Three-Dimensional Flow Distribution Based on Measurements In a 48-Inch Radial-Inlet Centrifugal Impeller", NACA TN 3101, 1954.

19. Lakshmlnarayana, B. and Horlock, J.H., "Generalised Expreselons for Secondary Vorticity using Intrinsic Coordinates", J. Fluid Mech., Vol. 59, PP 97-115, 1973.

20. Eckardt, D., "Detailed Flow Investigations wlthln a High-Speed Centrifugal Compressor Impeller", Trans. ASME, J. Fluids Eng. Vol. 98, pp 390-402, 1976.

21. Dean, R.C., Jr. and Senoo, Y., "Rotating Wakes In Vaneless Diffusers", Trane. ASME, J. Basic Eng., vol. 82, 1960, pp 563-574.

22. Johneton, J.P. and Dean, R.G., Jr., 'Losses in Vaneless Diffusers of Centrifugal Compressors and Pumps" , Trans. ASME, J. Eng. Power, Vol. 88, 1966, pp 49-60.

23. Senoo, Y. and Ishlda, M., "Behavior of Severely Asymmetric Flow in a Vaneless Diffuser", Trans. ASME, J. Eng. Power, Vol. 97, pp375-387, 1975.

24. Ellis, G.O., "A Study of Induced Vorticity in Centrifugal Compressors", Trans. ASME, J. Eng. Power, Vol. 06, pp63-76, 1964.

25. Rebernik, B., 'Investigation on Induced Vorticity in Vaneless Diffusers of Radial Flow Pumps', Proceedings of the Fourth Conference on Fluid Machinery, Budapest, 1972, pp 1129-1139.

26. Moore, J., "A Wake and an Eddy in a Rotating, Radial-Flow Passage (Part 1: Experimental Observations, Part 2: Flow Model)," J. Eng. for Power, Trans. ASME, Ser A, Vol. 95 No. 3, P!? 205-219, 1973.

27. Patankar, S.V. and Spalding, D.B., "A Calculation Procedure for Heat, Mass and Momentum Transfer In Three-Dimensional Parabolic Flows", Int. J. Heat and Mass Transfer, 15, pp 1787-1806, 1972.

28. Tsien, H.S., "The Equations of Gas Dynamics", Fundamentals of Gas Dynamics, ed. Emmons, H.W., Oxford Univ. Press, London, 1958.

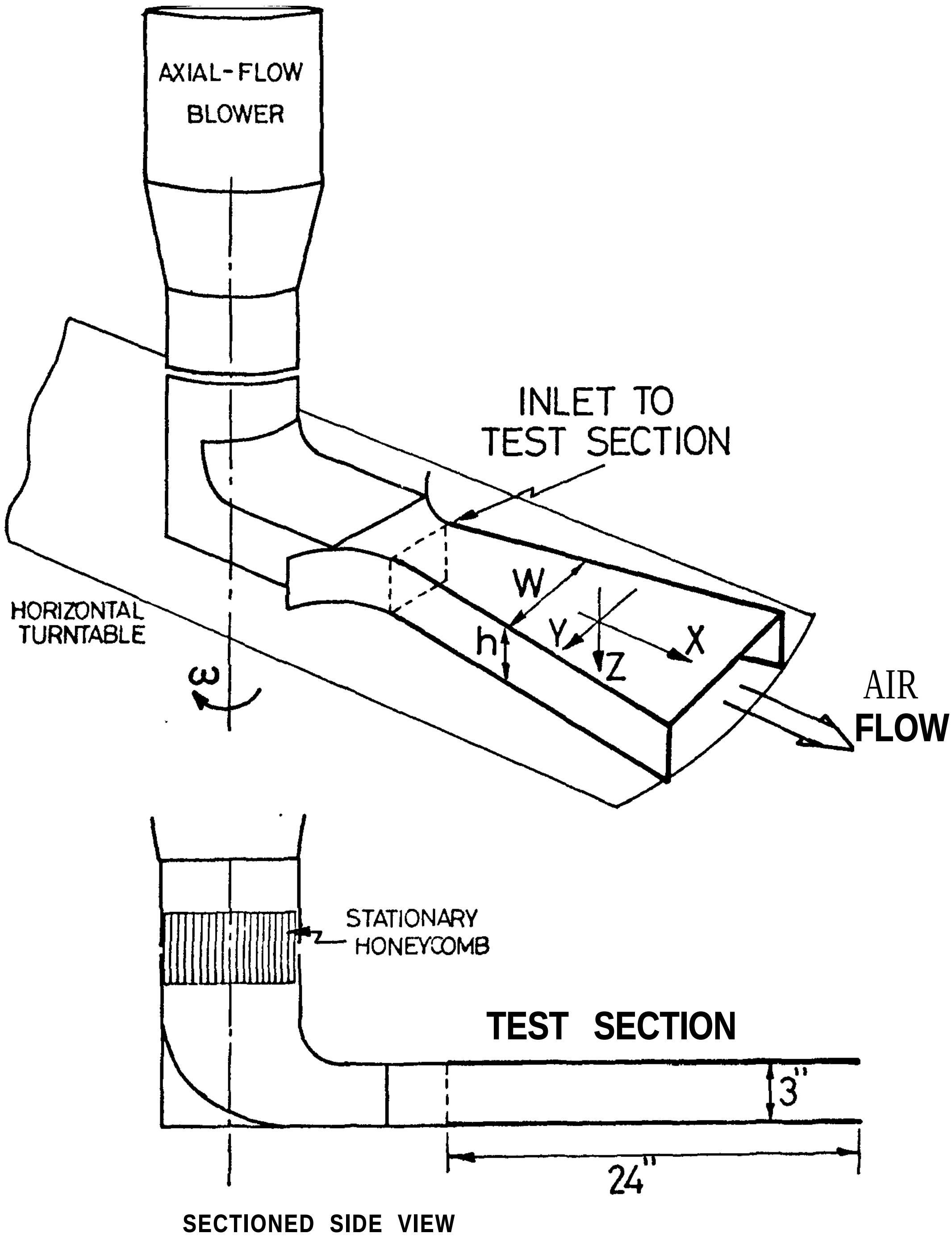


Figure 1. Schematic of Moore's test section.

