CR-152105

Department of Aerospace Engineering and Applied Mechanics
University of Cincinnati

THEORETICAL AND EXPERIMENTAL STUDY OF AERODYNAMICS, HEAT TRANSFER AND PERFORMANCE OF A RADIAL TURBINE

FINAL REPORT

BY

W. TABAKOFF

DECEMBER 1977

(NASA-CR-152105) THEORETICAL AND EXPERIMENTAL STUDY OF AERODYNAMICS, HEAT TRANSFER AND PERFORMANCE OF A RADIAL TURBINE Final Report, 1 Oct. 1973 - 30 Oct. 1977 (Cincinnati Univ.) 56 p HC A04/MF A01

N78-19155

Unclas

G3/07 08627

This work was sponsored by U.S. Army Air Mobility Research and Development Laboratory, Moffett Field, California 94035 under Contract Number NAS2-7850.

THEORETICAL AND EXPERIMENTAL STUDY OF AERODYNAMICS, HEAT TRANSFER AND PERFORMANCE OF A RADIAL TURBINE

FINAL REPORT

BY

W. TABAKOFF

DECEMBER 1977

This work was sponsored by U.S. Army Air Mobility Research and Development Laboratory, Moffett Field, California 94035 under Contract Number NAS2-7850.

ACKNOWLEDGEMENT

The author is very grateful and wishes to extend his sincere appreciation to Director John Acurio and Curtis Walker of the U.S. Army Air Mobility Research and Development Laboratory at NASA Lewis Research Center, Cleveland, Ohio, for their continued interest, the helpful discourse provided, encouragement and suggestions during the course of this work.

TABLE OF CONTENTS

INTRODUCTION

This is the final report covering the technical work performed by the Department of Aerospace Engineering and Applied Mechanics under Contract No. NAS2-7850 (U.S. Air Mobility Research and Development Laboratory - Moffett Field, California). The work was supervised by Dr. Widen Tabakoff, Professor in the Department of Aerospace Engineering and Applied Mechanics. The report covers work conducted from October 1, 1973 to October 30, 1977.

The contents of the technical reports written during the contract period have been incorporated in this report through the provision of abstracts. In addition, the technical publications in Scientific Journals and Papers are given with the first page of the corresponding publication. During the last year of this contract we started to develop a method of analyzing turbulent nonadiabatic viscous flow through radial turbomachine rotors. This work is now only in the progress stage.

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA CR 137633		

4. Title and Subtitle
A Two-Dimensional Finite-Difference Solution for the Temperature Distribution in a Radial Gas Turbine Guide Vane Blade

5. Report Date
September 1975

6. Performing Organization Code

7. Author(s)
W.M. Hosny and W. Tabakoff

8. Performing Organization Report No.

9. Performing Organization Name and Address
Department of Aerospace Engineering
University of Cincinnati
Cincinnati, Ohio 45221

10. Work Unit No.

11. Contract or Grant No.
NAS2-7850

12. Sponsoring Agency Name and Address
National Aeronautics and Space Administration
Washington, D.C. 20546

13. Type of Report and Period Covered
Contractor Report

14. Sponsoring Agency Code

15. Supplementary Notes
Project Manager, Major G. Godfrey, U.S. Army Air Mobility Research and Development Laboratory, Ames Research Center, Moffett Field, California 94035

16. Abstract

A two-dimensional finite-difference numerical technique is presented to determine the temperature distribution in a solid blade of a radial turbine guide vane.

A computer program is written in Fortran IV for IBM 370/165 computer. The computer results obtained from these programs have a similar behavior and trend as those obtained by experimental results.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Heat Transfer	Unclassified - unlimited

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price*
Unclassified	Unclassified	56	

*For sale by the National Technical Information Service, Springfield, Virginia 22151

1. Report No	2. Government Accession No.	3. Recipient's Catalog No.
NASA CR 137632		

4. Title and Subtitle	5. Report Date
Flow Behavior in Inlet Guide Vanes of Radial Turbines	October 1975
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No.
J. Sokhey, W. Tabakoff and W. Hosny	
	10. Work Unit No.
9. Performing Organization Name and Address	11. Contract or Grant No.
Department of Aerospace Engineering University of Cincinnati Cincinnati, Ohio 45221	NAS2-7850
	13. Type of Report and Period Covered
12. Sponsoring Agency Name and Address	Contractor Report
National Aeronautics and Space Administration Washington, D.C. 20546	14. Sponsoring Agency Code

15. Supplementary Notes

Project Manager, Major G. Godfrey, U.S. Army Air Mobility Research and Development Laboratory, Ames Research Center, Moffett Field, California 94035

16. Abstract

A brief discussion on the scroll flow is presented. Streamline pattern and velocity distribution in the guide vanes are calculated. The blade surface temperature distribution is also determined. The effects of the blade shapes and the nozzle channel width on the velocity profiles at inlet to the guide vanes are investigated.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Radial Turbine Scroll Guide Vane	Unclassified - unlimited

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price*
Unclassified	Unclassified	32	

* For sale by the National Technical Information Service, Springfield, Virginia 22151

AIAA PAPER
NO. 76-44

ORIGINAL PAGE IS
OF POOR QUALITY

TEMPERATURE DISTRIBUTION STUDY IN A COOLED RADIAL INFLOW TURBINE ROTOR

by
A. HAMED, E. BASKHARONE,
and
W. TABAKOFF
University of Cincinnati
Cincinnati, Ohio

Abstract

A numerical study to determine the temperature distribution in the rotor of a radial inflow turbine is presented. The study is based on the use of the finite element method in the three dimensional heat conduction problem. Different cooling techniques with various coolant to primary mass flow ratios are investigated. The resulting temperature distribution in the rotor are presented for comparison.

AIAA 14th Aerospace Sciences Meeting
WASHINGTON, D.C. / JANUARY 26-28, 1976

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA CR 137942		

4. Title and Subtitle	5. Report Date
An Investigation of Viscous Losses in Radial Inflow Nozzles	September 1976
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No.
I. Khalil, W. Tabakoff and A. Hamed	
	10. Work Unit No.
9. Performing Organization Name and Address	11. Contract or Grant No.
Department of Aerospace Engineering & Applied Mechanics University of Cincinnati Cincinnati, Ohio 45221	NAS2-7850
	13. Type of Report and Period Covered
12. Sponsoring Agency Name and Address	Contractor Report
National Aeronautics & Space Administration Washington, D.C. 20546, and U.S. Army Air Mobility Research & Development Lab. Moffett Field, California 94035	14. Sponsoring Agency Code

15. Supplementary Notes

Interim Report. Project Manager, LTC Dwain Moentmann, U.S. Army Air Mobility Research and Development Laboratory, Ames Research Center, Moffett Field, California 94035.

16. Abstract

A theoretical model is developed to predict losses in radial inflow turbine nozzles. The analysis is presented in two parts. The first one evaluates the losses which occur across the vaned region of the nozzle, while the second part deals with the losses which take place in the vaneless field.

In the vaned region, equations are derived to relate the losses to the boundary layer characteristic parameters on the vane surfaces and over the two end walls. The effects of vanes geometry, flow conditions and boundary layer characteristic parameters, at exit from the nozzle channel, on the level of losses are investigated. Results indicate that the portion of the losses incurred due to the end walls boundary layer may be significant especially when these boundary layers are characterized by a strong cross flow component.

In the vaneless field, the governing equations for the flow are formulated using the assumptions of the first order boundary layer theory. The viscous losses are evaluated through the introduction of a wall shear stress that takes into account the effects of the three dimensional end walls boundary layer. The resulting governing equations are solved numerically and the effects of some operating conditions and the nozzle geometry are studied.

It is concluded that the losses in the vaneless region may be looked at as a secondary factor in the determination of the overall efficiency of the turbine nozzle assembly.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Turbomachinery Losses Radial Turbine	Unclassified - unlimited

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price*
Unclassified	Unclassified	83	

* For sale by the National Technical Information Service, Springfield Virginia 22161

Reprinted from September 1976, Vol. 98, Journal of Fluids Engineering

ORIGINAL PAGE IS
OF POOR QUALITY

I. M. KHALIL
W. TABAKOFF
A. HAMED
Department of Aerospace Engineering,
University of Cincinnati,
Cincinnati, Ohio

Losses in Radial Inflow Turbines

A study was conducted to determine experimentally and theoretically the losses in radial inflow turbine nozzles. Extensive experimental data were obtained to investigate the flow behavior in a full-scale radial turbine stator annulus. A theoretical model to predict the losses in both the vaned and vaneless regions of the nozzle was developed. In this analysis, the interaction effects between the stator and the rotor are not considered. It was found that the losses incurred due to the end wall boundary layers can be significant, especially if they are characterized by a strong crossflow. The losses estimated using the analytical study are compared with the experimentally determined values.

Introduction

Due to the complexity of the flow patterns in radial inflow turbines, previous analytical investigations dealt with ideal flow, with relatively simple loss models. The viscous losses were assumed to be proportional to the average kinetic energy in reference [1],[1] while approximations based on flat plate and pipe flow analogies were used in reference [2]

In the present study, experimental measurements were obtained for the flow at the inlet and the exit of a full-scale radial turbine stator annulus Detailed data, which includes flow directions, total and static pressure surveys, have been used to gain an insight into the flow processes by which the distribution of the losses is affected. Results are then used to develop a better theoretical model for loss prediction in both the vaned and the vaneless field constituting the stator annulus. It is hoped that this model will assist in developing an optimization scheme which can be used for designing the stator annulus of a radial turbine

Experimental Investigation

A test facility was designed to incorporate a complete radial inflow turbine for stator and rotor testing using cold air The present phase of experimental investigation was concerned with evaluating the overall performance of the stator annulus and to gain a better understanding of the flow behavior in the radial nozzle This was accomplished by testing a full-scale stator assembly installed in the scroll housing of the turbine. The rotor was not assembled during these tests. Instead, a plastic body of revolution was installed to provide a smooth, continuous hub flow-path downstream the stator. The scroll which was located upstream the stator assembly was of the nonsymmetric circular type. The nozzle stator annulus consisted of 13 untwisted constant section vanes of 0 0482-m chord length The annulus had an

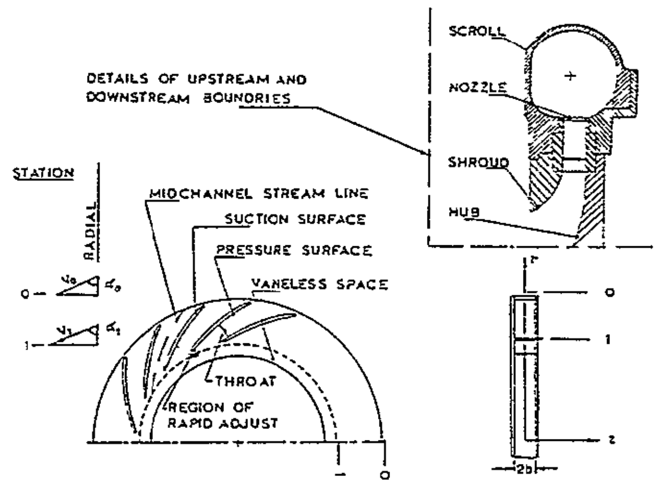


Fig. 1 Flow regions and stations designation

outer diameter of 0 191 m and an inner diameter of 0 139 m The distance between the two side walls was constant and equal to 0.0147 m. The facility, instrumentation, testing techniques and the stator design characteristics are described in reference [3] Fig 1 shows a cross-sectional sketch of the stator assembly

Surveys of total pressures, static pressures and the flow directions were made both upstream and downstream of the stator annulus, using calibrated wedge probes Outside the boundary layer regions, a three-dimensional cobra-probe was designed to measure the flow total pressure, static pressure, pitch and yaw angles The probe consisted of five 0 037 cm outside diameter tube with their meauring ends grounded to a total included angle of 70° Inside the boundary layer regions a pitot-static yawmeter was used to measure the air velocity and flow angles The yawmeter tube tip was flattened to approximately 0 015 cm inside height Measurements of vane surface pressure distributions were also obtained by means of ports of 0 043 cm diameter drilled in hypodermic tubes embedded in the body of two consecutive vanes The upstream measuring station, 0, of Fig 1 was located 1/4 chord radially outward from the leading edge, while the downstream station 2 was at 0 1 chord radially inward from the trailing edge (very close to station 1) Surveys were obtained through the spanwise and pitchwise movements

---

[1]Numbers in brackets designate References at end of paper

Contributed by the Fluids Engineering Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS and presented at the Gas Turbine and Fluids Engineering Conference, New Orleans, La, March 21-25, 1976 Manuscript received at ASME Headquarters January 5, 1976 Paper No 76-FE-9

1 Report No	2 Government Accession No	3 Recipient's Catalog No
NASA CR 137943		

4 Title and Subtitle	5. Report Date
Numerical Solution for the Temperature Distribution in a Cooled Guide Vane Blade of a Radial Gas Turbine	October 1976
	6 Performing Organization Code
7 Author(s)	8 Performing Organization Report No
W. Hosny and W. Tabakoff	
	10 Work Unit No
9 Performing Organization Name and Address	11 Contract or Grant No
Department of Aerospace Engineering University of Cincinnati Cincinnati, Ohio 45221	NAS2-7850
	13 Type of Report and Period Covered
12. Sponsoring Agency Name and Address	Contractor Report
National Aeronautics & Space Administration Washington, D.C. 20545, and U.S. Army Air Mobility Research & Development Lab. Moffett Field, California 94035	14 Sponsoring Agency Code

15 Supplementary Notes

Interim Report. Project Manager, LTC Dwain Moentmann, U.S. Army Air Mobility Research and Development Laboratory, Ames Research Center, Moffett Field, California 94035.

16 Abstract

A two-dimensional finite-difference numerical technique is presented to determine the temperature distribution of an internal cooled blade of radial turbine guide vanes.

A simple convection cooling is assumed inside the guide vane blade. Such cooling has relatively small cooling effectiveness at the leading edge and at the trailing edge. Heat transfer augmentation in these critical areas may be achieved by using impingement jets and film cooling.

A computer program is written in Fortran IV for IBM 370/165 computer.

17 Key Words (Suggested by Author(s))	18 Distribution Statement
Turbine Blade Heat Transfer	Unclassified - unlimited

19 Security Classif (of this report)	20 Security Classif (of this page)	21 No of Pages	22 Price*
Unclassified	Unclassified	58	

* For sale by the National Technical Information Service Springfield, Virginia 22161

NASA-C-168 (Rev 10-75)

77-200

ORIGINAL PAGE IS
OF POOR QUALITY

Heat Transfer in Cooled Guide Vane Blades

W. Tabakoff, R. Kotwal and A. Hamed,
University of Cincinnati, Cincinnati, Ohio

Abstract

A numerical study to determine the temperature distribution in the guide vanes of a radial inflow turbine is presented. A computer program has been developed to calculate the temperature distribution when the vanes are cooled internally using a combination of impingement and film cooling techniques. The study is based on the use of the finite difference method in a two dimensional heat conduction problem. The results are then compared to determine the best cooling configuration for a certain coolant to primary mass flow ratio.

