EXPERIMENTAL AND NUMERICAL INVESTIGATION OF STATOR EXIT FLOW FIELD OF AN AUTOMOTIVE TORQUE CONVERTER

B. V. Maratha, B. Lakshminarayana, and Y. Dong
Department of Aerospace Engineering
Pennsylvania State University
University Park, Pennsylvania

ABSTRACT

The objective of this investigation is to understand the nature of the complex flow field inside each element of the torque converter through a systematic experimental and numerical investigation of the flow field. A miniature five-hole probe was used to acquire the data at the exit of the stator at several operating conditions. The flow field is found to be highly three-dimensional with substantial flow deviations, and secondary flow at the exit of the stator. The secondary flow structure, caused by the upstream radial variation of the through flow, induces flow overturning near the core. Flow separation near the shell causes flow underturning in this region. The rate of decay of stator wake is found to be slower than that observed in the wakes of axial flow turbine nozzles. The flow predictions by a Navier-Stokes code are in good agreement with the pressure and the flow field measured at the exit of the stator at the design and the off-design conditions.

NOMENCLATURE

\[ C \quad \text{Chord length} \]
\[ C_{p0}, C_{pS} \quad 2(P_1 - P_{hub})/\rho \overline{V_x} \]
\[ H \quad \text{Radial distance from stator shell (hub) normalized by the stator blade span} \]
\[ P \quad \text{Pressure normalized by } \rho \overline{V_x}^2 \]
\[ P_{hub} \quad \text{Reference pressure on the hub} \]
\[ PS, SS \quad \text{Pressure and suction side} \]
\[ r \quad \text{Radius} \]
\[ S \quad \text{Blade spacing} \]
\[ SR \quad \text{Speed ratio (turbine rpm/pump rpm)} \]
\[ TR \quad \text{Torque ratio (turbine torque/pump torque)} \]
\[ V \quad \text{Absolute flow velocity/total velocity normalized by } \overline{V_x} \]
\[ V_c \quad \text{Defect in wake centerline velocity} \]
\[ \overline{V_x} \quad \text{Mass averaged axial velocity downstream of stator} \]
\[ U \quad \text{Blade Speed} \]
\[ Z \quad \text{Distance downstream of trailing edge} \]
\[ \alpha \quad \text{Pitch angle } \arctan \left( \frac{V_y}{V_z} \right) \]
\[ \beta \quad \text{Yaw angle (measured from the axial direction, } \arctan \left( \frac{V_y}{V_z} \right) \text{)} \]
\[ \xi \quad \text{Total pressure loss coefficient (Eqn. 3)} \]
\[ \rho \quad \text{Density of the fluid} \]

Presented at the International Gas Turbine and Aeroengine Congress and Exposition
The Hague, Netherlands — June 13–18, 1994
This paper has been accepted for publication in the Transactions of the ASME
Discussion of it will be accepted at ASME Headquarters until September 30, 1994

Copyright © 1994 by ASME
INTRODUCTION

Very little is known about the flow field inside the automotive torque converters. Lack of information on this flow field could be attributed to the severe limitations and difficulties associated with the size, configuration and closely coupled blade rows in these torque converters. Some early experiments conducted by Adrian (1985) were performed in a modified torque converter geometry. The blade surface pressure measurements performed by By and Lakshminarayana (1991) and By (1992) and LDV measurements performed by Bahr et al. (1990) and Flack (1993) indicate the presence of highly three-dimensional flow field with large flow separation zones inside the torque converter components. The torque converter flow field is complex due to highly three-dimensional flow path geometries, large variations in the operating conditions and the viscous interactions between closely spaced blade elements. Improvements in the performance of the torque converter could be achieved by a systematic investigation of the flow field in these torque converters.

The objective of the experimental investigation reported in this paper is to improve the performance of the torque converter through a systematic experimental and numerical investigation. The focus of the program is on the measurement of the steady and the unsteady flow field upstream and downstream of each element of the torque converter. These measurements will provide insight into the flow phenomena in each blade component. Specifically, they will provide information on losses, torque and the interaction effects between each component. The flow field information could be incorporated into the design procedure to improve the torque converter performance and efficiency.