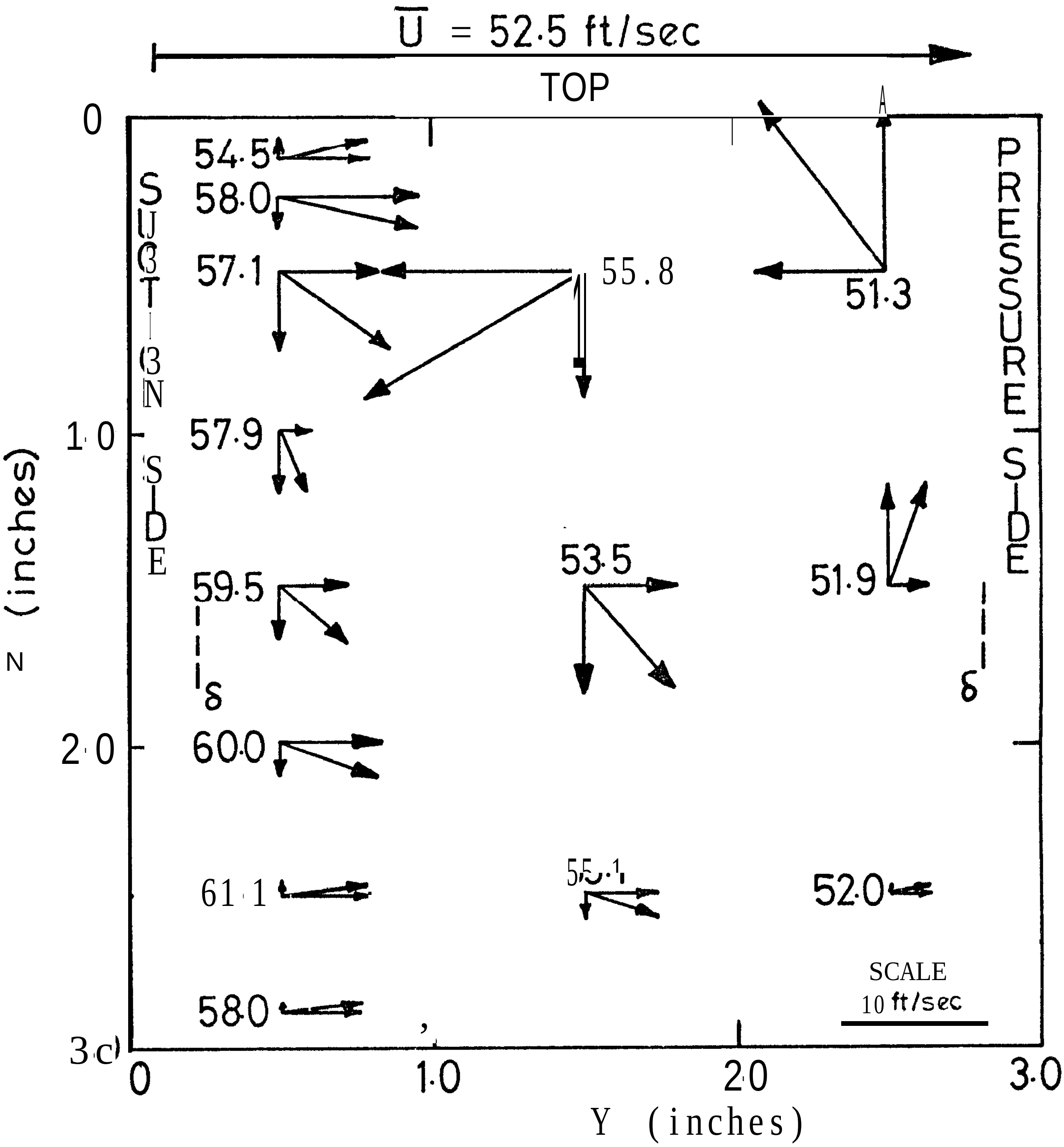
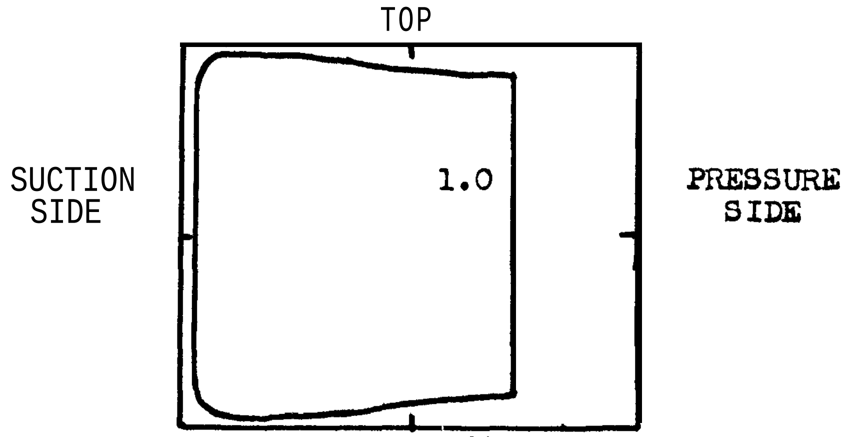
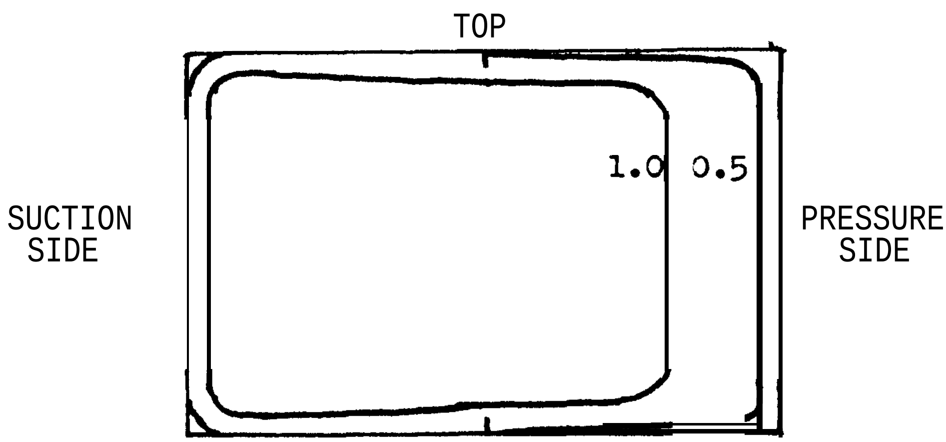


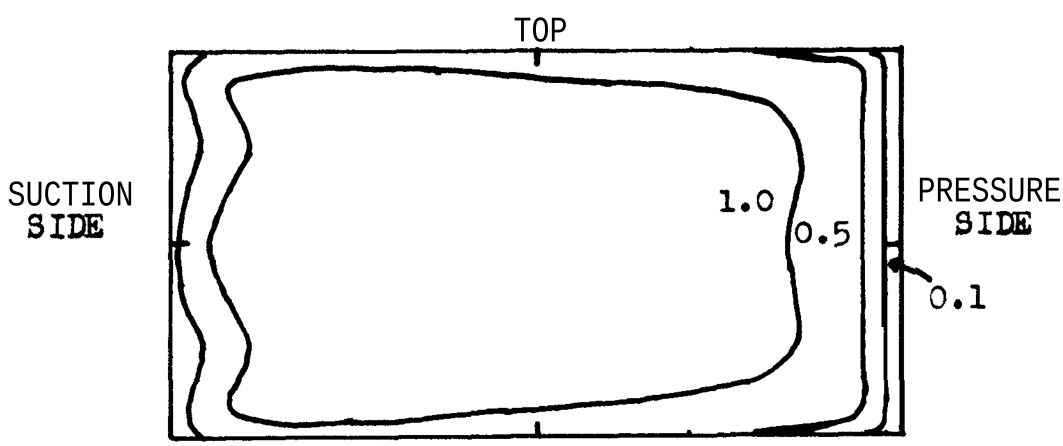
Figure 2. Measured secondary and radial velocities at x = 0.5 Inches.



