NASA Technical Memorandum 81644

NASA Contributions to Radial Turbine Aerodynamic Analyses

Arthur J. Glassman
Lewis Research Center
Cleveland, Ohio

(NASA-TM-81644) NASA CONTRIBUTIONS TO RADIAL TURBINE AERODYNAMIC ANALYSES (NASA) 20 p HC A02/MF A01 CSCL 01A

N81-13019

Unclas

G3/02 29464

Prepared for the
Automotive Technology Development Contractor Coordination Meeting
Dearborn, Michigan, November 11-13, 1980

NASA

NASA CONTRIBUTIONS TO RADIAL TURBINE AERODYNAMIC ANALYSES

Arthur J. Glassman

NASA Lewis Research Center
Cleveland, Ohio 44135

E-9356-1

ABSTRACT

This paper presents a review of available analytical methods and computer programs developed in-house or funded by NASA for the aerodynamic analysis of radial turbines. A brief description of the radial turbine and its analysis needs is followed by discussions of five analytical areas; design geometry and performance, off-design performance, blade-row flow, scroll flow, and duct flow. The functions of the programs, areas of applicability, and limitations and uncertainties are emphasized. Both past contributions and current activities are discussed.

INTRODUCTION

This paper provides a guide to NASA-developed methods and programs that are available for the aerodynamic analysis of radial turbines. The NASA Lewis Research Center has been involved in radial turbine research since the early sixties, when interest developed in Brayton Cycle space power systems. Since then, interest and efforts in radial turbines have extended to applications such as helicopter turbine engines, automotive turbine engines, and turbochargers for both automotive and general-aviation engines. A significant part of the past and present efforts has been and is still being devoted to the development of analytical techniques and associated computer programs for aerodynamic analyses of these radial turbines.

Only a few of the larger private companies can afford to support computer program development, and the results of this work usually remain proprietary. Most companies and consultants depend on the open literature for their analysis capabilities. The radial turbine analytical techniques and computer programs presented in the open literature are primarily the results of government (in-house and sponsored) and university work, with NASA providing a major and widely-used share.

Presented in this paper is a review of analytical methods and computer programs developed in-house or funded by NASA for the aerodynamic analysis of radial turbines. A brief description of the radial turbine and its analysis needs is followed by discussions of five analytical areas: design geometry and performance, off-design performance, blade-row flow, scroll flow, and duct flow. The functions of the programs, areas of applicability, and limitations and uncertainties are emphasized. Both past contributions and current activities are discussed.

TURBINE DESCRIPTION AND REQUIRED ANALYSES

Radial turbines are suitable for many applications where compact power sources are required. A general discussion of the radial turbine and its features and behavior can be found in reference 1. Turbines of this type have a number of desirable characteristics such as high efficiency potential, ease of manufacture, sturdy construction, and reliability (when made of metal). In order to realize the high efficiency potential, it is necessary to be able to relate design geometry to the application requirements and then to determine efficiency as a function of both. This is the function of the "design" analysis. Since engines often operate over wide ranges of speed and pressure ratio, it is necessary to estimate the performance of a given turbine over its full range of operation. It is the "off-design" analysis that does this.

Figure 1 shows a section through a typical radial turbine. The flow enters the stator radially and leaves the rotor axially. This turning of the flow takes place in the rotor passage, which is relatively long and narrow. Figure 2 shows the stator and rotor blading. The stator blade usually has little or no camber as a result of a tangential component of velocity developed in the inlet scroll. The low solidity and low aspect ratio typical of radial turbine stators can be seen. At the rotor inlet, where the flow velocity relative to the rotor is approximately radial, the rotor blades are usually straight and radial. This straight section of the rotor blade generally is rather highly loaded. In order to reduce this high loading, splitter or partial blades are sometimes placed between the full blades, as shown for the turbine of figure 2. At the rotor exit, the blades are curved to turn the flow so that the exit absolute velocity is approximately axial.

In order for the turbine to achieve the high efficiencies predicted by the "design" analysis, the stator and rotor blades must have the proper shapes and spacings so that the flow remains well behaved. To design the blades to do this, it is necessary to be able to analyze the velocity distribution in the blade-row passages and on the blade surfaces.

Flow usually enters the radial turbine by means of some type of pipe-fed inlet manifold rather than from a full annular passage. A scroll, sometimes called a volute and shown in figure 3, often surrounds the stator inlet. The scroll is fed by a tangential inlet pipe, which imparts a tangential component of velocity to the gas before it enters the stator blade row. It can be seen from figure 3 that the scroll diameter is considerably larger than the rotor diameter. Since it is the scroll that distributes the flow to the stator passages, it is necessary to analyze the velocity distribution in the scroll in order to achieve a design that provides an even distribution of clean flow to the turbine.

Small engine designs usually require turbine interspool and/or exit diffuser ducts to transport and condition the flow between components. Figure 4 illustrates a radial turbine with an exit diffuser duct. Interspool ducts exhibit less of an area change and usually do not turn the flow except as needed to provide a radius change. While interspool and exit ducts are not strictly parts of the turbine, they must be analyzed and designed in conjunction with the turbine if an optimum turbine system overall design is to be achieved. Therefore, we want to be able to analyze flow in these ducts to assure designs capable of providing clean flow and good performance.

DESIGN GEOMETRY AND PERFORMANCE

The "design" analysis is the first step in any turbine study. It is used to determine the overall size and shape of the turbine flowpath and the design-point efficiency. Design geometry (flowpath radii, heights, and angles) is related to the application requirements of speed, flow, and power. The resultant dimensions, velocities, and angles then serve to define turbine efficiency by means of some assumed loss model.

A generalized design analysis was presented by Rohlik (ref. 2), who used the dimensionless parameter specific speed

$$\text{Specific speed} = \frac{(\text{Rotative speed})(\text{Volume flow})^{1/2}}{(\text{Ideal work})^{3/4}}$$

to define the application requirements. A given specific speed can be satisfied by various combinations of geometric parameters, and reference 2 analyzes these to determine the optimum combinations for maximum static efficiency.