The steady and unsteady measurements were performed downstream of the stator using a miniature five-hole probe. However, only the steady state flow measurements are reported in this paper.

Experimental Facility and Instrumentation

The torque converter test facility used in this research is shown in Fig. 1.

Detailed description of the experimental facility and static pressure distribution on stator blade are given by By and Lakshminarayana (1991) and By (1992). This facility has five components: driver and absorption dynamometer, test section, oil system and control system. The driver dynamometer is a 30 HP dc motor which delivers the precise amount of torque or rpm specified by the control system and drives the pump. The turbine is connected to the absorption dynamometer, which is controlled to maintain constant torque or rpm. The cross section of the torque converter flow path is an elliptical torus formed by pump, turbine and stator blades. Inner boundary of the torus is called core and the outer boundary is called shell. It has been observed that high oil pressure prevents cavitation, and maintaining elevated temperatures enables stable flow conditions with repeatable performance. Hence an oil system is used to keep the oil pressure and temperature constant at 14 psi and 60°C, respectively. The performance of a torque converter is expressed in terms of torque ratio (TR) and speed ratio (SR), defined in the Nomenclature.
Tests were conducted at SR = 0.0, 0.2, 0.4, 0.6 and 0.8 in order to simulate all the conditions the torque converter experiences in actual automatic transmission. The design speed ratio of the torque converter is 0.6. It has a maximum efficiency at SR = 0.8 and minimum efficiency at SR = 0.0. Data at only these two speed ratios (0.0 and 0.8) will be reported here. The blade geometry, blade angles and inlet flow angles (obtained from one-dimensional analysis) at various speed ratios are presented in Fig. 2.

Fig. 2. Blade Profile, Inlet Flow Conditions and Measurement Stations

A miniature five-hole probe (1.67 mm tip diameter) was designed and fabricated to facilitate the measurements of the flow properties upstream and downstream of each of the blade rows. The measurements are carried out at 5 radial positions and 11 tangential positions in one blade passage at 0.36 axial chords downstream, as shown in Fig. 2. The radial position at the shell will be designated by H = 0.0 and the core will be designated by H = 1.0. Measurements were also performed at two additional tangential positions in an adjacent blade passage to check for periodicity of the flow field. Two static pressures on the stator were used as reference pressures for normalizing the total and static pressures obtained from the five-hole probe. The data processing technique for the five-hole probe is described by Treaster and Yocum (1979). A thermocouple was installed on the hub of the stator for accurate reference temperature. The probe calibration has an accuracy of ±1° in angle measurement. The pressure measurements are obtained to an accuracy of ±0.01 psi. The transducer drift introduces a maximum error of 0.5%. This results in a maximum 1-2% error in the velocities and 0.5% error in the static and total pressures. The correction to the probe due to blockage and the pressure gradient are incorporated into the data processing. The maximum cumulative error in the velocity and pressure measurements is 2-3%.

RESULTS AND DISCUSSION

The tangential and radial distribution of important flow parameters at the exit of stator 2 are discussed in this section. All the velocities are normalized by the mass averaged mean through flow (axial) velocity \( \bar{V}_x \). Total and static pressures measured by the five-hole probe are normalized by subtracting the reference hub static pressure and dividing it by the average dynamic head of the through flow \( \frac{1}{2} \rho \bar{V}_x^2 \).

Stagnation Pressure Coefficient at Stator Exit

The distribution of stagnation pressure coefficient \( C_{p_{st}} \) is shown in Fig. 3 for SR = 0.8. The presence of an inviscid core, where the stagnation pressure is nearly uniform is evident at this speed ratio. The shell region (H = 0.2) shows fully developed profile, caused by the thick boundary layers in the corner formed by the suction surface and the endwall. The secondary flow, discussed later in the paper, interacts with the blade boundary layer in the shell region, resulting in thick corner boundary layer or the corner separation (corner stall). The data presented later does not indicate any flow separation in this region. Hence the major phenomena here is the accumulation of low energy fluid swept by the secondary flow. However, substantial difference in the stagnation pressure distribution between H = 0.2 and 0.35 indicates that this phenomena does not extend beyond H = 0.35 and ends somewhere between these two locations. The flow periodicity is maintained at this speed ratio. The wake thickness varies from nearly 70% of passage at H = 0.2 (shell region) to about 50% of the passage at H = 0.5. The poor definition of wake near the core endwall can be attributed to the faster wake decay near the
core endwall. It is also possible that the boundary layer on the core endwall separates due to the low momentum fluid at the exit of the turbine near the core region. The wake width at midspan and shell is large compared to the wakes of nozzle vanes in other types of turbomachinery.