X=2.5 INCHES, $\overline{U}$ = 13.7 M/S

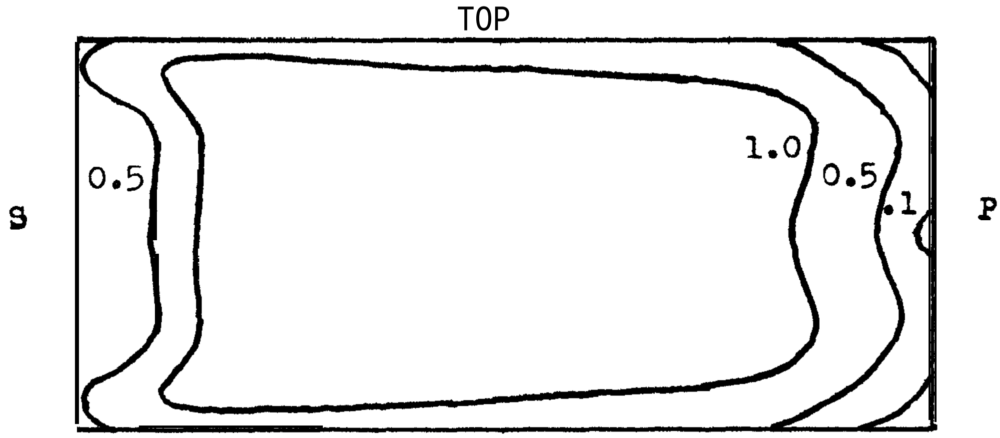


X = 6.5 INCHES, $\overline{U}$ = 10.6 M/S

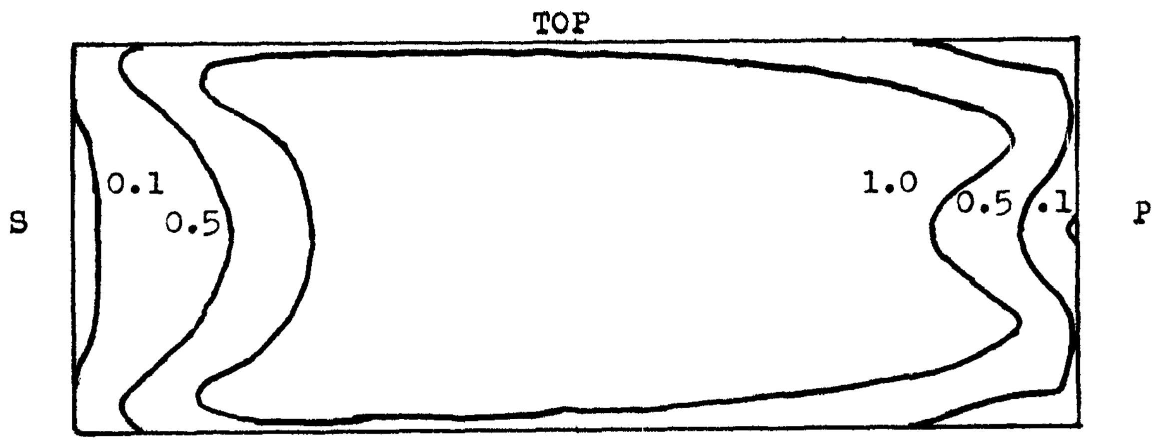


X = 10.5 INCHES, $\overline{U}$ = 8.6 M/S

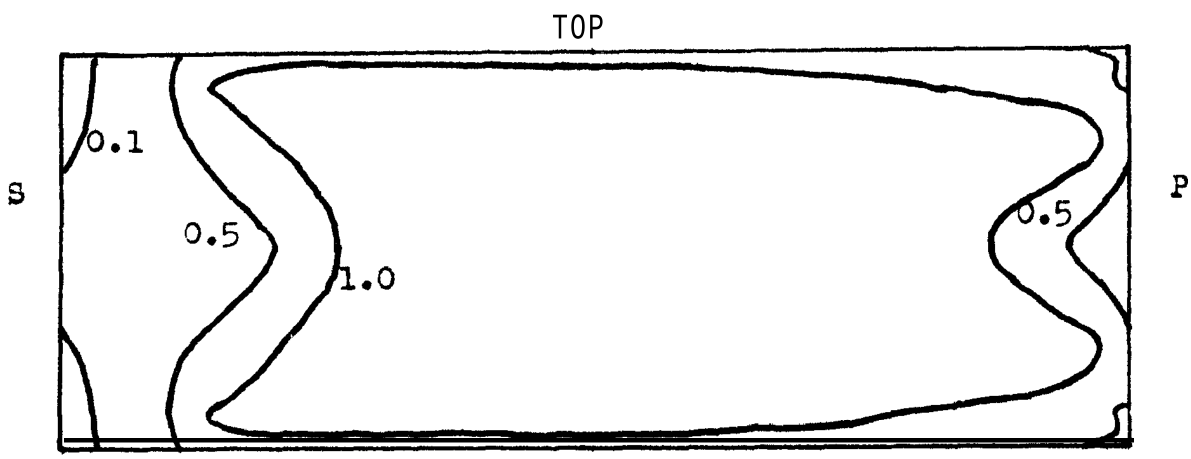
Figure 3. Contoure of calculated radial velocity, expressed as a fraction of the local mean velocity, for symmetric flow at x = 2.5, 6.5 and 10.5 Inches.



X = 14.5 INCHES, $\bar{U}$ = 7.3 M/S



X = 18.5 INCHES, $\bar{U}$ = 6.3 M/S



X = 20.5 INCHES, $\bar{U}$ = 5.9 M/S

Figure 4. Contours of calculated radial velocity, expressed as a fraction of the local mean velocity, for symmetric flow at x a14.5, 18.5 and 20.5 Inches.

