N72-32959

~~N72-27720~~

NASA TECHNICAL MEMORANDUM

NASA TM X-67959

NASA TM X-67959

QUASI-THREE-DIMENSIONAL CALCULATION OF VELOCITIES IN TURBOMACHINE BLADE ROWS

by Theodore Katsanis
Lewis Research Center
Cleveland, Ohio

TECHNICAL PAPER proposed for presentation at Winter Annual Meeting of the Americal Society of Mechanical Engineers
New York, New York, November 12-16, 1972

E-6666

QUASI-THREE-DIMENSIONAL CALCULATION OF VELOCITIES IN TURBOMACHINE BLADE ROWS

by Theodore Katsanis

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio

ABSTRACT

A practical way of obtaining an approximation to a three dimensional flow is to combine several two-dimensional solutions (quasi-three-dimensional solution). This paper discusses three basic types of two-dimensional solutions (meridional, blade-to-blade, and channel) and several ways of combining them. A centrifugal impeller is analyzed as an example. All recommended methods are based on available NASA general purpose computer programs.

INTRODUCTION

The design of blades for compressors or turbines requires analysis methods for flow that is usually three-dimensional. Because of the difficulty of obtaining a true three-dimensional solution, the usual approach is to combine several two-dimensional solutions. This is often called a quasi-three-dimensional solution. Since there are several choices of two-dimensional surfaces, and even more ways of combining them, there are many approaches to obtaining a quasi-three-dimensional solution. Most two-dimensional solutions are either on a blade-to-blade surface of revolution or on the meridional (radial-axial) plane. However, when three-dimensional effects are most important, neither a blade-to-blade nor a meridional plane solution can be expected to give good results alone. In this case, significant information can often be obtained from a solution on a passage cross-sectional surface (i.e. a surface normal to the flow). This is called a channel solution.

To obtain maximum information analytically for a three-dimensional flow, it is obviously desirable to combine different two-dimension solutions. This paper discusses these techniques, with an emphasis on using a channel solution where there are strong three-dimensional effects. Since the best technique depends on the blade aspect ratio, solidity and other factors, analysis procedures are suggested for several types of turbomachines. As an illustration of a quasi-three-dimensional analysis, a six to one pressure ratio centrifugal impeller is analyzed, using a combination of all three analysis methods.

GENERAL DESCRIPTION OF TWO-DIMENSIONAL METHODS

Generally there are three main types of two-dimensional analysis methods. These are the meridional plane analysis, the blade-to-blade analysis, and the channel analysis. All three methods have been programmed for computer solution. The two-dimensional meridional plane and blade-to-blade analysis methods have been widely used on many types of turbomachines. The channel analysis has been used extensively in the past for axial flow turbines. However, now the technique has been generalized and can be used on other types of turbines and compressors.

There are four fundamental assumptions for all the methods described here. These assumptions are

(1) The flow is steady relative to the blade.

(2) The fluid is a perfect gas with constant Cp.

(3) The fluid is nonviscous.

(4) The flow is absolutely irrotational.

Additional assumptions are made for each particular method.

Meridional Plane Solution

A meridional plane solution is a flow solution on a stream surface between the blades. The shape of this surface (see figure 1) is often taken to be the same as the mean blade surface, and the tangential thickness is taken to be the tangential space between blade surfaces. The solution can be obtained either by a finite difference solution (reference 1) or by using the quasi-orthogonal method (reference 2). The quasi-orthogonal solution, using the velocity gradient equation has been programmed for radial turbines with radial blade elements (reference 2) and for axial or centrifugal compressors of fairly general blade shape (reference 3). The quasi-orthogonal solution is not limited to subsonic flow, but may be locally supersonic. The most serious limitation of the quasi-orthogonal method is imposed by the difficulty in obtaining convergence of the iterative procedure with high aspect ratio blades. Presumably, this type of geometry could be analyzed by the finite-difference stream function method. At the present time there is not believed to be any generally available computer program of this type. Hopefully, this type of program will be available in the near future.

Blade-To-Blade Solution

Figure (2) shows a typical blade-to-blade surface of revolution, and figure (3) illustrates how the normal thickness can vary. The blade-to-blade solution can be obtained by means of a finite difference solution of the stream function equation. Several NASA computer programs are available for this purpose.