The distribution of stagnation pressure coefficient at SR = 0.0, shown in Fig. 3, reveals a large region of low stagnation pressure extending from the core to the shell. This low pressure region indicates massive flow separation near the suction side. The zone of low stagnation pressure and the wake region covers approximately 50% of the blade spacing at all radii. The inviscid core is less than 20% of the blade spacing at this speed ratio. In the inviscid core, the stagnation pressure increases slightly from the core to the shell, indicating an increase in the total velocity from the core to the shell. Lowest stagnation pressures and largest wake widths are observed near the midspan indicating a rapid deterioration of flow at midspan as compared to the core or the shell. Another important feature is that the center of the low pressure region is at a different tangential position for each radii, indicating that the wake is skewed. The reason for the skewness of the wake is discussed later in this paper. Due to high positive incidence angle (Fig. 2), the flow blockage as well as the flow turning are high (approximately 120°) at this speed. As a result, the blade boundary layers are thick on the suction side at this speed ratio. This is also confirmed by the numerical analysis, presented later. The flow separation near the trailing edge of the stator shifts the position of the wake. Also, the flow blockage due to separation affects the secondary flow structure induced by the radial gradient of through flow velocity upstream of the stator. The interaction of the blockage caused by the flow separation, secondary flow induced by the upstream normal vorticity and the secondary flow induced by endwall boundary layers generate a very complex flow structure at this speed ratio.

The radial distribution of the mass averaged stagnation pressure coefficient is plotted in Fig. 4.

Fig. 3 Profiles of Total Pressure Coefficient

Fig. 4. Radial Variation of Mass Averaged Pressure Coefficient ($C_{po}$ and $C_{ps}$)
Fig. 4. The average stagnation pressure coefficient at SR = 0.8 is nearly uniform, with the exception of the hub and the tip regions and this is caused by the shell and the core boundary layers. At SR = 0.0, the stagnation pressure coefficient decreases from the shell to the core, indicating high losses near the core. These losses could be due to the flow separation on the core endwall of the upstream turbine rotor. However, this hypothesis cannot be confirmed due to lack of information on the flow field upstream of the stator. The axial velocities in this region, described later, are also found to be small.

**Static Pressure Coefficient at Stator Exit**

The radial distribution of the mass averaged static pressure coefficient, plotted in Fig. 4, shows very non-uniform distribution. For SR = 0.8, the static pressure increases with radii. Very high static pressure gradients are observed between H = 0.2 and 0.35 and H = 0.5 and 0.65. At SR = 0.0, the static pressure coefficient increases from location H = 0.2 to 0.35 and decreases from H = 0.35 to 0.8. Physical reasoning for such a distribution is given below:

The fluid particles experience centrifugal forces in two directions as shown in Fig. 5. It should be noted that the streamlines have curvature in meridional as well as azimuthal directions. As a result, a simplified radial equilibrium equation cannot be used to determine the radial pressure gradient. A streamline curvature equation should be used to calculate the pressure gradient. This equation is given by:

\[
\frac{\partial P}{\partial r} = \rho \frac{V_0^2}{r} - \rho \frac{V_x^2}{(R_{nc} - r)} \cos \Psi_M
\]  

(1)

where \( R_{nc} \) is the distance between the center of torus (or center of curvature of the streamlines in torus plane, as shown in Fig. 5), and the axis of rotation; \( r \) is the radius from the axis of rotation and \( \Psi_M \) is the angle between the streamlines and axial flow direction at exit of the stator. This equation can be integrated to obtain the static pressure distribution.