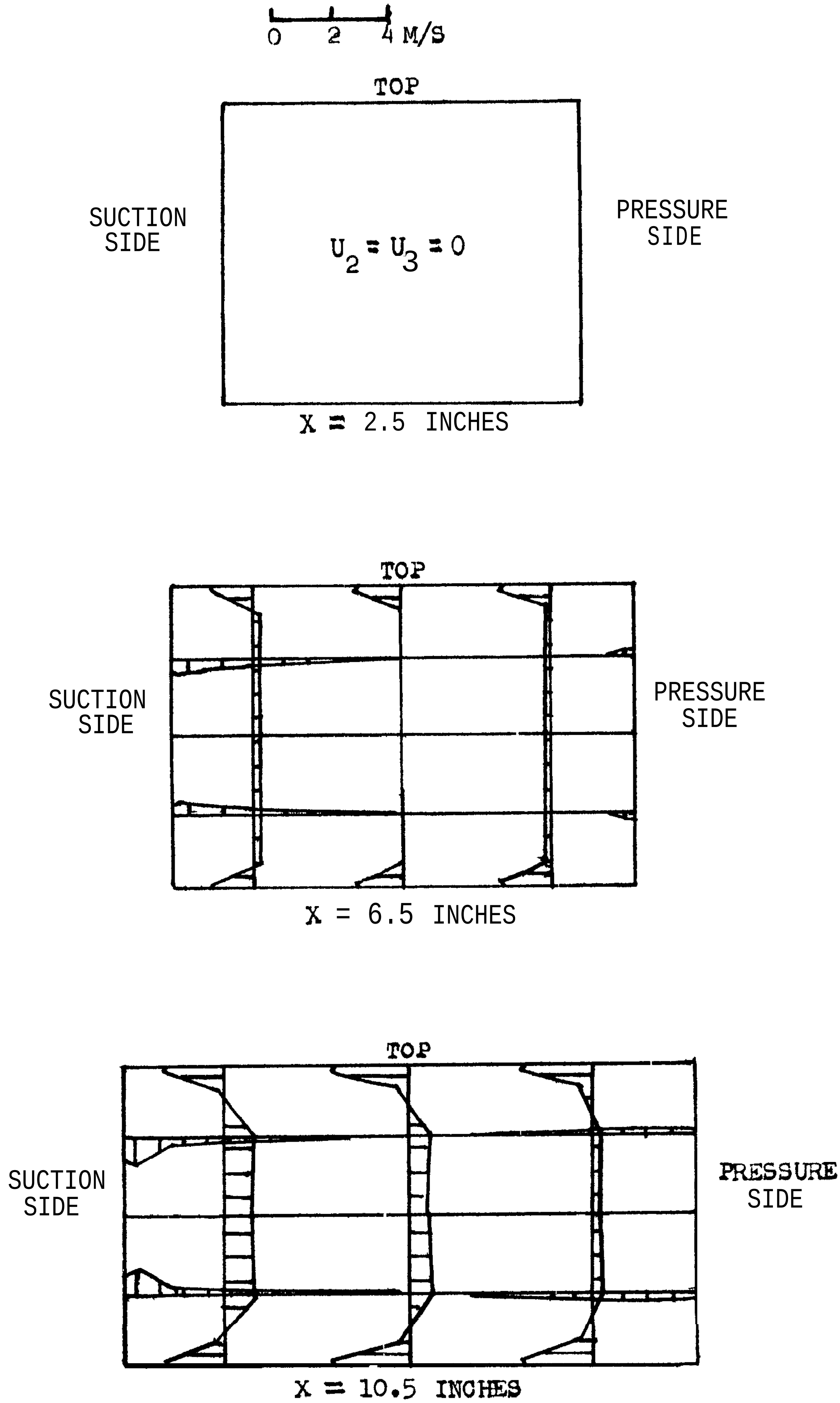


Figure . Calculated secondary velocity distributions for symmetric flow at $x = 2.5$, 6.5 and 10.5 inches.