The analysis related losses to mean-section flow properties, neglecting hub-to-shroud variations. The losses considered were those caused by the stator and rotor boundary layers, clearance, windage, and exit kinetic energy. Assumptions included the use of straight radial blades at the rotor inlet, a favorable velocity increase in the rotor with an associated low loss coefficient, axial flow out of the rotor, and practical limitations on geometric parameters.

The overall results of the loss analysis are shown in figure 5. Each shaded area represents all the results for a given stator-exit angle with the various combinations of dimension ratios. Each point along the curve of maximum static efficiency, therefore, corresponds to some optimum combination of geometric parameters. Plotted in reference 2 (also in ref. 1) as functions of specific speed are the optimum values of stator-exit angle, ratio of stator height to rotor-inlet diameter, ratio of rotor-exit- tip to rotor-inlet diameter, and blade-jet speed ratio. These graphs can be used to quickly size a radial turbine consistent with the assumptions of the analysis. Because of the many assumptions associated with the loss model, the computed values of efficiency are considered to be more uncertain than are the optimum geometries.

The analysis of a power or propulsion system involves many repetitive calculations of component performance and geometry over a range of conditions. Although the generalized graphs of reference 2 provide some convenience, such calculations are most easily and quickly done by a computer. A computer program for the design analysis of radial turbines was developed by Glassman (ref. 3). The flow analysis is one-dimensional at the stator inlet, stator exit, and rotor inlet, each of these calculation stations being at a constant radius. Straight radial blades are assumed at the rotor inlet, and the design point is restricted to the optimum, or zero-incidence loss, case. At the rotor exit, where there is a variation in flow-field radius, an axisymmetric two-dimensional analysis is made using constant height sectors. Simple radial equilibrium is used to establish the static pressure gradient between sectors.

Program input design requirements are power, mass flow rate, inlet temperature and pressure, and rotative speed. The design parameters that can be varied include stator-exit angle, rotor-exit-tip to rotor-inlet radius ratio, rotor-exit-hub to tip radius ratio, and the magnitude and radial distribution of rotor-exit tangential velocity. This latter parameter allows designs with exit swirl (non-axial exit flow). While this can reduce the required rotor tip speed for a zero-incidence design, it increases rotor exit section and downstream duct losses and the proper trade-off must be made.

The program output includes diameters, total and static efficiencies, and all absolute and relative temperatures, pressures, velocities, and flow angles at stator inlet, stator exit, rotor inlet, and rotor exit. At the rotor exit, these values are presented at the mid-radius of each of the radial sectors as well as at the hub and tip. Examples of computed rotor-exit radial gradients of relative and absolute critical velocity ratios and relative flow angle are shown in figure 6. The gradients are significant, especially for relative flow angle, which is reflected by the amount of rotor-exit blade twist. The gradients can be such that in many cases where a strictly mean-diameter analysis provides a solution consistent with certain assumptions, a two-dimensional analysis shows that there is no solution.

A major uncertainty associated with this computer program is the loss model, which is a modification and extension of the model used in reference 2. The stator and rotor viscous losses are the major losses for the turbine, and these have been calibrated on the basis of a very limited amount of data available from a single turbine of rather low pressure ratio. A larger well-documented data base is needed before a loss model, undoubtedly modified from the current one, can be used with any confidence.

Updated versions of the reference 3 program are currently in use at NASA Lewis Research Center. The analysis is no longer restricted to straight radial blades at the rotor inlet. We intend to remove the restriction to zero-incidence-loss designs. At the rotor exit, the two-dimensional sector analysis can now handle constant area or constant flow sectors as well as the original constant height sectors. Incorporated into one version of the program are idealized loading models and associated boundary-layer calculations for both stator and rotor. Also included is a vaneless space loss. These features should provide the loss model with some flexibility in evaluating designs of differing geometries and rotor reactions.

OFF-DESIGN PERFORMANCE

A turbine is designed for a single operating point, i.e., the design point. Since engines often operate over a wide range of conditions, the turbine is required to perform at conditions of speed and pressure ratio other than those of the design point. Under these different operating conditions, which affect turbine work output and flow rate, the turbine is said to be operating off-design. In order to maximize off-design performance, some turbines are made with a stator capable of area variation, either by pivoting vanes or moving endwalls. This area variation then becomes another facet of off-design operation.

The approach to off-design analysis methods is somewhat different from that used for design analysis. In the off-design analysis, the geometry is fixed and the independent variables are blade speed, pressure ratio and, sometimes, stator area. The dependent variables are efficiency and flow rate. Losses calculated for the stator and rotor depend on loss coefficients selected to force agreement between calculated and experimental or design values at the design point. The first radial turbine off-design performance analysis method developed at NASA Lewis Research Center is described by Futral and Wasserbauer (ref. 4), and the associated computer program is presented by Todd and Futral (ref. 5). These are mentioned here primarily for historical recognition. The loss modeling was based on the only experimental data available at that time, which was for one rather unconventional radial turbine. As more data became available from the NASA research program, the loss model was updated and the program modified by Wasserbauer and Glassman (ref. 6).

The analysis of reference 6 uses a one-dimensional solution of flow conditions through the turbine along a mean flowpath, neglecting hub-to-shroud variations at the rotor exit. This analysis is restricted to the case where the blades are straight and radial at the rotor inlet. Stator and rotor trailing-edge blockages are included in the analysis, with choking based on inside-the-trailing-edge areas. Program output includes performance and velocity-diagram parameters for all speeds over a range of pressure ratio.

Figure 7 illustrates results obtained from this computer program by presenting calculated performance over a range of speed and pressure ratio and comparing this with experimental performance. The mass flow estimation (fig. 7(a)) shows an accurate representation of the experimental variation with pressure ratio. The calculated efficiencies, shown in figure 7(b) plotted against blade-jet speed ratio, are generally within 1 percent of the experimental values. Blade-jet speed ratio is the rotor tip speed divided by the isentropic velocity corresponding to turbine total-to-static pressure ratio.