A comparison of the radial distribution of the static pressure coefficient derived from the streamline curvature equations and the experimental data, shown in Fig. 4, reveals good agreement between the estimated and the measured distribution of \( \bar{c}_{ps} \) for SR = 0.0. The positive pressure gradient between H = 0.2 and 0.35 and negative pressure gradient between H = 0.35 and 0.8 are predicted accurately for SR = 0.0. The estimation is poor for SR = 0.8. The pressure gradient for SR = 0.8 between H = 0.35 and 0.5 and H = 0.65 and 0.8 are estimated accurately, even though the magnitudes are not predicted well near the core. However, the estimation of the gradient between H = 0.2 and 0.35 and H = 0.5 and 0.65 is poor. This confirms the influence of both the meridional and the azimuthal curvatures on the radial static pressure gradient measured.

**Velocity Profiles at the Stator Exit**

The distribution of non-dimensionalized total velocity behind the stator blade is shown in Fig. 6 for SR = 0.8 and 0. The velocity distribution is nearly uniform over the entire blade passage at speed ratio 0.8. At SR = 0.8 the stator wake is thin due to thin boundary layers on the blade surface and the wake decays rapidly due to high turbulence and mixing of the flow downstream of the stator. As indicated earlier, the wake thickness is about 70% of passage at H = 0.2 and is relatively thin at other locations.

At SR = 0.0, the wake trajectory is skewed from the core to the shell (Fig. 6). The wake is
very thick at this speed and it occupies nearly 70-80% of the blade passage in the entire annulus. However, it should be noted that the flow mixes rapidly downstream of the blade and the velocity defect varies from 25-50% of the free stream velocity. It is also clear, despite the fully developed nature of the flow, the total velocity distribution is periodic in the blade-to-blade direction.

The radial distribution of the mass averaged non-dimensional total velocity (Fig. 7) shows that the average total velocity decreases from the shell to the core, especially at SR = 0. The low velocity region near the core is attributed to the separation on the core surface in turbine rotors. Flack et al. (1993) have also observed a similar variation of the flow velocity downstream of the pump and turbine in the torque converter. The dominant component at the stator exit is the tangential velocity, whose mass averaged values are also shown in Fig. 7. Large values of $V_t$ and $V$ in the shell region at SR = 0 are caused by secondary flow, which induces overturning. The low velocity region at midspan at SR = 0 is due to flow separation on the suction side as shown in Fig. 6. The tangential velocity distribution at SR = 0.8 indicates no major influence of secondary flow. It should be noted here that no measurements are taken from H = 0 to 0.2, and from H = 0.8 to 1.0. At SR = 0.0, the tangential velocity is lower at midspan and higher near the core and the shell.

The radial distribution of the mass averaged axial velocity (Fig. 7) indicates an almost linear variation in the axial velocity from the shell to the core for SR = 0.0 and 0.8. The mass averaged axial velocity decreases from the shell to the core. This observation is in conformity with the data acquired by Flack et al. (1993). The distribution of mass averaged axial velocity is identical for SR = 0.8 and
0.0, except at H = 0.2. The radial distribution of the mass averaged axial velocity was also found to be identical for the speed ratios 0.6, 0.4 and 0.2. Thus the change in incidence angle, speed ratio and the Reynolds number does not affect radial distribution of average axial velocity. Hence, it can be concluded that the non-dimensionalized through flow velocity distribution is dependent only on the geometric parameters and not on the operating conditions.

**Yaw Angle Profiles at Stator Exit**

The contours of yaw angle (measured from the axial direction) are shown in Fig. 8. As indicated in Fig. 2, the blade outlet angle is 63°. The distribution at SR = 0.8 indicates that the flow angles are nearly uniform across the passage, with large variations occurring across the wake. The turning is lower on the pressure side and higher on the suction side of the wake and this is consistent with measurements across the wake in a compressor reported by Prato and Lakshminarayana (1993).