These programs will obtain a compressible flow solution on any blade-to-blade surface of revolution. Reference 4 presents a FORTRAN program (TURBLE) which will give a solution for subsonic, compressible, nonviscous flow through either an axial, radial or mixed flow stator or rotor. Other programs have been published for tandem blades (TANDEM) (reference 5), and to obtain a detailed leading edge, or slot region solution (MAGNFY) (reference 6). A summary of these programs with some examples, is given in reference (7). Generally, the stream function solution can be obtained only if the flow is completely subsonic. However, by using velocity gradient equations, it is possible to extend a high subsonic solution to obtain a locally supersonic (transonic) solution. This technique has also been programmed. This program, called TSONIC, has been published (reference (8)).

A limitation of the blade-to-blade analysis is that the normal stream-sheet thickness must be specified throughout the passage. This thickness is determined by the three-dimensional nature of the flow, so that this thickness cannot be determined precisely with two-dimensional solutions. However, this thickness can often be estimated reasonably with a channel flow solution or with a meridional plane solution. This will be discussed further later on.

Channel Solution

The channel solution is a solution on a surface across the channel bounded by hub, shroud, and blade surfaces on all sides (fig. 4). The solution is based on using the velocity gradient equation both from blade-to-blade and from hub to tip. Although this solution is based on simplifying assumptions about how streamline curvature varies across the passage, good results can be obtained. The method has been used successfully in axial turbine design and analysis (ref. (9) and (10)). Recently the method has been generalized to be used on any guided turbine passage (ref. (11)). The solution procedure is simple, and there are no convergence problems. Also, slightly supersonic flow is no particular problem. However, shocks or Prandtl-Meyer expansion will not be indicated by this method.

The velocity distribution can be obtained in this way for the guided channel formed by the portion of the passage where the orthogonal surface is completely within the blades. The guided channel will not cover the entire suction surface. To obtain the velocities on the uncovered portion of the blade, the location of the stagnation streamline would have to be known. Therefore, the channel method cannot be used to obtain velocities on the uncovered portion of the blade. However, the blade-to-blade analysis can be used for this problem.

In addition to the basic assumptions of steady flow of a perfect, non-viscous gas, it is necessary to make assumptions as to how streamline curvature varies across the passage, both blade-to-blade and hub-to-tip. The usual assumption is that either the streamline curvature or radius of curvature varies linearly. The curvature at the boundaries, of course, corresponds to the curvature of the physical surface.

USE OF TWO-DIMENSIONAL METHODS

There are several combinations of 2-D analysis methods that will give good results. The technique to use depends on whether the flow is radial, axial, or mixed; on the blade aspect ratio, solidity, and wall curvatures; and on hub-to-tip radius ratio. To be helpful in obtaining maximum use from available NASA computer analysis programs, some suggested analysis procedures are presented below. These procedures are intended to be useful as an iterative design procedure. The starting point for any blade design should be accurate velocity diagrams. There are programs available for this important part of the blade design. For example, see ref. 12 for axial turbines, and ref. 13 for axial compressors.

Accurate results from any of the analysis methods discussed depends on the accurate estimation of losses. There are two main approaches to estimating losses. One approach is to estimate the overall loss for the entire cross section. Such loss estimates can be obtained from performance data correlated with parameters such as blade jet speed ratio, flow coefficient and reaction; e.g. ref. 14 and 15. The other approach is to estimate the boundary layer growth, and use the displacement thickness to adjust the physical boundaries. For rotating blades, centrifugal force has a significant effect on the boundary layer, so that a 3-D boundary layer analysis becomes desirable.

Because of the difficulties with a realistic boundary layer analysis, it is more usual to use the overall loss, expressed as a loss in relative total pressure. Within the blade, the loss can be assumed distributed linearly from the blade leading edge to trailing edge.

Pure Axial Flow

For pure axial flow, the flow can be analyzed with the use of the TSONIC program of reference 8. For the analysis problem it is necessary to make a layout to determine enough geometrical data to define the blade shape adequately. Often there will be a significant radial streamline shift through the blade row, even with constant radius hub and shroud. To allow for this effect, velocity diagram information should be used to determine the radial shift and change in streamsheet thickness through the blade row. The analysis may be done at three or more sections from hub to tip to determine a complete blade surface velocity distribution.