A word of caution is in order with regard to the program procedure (called the design option in ref. 6) that attempts to compute stator and rotor loss coefficients satisfying the design conditions of flow, efficiency, and pressure ratio. Often, the design option is unable to produce any solution. Sometimes a solution is found where the coefficients are not reasonable. It has been our experience that it is best not to use the design option. We recommend that a reasonable value, the design value if available, be used for the stator loss coefficient and the rotor loss coefficient be determined on the basis of matching design-point efficiency and pressure ratio. The input flow areas, which are usually not accurately known for fabricated hardware, can then be adjusted if necesasary to match flow rate.

Although the program of reference 6 can and has been used for estimating off-design performance due to stator area variation, there was originally no loss modeling provided for variable area. An off-design performance loss model for a radial turbine with variable-area pivoting-vane stators was developed by Meitner and Glassman (ref. 7). A viscous loss model is used for the variation in stator loss coefficient with setting angle. Stator vane end-clearance leakage effects are predicted by a clearance flow model. The variation in rotor loss coefficient with stator setting angle (for no stator clearance) was obtained by analytical matching of experimental data for a rotor previously tested with six stators having throat areas from 20 to 144 percent of design area. Figure 8 shows the effect of incorporating this new loss model into the off-design performance calculation. Comparison of calculated and experimental efficiencies shows that a significant improvement in predicted efficiency at off-design stator areas is obtained with the new loss model.

A new computer program for radial turbine off-design analysis is currently being written at NASA Lewis Research Center. The variable-stator loss model will be included in the program. At the rotor inlet, the analysis will accommodate non-radial (swept) as well as radial blading. A two-dimensional sector analysis will replace the one-dimensional mean-section analysis at the rotor exit. Rotor clearance and disk friction losses are being added to the loss model. These features will make the off-design program more compatible with the design program.

BLADE-ROW FLOW

In order to insure good performance from turbine blades, it is necessary to determine and control the flow distribution throughout the blading flow passages. Velocity gradients occur from blade to blade as a result of the static pressure difference required to turn the flow. Variations in velocity from hub to shroud (when a radius change occurs between hub and shroud) occur as a result of radial equilibrium and effects of changing blade speed. The design of a proper blade profile, therefore, depends on our ability to calculate this three-dimensional flow field and determine the velocities at the blade surfaces.

Flow in a turbomachine involves three-dimensional viscous unsteady flow through geometrically complex passages. The design tools of today cannot handle such a flow problem. In order for the designer to calculate a velocity distribution with reasonable dispatch, the analysis methods used are based on a number of simplifying assumptions. First of all, three-dimensional flow is simplified to two-dimensional flow to provide solutions in the hub-to-shroud, blade-to-blade, and orthogonal surface of blade passages. These surfaces for a radial turbine rotor are illustrated in figure 9. In addition, the flow is assumed to be steady relative to the blades, isentropic, inviscid, and ideal. As will be discussed later in this section, more rigorous analysis methods are being developed.

Two basic computation methods have been used in the past at NASA Lewis Research Center to obtain the currently operational two-dimensional flow solutions; a stream function method, which gives results for the entire flow passage but is limited to subsonic flow regions, and a velocity gradient method, which gives results only for the guided part of a passage but can be used for all flow regimes. The computer programs currently being used most at NASA Lewis Research Center for radial turbine blade row flow are the hub-to-shroud analysis (MERIDL) of Katsanis and McNally (ref. 8) and the blade-to-blade analysis (TSONIC) of Katsanis (ref. 9). Subsonic solutions are obtained directly by the stream function method. Transonic solutions are obtained by a velocity gradient approximation that uses information from a stream function solution at a reduced mass flow.

The TSONIC program has been extensively modified as reported by Wood (ref. 10), since its publication more than 10 years ago. The velocity gradient approximation for transonic solutions has been replaced by a stream-sheet thickness modification. The solution is more stable and the results more reliable than those of the original program, and all input for the current version is directly provided in card format by the output from the MERIDL program of reference 8. These analyses have been extensively verified for axial-flow blade rows, but appropriate experimental data is lacking for radial turbine blade rows.

Hub-to-shroud and blade-to-blade solutions can also be obtained using velocity gradient methods alone. The velocity gradient, or stream filament, method can only give solutions within a passage where both ends of all streamline orthogonals intersect a solid boundary. Therefore, the usefulness of this method depends on the degree of flow guidance provided by the blades. For a radial turbine stator, where there is usually a large amount of unguided pasasage, the velocity gradient method is not very useful.

Velocity gradient analyses specifically developed for radial turbine rotors are the hub-to-shroud (ref. 11) and blade-to-blade (ref. 12) analyses of Katsanis. These analyses are based on use of the velocity gradient equation along fixed straight lines, called quasi-orthogonals, rather than along orthogonals, whose length and location are not known in advance. The hub-to-shroud analysis of reference 12 also includes an approximate calculation of blade surface velocities based on a linear velocity distribution between blades. This program is particularly useful for initial screening studies and in smaller computer installations because it requires much less storage and much shorter running times as compared to the stream function programs. A comparison of rotor blade surface velocity distributions as computed by the approximate method of reference 11 and the more rigorous method of reference 9 is shown in figure 10. The surface velocities agree fairly well over the mid-portion of the blade. An appreciable difference, however, occurs at the leading and trailing edge regions because of the differing assumptions.

Velocity gradient methods were also used by Katsanis (ref. 13) to determine velocity variations from hub to tip along meridional streamline orthogonals and from blade to blade along hub, mean, and tip streamline orthogonals. This results in a flow solution for an orthogonal surface (fig. 9(c)). Computations are made for a number of these surfaces along the blade passage. This program not only yields surface velocities, but also yields a two-dimensional estimate for design and choking mass flows.