The radial distribution of the average yaw angles (Fig. 9), calculated from the mass averaged velocities (Fig. 7), indicates a smooth variation at SR = 0.8. The radial distribution of yaw angles at SR = 0.0 indicates high yaw angles at both the core and the shell regions and low yaw angles at H = 0.35. This type of distribution can be attributed to the flow separation zone on suction surface of the blade near H = 0.35. Also the secondary flow generated by the shell boundary layer causes overturning at H = 0.2 and underturning at H = 0.35. The details of this flow structure are discussed earlier in the paper and will not be repeated here.

**Radial Velocity Profiles at Stator Exit**

The contour plots of radial velocity are shown in Fig. 10. For SR = 0.8, negative radial velocities (inward) are observed in the wake regions. These velocities are induced by a strong positive
static pressure gradient (toward the shell) discussed earlier in the paper (Fig. 4). Small positive radial velocities are observed in the inviscid core. These velocities result from primary inviscid flow features. The radially positive velocities in the inviscid core region and the radially negative velocities inside the wake region result in a strong secondary flow pattern near the blade regions.

The radial velocity distribution at SR = 0 shows some complex features. The negative radial velocities induced by the secondary flow are observed only in a small region located near the core, near the pressure surface. Most of the blade passage shows high positive radial velocities (towards the core). The blockage, generated by the flow separation, causes the flow to go around the separation zone inducing high radial velocities in the inviscid core. Also, since the pressure gradient is negative (toward the core), positive radial velocities are induced in the separation and the wake regions.

As a result, the radial distribution of the mass averaged radial velocity, shown in Fig. 7, are much higher at SR = 0.0 than at SR = 0.8. For SR = 0.8, the mass averaged radial velocity increases linearly with radius from the shell to the core. This increase is due to the differences in the torus curvature from the shell to the core. For SR = 0.0, the radial distribution of average radial velocity is uniform except at H = 0.35. This discrepancy is attributed to high radial velocities in the region surrounding the separation zone. For SR = 0 at H = 0.35, \( \frac{\partial p}{\partial r} \) changes sign. It is positive for H < 0.35 and negative for H > 0.35 (Fig. 4).

The radial distribution of average pitch angles calculated from the mass averaged velocities are shown in Fig. 9. The average pitch angles increase from the shell to the core due to differential bending of the stream lines in torus plane. The pitch angles derived from mass averaged velocity vary by approximately 10° from the shell to the core for both SR = 0.8 and SR = 0.0. The pitch angles are generally higher for SR = 0.0.

As mentioned earlier, the radius of the torus introduces three-dimensionality as evidenced by the magnitude of the radial velocities and pitch angles. Hence, some of the three-dimensionality is due to inviscid effects. The fact that the pitch angles are substantially higher at SR = 0 indicates the presence of highly three-dimensional nature of the flow at zero speed ratio.

Wake Decay

The defect in velocity inside the wake depends on several factors including the axial distance from the trailing edge, turbulence level, Reynolds number, secondary flows and the pressure gradients. A knowledge of the stator wake is essential for improvement in the performance of torque converters, as these interact with the pump rotor resulting in unsteady flow and vibration. The wake width and the maximum velocity defect are of interest. Magnitude of wake width was discussed earlier. The maximum velocity defect (normalized the local maximum velocity) is shown plotted as a function of distance from trailing edge in Fig. 11. It is also compared with Raj and Lakshminarayana's (1973) correlation for a cascade, Zaccaria et al. (1991) nozzle wake data, Sitaram and Govardhan's (1986) rotor cascade data, as well as Goldman and Seaholtz (1992) data on annular turbine cascade. The wake decay at SR = 0.8 follows the trend observed in other cascades, but the decay of the wake is slower at SR = 0.6. This is due to high inlet incidence and blade loading.
Composite Flow Field at the Exit of the Stator

A schematic of the composite flow field, based on the data presented earlier, is shown in Fig. 12. The flow field at SR = 0.8 is well behaved. The wake is thin and uniform at all radii except near the shell at H = 0.2. The secondary flow causes flow overturning near the core. The underturning near the shell is due to possible corner separation. The positive static pressure gradient induces radial inward velocities in the wake and curvature of the streamlines in azimuthal direction causes radial outward velocities in the inviscid core region. As a result, a weak secondary flow structure could be observed, as shown in Fig. 12.