While the TSONIC program is an analysis program, it is easier in many ways to use it as a design program. No blade layout is necessary in the design problem, since the geometrical input data can be obtained without a layout. Minimum geometrical input is recommended for blade design. The minimum geometrical input consists of blade axial chord, leading- and trailing-edge radii, stagger, and tangent angles to leading- and trailing-edge radii for both suction and pressure surfaces. Usually, this information can be determined without a layout so that a satisfactory velocity distribution results; i.e. proper diffusion is obtained and surface velocity curves for suction and pressure surfaces close at leading and trailing edge. In some cases it may be necessary to specify a point on one blade surface to control a throat dimension. If this procedure is followed at three or more sections from hub to tip, a complete blade design is obtained.

After the complete analysis is obtained, the choking mass flow should be checked for any blade that is choked, or close to it. This cannot be readily determined from the blade-to-blade analysis, since the radial distribution of flow becomes critical. Therefore, the channel flow analysis (CHANEL program, ref. 11) should be used for this purpose. This will require a blade layout to determine the lengths of blade-to-blade streamline orthogonals. It is usually necessary to check only a few orthogonals to verify the choking mass flow and to see if the normal thickness used in TSONIC is correct. Any discrepancy may be corrected by rerunning the TSONIC program with the necessary corrections.

If desired, a blade designed or analyzed with the TSONIC program may have offset coordinates calculated with respect to the true blade chord, using the program of reference 16.

Mixed Axial Flow With Low Aspect Ratio Blades

When there is substantial change in radius through the blade row (either stator or rotor) it is more difficult to determine the proper normal streamsheet thickness to use for the blade-to-blade analysis. This thickness could be obtained from the CHANEL program, but this necessitates the layout of the blade shape at all sections from hub-to-tip. Rather than do this at an early stage, it would appear better to use a meridional plane analysis, such as the quasi-orthogonal analysis of reference 2 or 3. If the aspect ratio is not high, and wall curvatures are not too large, this will converge to a solution with meridional streamline locations. This gives sufficient information on the normal streamsheet thickness to be able to use the blade-to-blade TSONIC program. From this point the procedure is very similar to that for a pure axial blade.

It should be cautioned that the layout procedure to determine input information for the CHANEL program can become difficult if there is a large deviation of the flow from the axial direction combined with substantial hub or tip wall curvature. In this case the blade-to-blade surface is not developable and cannot be laid out on a flat surface with correct angles and linear measurements. There are several ways that a nondevelopable blade-to-blade surface of revolution can be laid out. However, these layouts are distorted, so that corrections must be made to determine orthogonal directions; also, distance measurements from the layout may require a graphical integration procedure. Although these calculations could be done with a computer program requiring no layouts such a program is not available at present.

Mixed Axial Flow With High Aspect Ratio Blades

The problem becomes more difficult when there is a high aspect ratio, combined with substantial radius change. It is desirable to start with a meridional plane solution, but the Q.O. program of ref. 2 or 3 most likely would not converge. A finite difference solution in the meridional plane (ref. 1) would be desirable, but such a program is not generally available at present. The only alternative is to work between the TSONIC program and the CHANEL program. This may be difficult, since there may be very little or no guided channel. Ultimately, you may be reduced to guesswork.

Mixed Radial Flow Compressor or Turbine

The most difficult analysis problem is the mixed radial flow compressor or turbine, because the flow is highly three dimensional with strong secondary flows. However, a rough analysis is possible using the quasi-orthogonal meridional plane solution (references 2 and 3). The blade-to-blade analysis (reference 6) can be used then, if desired. The quasi-orthogonal meridional solution provides information of limited accuracy because the blade-to-blade variation of velocity is not considered in satisfying continuity. The channel analysis is useful for this purpose since both the blade-to-blade and the hub to tip variation in velocity is considered. Especially if there is any possibility of choking, the channel analysis should be used. The limitation in the accuracy of predicting choking mass flow is controlled by the accuracy with which boundary layer displacement thickness can be estimated. It is possible, of course, that the boundary layer may be separated, so that this is a difficult problem. Reference 17 presents a theory on separated flow in centrifugal compressors, with suggestions on analysis procedures.

An example 6 to 1 pressure ratio centrifugal compressor will be analyzed to show the results which can be obtained.