AIAA 15th AEROSPACE SCIENCES MEETING

Los Angeles, Calif /January 24-26, 1977

Reprinted from JOURNAL OF AIRCRAFT, Vol. 14, No 2, February 1977, 173–176

Temperature Distribution Study in a Cooled Radial Inflow Turbine Rotor

A. Hamed,* E. Baskharone,† and W. Tabakoff‡
University of Cincinnati, Cincinnati, Ohio

In this work, the three dimensional temperature distribution in the cooled rotor of a radial inflow turbine is determined numerically using the finite element method. Some of the results obtained using different cooling techniques with various coolant to primary mass flow ratios are presented for comparison. In this analysis, the complicated geometries of the hot rotor and coolant passage surfaces are handled easily, and the temperatures are determined without loss of accuracy at these convective boundaries. The present work can easily be used in combination with a finite element stress analysis, to investigate the thermal stresses corresponding to the different cooling arrangements.

Nomenclature

C_f = skin friction coefficient
D = internal cooling passage hydraulic diameter (ft)
g''' = heat generation rate per unit volume (Btu/ft^3 hr)
$[G]$ = heat generation matrix, Eq. (9)
h = convective heat transfer coefficient (Btu/ft hr°F)
$[H]$ = surface convection matrix
k = coefficient of thermal conductivity (Btu/ft hr °F)
$[K]$ = thermal stiffness matrix
Nu = Nusselt number
$\hat{n}$ = outward normal unit vector from the rotor surface
Pr = Prandtl number
$\{P\}^t$ = row vector representing the spatial distribution within the element, Eq. (5)
q = heat flux density (Btu/ft^2 hr)
$[R]$ = element nodal spatial matrix, Eq. (6)
Re = Reynolds number
$[SR]$ = thermal boundary load matrix, Eq. (9)
S_h = rotor surface exchanging heat with the hot gas or coolant flow
S_q = rotor surfaces on which heat flux is specified, including adiabatic surfaces
T = temperature (°F)
V = the volume of the tetrahedral element in the finite element representation (ft^3)
$\{\alpha\}$ = column vector of coefficients, Eq. (5)
ν = kinematic viscosity (ft^2/hr)
ω = rotor angular velocity (radiance/hr)

Subscripts
x,y,z = refer to partial derivation in these directions, respectively
∞ = flow conditions generally in hot gas or coolant flow

Superscripts
t = transpose of a tensor

Introduction

In the last decade internal cooling has been used extensively resulting in the continuous increase of turbine inlet temperatures Most internal cooling systems are designed using semi-empirical methods to achieve the highest possible effectiveness. Reference 1 presents a review of the present state-of-the-art for the internal cooling of turbine nozzles in aircraft applications.

Experimental studies of air cooled radial inflow turbine rotors are reported in Refs. 2 and 3 Branger[2] investigated the effectiveness of veil cooling on the hub side of the rotor. It was found that the cooling effectiveness was larger at the rotor tip, and decreased as the cooling film is heated and mixed with the hot turbine flow Petrick and Smith[3] studied the effect of cooling the backside of the rotor on its temperature distribution. While veil and rotor backside cooling techniques are effective for the rotor disk, they induce little variation in the blade temperature. Internal cooling on the other hand can be used to produce the desired reduction in the blade temperature as reported in Ref. 4

Considering the time and cost associated with experimental studies, computational methods can provide valuable design and development information, concerning the temperature and stress distributions associated with the different turbine cooling techniques. Several studies are available for the temperature distribution in the cooled blades of the axial gas flow turbine blades. They are generally based on two dimensional temperature computations with a spanwise variation in the blade surface convective heat transfer coefficient. In the radial inflow turbine rotor, a similar approach is not possible and the real three dimensional temperature field must be investigated. The finite element method is particularly suitable for handling such complicated geometries without loss of accuracy at the boundaries.

In the present study a numerical method is developed to determine the temperature field in the rotor of a cooled radial inflow turbine The computations are based on the use of the finite element method, with a variational statement of the three dimensional conduction problem. The various boundary conditions, at the different external and internal cooling surfaces of the rotor are discussed. Different cooling methods and coolant flow rates are investigated and the resulting cooling effectiveness and temperature distributions are reported. The data obtained from the present analysis were found to be in agreement with the available experimental measurements.

Governing Equations

The field equation for the steady state three dimensional heat conduction problem in an isotropic medium can be expressed as

$$\nabla \cdot (k \nabla T) + g''' = 0$$

Presented as Paper 76-44, AIAA 14th Aerospace Sciences Meeting, Washington, D C , Jan. 26-28, 1976; submitted Feb 11, 1976, revision received June 23, 1976

*Associate Professor, Department of Aerospace Engineering, University of Cincinnati, Cincinnati, Ohio Member AIAA
†Graduate Research Assistant, Department of Aerospace Engineering, University of Cincinnati, Cincinnati, Ohio
‡Professor, Department of Aerospace Engineering, University of Cincinnati, Cincinnati, Ohio Associate Fellow AIAA

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA CR 137951		
4. Title and Subtitle: A NUMERICAL STUDY OF THE TEMPERATURE FIELD IN A COOLED RADIAL TURBINE ROTOR		5. Report Date: March 1977
		6. Performing Organization Code
7. Author(s): A. Hamed, E. Baskharone and W. Tabakoff		8. Performing Organization Report No.
		10. Work Unit No.
9. Performing Organization Name and Address: Department of Aerospace Engineering & Applied Mechanics, University of Cincinnati, Cincinnati, Ohio 45221		11. Contract or Grant No.: NAS2-7850
		13. Type of Report and Period Covered: Contractor Report
12. Sponsoring Agency Name and Address: National Aeronautics & Space Administration, Washington, D.C. 20546 and U.S. Army Air Mobility Research & Development Laboratory, Moffett Field, California 94035		14. Sponsoring Agency Code

15. Supplementary Notes

Interim Report. Project Manager, LTC Dwain Moentmann, U.S. Army Air Mobility Research and Development Laboratory, Ames Research Center, Moffett Field, California 94035.

16. Abstract

In this work, the three dimensional temperature distribution in the cooled rotor of a radial inflow turbine is determined numerically using the finite element method. Through this approach, the complicated geometries of the hot rotor and coolant passage surfaces are handled easily, and the temperatures are determined without loss of accuracy at these convective boundaries. Different cooling techniques with given coolant to primary flow ratios are investigated, and the corresponding rotor temperature fields are presented for comparison. The data obtained from the present analysis were found to be in agreement with the available experimental measurements.

The present work can be used in combination with a finite element stress analysis to investigate the thermal stresses corresponding to the different cooling arrangements. This can provide valuable information concerning the critical locations of possible creep, rupture or fatigue, for a given centrifugal, thermal and aerodynamic loading.

ORIGINAL PAGE IS
OF POOR QUALITY

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Turbomachinery Cooling Radial Turbine	Unclassified - unlimited

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price*
Unclassified	Unclassified	67	

*For sale by the National Technical Information Service, Springfield, Virginia 22161

NASA-C-168 (Rev. 10-75)

$3.00 PER COPY
$1[redacted]0 TO ASME MEMBERS

Cooling Considerations for Design of a Radial Inflow Turbine*

A. HAMED

Associate Professor
Mem ASME

Y. SHEORAN

Graduate Research Assistant

W. TABAKOFF

Professor
Mem ASME

Department of Aerospace Engineering
and Applied Mechanics,
University of Cincinnati,
Cincinnati, Ohio

A numerical study to determine the temperature distribution in the rotor of a radial inflow turbine is presented Internal cooling passages are modeled in the present formulation in order to carry out solid and coolant temperature computations simultaneously resulting in a considerable computer time savings The stresses due to centrifugal and thermal loadings are determined in an actual rotor and the effect of cooling design on its mechanical integrity is discussed

*This research work was sponsored by NASA Contract No NAS2-7850, Ames Research Center, Moffett Field, Calif

Contributed by the Gas Turbine Division of The American Society of Mechanical Engineers for presentation at the Gas Turbine Conference & Products Show, Philadelphia, PA, March 27-31, 1977 Manuscript received at ASME Headquarters December 7, 1976.

Copies will be available until December 1, 1977

77-GT-83

$3.00 PER COPY
$1.50 TO ASME MEMBERS

Computation of the Temperature Distribution in Cooled Radial Inflow Turbine Guide Vanes

W. TABAKOFF

Professor
Mem ASME

W. HOSNY

Postdoctoral Fellow

A HAMED

Associate Professor
Mem ASME

Department of Aerospace Engineering
and Applied Mechanics,
University of Cincinnati,
Cincinnati, Ohio

A two-dimensional finite-difference numerical technique is presented to determine the temperature distribution of an internally-cooled blade of radial turbine guide vanes A simple convection cooling is assumed inside the guide vane Such an arrangement results in relatively small cooling effectiveness at the leading edge and at the trailing edge Heat transfer augmentation in these critical areas may be achieved by using impingement jets and film cooling A computer program is written in Fortran IV for IBM 370/165 computer

Contributed by the Gas Turbine Division of The American Society of Mechanical Engineers for presentation at the Gas Turbine Conference & Products Show, Philadelphia, PA, March 27-31, 1977. Manuscript received at ASME Headquarters December 8, 1976

Copies will be available until December 1, 1977

AMERICAN SOCIETY OF MECHANICAL ENGINEERS, UNITED ENGINEERING CENTER, 345 EAST 47th STREET, NEW YORK, N Y 10017

1 Report No CR No. to be assigned.	2 Government Accession No	3 Recipient's Catalog No
4 Title and Subtitle AN INVESTIGATION OF TEMPERATURE DISTRIBUTION IN COOLED GUIDE VANES		5 Report Date September 1977
		6 Performing Organization Code
7 Author(s) R. Kotwal, W. Tabakoff and A. Hamed		8 Performing Organization Report No
9 Performing Organization Name and Address Department of Aerospace Engineering & Applied Mechanics University of Cincinnati Cincinnati. Ohio 45221		10 Work Unit No
		11 Contract or Grant No NAS2-7850
12 Sponsoring Agency Name and Address National Aeronautics & Space Administration Washington, D.C. 20546 U.S. Army Air Mobility Research & Development Laboratory Moffett Field, California 94035		13 Type of Report and Period Covered
		14 Sponsoring Agency Code
15 Supplementary Notes		

16 Abstract

A numerical study to determine the temperature distribution in the guide vane blades of a radial inflow turbine is presented. A computer program has been developed which permits the temperature distribution to be calculated when this blade is cooled internally using a combination of impingement and film cooling techniques. The study is based on the use of the finite difference method in a two dimensional heat conduction problem. The results are then compared to determine the best cooling configuration for a certain coolant to primary mass flow ratio.

17 Key Words (Suggested by Author(s))	18 Distribution Statement
Heat Transfer Radila Turbine	Unclassified - unlimited.

19 Security Classif (of this report)	20 Security Classif (of this page)	21 No of Pages	22 Price*
Unclassified	Unclassified		

* For sale by the National Technical Information Service, Springfield Virginia 22161

NASA-C-168 (Rev 10-75)

1 Report No CR No. to be assigned.	2 Government Accession No	3 Recipient's Catalog No
4 Title and Subtitle OPTIMIZATION STUDY FOR HIGH SPEED RADIAL TURBINE WITH SPECIAL REFERENCE TO DESIGN VARIABLES		5 Report Date October 1977 6 Performing Organization Code
7 Author(s) I. Khalil and W. Tabakoff		8 Performing Organization Report No 10 Work Unit No
9 Performing Organization Name and Address Department of Aerospace Engineering & Applied Mechanics University of Cincinnati Cincinnati, Ohio 45221		11 Contract or Grant No NAS2-7850 13 Type of Report and Period Covered
12 Sponsoring Agency Name and Address National Aeronautics & Space Administration Washington, D.C. 20546 and U.S. Army Air Mobility Research & Development Laboratory Moffett Field, California 94035		14 Sponsoring Agency Code
15 Supplementary Notes		

16 Abstract

Numerical results of a theoretical investigation are presented to provide information about the effect of variation of the different design and operating parameters on radial inflow turbine performance. The effects of variations in the mass flow rate, rotor tip Mach number, inlet flow angles, number of rotor blades and hub to shroud radius ratio, on the internal fluid dynamics of turbine rotors, was investigated. A procedure to estimate the flow deviation angles at the turbine exit is also presented and used to examine the influence of the operating conditions and the rotor geometrical configuration on these deviations. The significance of the results obtained are discussed with respect to improved turbine performance.

17 Key Words (Suggested by Author(s))	18. Distribution Statement
Radial Turbine Internal Fluid Dynamics in Radial Turbines	Unclassified - unlimited

19 Security Classif (of this report)	20 Security Classif (of this page)	21 No of Pages	22 Price*
Unclassified	Unclassified	62	

*For sale by the National Technical Information Service, Springfield, Virginia 22161

NASA-C-168 (Rev 10-75)

Abstract

With increased turbine inlet temperatures, numerical methods of thermal and stress analysis are becoming more valuable in the design of air-cooled turbines. This paper presents a study of the stresses associated with different cooling patterns in a radial inflow turbine rotor. The finite element method is used in the stress calculations taking into consideration centrifugal, thermal and aerodynamic loading. The effects of temperature distribution and the presence of internal cooling passages are discussed.

78-94

STRESS ANALYSIS STUDY IN COOLED RADIAL INFLOW TURBINE

A. Hamed, Y. Sheoran and W. Tabakoff
University of Cincinnati, Ohio

AIAA 16TH AEROSPACE SCIENCES MEETING

Huntsville, Alabama/January 16-18, 1978

VISCOUS FLOW ANALYSIS IN TURBOMACHINE ROTORS

I.M. Khalil and W. Tabakoff

A method of analyzing the turbulent nonadiabatic viscous flow through turbomachine rotors is now in the progress stage. The field analysis is based upon the numerical integration of the full Navier-Stokes equations, together with the energy equation, over the rotor blade-to-blade stream surfaces. The numerical code used to solve the elliptic governing equations will employ a curvilinear body-oriented coordinate system that suits the most complicated blade geometries. Turbulence will be represented by a two-equation model, one expressing the development of the turbulent kinetic energy, and the other its dissipation rate.

The flow analysis will be applied to various types of turbomachine rotors. The flow characteristics within the different rotor passages will be investigated over a wide range of operating conditions. The development of the viscous boundary layers over the blade surfaces will be also studied for different boundary conditions. Calculated results will be compared to experimental data. It is felt that the method of this analysis is quite general and can deal with wide range of applications.

Preliminary computations, using the computer programs developed during this period of study, were performed for different flow configurations. For example, the viscous flows within the rotors of a radial bladed compressor and a mixed flow turbine are investigated under different operating conditions. Excellent results were obtained when compared with existing experimental data over a wide range of Reynolds numbers.