Currently there are numerous in-house, contract, and grant activities at NASA Lewis Research Center in the field of computational fluid mechanics for turbomachinery. A recently developed blade-to-blade analysis program, called QSONIC, uses a full potential method to provide a more valid transonic solution, in about twice the running time, than that obtained from TSONIC. This analysis calculates shocks and is considered valid for Mach numbers up to about 1.4. Another recently developed program uses a panel method to provide blade-to-blade solutions almost as good as those from TSONIC, but in one-tenth or less the running time. At present, this program does not permit a change in radius; however, the addition of such a capability is not a difficult problem. Both of these recent developments will be published.

Two-dimensional viscous, three-dimensional inviscid, and three-dimensional viscous analysis methods are also being explored and developed. In time, computer programs based on these methods will achieve operational status and find their way into use for radial blade-row flow analysis.

SCROLL FLOW

A radial turbine scroll, shown in figure 3, is usually designed in a helical configuration to discharge uniformly into the stator vanes or, in the case of a vaneless turbine, into the rotor. This is shown by the scroll-nozzle stream surface in figure 11. The actual flow in a scroll is three-dimensional, compressible, and viscous. Even in the absence of viscous effects, three-dimensional flow is caused by the continuous discharge of flow into the vanes. Studies have shown that the performance of the stator vanes and the flow distribution are greatly influenced by the flow incidence angle at the vane inlet and this can vary from vane to vane. Therefore, knowledge and control of the flow conditions at the scroll exit is essential in order to predict as well as to maximize stator performance.

Most radial turbine design and performance analyses conducted in the past have either ignored the scroll, combined it with the stator loss coefficient for purposes of performance prediction, or for flow analysis assumed one-dimensional flow and conservation of mass and angular momentum. In order to improve our knowledge of scroll flow and our ability to predict it, a grant program was undertaken with the University of Cincinnati in 1975. This program was both analytical and experimental in nature and the analytical efforts will be summarized herein.

The three-dimensional inviscid flow field in the scroll was formulated in terms of the velocity potential. Because of the geometrical complexity of the flow field, considerations of storage requirements and running time, and solution method limitations, the problem was broken down and studied primarily in terms of incompressible two-dimensional flow fields in order to get a quasi-three-dimensional picture of the flow. The two-dimensional flow field on a scroll-nozzle stream surface, shown in figure 11, was investigated by Baskharone, Hamed, and Tabakoff (ref. 14) using the finite element method for solution. The report presents the computer program along with the analysis results. The calculated velocity field, an example of which is shown in figure 12, is used by the program to determine the variation in mass flow between the different stator channels. For the velocity field illustrated in figure 12, the mass flow through the individual stator channels varied from about 8 percent below to about 15 percent above the average value.

A study was performed to investigate the effects of scroll shape and throughflow velocity profile on the flow velocities in the cross-sectional planes, shown in figure 13, of the scroll. The analysis and the computer program are presented by Abdallah, Hamed, and Tabakoff in references 15 and 16, respectively. A finite difference solution procedure was used for this effort.

Hamed and Baskharone (ref. 17) present an analysis using the finite element method for the three-dimensional flow field in a vaneless scroll-nozzle assembly. A comparison of analytical and experimental results, some of which is shown in figure 13, is presented for the throughflow velocities in cross-sectional planes. Although the experimental data was for a scroll followed by nozzle vanes, as opposed to the analysis which was for a vaneless case, the comparison is made because this was the only data available. Considering the aforementioned difference along with the fact that secondary flow and viscosity were not included in the analysis, the comparison is encouraging.

In order to extend and improve the analysis methods and computer programs, a follow-on grant effort with the University of Cincinnati was initiated earlier this year along with a companion effort to provide experimental verification. Both the analysis and experiments will be done with the same scroll geometry.

DUCT FLOW

Design methods for diffusers have long been based on empirical correlations. These empirical criteria, however, are not adequate for designing curved wall diffusing ducts. The first design method used for diffuser duct flow analysis divided the flow field into inviscid and viscous parts. The inviscid solution, such as provided by the analysis of Katsanis and McNally (ref. 8), and the boundary layer solution, such as provided by the analysis of McNally (ref. 18), must be iterated to convergence. Problems encountered with this type of analysis used for diffuser ducts were frequent failure to converge and unsatisfactory prediction of separation.

An improved analysis method and program developed by Anderson is presented in reference 19. The method for solving the axisymmetric compressible swirling flow problem was based on a single set of equations for the entire flow field in the diffuser, thereby eliminating the iterations between inviscid and boundary layer solutions. The analysis can treat flows with arbitrary distributions of heat and mass transfer at the walls, and can include the effects of struts or blades in the duct passage. It was found that the method of reference 19 yielded good results when the curvature on both walls was small and nearly the same; however, the solution sometimes failed or yielded significant errors when the curvature varied significantly from wall to wall or when larger curvatures were encountered.

In an Army sponsored follow-on program, Anderson (ref. 20) extended the analysis and program to include ducts with larger curvature, slot-cooled walls, and flows downstream of separation. A turbine interspool duct (designed for an inlet Mach number of 0.5 and an inlet flow angle of 20°) tested at NASA Lewis Research Center is shown in figure 14. Shown in the figure is the survey plane where measurements of total pressure and flow angle were made. Figure 15 presents the measured radial distributions of total pressure and flow angle at the survey plane and compares them with analytical results from the program of reference 20. Excellent agreement is obtained even though the flow is considerably separated at the measurement plane.

A DOE-funded program with United Technologies Research Laboratories under contract to NASA Lewis Research Center was very recently initiated to further extend the Anderson program to be more compatible with small engine requirements. The computer program is to be modified to: (1) accommodate ducts with 90° or more of turning (such as that illustrated in figure 4; (2) include loss correlations more suitable to small turbines; (3) allow more general strut and vane geometries; and (4) provide detailed output at planes within the duct. The updated program should be available in about 1 year.