CENTRIFUGAL IMPELLER ANALYSIS

One of the areas where flow analysis is difficult is in a centrifugal impeller. The flow is highly three dimensional, since the flow changes from axial to radial in a short distance. To obtain a high pressure ratio it is also necessary to have high blade loading and transonic velocities with a high tip speed. For optimum efficiency it is also necessary to have backswept blades. The resulting design is difficult to analyze. To illustrate this type of problem and show what results can be obtained, a backswept centrifugal impeller with a 6 to 1 pressure ratio is discussed here. This example is taken from reference 3. Figure 5 shows the meridional profile and figure 6 shows the mean camber line for the blade at the hub and shroud. This blade was designed so that the contours of constant θ in the meridional plane are straight lines.

The hub and shroud blade surface velocities which were obtained from the QUAC program of reference 3 are shown in figure 7. Only the mean velocities are obtained by the meridional analysis. From the mean velocities, the blade loading is approximated by Stanitz' method (reference 18). Stanitz' method is based on the assumption of linear variation of velocity from blade-to-blade with absolutely irrotational flow.

A blade-to-blade analysis at hub and shroud was made using the TSONIC program of reference 8. The results of this analysis is also shown in figure 7. Several problems arise in using the TSONIC program here. First, the exit flow angle must be specified as input to the TSONIC program. Because of slip the exit flow angle will deviate substantially from the blade exit angle. The exit flow angle for this example was estimated using a slip factor calculated by Wiesner's method, reference 19. Second, at the shroud, velocities for the inlet portion of the blade become fully supersonic (not just transonic). Therefore, the TSONIC program did not obtain the proper solution for the first part of the blade at the shroud. This is the reason why the velocity distribution from TSONIC is incomplete on figure 7(b). Finally, the velocities obtained by TSONIC depend strongly on the normal stream sheet thickness. This is not known accurately, but is only approximated from the meridional plane solution. Since the inlet is nearly choked it is critical to get the right meridional streamline spacing.

For these reasons it was better to get the blade loading and final velocities by using the CHANEL program within the guided portion of the blades. Because of the complex geometry it was required to make detailed layouts of the blade at the hub, mean and tip to determine blade-to-blade streamline normal lengths. The meridional normal lengths were obtained from the meridional profile, figure 5. The blade-to-blade and hub-to-shroud normal lengths together determine the blade passage flow area.

Also required as input for the CHANEL program was the hub and shroud wall curvatures, and the blade surface curvatures. The hub and shroud wall curvatures were obtained as part of the QUAC program output. The blade surface curvatures were obtained from the output of the TSONIC program, interpolating as necessary. These curvatures help determine the blade loading and the variation of velocities from hub to tip.

The results of the CHANEL program analysis are shown in figure 7. It will be noted that the blade loading increases right up to the end of the passage. This is not realistic, since in a radial impeller there is slip near the tip, resulting in a reduced loading at the last portion of the blade. This reduced loading is not easily predicted by the CHANEL program. However, qualitative results can be obtained by referring to the TSONIC output shown in figure 7. Here, by using the proper outlet flow angle as input, the blade loading is predicted to fall rapidly near the trailing edge. The fact that the loading tends to zero at the trailing edge is an indication that the correct exit flow angle was used.

Since the CHANEL program considers variation in both hub-to-shroud and blade-to-blade directions simultaneously, it is considered to be closest to a theoretical three-dimensional solution. The TSONIC program shows qualitatively the rapid unloading of the blade near the exit. Overall, there is a fair agreement between the three methods as to the general behavior of the flow, but with fairly large differences in some areas. These differences occur because of the three dimensional nature of the flow and are indicative of the uncertainty of the analysis of three dimensional flow by two dimensional methods.

CONCLUDING REMARKS

Three types of two-dimensional analysis methods have been described. The proper use of these methods will give useful information on flows which are not two-dimensional. Particularly useful for three dimensional flows is the channel method which gives very good results in a well-guided channel, even with strong three-dimensional flow, if losses can be estimated reasonably. In other cases, proper combinations of two-dimensional solutions can be very helpful in the absence of a true three dimensional analysis computer program.

Because of the complexity of the solution procedures for any of the two-dimensional methods, it is not practicable for each blade designer to write his own computer program for each method. Hence, the methods proposed for actual use is limited to those for which a general purpose computer program is available.