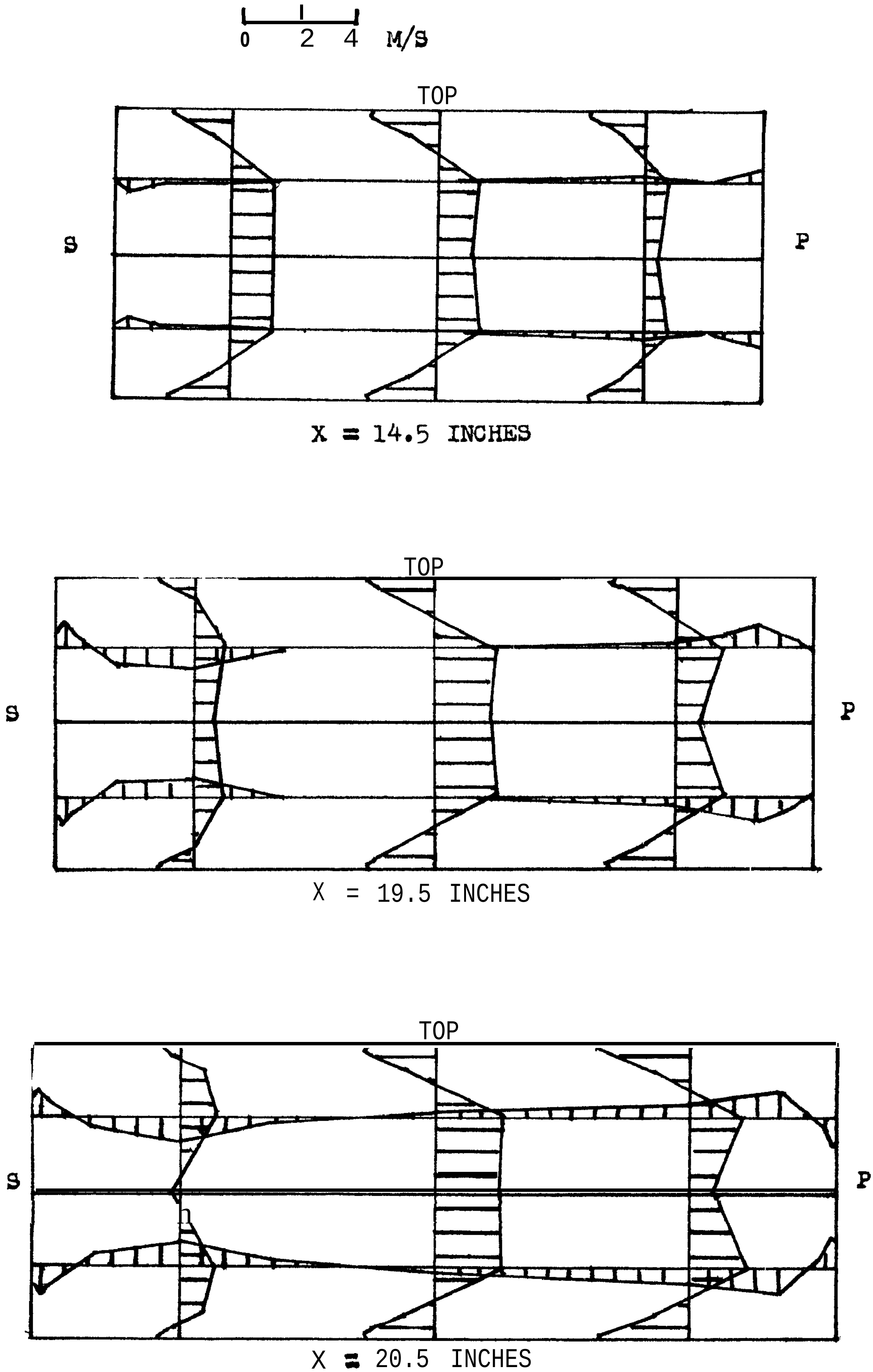


Figure 6. Calculated secondary velocity distributions for symmetric flow at x = 14.5, 18.5 and 20.5 inches.

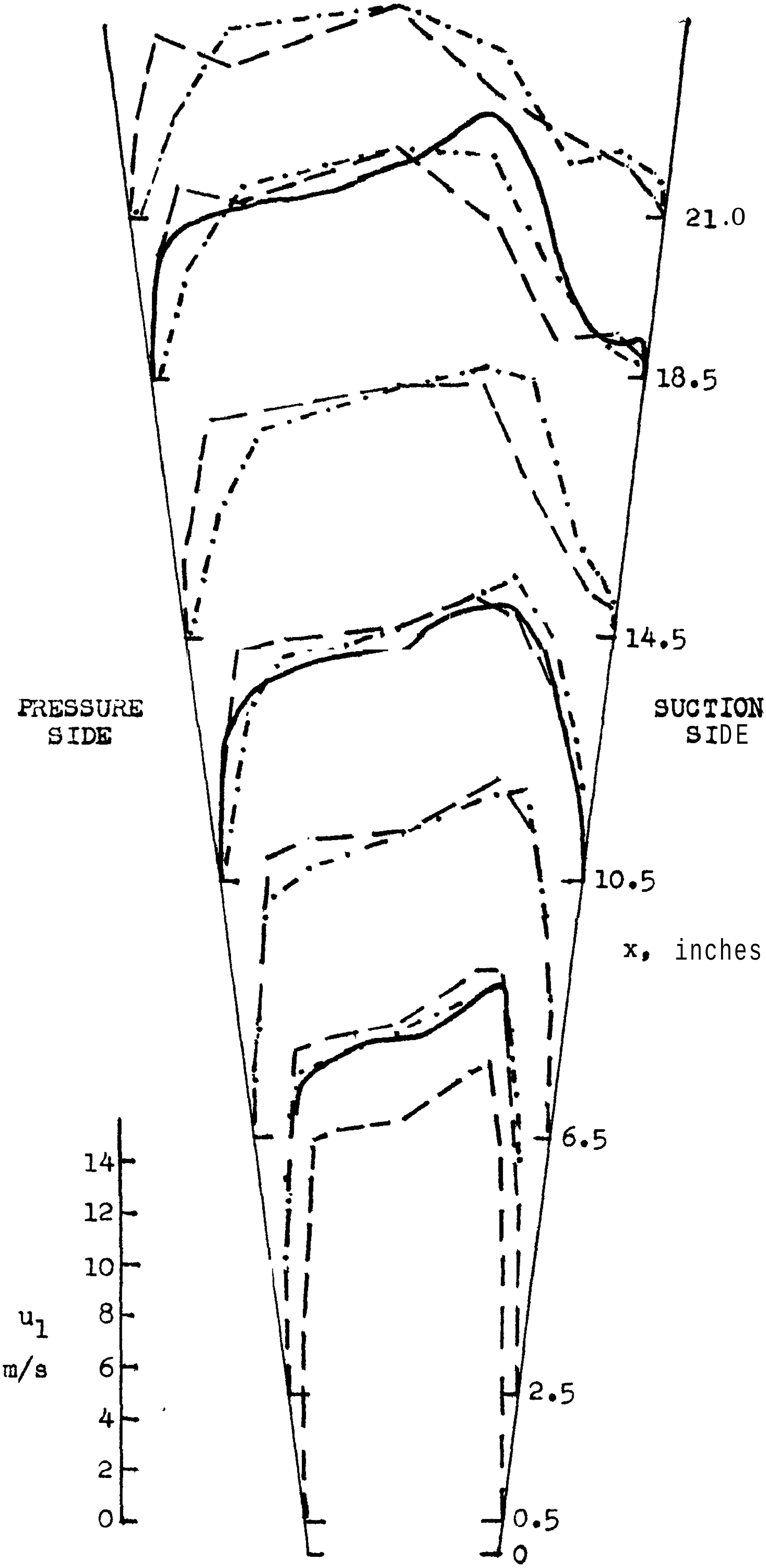
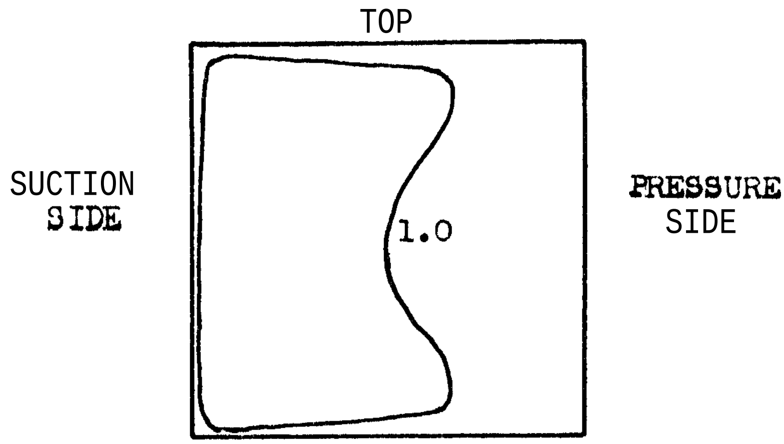
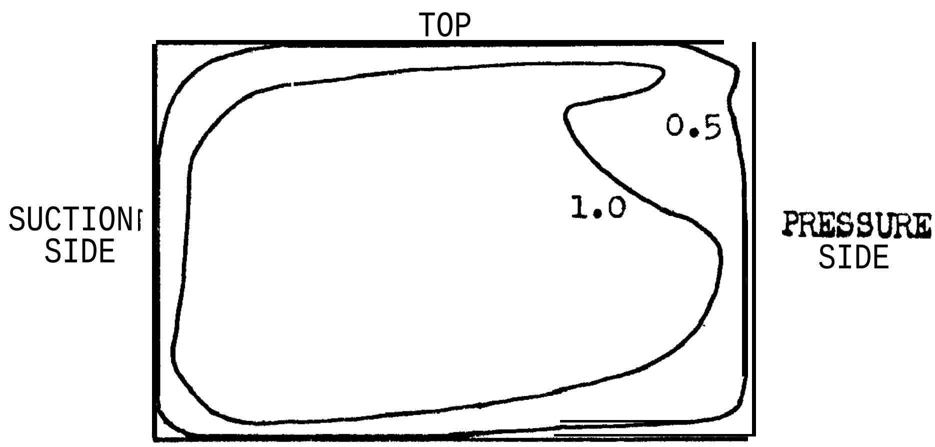