CONCLUDING REMARKS

This paper reviewed analytical methods and computer programs developed in-house or funded by NASA for the aerodynamic analysis of radial turbines. Five analytical areas were discussed: design geometry and performance, off-design performance, blade-row flow, scroll flow, and duct flow. The NASA contributions are certainly not the only analytical methods and programs that have been developed or even published; however, they do constitute a major share of the analytical tools openly available to the technical community. Most of the computer programs referenced in this paper can be obtained from the Computer Software Management Information Center (COSMIC), Barrow Hall, University of Georgia, Athens, Georgia 30601. Inquiries for general information concerning these programs can be directed to any of the referenced authors or to me.

REFERENCES

1. Rohlik, Harold E.: Radial-Inflow Turbines. Ch. 10 of Turbine Design and Application, Vol. 3, Arthur J. Glassman, ed., NASA SP-290, 1975, pp. 32-58.
2. Rohlik, Harold E.: Analytical Determination of Radial Inflow Turbine Design for Maximum Efficiency. NASA TN D-4384, 1968.
3. Glassman, Arthur J.: Computer Program for Design Analysis of Radial-Inflow Turbines. NASA TN D-8164, 1976.
4. Futral, Samuel M., Jr.; and Wasserbauer, Charles A.: Off-Design Performance Prediction with Experimental Verification for a Radial-Inflow Turbine. NASA TN D-2621, 1965.
5. Todd, Carroll A.; and Futral, Samuel M., Jr.: A FORTRAN IV Program to Estimate the Off-Design Performance of Radial-Inflow Turbines. NASA TN D-5059, 1969.
6. Wasserbauer, Charles A.; and Glassman, Arthur J.: FORTRAN Program for Predicting Off-Design Performance of Radial-Inflow Turbines. NASA TN D-8063, 1975.
7. Meitner, Peter L.; and Glassman, Arthur J.: Off-Design Performance Loss Model for Radial Turbines with Pivoting Variable Area Stators. NASA TP-1708, 1980.
8. Katsanis, Theodore; and McNally, William D.: Revised FORTRAN Program for Calculating Velocities and Streamlines on the Hub-Shroud Midchannel Stream Surface of an Axial-, Radial-, or Mixed-Flow Turbomachine or Annular Duct. I-User's Manual. NASA TN D-8430, 1977.
9. Katsanis, Theodore: FORTRAN Program for Calculating Transonic Velocities on a Blade-to-Blade Stream Surface of a Turbomachine. NASA TN D-5427, 1969.
10. Wood, Jerry R.: Improved Method for Calculating Transonic Velocities on Blade-to-Blade Stream Surfaces of a Turbomachine. NASA TP-1772, 1980.
11. Katsanis, Theodore: Use of Arbitrary Quasi-Orthogonals for Calculating Flow Distribution in the Meridional Plane of a Turbomachine. NASA TN D-2546, 1964.
12. Katsanis, Theodore: Use of Arbitrary Quasi-Orthogonals for Calculating Flow Distribution on a Blade-to-Blade Surface in a Turbomachine. NASA TN D-2809, 1965.
13. Katsanis, Theodore: FORTRAN Program for Quasi-Three-Dimensional Calculation of Surface Velocities and Choking Flow for Turbomachine Blade Rows. NASA TN D-6177, 1971.
14. Baskharone, E.; Hamed, A.; and Tabakoff, W.: Flow in Nonrotating Passages of Radial Inflow Turbines. NASA CR-159679, 1979.
15. Hamed, A.; Abdallah, S.; and Tabakoff, W.: Analysis of the Cross Flow in a Radial Inflow Turbine Scroll. NASA CR-135320, 1977
16. Hamed, A.; Abdallah, S.; and Tabakoff, W.: Computer Program for the Analysis of the Cross Flow in a Radial Inflow Turbine Scroll. NASA CR-135321, 1977.

17. Hamed, A.; and Baskharone, E.: Analysis of the Three Dimensional Flow in a Turbine Scroll. Flow in Primary Non-Rotating Pasasages in Turbomachines, Proceedings of Winter Annual Meeting, ASME, 1979, pp. 99-105.
18. McNally, William D.: FORTRAN Program for Calculating Compressible Laminar and Turbulent Boundary Layers in Arbitrary Pressure Gradients. NASA TN D-5681, 1970.
19. Anderson, O. L.: Finite-Difference Solution for Turbulent Swirling Compressible Flow in Axisymmetric Ducts with Struts. NASA CR-2365, 1974.
20. Anderson, O. L.: User's Manual for a Finite-Difference Calculation of Turbulent Swirling Compressible Flow in Axisymmetric Ducts with Struts and Slot Cooled Walls, Vol. I. UL-7500245, United Aircraft Research Laboratories, 1974.

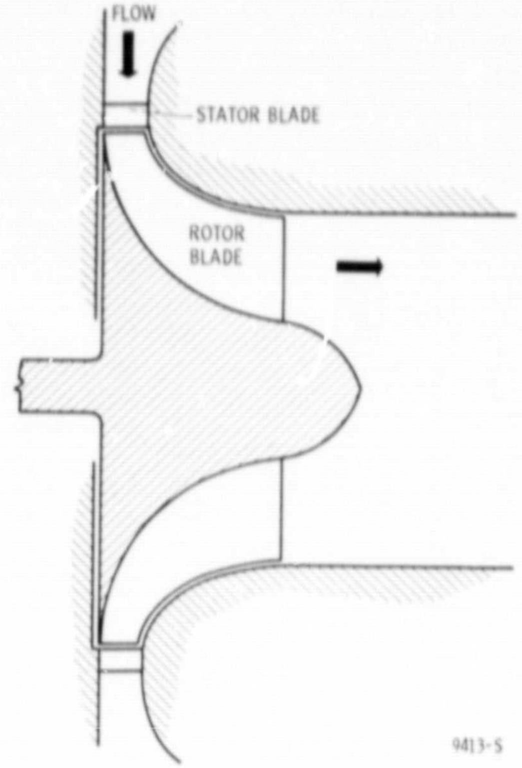


Figure 1. - Schematic cross section of radial turbine.