Currently, active steps are being carried out to incorporate Thompson approach (Ref. 1) for transformation in our main code such that the complicated blade geometries can be dealt with. Up till now, we are faced with minor problems in the transformation process and procedures are being devised to solve these problems. Specifically, the mesh resolution near the leading and trailing edge needs special attention. The Jacobian of transformation for these regions varies largely and leads to less accurate results if small step sizes are not considered.

FLOW ANALYSIS

The partial differential equations that govern the flow behavior within the turbomachine rotor passages are presented. A transformation of the three dimensional flow equations to several particular two dimensional forms, on predetermined stream surfaces, is then outlined. The details of the mathematical formulation used to perform the transformation is not reported here, but the final results are summarized. The resulting flow equations are further expressed in the form of conservation law by using the vorticity-stream function formulation. Finally, the boundary conditions necessary to obtain a unique solution to the problem are discussed.

Fundamental Aerothermodynamic Relations

The three dimensional flow of viscous, compressible fluid through turbomachinery is governed by the following set of laws.

Conservation of Mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{W}) = 0 \tag{1}$$

Conservation of Momentum:

For a blade row rotating at a constant angular velocity ($\bar{\Omega}$) about its axis, Newton's second law of motion when combined with Stokes hypothesis can be written as

$$\rho\left(\frac{\partial \bar{W}}{\partial t} + \bar{W}\cdot\nabla\bar{W} + 2\bar{\Omega}\times\bar{W} - \Omega^2\bar{R}\right) = -\nabla p - \nabla\times[\mu(\bar{\nabla}\times\bar{W})] + \frac{4}{3}\nabla(\mu\nabla\cdot\bar{W}) \tag{2}$$

Conservation of Energy:

In the absence of heat sources or sinks, the first law of thermodynamics for a heat conducting fluid with a thermal conductivity (K) can be written as:

$$\rho\left(\frac{\partial h}{\partial t} + \bar{W}\cdot\nabla h\right) = \frac{\partial p}{\partial t} + (\bar{W}\cdot\nabla)p + \mathcal{D} + \nabla\cdot(K\nabla T) \tag{3}$$

where p = static pressure,
ρ = static density,
T = static temperature,
h = static enthalpy
$\bar{W}$ = relative velocity vector,
$\bar{R}$ = position vector
μ = kinematic viscosity coefficient,
$\mathcal{D}$ = viscous dissipation function.

The relations among the flow state variables are those of the ideal gas equations:

$$p = \rho RT \tag{4}$$

and

$$dh = C_p \, dT \tag{5}$$

ORIGINAL PAGE IS OF POOR QUALITY

In order that the above equations represent a turbulent flow situation, the mass-averaged variables principle is utilized. According to this principle, the components of the velocity vector $\bar{W}$, and the temperature T, are expressed on a mass-averaged basis, while the pressure and the density ρ are expressed by their mean values only. For any variable $Q(x_1,x_2,x_3,t)$, its mass-average is given as follows:

$$q(x_1,x_2,x_3) = \overline{Q(x_1,x_2,x_3,t) \; P(x_1,x_2,x_3,t)}/\rho$$

where $P(x_1,x_2,x_3,t)$ is the instantaneous value of the density at any point whose location is defined by the coordinate (x_1,x_2,x_3). The over bar in the above equation represents the conventional time average, thus $\bar{P} \equiv \rho$. An effective viscosity μ_e is also used instead of the kinematic viscosity μ. This effective viscosity is assumed to describe the effects of Reynolds stress. In the present work, a two equation turbulence model will be employed to calculate μ_e.

Preliminaries

An approach is taken to reduce the spatial dimensions of the problem from the general three dimensional form to several particular two dimensional forms. The object is to obtain a solution to the flow governed by the equations (1) to (5) through an appropriate combination of two dimensional flow solutions. The reduction in spatial dimension is achieved through a consideration of the blade-to-blade stream surface concept. The blade-to-blade stream surface, S_1, may be described as the surface that would extend from the pressure surface of a blade to the suction surface of the next blade as shown in Figure 1a. If desired, this surface may be considered to be a surface of revolution generated by a meridional streamline obtained from a meridional flow analysis. On this stream surface, the flow equations (1), (2) and (3) are transformed to several two dimensional mathematical expressions. This results from the fact that the S_1 surface provides relations amongst the coordinate variables, such that the variation of flow properties over the surface may be described in terms of two space variables only. If a solution to the resulting two dimensional equations is obtained, then with several such blade-to-blade solutions, the flow properties throughout the three dimensional field may be evaluated.

Stream Surfaces Equations:

Consider a flow annulus as shown in Figure 1. The curvilinear distance along the intersection of the mid-line of the annulus with a meridional plane is denoted by m. The distance normal to the mid-line in a meridional plane is labeled by n. The circumferential coordinate ϕ is considered positive in the counterclockwise direction when viewed down the positive z axis. The thickness of the annulus, b, is assumed to be small compared to the radius r. Hence, the n component of the velocity vector and all variations in the n direction are neglected (except for the regions near the hub and casing). Accordingly, the annulus is termed a stream surface. The transformation of the governing equations to the two-dimensional stream surface coordinate system gives the following results:

Continuity:

$$br \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial m} (br\rho W_m) + \frac{\partial}{\partial \phi} (b\rho W_\phi) = 0 \qquad (6)$$

Meridional Momentum:

$$\rho \frac{\partial W_m}{\partial t} + \rho (W_m \frac{\partial W_m}{\partial m} + \frac{W_\phi}{r} \frac{\partial W_m}{\partial \phi} - \frac{W^2{}_\phi}{r} \sin\alpha - \Omega^2 r \sin\alpha - 2\Omega W_\phi \sin\alpha)$$

$$= - \frac{\partial p}{\partial m} - \frac{2}{3} \frac{\partial}{\partial m} [\mu_e (\frac{\partial W_m}{\partial m} + \frac{W_m}{r} \sin\alpha + \frac{1}{r} \frac{\partial W_\phi}{\partial \phi})]$$

$$+ \frac{1}{r} \{\frac{\partial}{\partial m} (2\mu_e r \frac{\partial W_m}{\partial m}) + \frac{\partial}{\partial \phi} [\mu_e (\frac{\partial W_\phi}{\partial m} - \frac{W_\phi}{r} \sin\alpha + \frac{1}{r} \frac{\partial W_m}{\partial \phi})]\}$$

$$- 2 \frac{\mu_e}{r} [\frac{1}{r} \frac{\partial W_\phi}{\partial \phi} + \frac{W_m}{r} \sin\alpha] \sin\alpha \qquad (7)$$

Tangential Momentum:

$$\rho r \frac{\partial W_\phi}{\partial t} + \rho (rW_m \frac{\partial W_\phi}{\partial m} + W_\phi \frac{\partial W_\phi}{\partial \phi} + W_m W_\phi \sin\alpha + 2\Omega r W_m \sin\alpha)$$

$$= - \frac{\partial p}{\partial \phi} - \frac{2}{3} \frac{\partial}{\partial \phi} [\mu_e (\frac{\partial W_m}{\partial m} + \frac{W_m}{r} \sin\alpha + \frac{1}{r} \frac{\partial W_\phi}{\partial \phi})] + \frac{\partial}{\partial m} [\mu_e (r \frac{\partial W_\phi}{\partial m}$$

$$- W_\phi \sin\alpha + \frac{\partial W_m}{\partial \phi})] + \frac{\partial}{\partial \phi} [2\mu_e (\frac{1}{r} \frac{\partial W_\phi}{\partial \phi} + \frac{W_m}{r} \sin\alpha)] + \mu_e \sin\alpha [\frac{\partial W_\phi}{\partial m}$$

$$- \frac{W_\phi}{r} \sin\alpha + \frac{1}{r} \frac{\partial}{\partial \phi} W_m] \qquad (8)$$

where

W_m = relative meridional mean velocity component,

W_ϕ = relative tangential mean velocity component,

α = $\sin^{-1} \frac{dr}{dm}$ = angle between meridional streamline and axis of rotation, fig. 1.

The system of the differential equations (6), (7) and (8), can be written alternatively in terms of a stream function ψ and a vorticity ω. In the present study, the stream function-vorticity formulation is considered for the purpose of simplifying the handling of the boundary conditions. Among other advantages, this formulation offers the ability to express the governing equations in the conservation law form. By so doing, the finite difference approximation of the partial differential equations will not allow excessive accumulation of the errors in the fluxes of the conserved quantities.

For steady state flow situation, and for the stream channel configuration shown in Figure 1, the continuity equation is satisfied by specifying a stream function (ψ), with the following relations:

$$W_m = \frac{\dot{M}}{b} \frac{1}{\rho r} \frac{\partial \psi}{\partial \phi}$$

and
$$W_\phi = - \frac{\dot{M}}{b} \frac{1}{\rho} \frac{\partial \psi}{\partial m} \qquad (9)$$

where $\dot{M}$ is the rate of mass flow passing through the stream channel. If the pressure terms from equations (7) and (8) are eliminated by cross-differentiation and the mean vorticity variable ω is introduced, one obtains the vorticity transport equation:

$$\frac{\partial}{\partial m} \left(\frac{\dot{M}}{b} \frac{\partial \psi}{\partial \phi} \omega\right) - \frac{\partial}{\partial \phi} \left(\frac{\dot{M}}{b} \frac{\partial \psi}{\partial m} \omega\right) + \frac{\partial \rho}{\partial m} \frac{\partial W^2/2}{\partial \phi} - \frac{\partial \rho}{\partial \phi} \frac{\partial W^2/2}{\partial m}$$

$$- \frac{\partial}{\partial m} \left[r \frac{\partial}{\partial m} (\mu_e \omega)\right] + \frac{\partial}{\partial \phi} \left[\frac{1}{r} \frac{\partial}{\partial \phi} (\mu_e \omega)\right] + G_1 = 0 \qquad (10)$$

where $$W^2 = W_m^2 + W_\phi^2 \tag{11}$$

$$G_1 = 2\Omega r(\sin\alpha)\ W_m \frac{\partial\rho}{\partial m} + 2\Omega(\sin\alpha)\ W_\phi \frac{\partial\rho}{\partial\phi} + 2\Omega r(\sin\alpha)\rho \frac{\partial W_m}{\partial m}$$

$$+ 2\Omega\rho(\sin\alpha) \frac{\partial W_\phi}{\partial\phi} + 2\Omega(\sin^2\alpha)\ \rho W_m + 2\Omega r\rho W_m \frac{\partial \sin\alpha}{\partial m}$$

$$+ \Omega^2 r \sin\alpha \frac{\partial\rho}{\partial\phi} \tag{12}$$

and ω is defined by

$$\omega = \frac{1}{r}\left[\frac{\partial}{\partial m}(rW_\phi) - \frac{\partial}{\partial\phi}(W_m)\right] \tag{13}$$

When the velocities W_m and W_ϕ in equation (13) are expressed in terms of the stream functión variable, as defined in equation (9), equation (13) would reduce to the stream function equation

$$\omega = -\frac{1}{r}\left[\frac{\partial}{\partial m}\left(\frac{\dot{M}r}{b\rho}\frac{\partial\psi}{\partial m}\right) + \frac{\partial}{\partial\phi}\left(\frac{\dot{M}}{b\rho r}\frac{\partial\psi}{\partial\phi}\right)\right] \tag{14}$$

For a turbomachine rotor it is convenient to express the energy equation in terms of the total enthalpy (H) of the gas, besides it velocity components. The total enthalpy for turbulent flow is expressed as follows:

$$H = h + \frac{V^2}{2} + E$$

or

$$H = h + \frac{W^2}{2} + \Omega W_\phi r + \frac{\Omega^2 r^2}{2} + E \tag{15}$$

where $\bar{V}$ = absolute velocity vector

E = kinetic energy of turbulence.

Therefore, the energy equation (3), when transformed to the two-dimensional stream surface coordinate system, results in

$$\frac{\partial}{\partial m}\left(\frac{\dot{M}}{b} H \frac{\partial \psi}{\partial \phi}\right) - \frac{\partial}{\partial \phi}\left(\frac{\dot{M}}{b} H \frac{\partial \psi}{\partial m}\right) - \frac{\partial}{\partial m}\left(\frac{K}{C_p} r \frac{\partial H}{\partial m}\right) - \frac{1}{r}\frac{\partial}{\partial \phi}\left(\frac{K}{C_p}\frac{\partial H}{\partial \phi}\right)$$

$$+ \frac{\partial}{\partial m}\{\mu_e r \left[\frac{1}{P_r}\frac{\partial W^2/2}{\partial m} - \left(\frac{1}{S_{ch}} - \frac{1}{P_r}\right)\frac{\partial E}{\partial m}\right]\}$$

$$+ \frac{\partial}{\partial \phi}\{\frac{\mu_e}{r}\left[\frac{1}{P_r}\frac{\partial W^2/2}{\partial \phi} - \left(\frac{1}{S_{ch}} - \frac{1}{P_r}\right)\frac{\partial E}{\partial \phi}\right]\} - W_\phi \frac{\partial}{\partial m}(\mu_e \omega)$$

$$+ \frac{W_m}{r}\frac{\partial}{\partial \phi}(\mu_e \omega) - Dr + G_2 = 0 \tag{16}$$

where P_r = turbulent Prandtl number = $(C_p \mu_e / K)$

S_{ch} = turbulent Schmidt number = (μ_e / Γ_E)

Γ_E = exchange coefficient of the kinetic energy of turbulence

$$G_2 = -\Omega \frac{\dot{M}}{b}\{\frac{\partial}{\partial m}\left[(W_\phi r + r^2\Omega)\frac{\partial \psi}{\partial \phi}\right] - \frac{\partial}{\partial \phi}\left[(W_\theta r + r^2\Omega)\frac{\partial \psi}{\partial m}\right]\}$$

$$+ \Omega\{\frac{\partial}{\partial m}\left[\frac{\mu_e}{P_r} r \frac{\partial}{\partial m}\left(W_\phi r + \frac{\Omega r^2}{2}\right)\right] + \frac{1}{r}\frac{\partial}{\partial \phi}\left[\frac{\mu_e}{P_r}\frac{\partial}{\partial \phi}\left(W_\phi r + \frac{\Omega r^2}{2}\right)\right]\} \tag{17a}$$

and the dissipation function, D, is given by

$$D = 2\mu_e \{\left(\frac{\partial W_m}{\partial m}\right)^2 + \left(\frac{1}{r}\frac{\partial W_\phi}{\partial \phi} + \frac{W_m}{r}\sin\alpha\right)^2\}$$

$$+ \mu_e \{\frac{\partial W_\phi}{\partial m} + \frac{1}{r}\frac{\partial W_m}{\partial \phi} - \frac{W_\phi}{r}\sin\alpha\}^2 \tag{17b}$$

The properties of flow passing through the stream surface S_1, are completely defined by equations (4), (9), (10), (14) and (16) together with the known variations of μ_e, K and the given boundary conditions. The kinetic viscosity, μ_e, is calculated from a two equation model, one expressing the development of the turbulent kinetic energy, E, and the other its dissipation rate, ε. These equations are expressed as follows.