REFERENCES

1. Marsh, H., "Digital Computer Program for the Through-Flow Fluid Mechanics in an Arbitrary Turbomachine Using a Matrix Method," ARC R&M No. 3509, 1968, Aeronautical Research Council, Great Britain.

2. Katsanis, T., "Use of Arbitrary Quasi-Orthogonals for Calculating Flow Distribution in the Meridional Plane of a Turbomachine," TN D-2546, 1964, NASA, Cleveland, Ohio.

3. Vanco, M. R., "FORTRAN Program for Calculating Velocities in the Meridional Plane of a Turbomachine. Part I. Centrifugal Compressor," TN D-6701, 1972, NASA, Cleveland, Ohio.

4. Katsanis, T. and McNally, W. D., "Revised FORTRAN Program for Calculating Velocities and Streamlines on a Blade-to-Blade Stream Surface of a Turbomachine," TM X-1764, 1969, NASA, Cleveland, Ohio.

5. Katsanis, T. and McNally, W. D., "FORTRAN Program for Calculating Velocities and Streamlines on a Blade-to-Blade Stream Surface of a Tandem Blade Turbomachine," TN D-5044, 1969, NASA, Cleveland, Ohio.

6. Katsanis, T. and McNally, W. D., "FORTRAN Program for Calculating Velocities in a Magnified Region on a Blade-to-Blade Stream Surface of a Turbomachine," TN D-5091, 1969, NASA, Cleveland, Ohio.

7. Katsanis, T. and McNally, W. D., "Programs for Computation of Velocities and Streamlines on a Blade-to-Blade Surface of a Turbomachine," Paper 69-GT-48, 1969, ASME, New York, N.Y.

8. Katsanis, T., "FORTRAN Program for Calculating Transonic Velocities on a Blade-to-Blade Stream Surface of a Turbomachine," TN D-5427, 1969, NASA, Cleveland, Ohio.

9. Stewart, W. L., Whitney, W. J., and Schum, H. J., "Three-Dimensional Flow Considerations in the Design of Turbines," Paper 59- Hyd -1, 1959, ASME, New York, N.Y.

10. Katsanis, T. and Dellner, L. T., "A Quasi-Three-Dimensional Method for Calculating Blade Surface Velocities for an Axial Flow Turbine Blade," TM X-1394, 1967, NASA, Cleveland, Ohio.

11. Katsanis, T., "FORTRAN Program for Quasi-Three-Dimensional Calculation of Surface Velocities and Choking Flow for Turbomachine Blade Rows," TN D-6177, 1971, NASA, Cleveland, Ohio.

12. Carter, A. F. and Lenherr, F. K., "Analysis of Geometry and Design-Point Performance of Axial-Flow Turbines Using Specified Meridional Velocity Gradients," CR-1456, 1969, NASA, Washington, D.C.

13. Creveling, H. F. and Carmody, R. H., "Axial Flow Compressor Design Computer Programs Incorporating Full Radial Equilibrium. Part I: Flow Path and Radial Distribution of Energy Specified," EDR-5845, NASA CR-54532, June 1968, General Motors Corp., Indianapolis, Ind.

14. Stewart, W. L., "A Study of Axial-Flow Turbine Efficiency Characteristics in Terms of Velocity Diagram Parameters," Paper 61-WA-37, 1961, ASME, New York, N.Y.

15. Smith, S. F., "A Simple Correlation of Turbine Efficiency," Journal of the Royal Aeronautical Society, Vol. 69, No. 655, July 1965, pp. 467-470.

16. Katsanis, T., "FORTRAN Program for Calculating Axial Turbomachinery Blade Coordinates," TM X-2061, 1970, NASA, Cleveland, Ohio.

17. Dean, R. C., Jr., Wright, D. D., and Runstadler, P. W., Jr., "Fluid Mechanics Analysis of High-Pressure-Ratio Centrifugal Compressor Data," USAAVLABS-TR-69-76, AD-872161, Feb. 1970, Creare Inc., Hanover, N. H.

18. Stanitz, J. D. and Prian, V. D., "A Rapid Approximate Method for Determining Velocity Distribution on Impeller Blades of Centrifugal Compressors," TN 2421, 1951, NACA, Cleveland, Ohio.

19. Wiesner, F. J., "A Review of Slip Factors for Centrifugal Impellers," Journal of Engineering for Power, Vol. 89, No. 4, Oct. 1967, pp. 558-572.