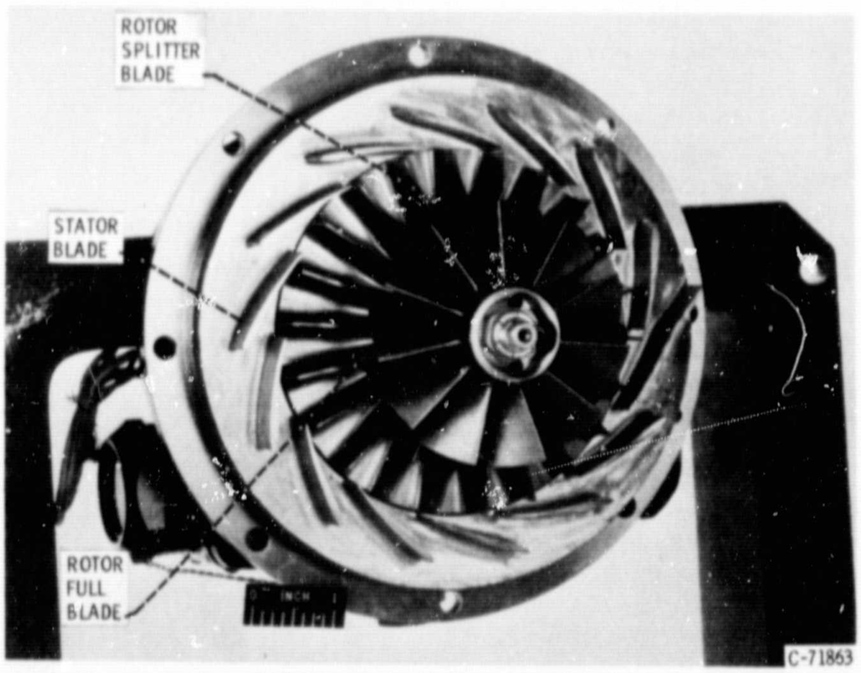


Figure 2. - Turbine stator and rotor blading.

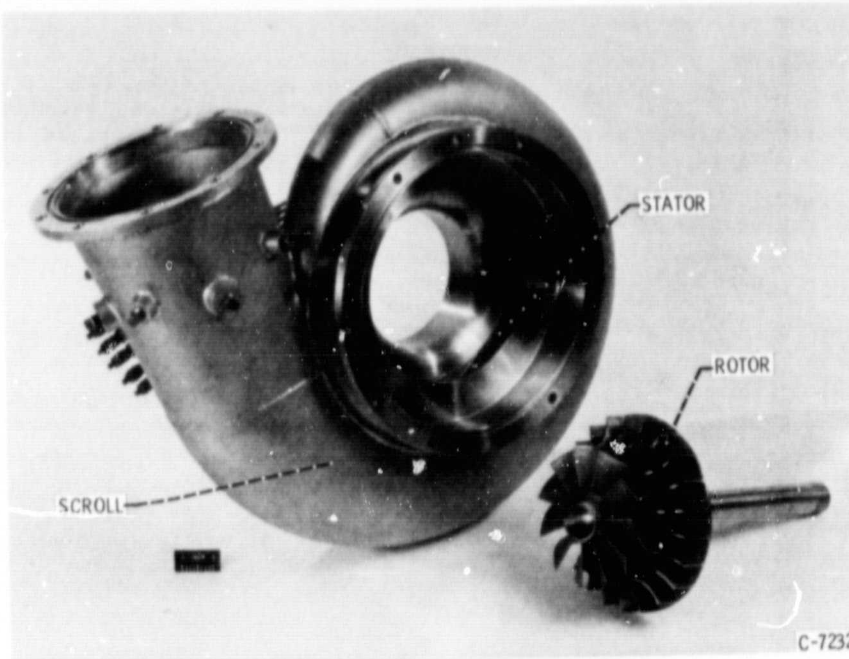


Figure 3. - Radial turbine with scroll.

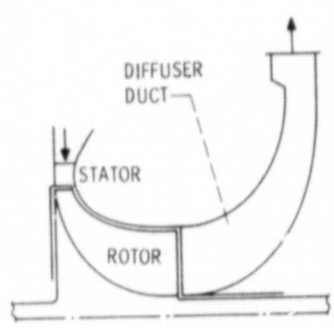


Figure 4. - Radial turbine with diffuser duct.

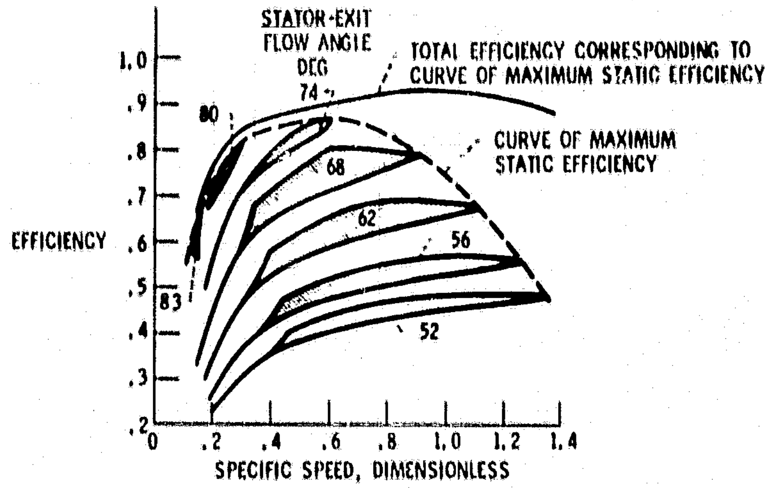


Figure 5. - Effect of specific speed on computed design-point efficiency.