At SR = 0.0, the flow field can be generally divided into four regions. The wake region, with possible separation, is the largest region near the suction surface. The secondary flow induced by the upstream radial gradient in velocity interacts with the wake near the core to generate an interaction zone where radial inward velocities and flow overturning are observed. The flow mixing adjacent to the wake entrains the main flow, causing large secondary flow and mixing. These zones are marked as wake mixing zones. The inviscid core is small and exhibits high radial velocities due to the curvature of streamlines in the meridional direction. The displacement of streamlines, due to the flow blockage, also contributes to the radial velocities.

Fig. 11 Decay of Total Velocity Defect with Streamwise Distance (H=0.5)

Fig. 12 Schematic of Secondary Flow Pattern

The flow field at the stall condition (SR = 0.0) is highly three-dimensional at the exit of the stator. The incidence angle is approximately 60° and the turning angle is about 120°. The mass flow rate is maximum and the secondary flow is very strong, the flow separation and wake dominate the flow in the entire passage at this speed ratio. The flow near the middle span is no longer two-dimensional, but the flow properties at the inlet of the stator are more uniform than the design condition since the turbine is stationary.

PREDICTIONS FROM NAVIER-STOKES ANALYSIS

A two-dimensional steady, incompressible Navier-Stokes code was used to predict the flow field at midspan of the stator. The analysis and code, which utilizes a pressure based method, was developed by Basson and Lakshminarayana (1993) at Penn State. The artificial dissipation terms are included directly and controlled for each case. A k-ε low Reynolds number turbulence model, due to Chien (1982) was employed. The flow field was computed for all speed ratios. The prediction for the design point (SR = 0.6) and extreme off-design
conditions (SR = 0) are presented and interpreted in detail. The incidence angle and velocity are specified from the one-dimensional analysis. It is assumed to be steady and uniform across the inlet plane located far upstream. The computation does not include unsteadiness in turbine exit flow field.

The incidence angle is 9° at the design point (SR = 0.6), the turning angle is about 65° and the secondary flow is relatively weak. The flow near the leading edge is relatively smooth and there is no flow separation. Furthermore, the wake is thin and the flow is nearly two-dimensional in the middle third of the span. Hence the predictions from the two-dimensional Navier-Stokes code should be valid near the midspan at design conditions.

The blade surface pressure distribution at the design (SR = 0.6) and the stall conditions (SR = 0.0) are shown in Fig. 13. The agreement between the data (By and Lakshminarayana, 1991) and the prediction is excellent for both the design and the off-design conditions. Small discrepancies near the leading and trailing edges are caused mainly by the lack of sufficient grid points. The grid size was systematically decreased to obtain improved performance.
The measured and predicted stagnation and static pressure coefficients for SR = 0.6 are compared in Fig. 14. The agreement between measured and predicted \( C_{p0} \) is very good except in the wake region. The stagnation pressures are captured exactly in the inviscid region. The static pressure on the other hand was predicted accurately in the wake region and discrepancies exist in the inviscid region. Both the data and the computation show that the static pressure is almost uniform in the tangential direction at this location (0.36 axial chord length downstream). Considering the complexity of the flow, these predictions provide confidence in the ability of Navier-Stokes codes to simulate the flow field.

The stagnation and static pressure coefficients at the exit of the stator at SR = 0 are shown in Fig. 14. The stagnation pressure coefficients are predicted reasonably well including the wake region which covers nearly 70% of the passage. The prediction of the inviscid region agrees well with the data, but the maximum defect in the wake is overpredicted. The static pressure coefficient is not predicted well.

The measured and computed velocity profiles at SR = 0.6 are shown in Fig. 15. Here again, the agreement between predicted and measured velocity components is excellent. The flow angle, the axial, tangential and the total velocity components are predicted well both in the inviscid region and in the wake. The velocities are nearly uniform in the inviscid region at this operating condition. The axial, tangential and the total velocity profiles for SR = 0 are shown plotted in Fig. 15. The agreement with axial velocity is excellent and surprisingly, the total velocity prediction is good in the wake but deviates appreciably in the inviscid region.