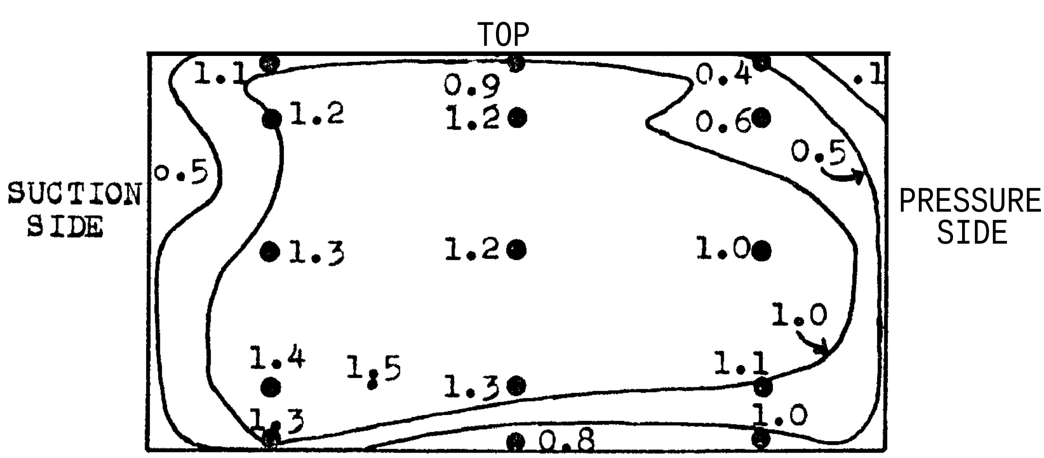
Figure 7. Comparison of calculated and measured radial velocity profiles at channel aid-height. ——— measured, -·-·-·- calculated symmetric flow, -- ▬ -calculated asymmetric flow.