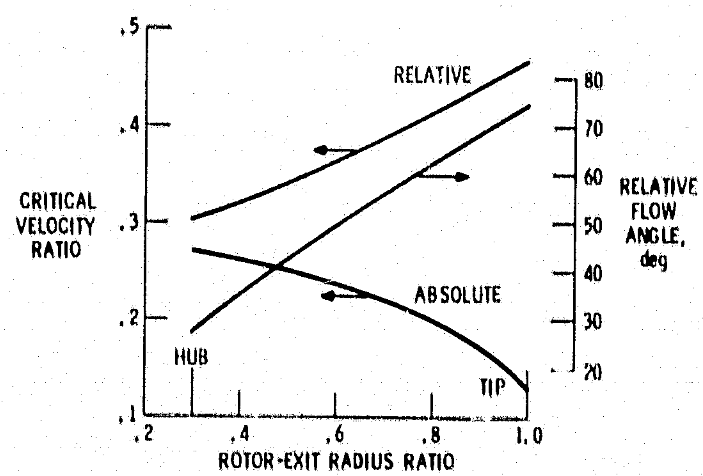
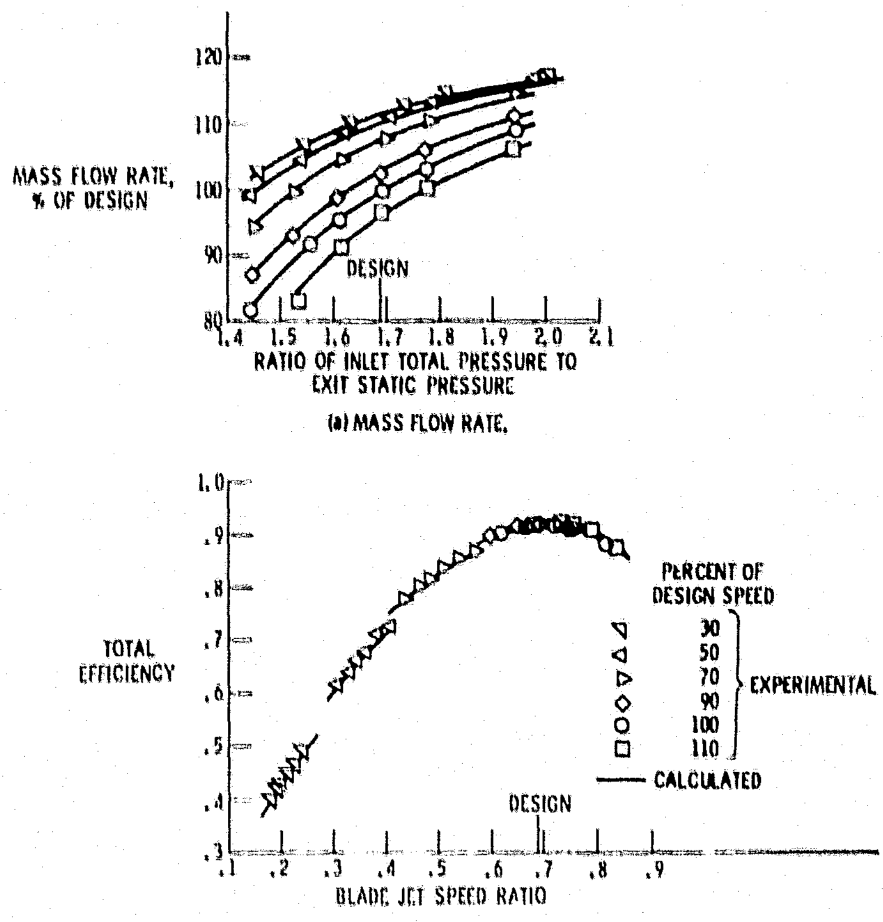


Figure 6. - Typical rotor-exit radial gradients from design computer program.



(a) MASS FLOW RATE.

(b) EFFICIENCY.

Figure 7. - Comparison of calculated and experimental performance for off-design operation.

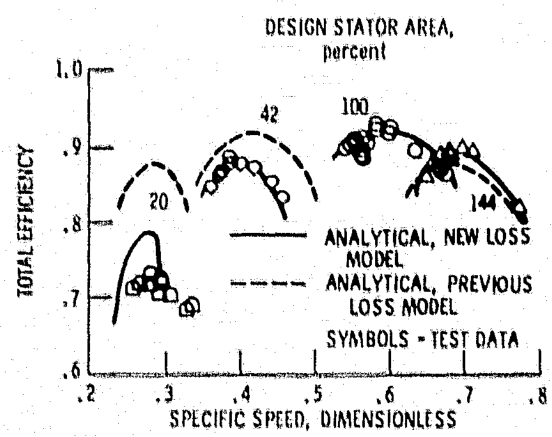
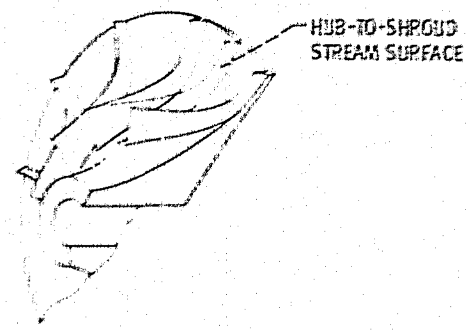
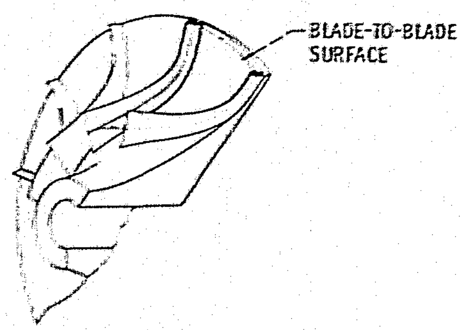


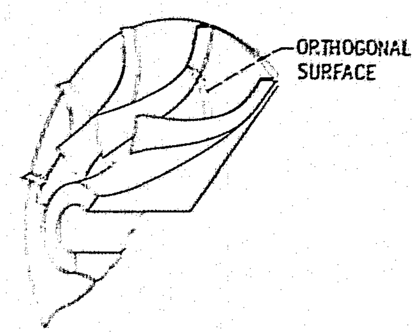
Figure 8. - Loss model comparison for variable-stator radial turbine.