The flow field in a torque converter is inherently unsteady due to close coupling of stator-pump, pump-turbine and turbine-stator. One of the major causes of unsteadiness is the wake of stator interacting with the pump rotor, and hence an attempt is made to assess the predictive capability of the code to resolve the wake structure. The maximum velocity defect in the wake at SR = 0.6 is shown compared with data from other turbine stators, nozzles and cascades and correlations in Fig. 11. The predicted wake defect compares very well with the measured data at SR = 0.6. No attempt is made to carry out such correlation for SR = 0, as the flow is highly three-dimensional and two-dimensional codes are not accurate at this speed.
The predicted stagnation pressure coefficients and velocities are used to compute the overall stagnation pressure loss at mid radius, using the equation:

\[
\xi = \frac{1}{P_{01}} \int_0^1 P_{01} V_{x_1} \, d(S) - \frac{1}{P_{02}} \int_0^1 P_{02} V_{x_2} \, d(S)
\]

\[
\int_0^1 P_{01} V_{x_1} \, d(S)
\]

The predictions are shown compared with the data (based on \(P_{01} = P_{o_{max}}\) at station 2) in Fig. 16. The agreement near the design condition (SR = 0.8 and 0.6) is excellent as the code is exactly valid at this speed ratio, as the flow is two-dimensional at mid radius. The agreement is poor at SR = 0.4, 0.2 and 0 and one of the major reasons for this is the fact that the flow is highly three-dimensional even at mid radius at this speed ratio.

**CONCLUSIONS**

A comprehensive investigation of the flow field at the exit of the stator is reported in this paper. The following conclusions are drawn from the data presented:

1) At low flow turning angles (9° at SR = 0.8), the growth of the blade surface boundary layer and the resulting wakes are thin. The wake decays rapidly before it reaches the measurement plane. At higher flow turning angles (120° at SR = 0.0) the boundary layer growth on the suction surface is large and the wake width covers a large extent of the blade passage. The defect in maximum velocity in the wake agrees well with the data from similar blade rows. The wake width is found to be a substantial portion of the blade passage, and this will generate large unsteadiness in the pump blade passage.

2) The through flow velocities are highest near the shell and lowest near the core. This type of through flow velocity distribution is similar to those observed in other mixed flow turbomachinery. The core-to-shell distribution of the through flow velocity is identical at the design and the off-design conditions.
3) The flow overturning and underturning are observed adjacent to wakes at both the speed ratios.

4) The stagnation pressure, static pressure and velocities are found to be nearly uniform from pressure to suction surface, with the exception of wakes, at SR = 0.8 and 0.6. Substantial variation is observed only near the shell. On the contrary, large non-uniformities are observed in distribution of flow property for SR = 0.0.

5) Negative radial velocities are observed inside the wake and this is due to radial gradient in static pressure. Positive radial velocities are observed in the inviscid core. This is due to the meridional streamline curvature. The radial velocity and hence three-dimensionality in the flow are found to be appreciable for SR = 0.8 and substantial for SR = 0. The maximum variation in the radial velocity for the design condition is found to be 10% of the total velocity.

6) The radial distribution of static pressure is influenced not only by the swirl velocity but also by the meridional streamline curvature.

7) The flow at the midspan is predicted accurately at the design condition by the two-dimensional Navier-Stokes code. The blade-to-blade distribution of the stagnation and static pressure coefficients, axial and tangential velocities are predicted accurately. At off-design conditions (SR = 0), the axial velocity and stagnation pressure are predicted reasonably well, with qualitative prediction for the static pressure. The computed and measured losses at midspan are in agreement with the data at SR = 0.8 and 0.6, but the comparison is not good at SR = 0.0.

ACKNOWLEDGEMENT

This project was sponsored by Hydromatic Division and Technical Center of General Motors Corporation. Helpful discussions and assistance in the experimental program by D. Maddock and R. By are gratefully acknowledged. J.F. Gallardo's help in computational effort is also acknowledged.

REFERENCES


Flack R. et al., 1993, Personal Communication, University of Virginia