Turbulent Kinetic Energy Equation:

$$\frac{\partial}{\partial m}\left(\frac{\dot{M}}{b} E \frac{\partial\psi}{\partial\phi}\right) - \frac{\partial}{\partial\phi}\left(\frac{\dot{M}}{b} E \frac{\partial\psi}{\partial m}\right) - \frac{\partial}{\partial m}\left(\frac{\mu_e}{S_{ch}} r \frac{\partial E}{\partial m}\right)$$

$$+ \frac{\partial}{\partial\phi}\left(\frac{\mu_e}{S_{ch}} \frac{\partial E}{r\partial\phi}\right) - r\phi \div \rho\varepsilon r = 0 \tag{18}$$

Dissipation Rate Equation:

$$\frac{\partial}{\partial m}\left(\frac{\dot{M}}{b} \varepsilon \frac{\partial\psi}{\partial\phi}\right) - \frac{\partial}{\partial\phi}\left(\frac{\dot{M}}{b} \varepsilon \frac{\partial\psi}{\partial m}\right) - \frac{\partial}{\partial m}\left(\frac{\mu_e}{S_{ch}} r \frac{\partial\varepsilon}{\partial m}\right)$$

$$+ \frac{\partial}{\partial\phi}\left(\frac{\mu_e}{S_{ch}} \frac{1}{r}\frac{\partial\varepsilon}{\partial\phi}\right) - C_1 \frac{\varepsilon}{E} r\phi + C_2 \frac{\varepsilon^2}{E} \rho r = 0 \tag{19}$$

where C_1 and C_2 are constants to be determined from available experimental data. The relation amongst E, ε and μ_e is given by:

$$\mu_e = C_D \frac{\rho E^2}{\varepsilon} \tag{20}$$

In general, equations (4) through (19) are valid for any turbomachine geometry or any number of stream surfaces except for the two stream surfaces S_{1_0} and $S1_N$ as shown in Figure 2, which contain the hub and shroud profiles. This exception may be attributed to the existence of a large variation in flow properties along the normal, $\bar{n}$, to these two surface resulting from the presence of the solid boundaries.

The solution of the above system of equations within the turbomachine passages is carried out numerically. One can observe from equations (10), (14), (16), (18) and (19), that they are coupled elliptic partial differential equations. The highest order derivatives in those equations are the second derivatives of ψ, ω, H, E, and ε and as such ψ, ω, H, E and ε are the dependent variables. From the nature of the problem, none of the terms are negligible in the governing equations. The convective terms introduce nonlinearity and also instability if the proper differences are not taken into account. Once a solution for these variables has been obtained, the velocity distribution can be determined from equation (9). The pressure distribution can then be evaluated from either equations (7) or (8). In order to solve the elliptic equations by the usual numerical methods, it is necessary to define a selected region in the physical domain with boundary conditions specified for all the dependent variables.

Description of the Computational Domain in the Physical Space

The flow region of interest, as shown in Fig. 3, contains the blade row and segments of the stream surface, S_1, extending upstream and downstream of the row. Due to the circumferential periodicity in turbomachine passages, the selected domain need to encompass only that fraction of the flow annulus containing a single blade to blade passage. The shape and location of the periodic boundaries (AB, NM, IH and FG) may be defined arbitrarily so long as their spacing corresponds to the blade pitch $2\pi/Z$. The upstream and downstream boundaries (AN, GH) are located sufficiently far from the blade so that tangential variation along them are ignored. The flow properties are consequently considered to be uniform along the boundary AN and GH.

Boundary Conditions

In specifying the boundary conditions, two flow situations are investigated. Preliminarily, only the case of the laminar flow is considered. The turbulent flow case is now in progress. Accordingly, in the following specification of boundary conditions, no assignment for the boundary values of E and ε in equations (18) and (19) is needed. Moreover, the flow properties within the turbomachine channels is completely defined through the simultaneous solution of only equations (10), (14) and (16).

Upstream flow boundary AN:

It is a common practice in turbomachine flow calculations that the magnitude and the direction of the flow velocities, the total temperature and the total pressure or density are defined at the engine inlet. Therefore, along the boundary AN the values of ψ, ω, H or their derivatives can be evaluated using the defined flow properties. The known magnitude of the inlet relative velocity and its direction, as shown in Figure 3, specify the values of $\partial\psi/\partial m$ and $\partial\psi/\partial\phi$ according to the relations

$$\frac{\partial \psi}{\partial m} = - \frac{b\rho}{\dot{M}} W_{\phi} \, , \qquad \frac{\partial \psi}{\partial \phi} = \frac{b\rho r}{\dot{M}} W_{m} \tag{21}$$

Since the inlet stream of the gas is considered to be uniform, the absolute value of vorticity ω has to be zero along the boundary AN. In a rotating frame of reference, as it is in the present case, the relative value of ω is given by the following expression

$$\omega = - \, [2\Omega \, \sin\alpha]_{\text{inlet}}$$

The value for the total enghalpy, H, can be defined using the specified flow properties.

<u>The periodic flow boundaries AB, NM and FG, IH:</u>

The periodicity condition requires that the magnitude and direction of the flow velocity as well as other fluid properties be equal at every two corresponding points along AB and MN. Similarly, the same condition should apply at every two corresponding points along FG and IH. The periodicity condition in terms of the dependent variables is satisfied through the following conditions. First, equating ω, $\partial\omega/\partial\phi$, $\partial\psi/\partial\phi$, H, and $\partial H/\partial\phi$ values on each corresponding points. Second, ensuring that the ψ values are differed by unity between the corresponding points.

<u>The blades surfaces boundary (MI and BF):</u>

For the laminar flow case, two boundary conditions over the blades surfaces are usually specified. These are the non-slip condition and the impermeability of the surface in the case of blades with no injection. The non-slip condition requires that

$$\frac{\partial \psi}{\partial \eta} = 0 \tag{23}$$

where η is the normal to the blade surface. On the other hand the impermeability condition requires that no component of velocity exist in the direction normal to the blade surface. Therefore, the blade surfaces are treated as streamlines with the ψ values specified as zero on the (MI) surface and unity on the (BF) surface. On either MI or BF surfaces, one has therefore, two boundary conditions for ψ but not boundary condition for ω. It is a well accepted fact in computational fluid mechanics to rely on a modified evaluation of equation (14) to determine the boundary condition for ω. The modification is introduced in an attempt to insure that equation (23) holds, that is, to satisfy the no slip condition. This approach is utilized in the current study to determine the value of the vorticity, ω, over the blade surfaces.

In regard to the thermal boundary conditions either the blade surface temperature is known or the normal derivative $\partial T/\partial \eta$ is specified as zero for the adiabatic wall conditions. In either case, equation (15) is used to determine the value of H or its derivative that is used as boundary condition for equation (16).

The downstream boundary GH:

Two basic problems arise in the specification of the boundary functions for the dependent variables along GH. The first concerns the behavior of the two dependent variables ω and H. The nature of the problem is that, physically, ω and H are known only if the boundary is located at an arbitrarily large distance from the blade surface. In this case the absolute value of vorticity, ω is zero, while the value of H is that of the corresponding surroundings. The placement of GH at exceedingly large distances from the blade boundary is quite obviously not possible for numerical considerations. Therefore, one has to employ some auxiliary condition, usually obtained by experience, to define ω and H implicitly. The conditions of zero gradient in the meridional direction, i.e.

$$\frac{\partial \omega}{\partial m} = \frac{\partial H}{\partial m} = 0 \tag{24}$$

seems to be appropriate and therefore is employed in the current work.

More important than the specification of the remote boundary functions of ω and H along GH is the determination of the ψ values at the same boundary. The downstream flow velocities, which may be used to determine ψ derivatives along GH, and that guarantee a unique solution to the problem are not known in general apriori. Therefore, one has to introduce a supplementary condition, generally resulting from physical intuition, to define the ψ derivatives. Generally, two approaches have been utilized by investigators working in the inviscid flow area to deal with this problem. Both of them are based on an iteration procedure, through which the Kutta condition for tangency of the flow at the blade trailing edge is satisfied. This is equivalent to specify the unique solution to the problem. Unfortunately, the Kutta condition cannot be applied in a realistic manner to solve the present flow problem due to its viscous nature. The conservation of angular momentum principle is employed as an alternative supplementary condition that results in the required unique solution. The procedure is as follows:

Estimated exit flow angles, β_{exit}, along GH are used to specify the values of ψ derivatives in m direction through the following relation

$$\frac{\partial \psi}{\partial m} = \frac{1}{2\pi/z} \frac{\tan\beta_{exit}}{r_{exit}} \tag{25}$$

The flow field equations are then solved for the boundary function of ψ given by equation (25) to obtain the velocity and the pressure distribution throughout the turbomachine channel. An evaluation of the torque developed by the channel is obtained through the integration of the difference in pressure and shear forces acting on the blade surfaces. The change in the angular momentum between the known inlet and the estimated exit flow conditions is calculated. If the value of the predicted torque was not equal to the rate of change of the angular momentum then the direction of the exit flow velocity is altered. The whole procedure is repeated until a satisfactory result is obtained.

PRELIMINARY RESULTS AND DISCUSSION

In the present phase of the study, where the case of laminar flow is considered, the governing differential equations (10), (14) and (16) are reduced to a set of simultaneous algebraic finite difference equations. These algebraic expressions are then recasted into a suitable form for solution by an iterative successive substitution technique. A computer program is developed for the numerical solution of the resulting algebraic equations subjected to the prescribed boundary conditions.

The program is arranged to handle general flow situations within turbomachinery. These may be axial, radial or mixed flow types. In general, the program requires as an input the shape and the configuration of the stream surface S_1, the blades geometry, the inlet flow conditions, and the rotational speed of the machine. The configuration of the stream surface S_1, is usually generated from a meridional flow analysis. The output consists of the distribution of the stream function, the vorticity and the static pressures within the blade passages. The variation of meridional and tangential velocity components from blade-to-blade and from the inlet of the machine to its exit is also generated. In cases where blade cooling is considered, the program has the capability to generate the temperature distribution and the local heat transfer coefficients along the blade surfaces.

Six flow cases are investigated using the developed program. The main purpose was to check the accuracy of the present method of analysis in predicting the actual flow behavior within turbomachine channels. The accuracy of the method was confirmed by a comparison with available experimental data. Five of those investigated cases were concerned with inward flow situation while the sixth one dealt with an outward flow case. In all cases investigated, the flow was considered to be incompressible and having a constant kinematic viscosity μ_e of 1.789×10^{-5} N - Sec/mt^2. The blade surfaces are assumed to be adiabatic with no heat sources or sinks.

Inward Flow Cases

These flow cases are those of a radial inflow turbine whose rotor consists of 8 radial straight blades. A full description of the rotor geometry is given in Fig. 4. The flow patterns on a blade-to-blade stream surface are presented over a wide range of operating conditions. The blade-to-blade stream surface S_1 is located midway the passage depth of the rotor as shown in Fig. 4. Table 1 summarizes the range over which the parameters for the numerical solution are varied. The results are presented as stream function contour plots, velocity profiles across the rotor passages together with some information concerning the pressure distribution within these passages.

The streamline contours for the five inward solution cases are shown in Fig. 5. The streamlines are plotted for the region between a pair of blades, represented by the heavy thick lines. The streamlines are designated by a stream function ratio ψ/ψ_{total} such that the value of a streamline indicates the ratio of flow through the passage between the streamline and the pressure surface of the blade. Thus, for the given channel configuration, the streamline spacing is indicative of the velocity relative to the rotor, with close spacing indicating high velocities and wide spacing indicating low velocities. For the operating conditions corresponding to case 2, as shown in Fig. 5a, it is observed that a recirculating eddy began to form near the pressure surface of the blades. As the rotating speed increases, the recirculating zone grows much larger, as shown for case 4 in

Fig. 5b. The relative velocity near the suction side of the blades increases in the later case. This may be attributed to the fact that the effective sectional area of the rotor decreases with the growth of the recirculating zone. In actual practice, large recirculatory zones are expected to cause higher losses in total pressure. Hence, it is desirable to avoid them through an efficient design of the rotor and by selecting a proper operating conditions. From inspection of Fig. 5 it is clear that the size of the recirculating eddy depends upon the relative magnitude of the flow rate ($\dot{M}\sqrt{T}/p$) and the rotor tip speed U_{tip}. In general, these zones can be reduced by increasing the mass flow rate or decreasing the rotor tip speed.

The most remarkable feature of the present results is the good agreement obtained between the predicted flow behavior and the experimental evidence given in reference 2. In all cases studied, the size and the extent to which the recirculating zone grows compares favorably well with the experimental data, as indicated in Figs. 5a, 5b and 5c.

Figures 6a, 6b and 6c show the predicted meridional velocity distribution across the blade passages at three different radial locations. These locations are selected to correspond to a radius of 23.2, 15 and 6.8 cms, respectively. The dotted line shown on each figure represents the velocity distribution for the inviscid case. From the profile distributions in the previous figures, it is evident that near the turbine inlet the boundary layers formed over the blade pressure surfaces are thicker compared to those formed on the blade suction surfaces. One of the unique features of the viscous flow in rotating channels is the large variation in the meridional velocity profiles as the flow travels downstream in the channel. The profile distribution, at stations located away from the turbine inlet, shows that regions of high meridional velocities are shifted towards the blades suction surface, as shown in Figures 6a, 6b and 6c. While, regions of high relative meridional velocities are observed to exist near the blades pressure surface at subsequent downstream stations. The inviscid flow solution gives a completely different flow behavior, in this respect, when compared with the viscous calculations. Moreover, in some flow situations where severe changes take place near the rotor tip as in case 5 in Fig. 6c, the inviscid flow solution fails completely even in predicting the flow characteristics. All these factors, in addition to the appearance of the reversed flow regions made it clear why the inviscid solutions always fail to produce a realistic prediction of boundary layer characteristic parameters in rotating machines when used in conjunction with standard boundary layer interacting techniques.

Figures 7a, 7b and 7c show the pressure variation between blades at different radial locations. These radial locations correspond to radius, r = 13.1, 16.3, 19.5 and 22.7 cms. The static pressure, p, is plotted in these figures using the nondimensional quantity, $C_p = p_1 - p / \frac{\rho_1}{2g} U_{tip}^2$, where p_1, ρ_1 are the mean static pressure and density at rotor inlet respectively. As with the stream function and velocity results, the present method of analysis provides an excellent prediction of the pressure distribution over a wide range of operating conditions. Correlation with experimental data of reference 2 as shown in the previous figures verify this statement. On the whole, the value of C_p is observed to be larger on the suction side of the blades and decreases near the pressure surface. The rate of decrease of C_p values is smaller when the rotational speed is increased. Also, in the region lying between the center of the passage and the pressure surface, the C_p values become smaller as the rotational speed decreases. All these observations imply that the static pressure drop from the rotor inlet increases with high rotational speed. Near the suction surface, the pressure drop increases with the decrease of rotational speed. Such a tendency is remarkable, particularly near the rotor inlet.