E-6666

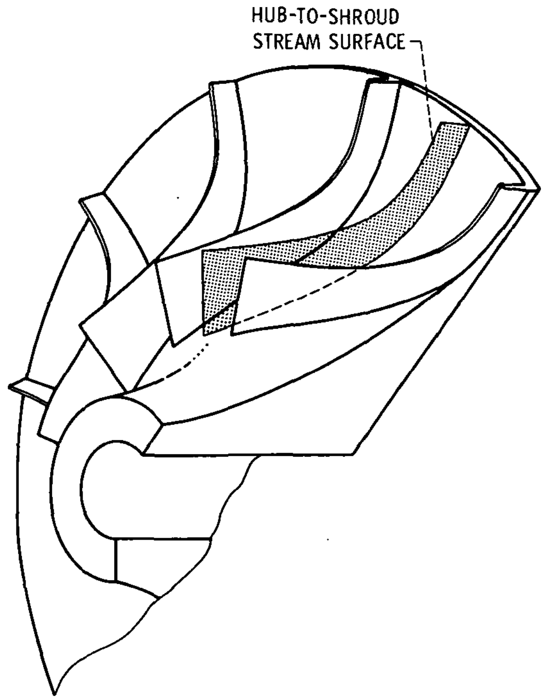


Figure 1. - Hub-to-shroud stream surface.

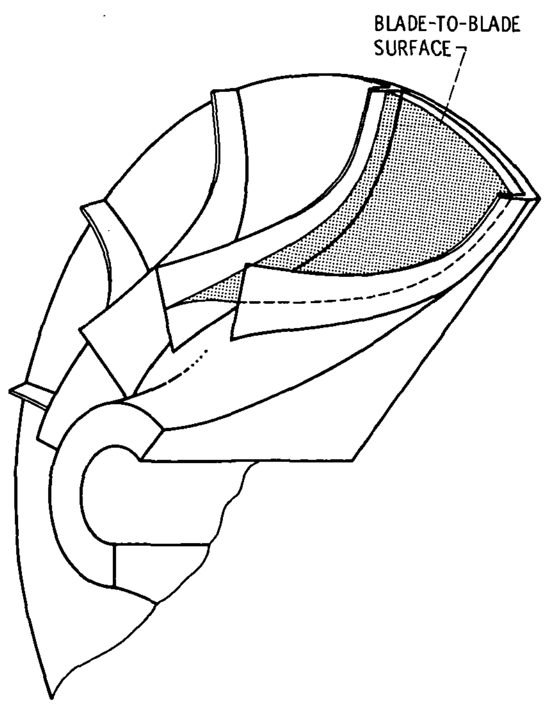


Figure 2. - Blade-to-blade surface of revolution.

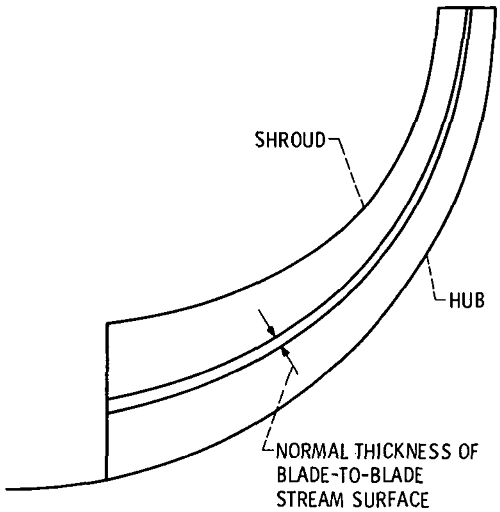


Figure 3. - Blade-to-blade stream channel in meridional plane.