(a) HUB-TO-SHROUD STREAM SURFACE

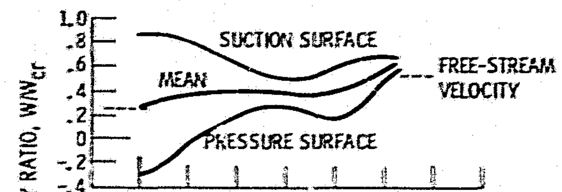


(b) BLADE-TO-BLADE SURFACE OF REVOLUTION.

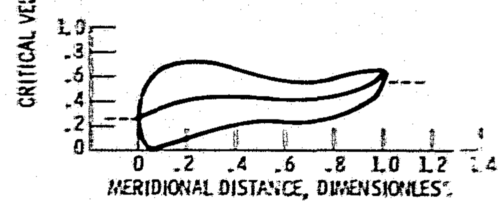


(c) ORTHOGONAL SURFACE ACROSS FLOW PASSAGE.

Figure 9. - Surfaces used for velocity distribution calculations.



(a) VELOCITY GRADIENT SOLUTION.



(b) STREAM FUNCTION SOLUTION.

Figure 10. - Comparison of rotor-blade surface-velocity distributions computed by different methods.

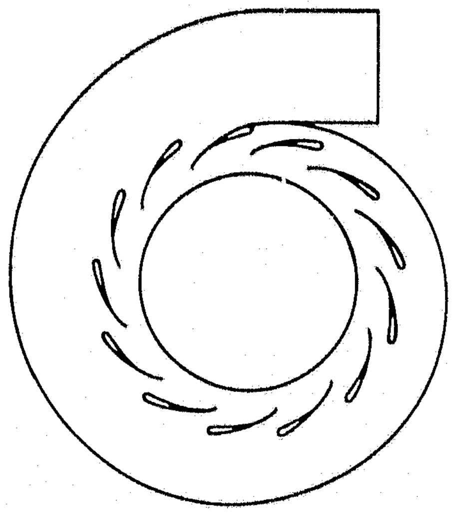


Figure 11. - Radial turbine scroll-nozzle stream surface.

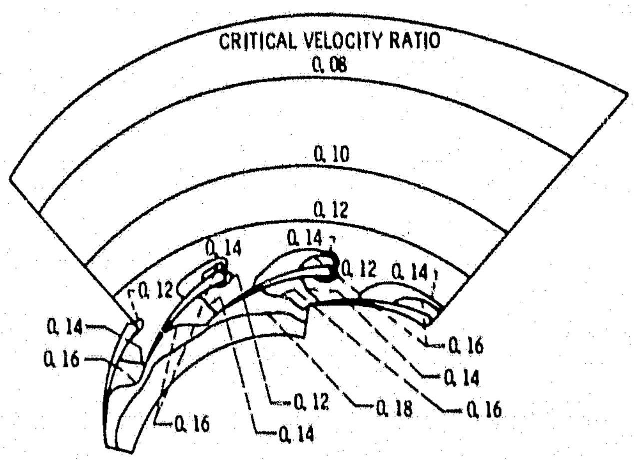


Figure 12. - Velocity distribution on scroll-nozzle stream surface.

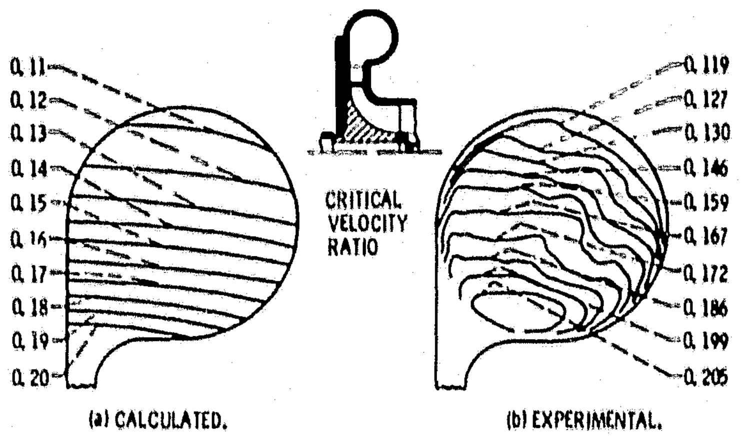


Figure 13. - Comparison of calculated and experimental throughflow velocities on scroll cross-sectional plane.

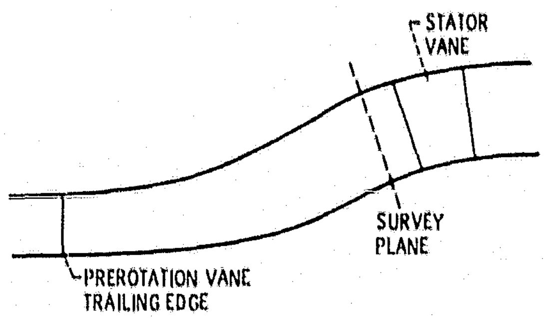


Figure 14. - Interstage duct flow path.

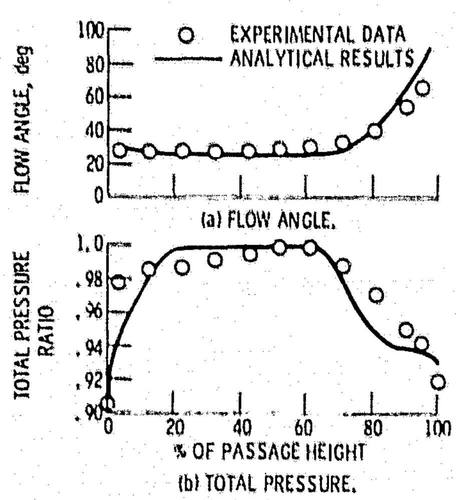


Figure 15. - Radial variation of duct exit flow conditions.