Outward Flow Case

The capability of the method of analysis to deal with flow in diffusion cascades is demonstrated by studying the flow behavior within the rotor of a radial bladed compressor. The compressor rotor under consideration has 12 straight radial blades. Other data pertaining to the rotor geometry is shown in Fig. 8. The input data for this case is presented also in Table 1. A typical distribution of the flow properties on the stream surface located midway (between the shroud and the hub) as shown in Fig. 8 are calculated and the corresponding results are presented in Figures 9 and 10. As with the inward flow cases, the stream function profile of Fig. 9 reflects the appearance of the recirculatory eddy. The inviscid flow solution, although shows the existance of the negative flow regions as illustrated in Fig. 10b, it overestimates the size of the recirculating eddy. This overestimation is supported by the existance of large negative meridional velocities near the blades suction surface. In an actual case, boundary layer phenomena are expected to reduce the effective flow area of the passage, thus increasing the volume flow rate per unit area through the effective area and thereby reducing the size of the eddy. This is exactly the same result obtained using the present viscous flow solution

In conclusion, it may be stated that the present method of analysis gives a better representation of the actual flow situation within the passage of turbomachine rotors. The preservation of the ellipticity of the flow problem is believed to be the major reason that results in this better representation. The ellipticity is preserved through the consideration of all the diffusion terms of the governing equations during the solution procedures. The method proves also to be accurate and provides invaluable information on the rotor flow characteristics. This is evidenced by the good agreement obtained between the predicted results and the available experimental data. The preliminary results obtained for the radial bladed passages of turbine and compressor had provided a great insight into the flow phenomena and this should encourage the efforts undertaken currently to deal with more complicated blade shapes. Such development in addition to the inclusion of the turbulence model in the method of solution are undoubtably essential for further aerodynamic improvement and performance prediction of turbomachines.

TABLE 1. PARAMETER FOR THE NUMERICAL SOLUTIONS

Case No.	Designation	$\dot{M}\sqrt{T_o}/P_o$	$N/\sqrt{T_o}$	U_{tip} (mt/sec)	β at upstream boundary AN, degrees	β at downstream boundary HG, degrees	Reynolds No. $=W^{\dagger}_{min}\cdot r_{tip}/\nu$	T_o °K	P_o kg/m^2	Remarks*
1	Inward	0.0011	58.5	26.2	66.67°	52.3°	1.113×10^5	288	10500	
2	Inward	0.0011	87.9	39.3	72.27°	33.7°	1.175×10^5	288	10540	
3	Inward	0.0012	87.3	39.3	74.25°	37.7°	1.208×10^5	288	10600	
4	Inward	0.0012	116.6	52.0	79.77°	10.5°	1.219×10^5	288	10670	
5	Inward	0.0012	145.6	65.4	81.28°	6.3°	1.275×10^5	288	10750	
6	Outward	0.00343	589.25	261.8	44.5°	81.2°	1.370×10^5	288	10330	

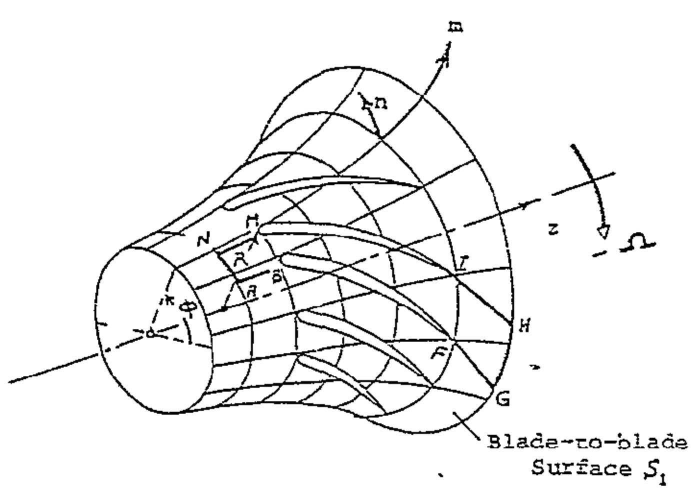
* In all cases studied, the upstream boundary AN of Fig. 3 is located at radius ratio r/r_{tip} of 1.35, while the downstream boundary HG is located at radius ratio r/r_{tip} of 0.254.

† W_{min} ≡ Meridional Velocity at Inlet.

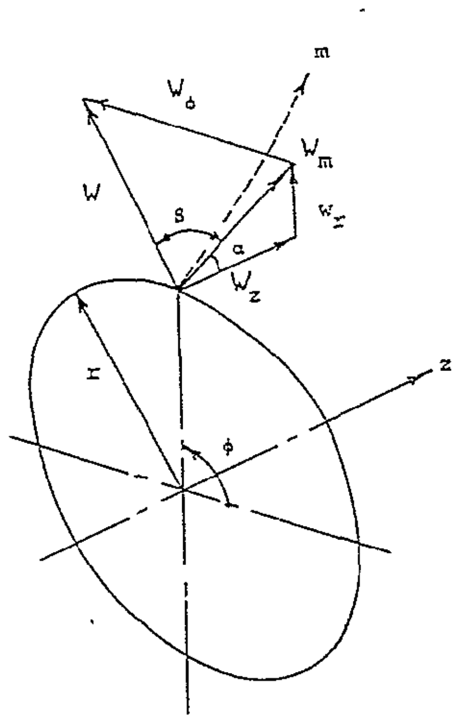
References

1. Thompson, J.F., Thames, F.C. and Mastin, C.W., NACA CR-2729, (1977).

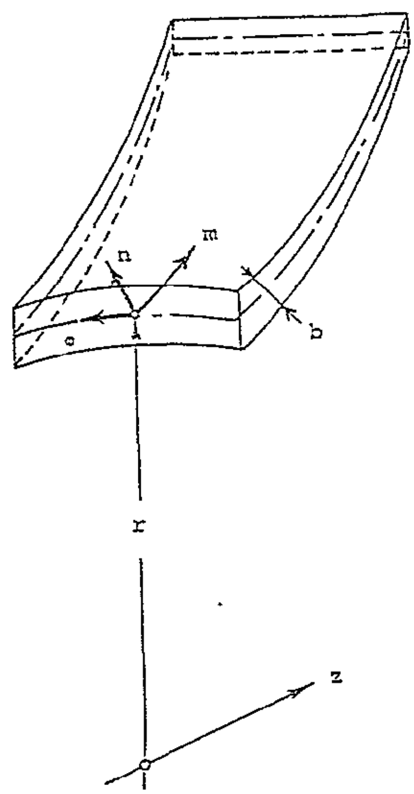
2. Watanabe, I., Ariga, I., and Fujie, K., "A Study of the Flow Patterns in an Impeller Channel of a Radial Turbine," Transactions of the ASME, Journal of Engineering for Power, October 1967, p. 467.



a) BLADE ROW INTERSECTION WITH A STREAM SURFACE.



b) COORDINATE SYSTEM AND VELOCITY COMPONENTS.



c) DETAILS OF STREAM SURFACE COORDINATE SYSTEM WITH FINITE THICKNESS SHEET.

FIG. 1. BLADE TO BLADE STREAM SURFACE S_1.

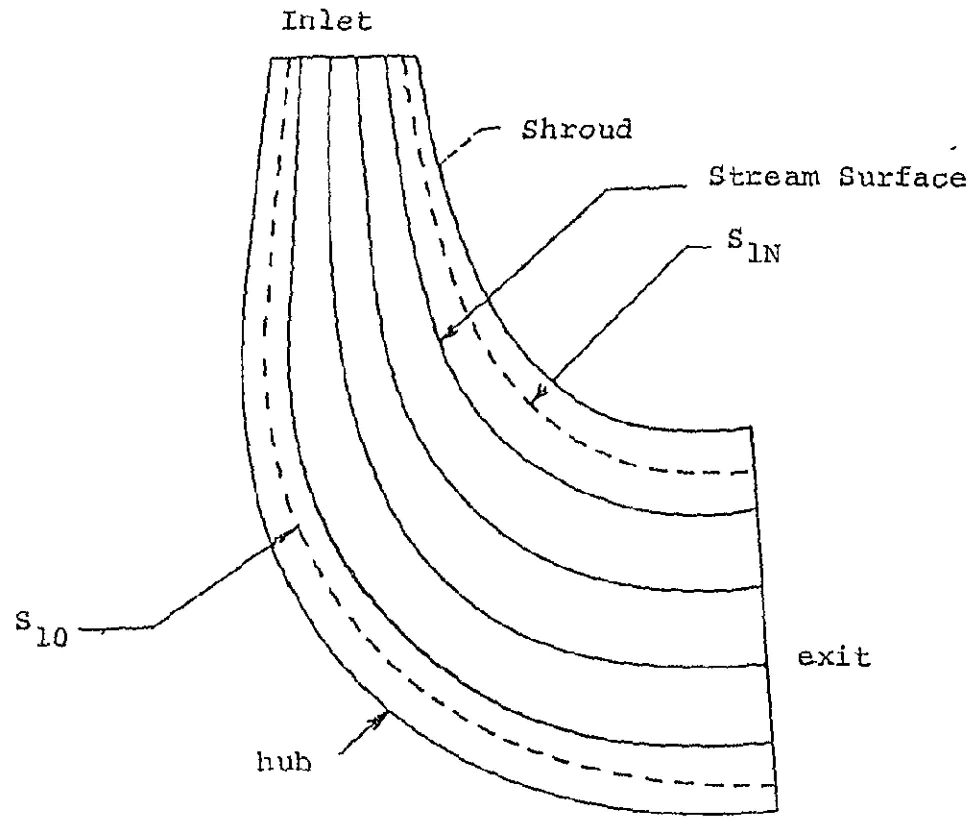


FIG. 2 . STREAM SURFACES CONTOURS, OBTAINED FROM A MERIDIONAL FLOW ANALYSIS, FOR A MIXED FLOW MACHINE.

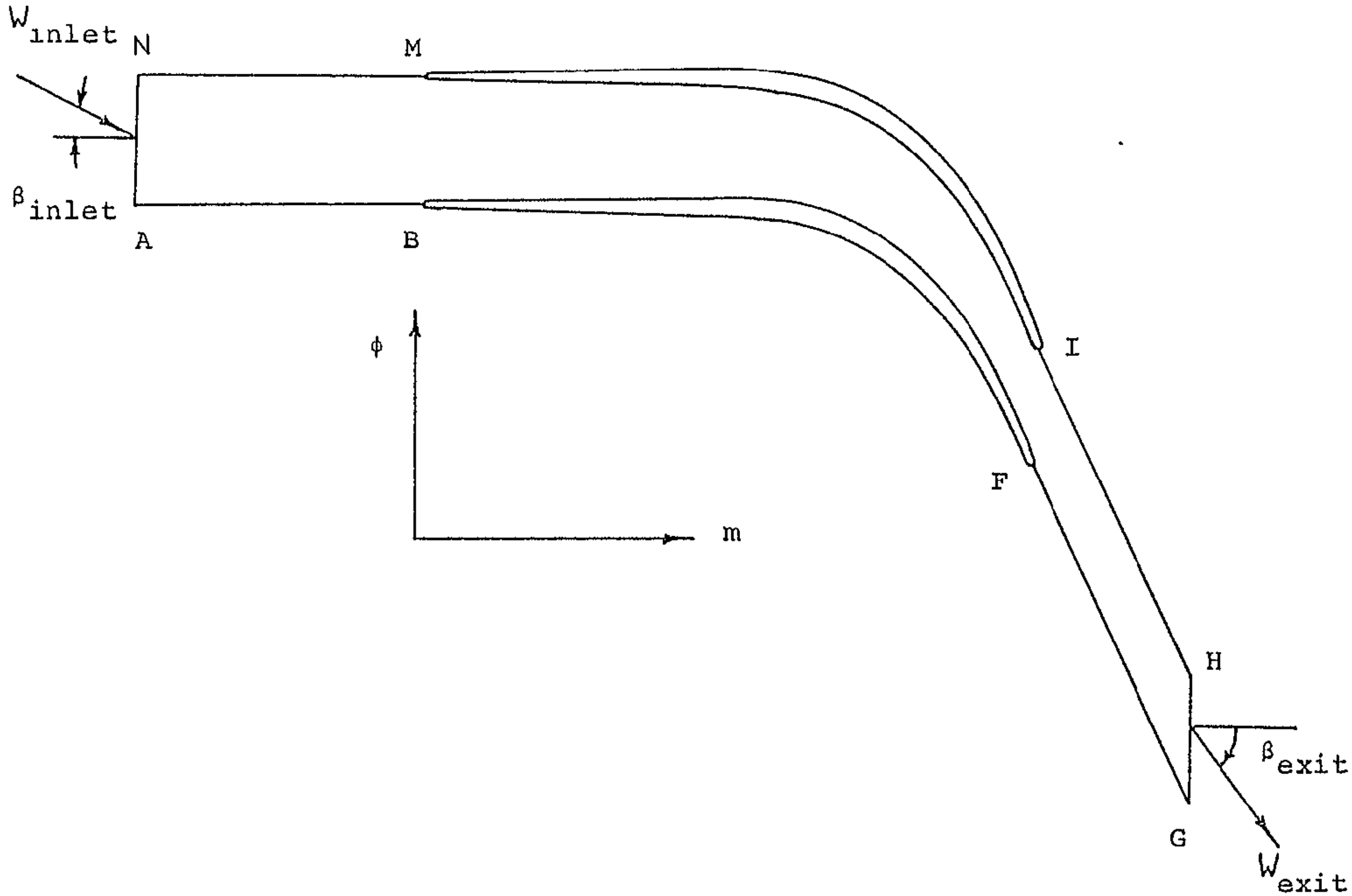
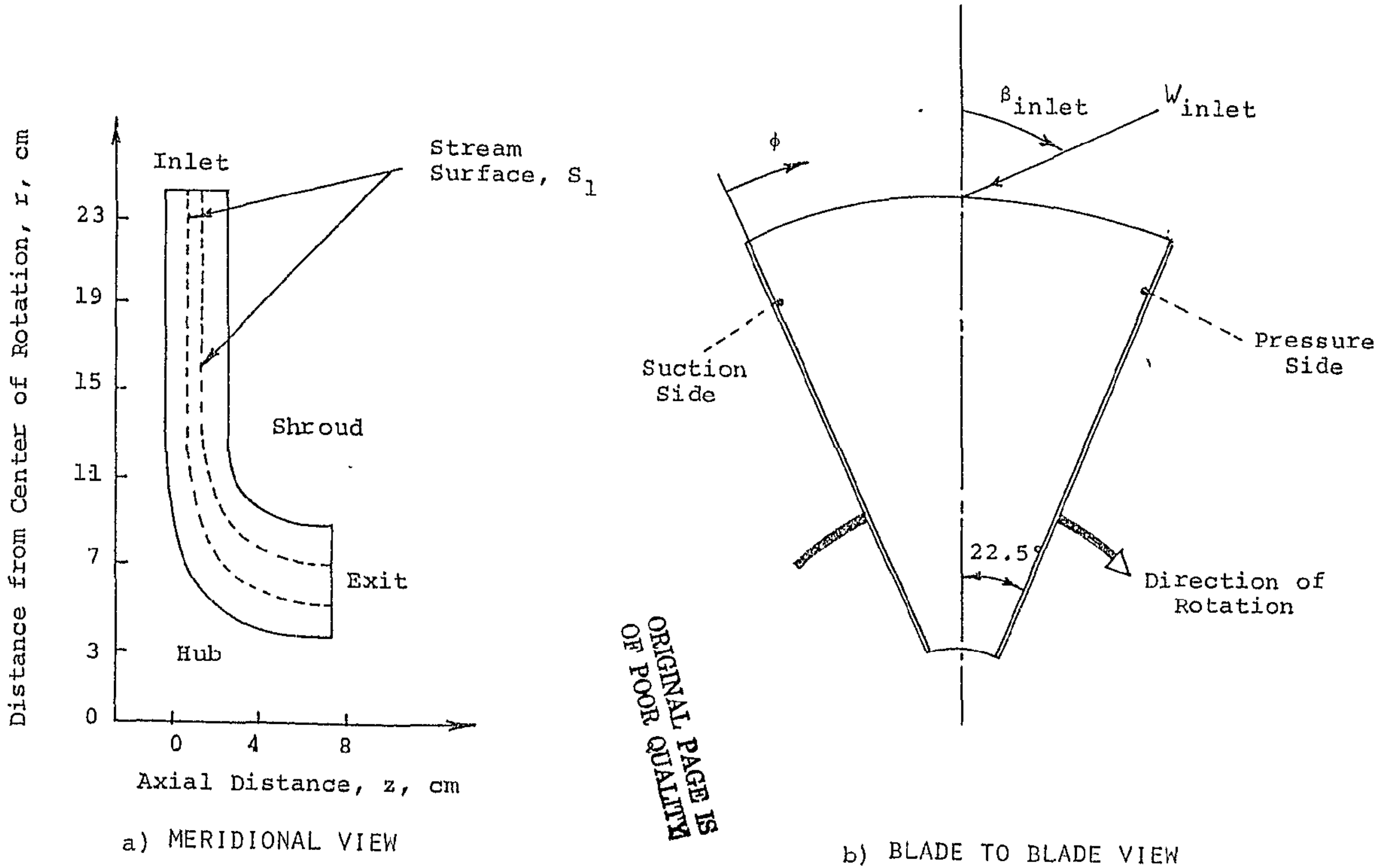