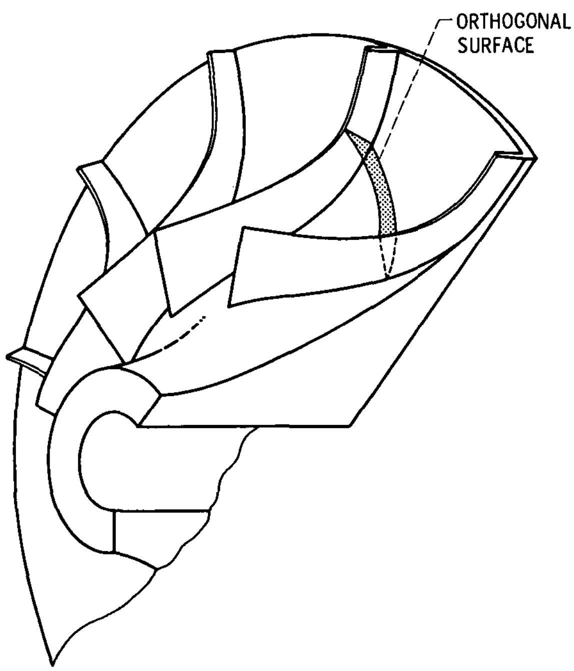


Figure 4. - Three-dimensional orthogonal surface across flow passage.

E-6666

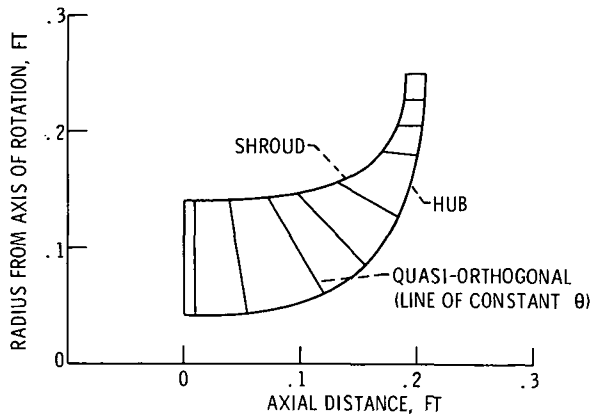


Figure 5. - Hub-to-shroud profile of backswept centrifugal impeller.

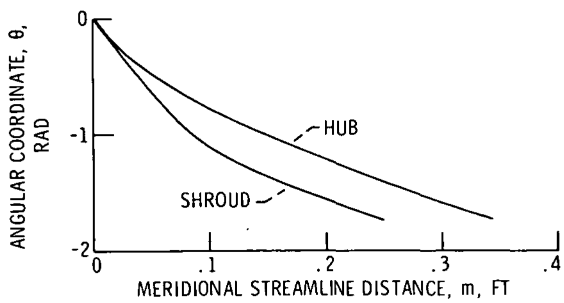
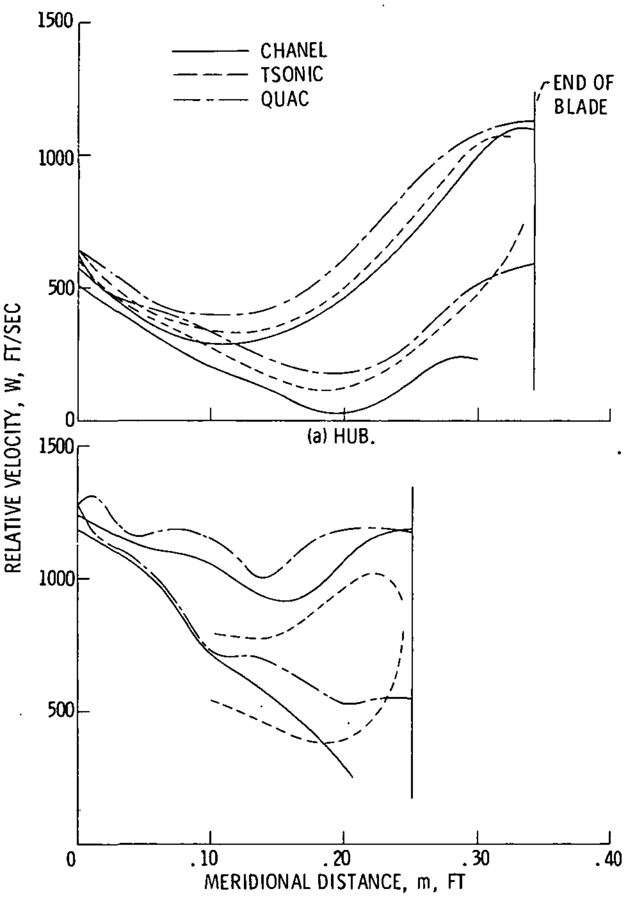


Figure 6. - Mean blade shape of backswept centrifugal impeller.

E-6566



(b) SHROUD.

Figure 7. - Blade surface velocities by three methods.