X = 0.5 INCHES, $\bar{U}$ = 16.3 M/S

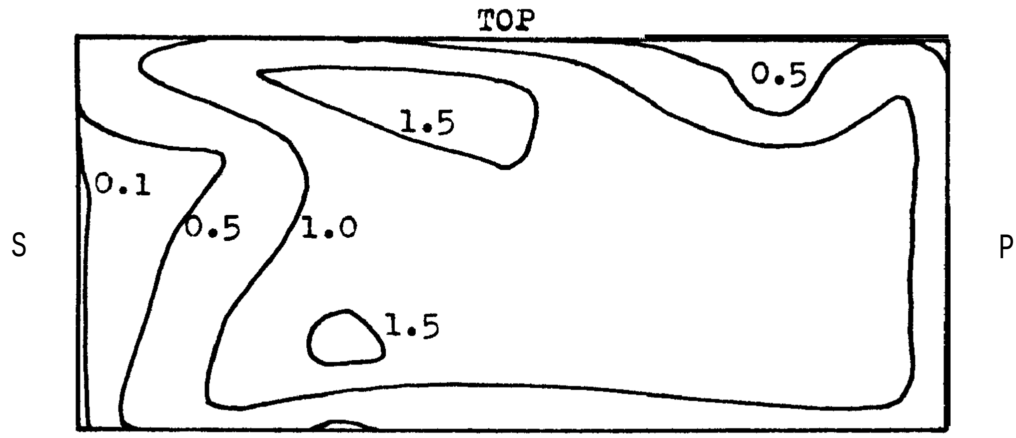


X = 6.5 INCHES, $\bar{U}$ = 10.9 M/S

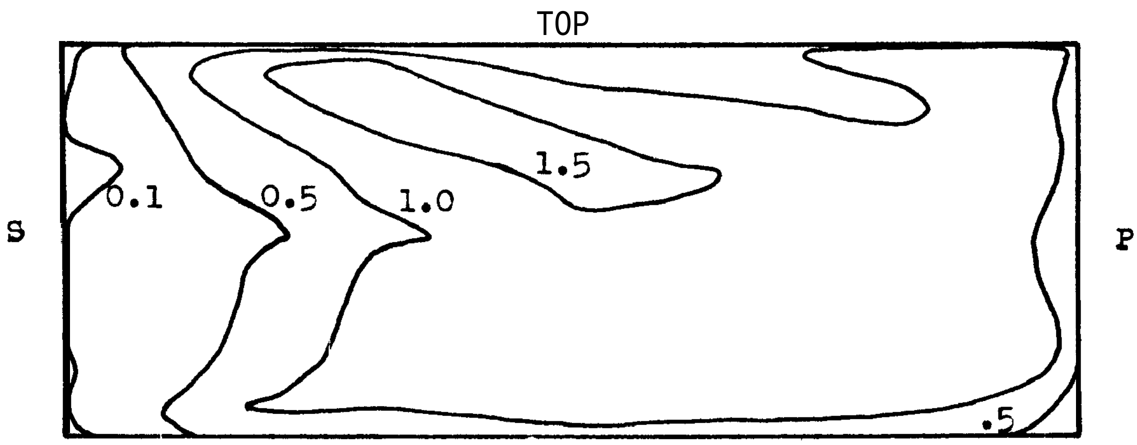


X = 10.5 INCHE3, $\bar{U}$ = 8.9 M/S

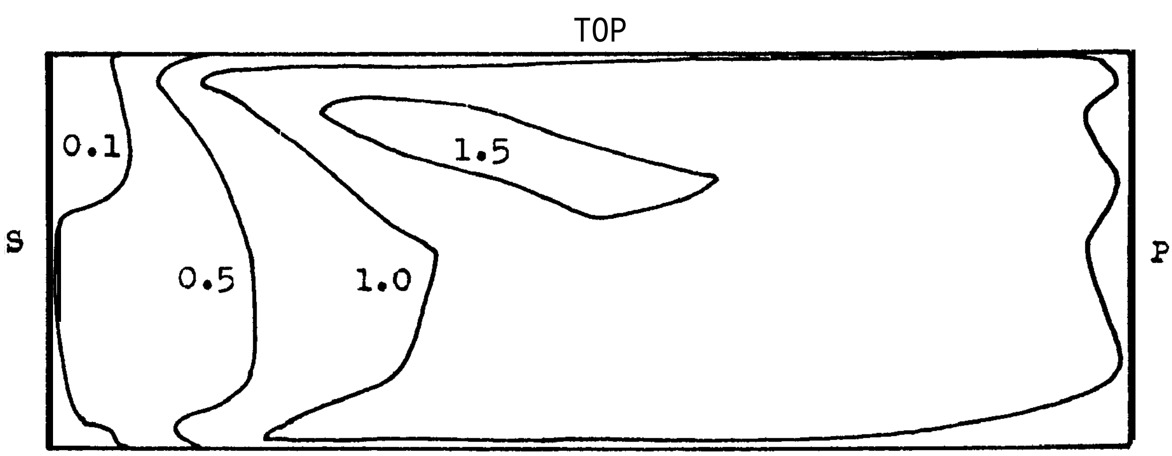
Figure 8. Contours of calculated radial velocity, expressed as a fraction of the local mean velocity, for **asymmetric flow** at x = 0.5 and 6.5 inches and at 10.5 inches compared with measured radial velocities at 11 inches. • measured velocities $u_1/\bar{u}$.



X = 14.5 INCHES, $\bar{U}$ = 7.5 M/S



X = 18.5 INCHES, $\bar{U}$ = 6.5 M/S



X = 20.5 INCHES, $\bar{U}$ = 6.1 M/S

Figure 9. Contours of calculated radial velocity, expressed as a fraction of the local mean velocity, for asymmetric flow at x = 14.5, 18.5 and 20.5 inches.

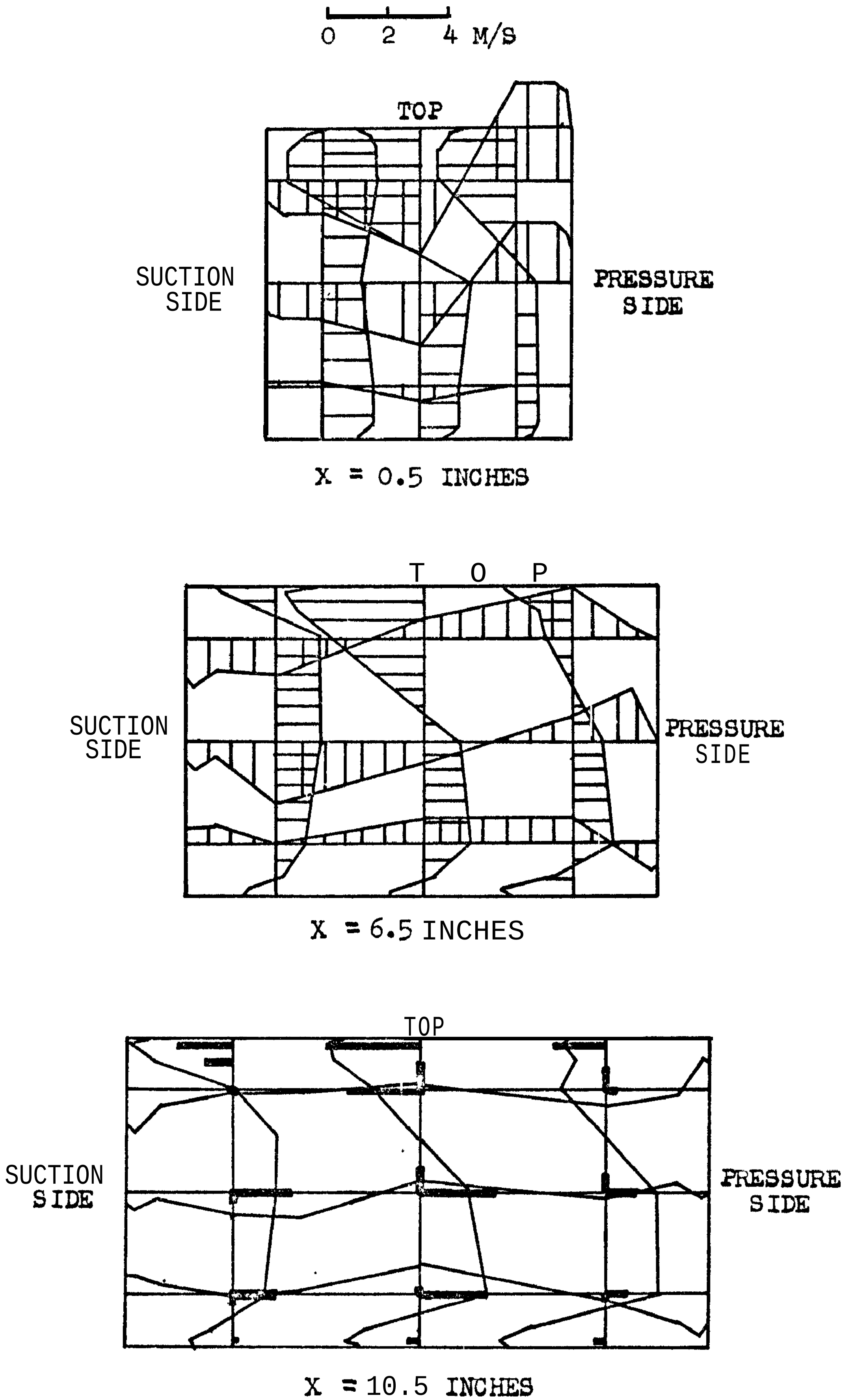


Figure 10. Calculated secondary velocity distributions for asymmetric flow at x = 0.5 and 6.5 inches and at x = 10.5 Inches compared with measured secondary velocities at 11 Inches. ▬ measured.