FIG. 3 . COMPUTATIONAL DOMAIN IN THE PHYSICAL SPACE.



ORIGINAL PAGE IS
OF POOR QUALITY

FIG.4 . HUB-SHROUD PROFILE WITH THE STREAM SURFACE S_1, USED FOR BLADE-TO-BLADE ANALYSIS.

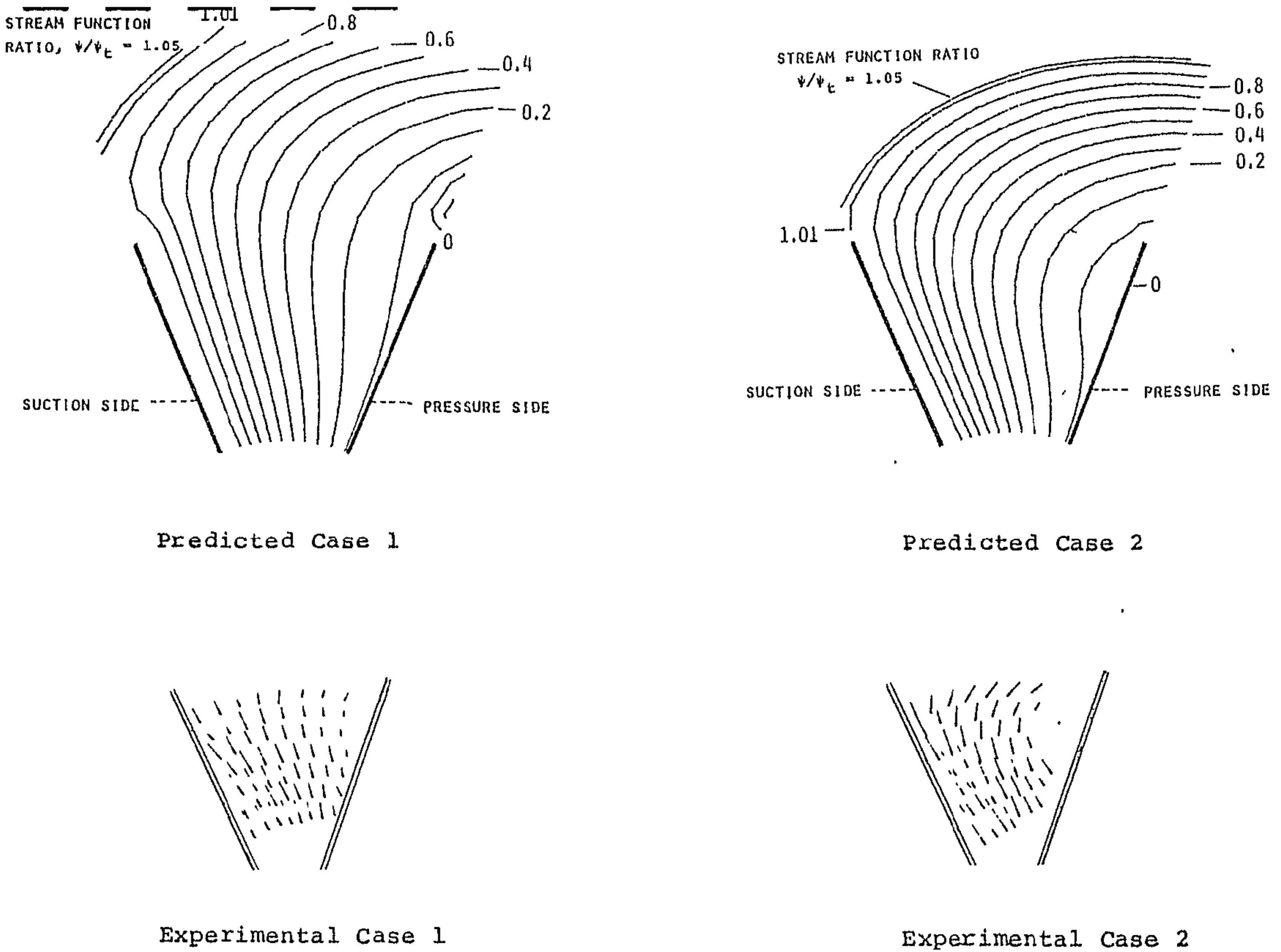


FIG.5a. RELATIVE STREAMLINES FOR FLOW THROUGH MIXED FLOW TURBINE.

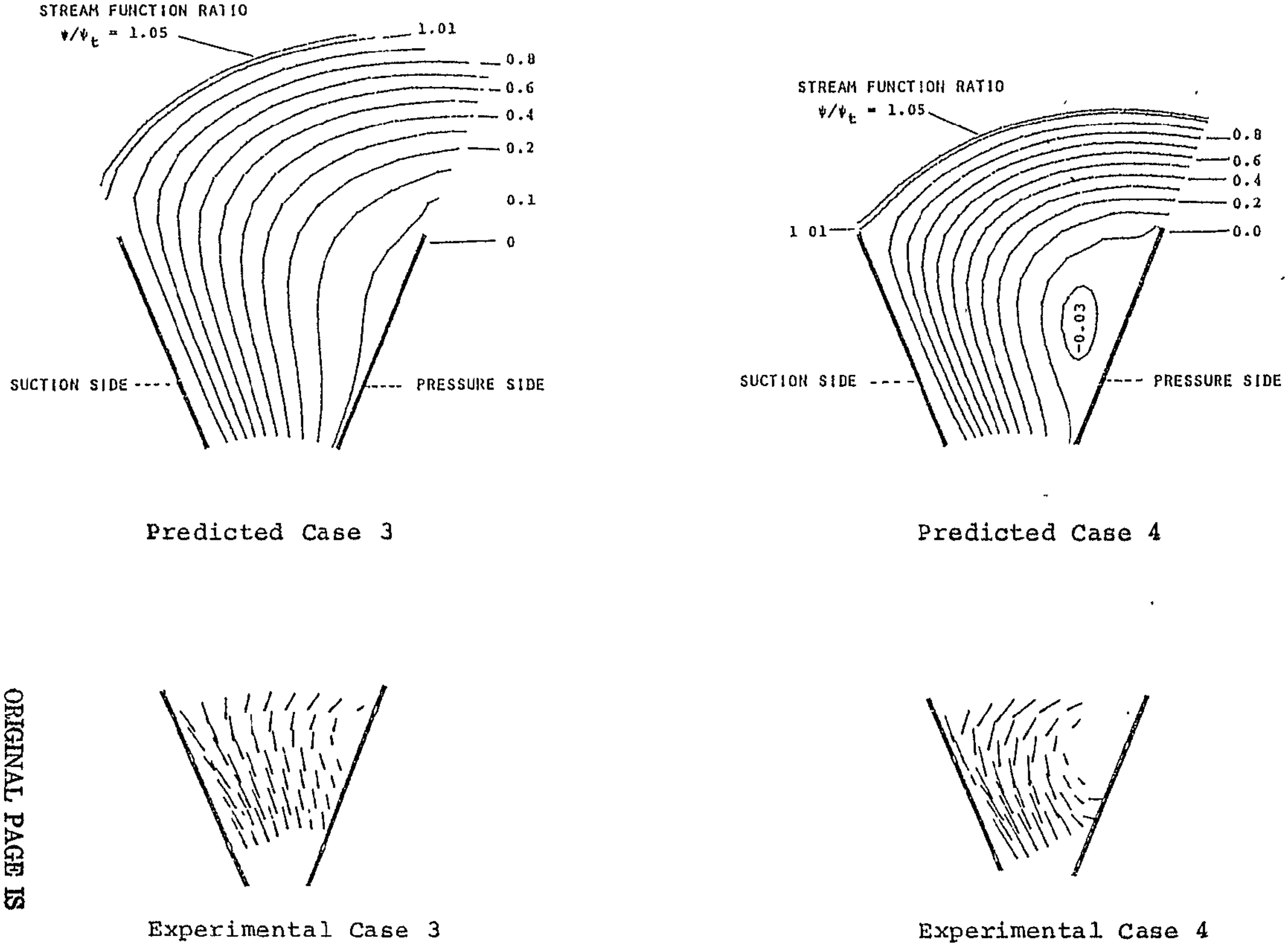
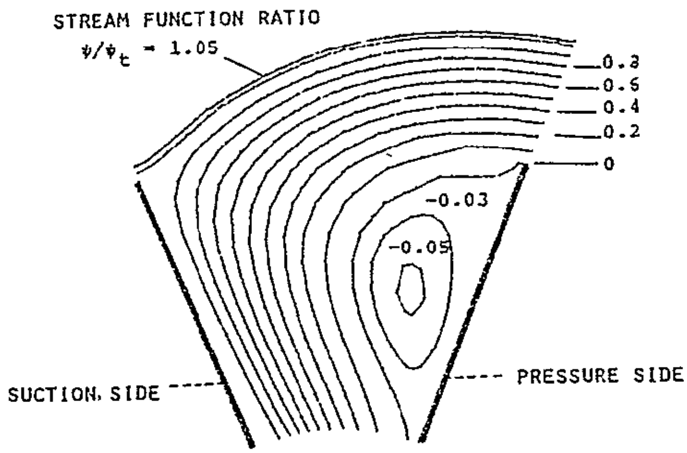
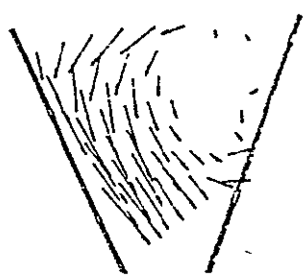


FIG.5b. RELATIVE STREAMLINES FOR FLOW THROUGH MIXED FLOW TURBINE.



PREDICTED CASE 5



EXPERIMENTAL CASE 5

FIG. 5c. RELATIVE STREAMLINES FOR FLOW THROUGH MIXED FLOW TURBINE.

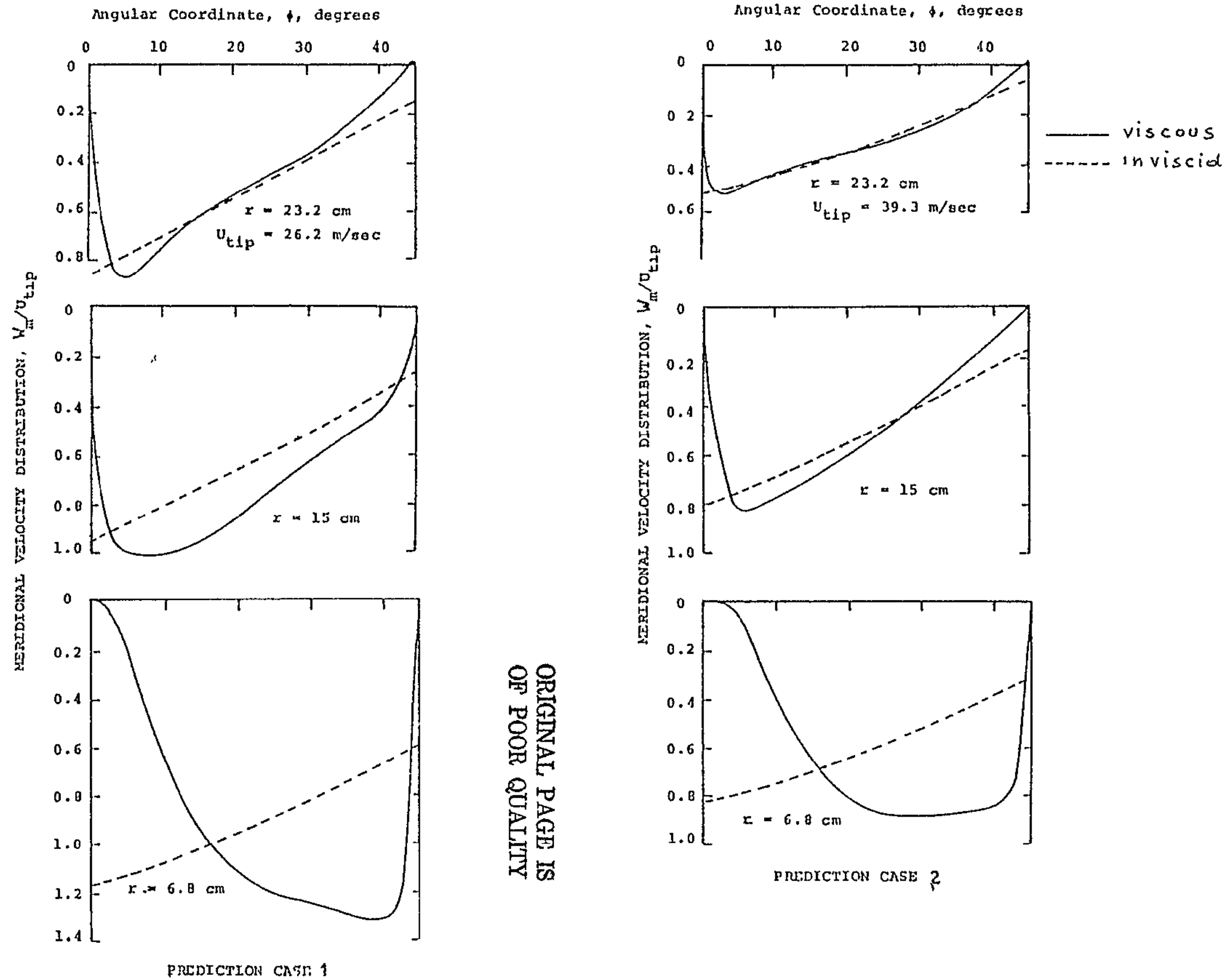


FIG.6 a NON-DIMENSIONAL VELOCITY DISTRIBUTION AT DIFFERLNT RADIAL LOCATIONS.

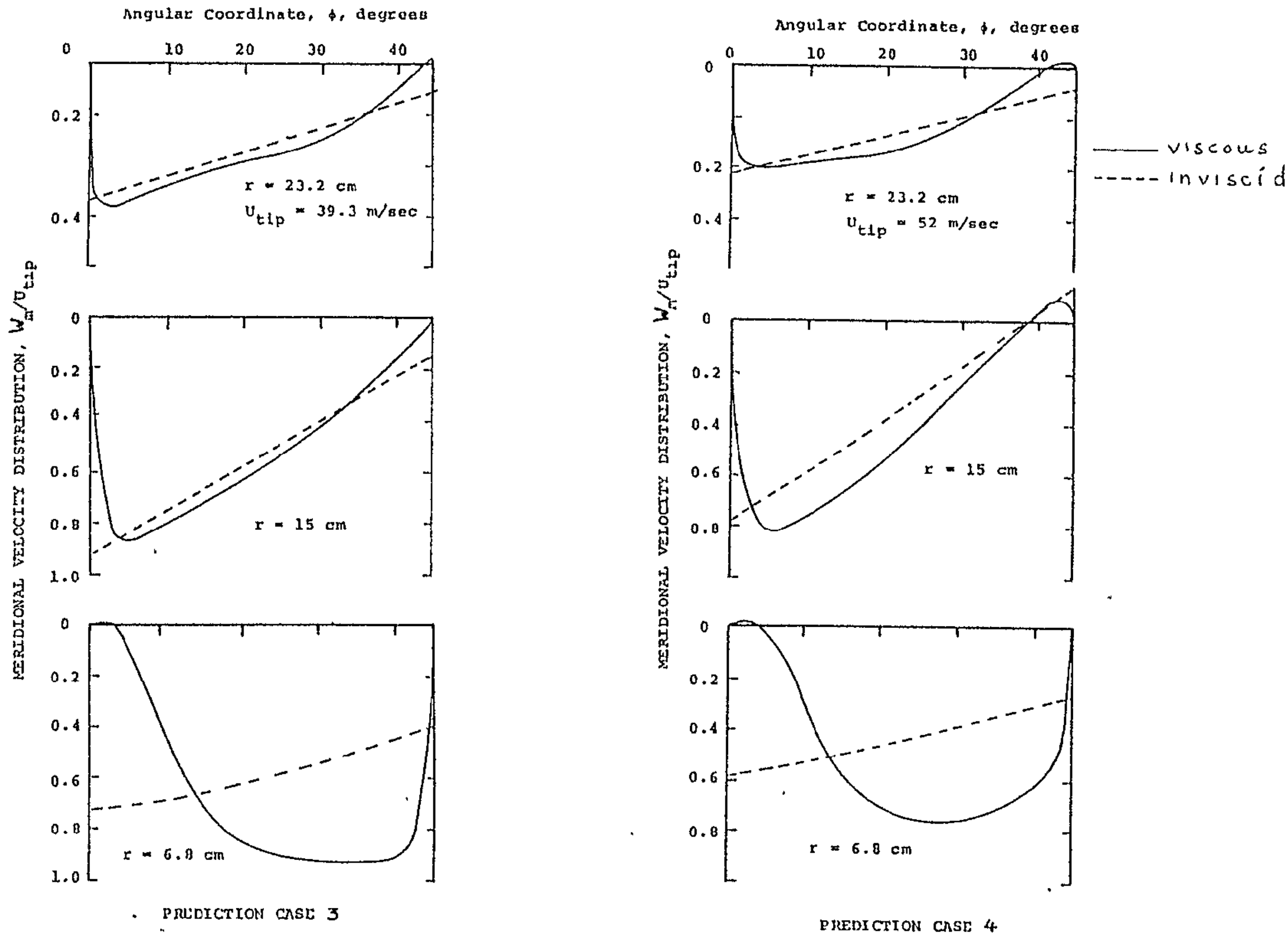


FIG. 6b. NON-DIMENSIONAL VELOCITY DISTRIBUTION AT DIFFERENT RADIAL LOCATIONS.

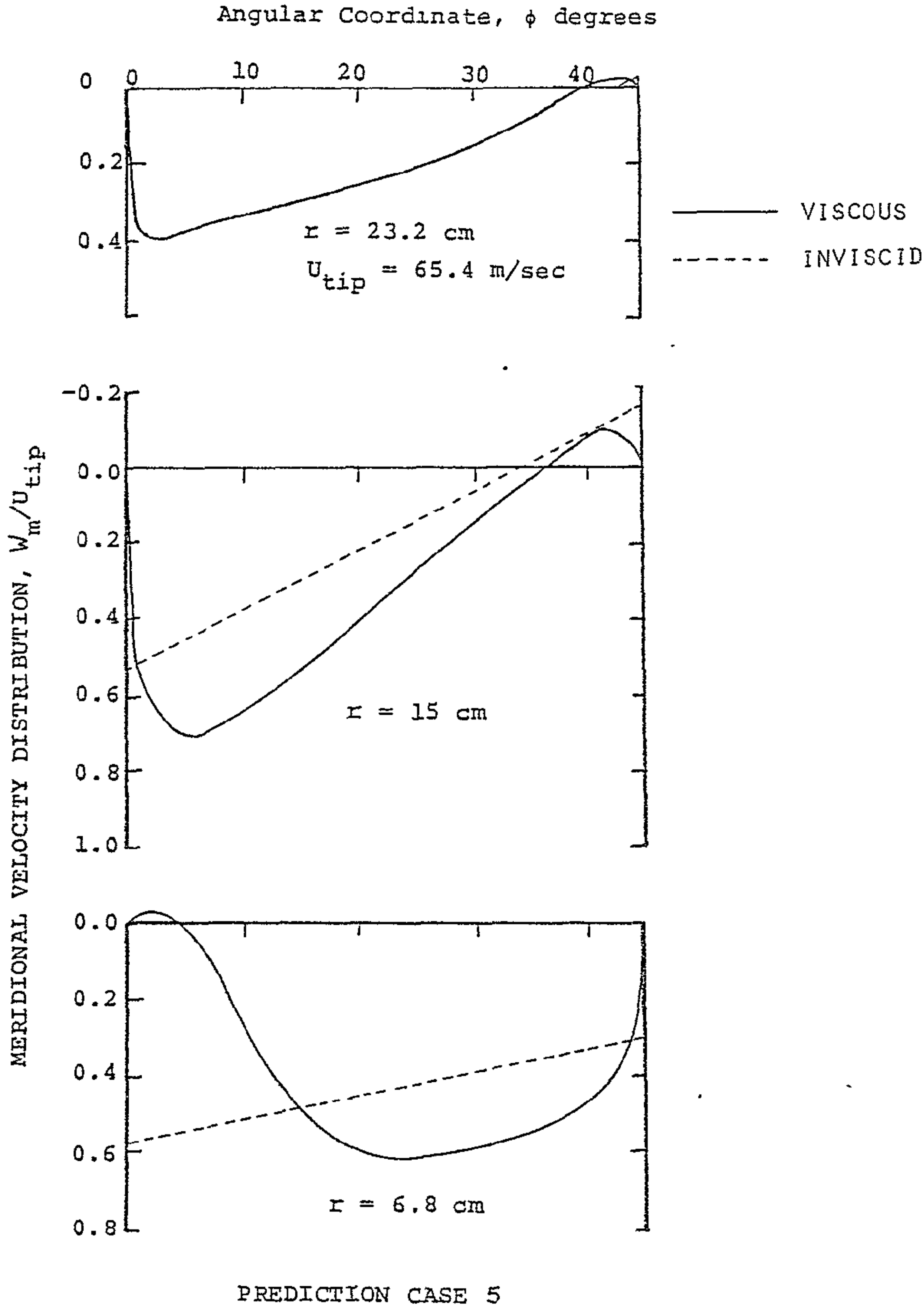
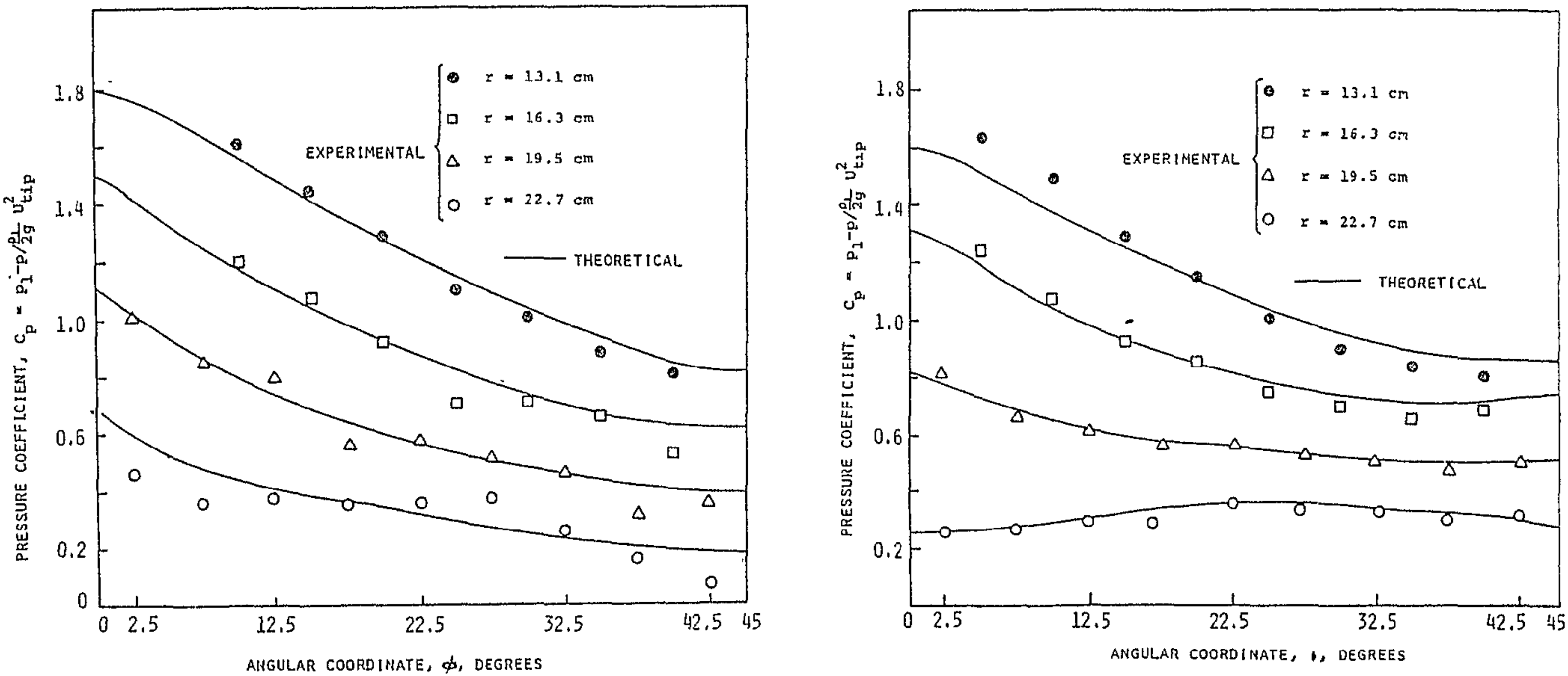


FIG. 6c. NONDIMENSIONAL VELOCITY DISTRIBUTION AT DIFFERENT RADIAL LOCATIONS.

ORIGINAL PAGE IS OF POOR QUALITY



CASE 1

CASE 2

FIG. 7α. COMPARISON OF PREDICTED STATIC PRESSURE DISTRIBUTION WITH EXPERIMENTAL DATA OF REFERENCE 2.