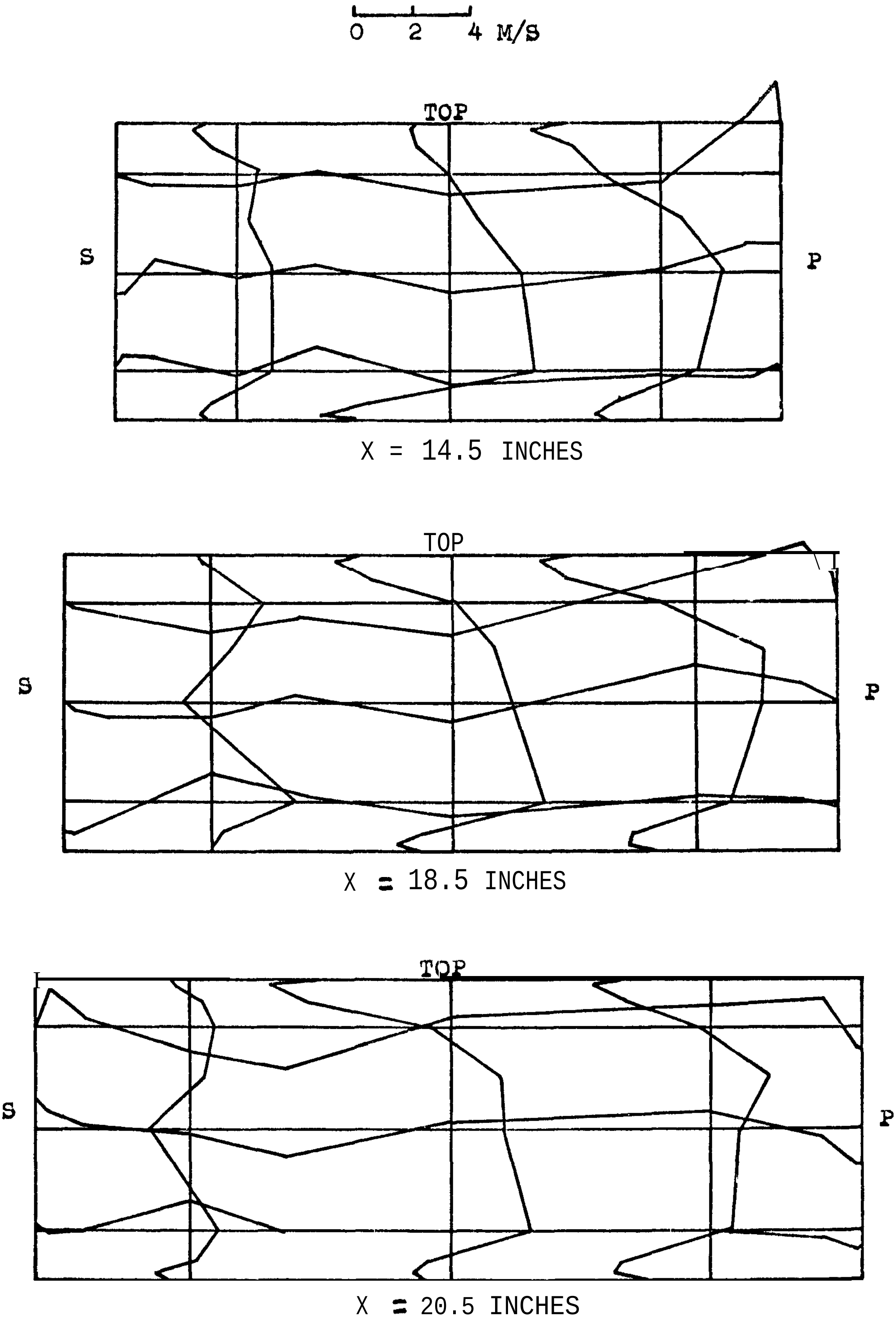


Figure 11. Calculated secondary velocity distributions for asymmetric flow at X = 14.5, 18.5 and 20.5 Inches.

Produced in England by Her Majesty's Stationery Office, Reprographic Centre, Basildon
Bas 40697/587483 K4 10/77 P

ARC CP No.1384
February 1977
Xoore, J., Moore, J.G. and Johnson, H.W.

ON THREE-DIMENSIONAL FLOW IN CENTRIFUGAL IMPELLERS

Evidence of non-uniform flow at the exit of centrifugal impeller passages is discussed and a Rossby number $W/\omega R_n$ which governs the stable location of wake flow in the exit plane of an impeller is presented. In impellers with large Rossby numbers the stable location of the wake is on the shroud wall; wakes *on* the suction side wall are 8table in impeller8 with low Rossbynumbers,

The/

ARC CP No.1384
February 1977
Xoore, J., Xoore, J.G. and Johnson, M.W.

ON THREE-DIMENSIONAL FLOW IN CENTRIFUGAL IMPELLERS

Evidence of non-uniform flow at the exit of centrifugal impeller passages is discussed and a Rossby number $W/\omega R_n$ which governs the stable location of wake flow in the exit plane of an iapeller is presented. In impellers with large Rossby number8 the stable location of the wake is on the shroud wall; wakes on the suction side wall are stable in impellers with low Rossby numbers.

The/

ARC CP No.1384
February 1977
Xoore, J., Moore, J.G. and Johnson, M.W.

ON THREE-DIMENSIONAL FLOW IN CENTRIFUGAL IMPELLERS

Evidence of non-uniform flow at the exit of centrifugal impeller passages is discussed and a Rossby number $W/\omega R_n$ which governs the stable location of wake flow in the exit plane of an impeller is presented. In impellers with large Rossby numbers the-stable location of the wake is on the shroud wall; wake8 on the suction side wall are stable in impeller8 with low Rossby numbers.

The/

ARC CP No.1384
February 1977
Moore, J., Xoore, J.G. and Johnson, M.W.

ON THREE-DIMENSIONAL FLOW IN CENTRIFUGAL IMPELLERS

Evidence of non-uniform flow at the exit of centrifugal impeller passages is discussed and a Rossby number $W/\omega R_n$ which governs the 8table location of wake flow in the exit plane of an impeller is presented. In impellers with large Rossby numbers the stable location of the wake is on the shroud wall; wakes on the suction side wall are stable in impeller6 with low Rossby numbers.

The/

DETACHABLE ABSTRACT CARDS

The ability of a marching-integration procedure to compute a three-dimensional rotating flow with large secondary velocities leading to the formation of a wake is demonstrated. A possible approach to the calculation of three-dimensional impeller flow is suggested.

The ability of a aarching-integration procedure to compute a three-dimensional rotating flow with large secondary velocities leading to the formation of a wake is demonstrated. A possible approach to the calculation of three-dimensional impeller flow is suggested.

The ability of a marching-integration procedure to compute a three-dimensional rotating flow with large secondary velocities leading to the formation of a wake is demonstrated. A possible approach to the calculation of three-dimensional impeller flow is suggested.

The ability of a **marching-integration** procedure to compute a three-dimensional rotating flow with large secondary velocities leading to the formation of a wake is demonstrated. A possible approach to the calculation of three-dimensional impeller flow is **suggested.**

C.P. No. 1384

HER MAJESTY'S STATIONERY OFFICE

Government Bookshops

49 High Holborn, London WC1V 6HB
13a Castle Street, Edinburgh EH2 3AR
41 The Hayes, Cardiff CF1 1 JW
Brazennose Street, Manchester M60 8AS
Southey House, Wine Street, Bristol BS1 2BQ
258 Broad Street, Birmingham B1 2HE
80 Chichester Street, Belfast BT1 4JY

Government publications are also available through booksellers

C.P. No. 1384

ISBN 0 11 47 1130 5