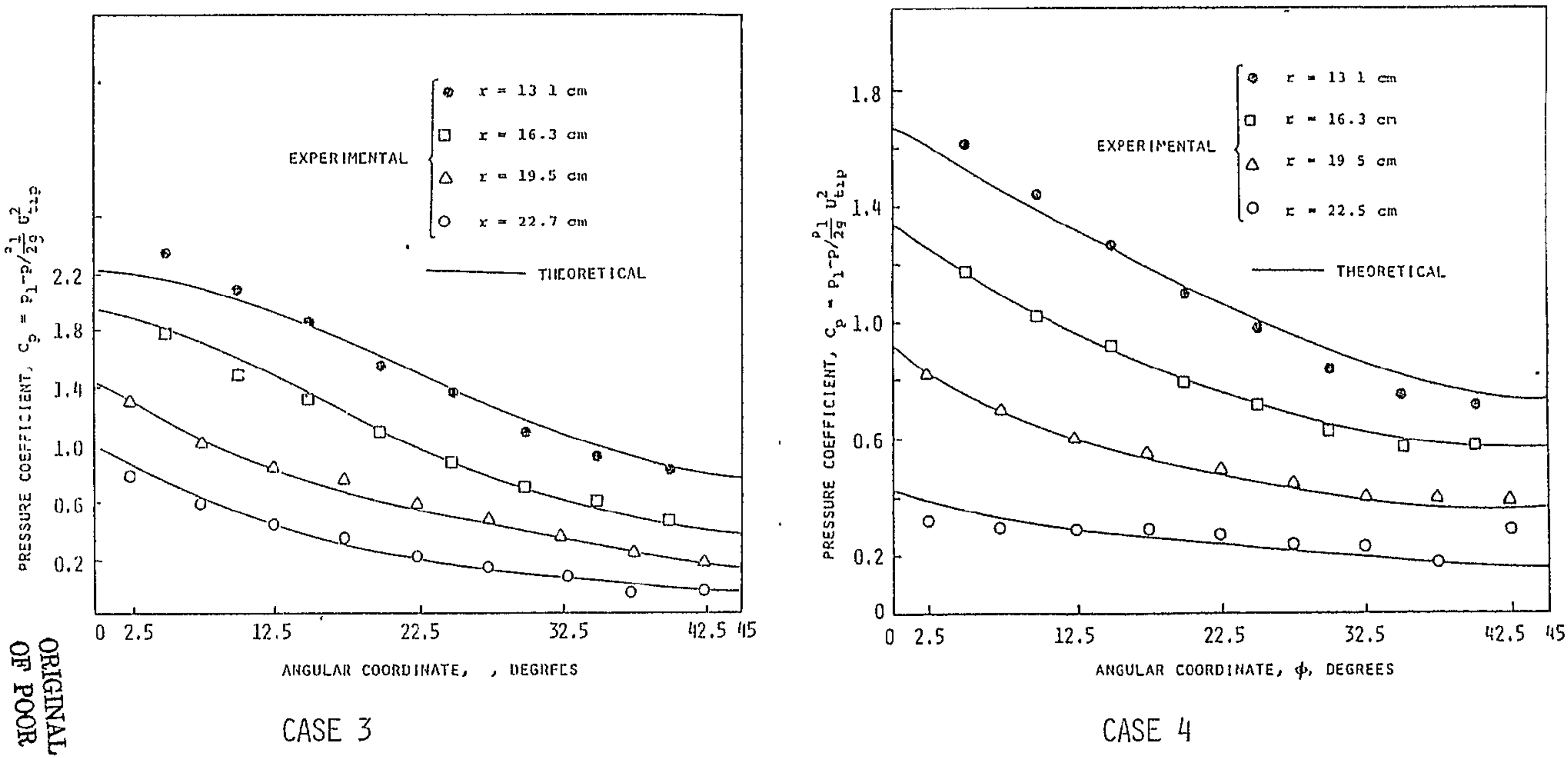
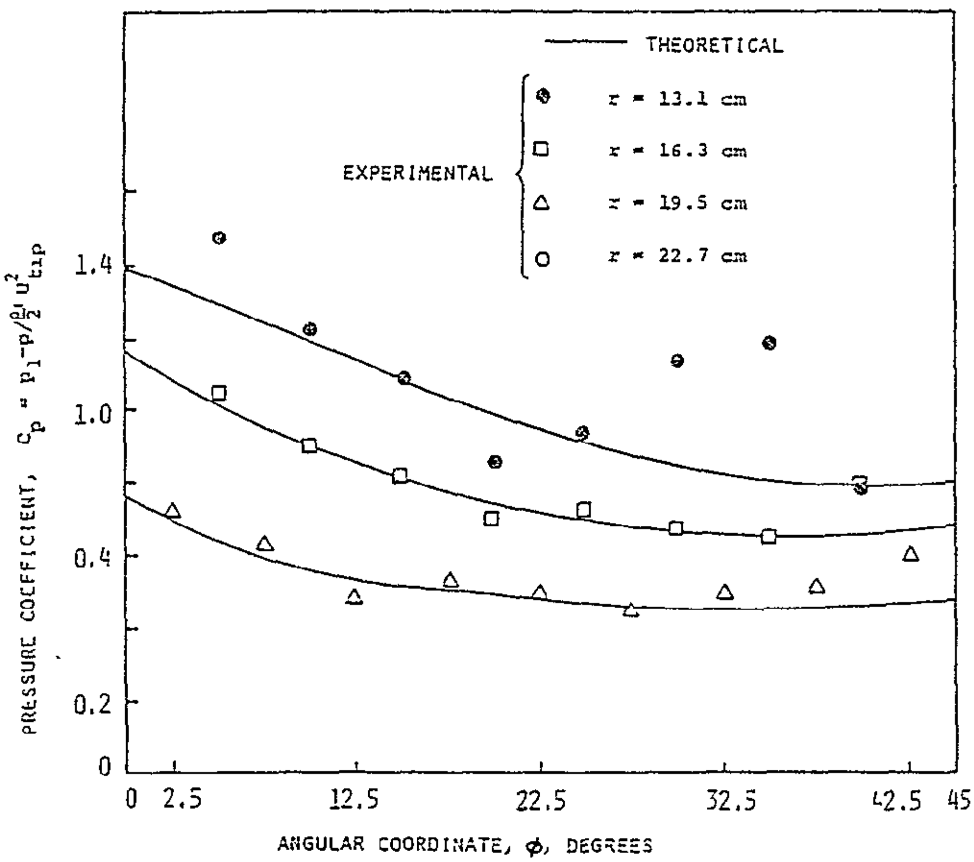
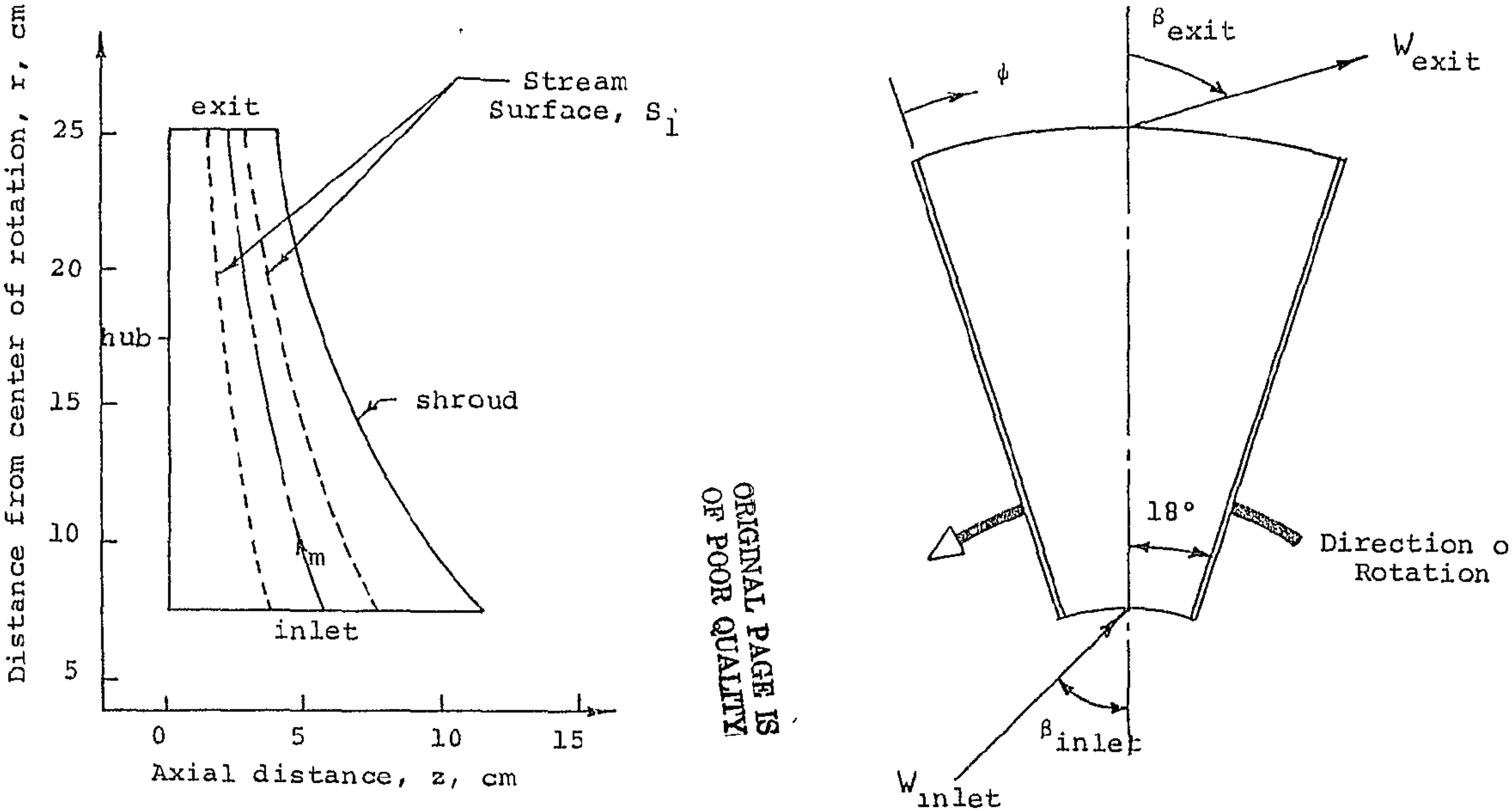


FIG. 7b. COMPARISON OF PREDICTED STATIC PRESSURE DISTRIBUTION WITH EXPERIMENTAL DATA OF REFERENCE 2 .



CASE 5

FIG. 7c. COMPARISON OF PREDICTED STATIC PRESSURE DISTRIBUTION WITH EXPERIMENTAL DATA OF REFERENCE 2.



a) MERIDIONAL VIEW

b) BLADE-TO-BLADE VIEW

FIG. 8 . HUB-SHROUD PROFILE WITH THE STREAM SURFACE, S_1, USED FOR BLADE-TO-BLADE ANALYSIS (COMPRESSOR CASE).

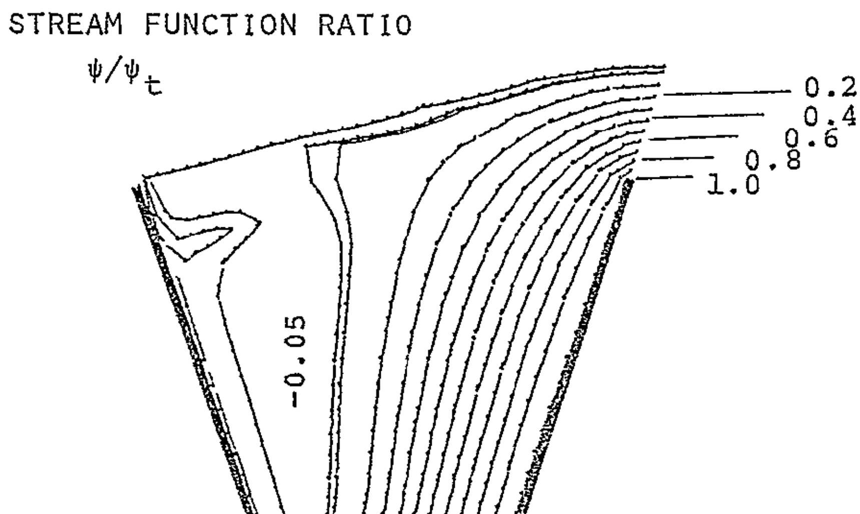


FIG. 9. RELATIVE STREAMLINES FOR FLOW THROUGH RADIAL COMPRESSOR.

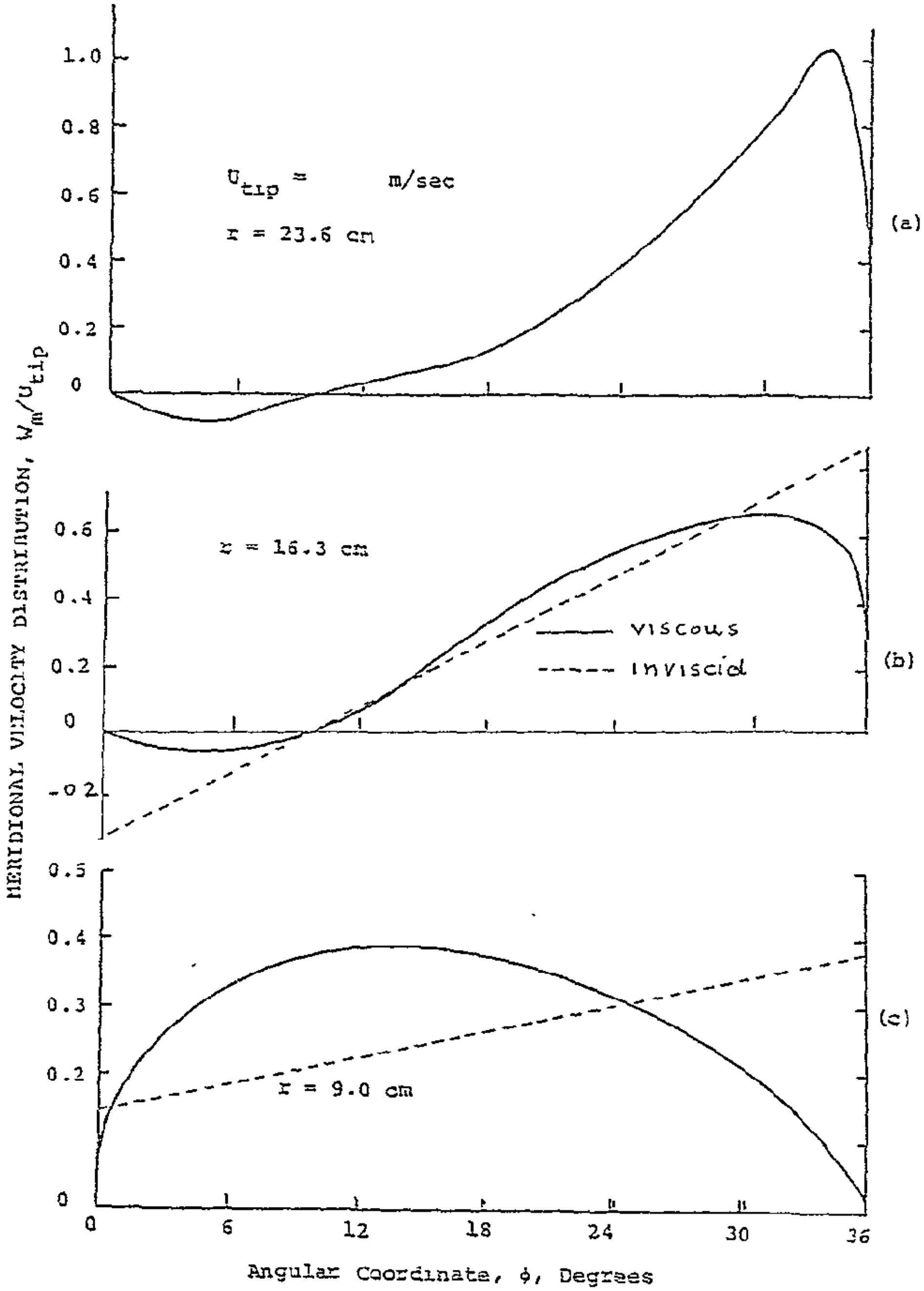


FIG. 10. VELOCITY PROFILES ACROSS THE COMPRESSOR PASSAGES AT DIFFERENT RADIAL LOCATIONS.