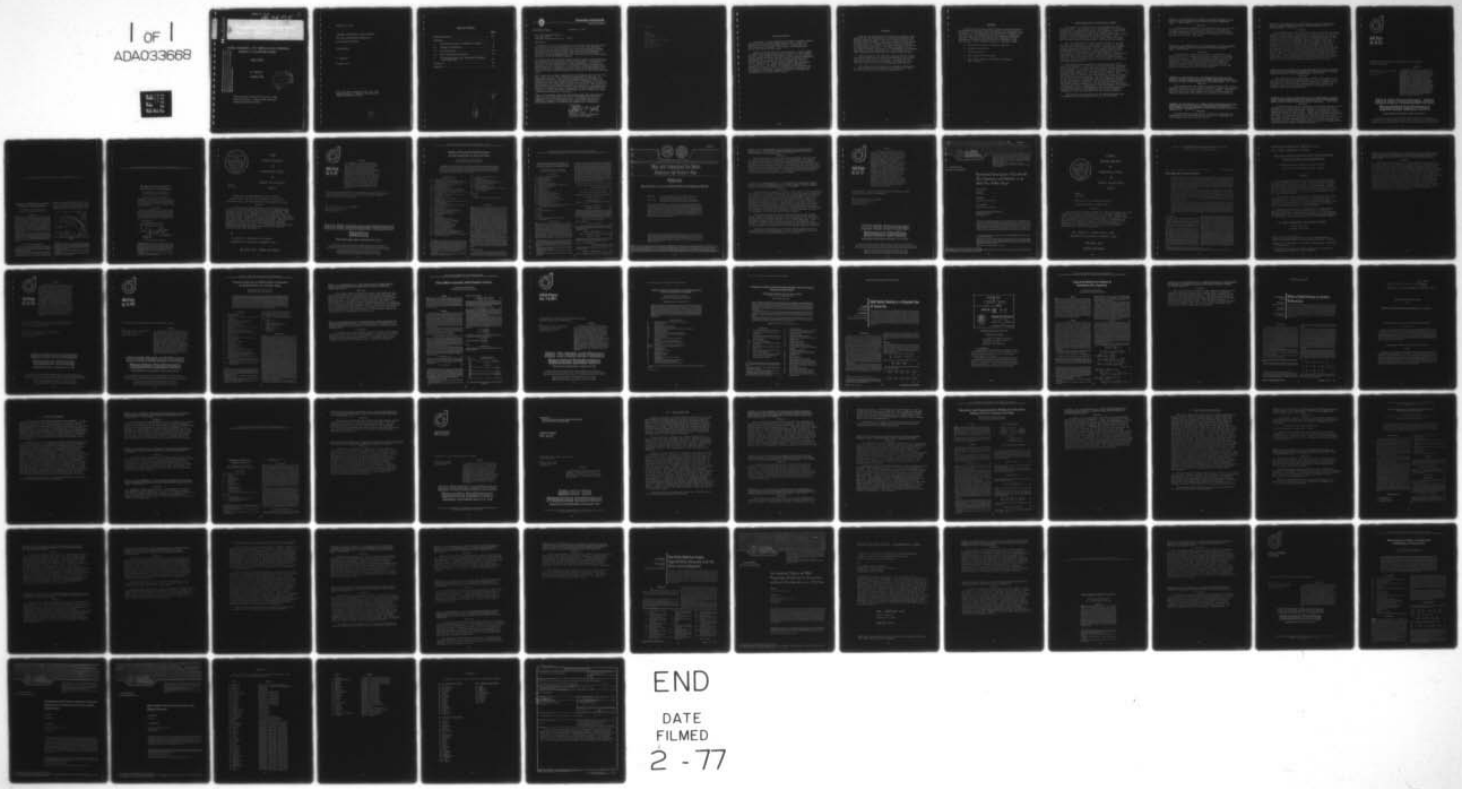
AD-A033 668 CINCINNATI UNIV OHIO DEPT OF AEROSPACE ENGINEERING F/G 21/5
INTERNAL AERODYNAMICS, HEAT TRANSFER AND HIGH TEMPERATURE MATER--ETC(U)
OCT 76 W TABAKOFF DAAG29-69-C-0016

UNCLASSIFIED 76-50 ARO-T-4.78-E NL

1 OF 1
ADA033668



END

DATE
FILMED
2 - 77

AD A033668

REPORT NO. 76-50

ARO. F-4. 78-E FG

INTERNAL AERODYNAMICS, HEAT TRANSFER AND HIGH TEMPERATURE MATERIALS IN AIR-BREATHING ENGINES

FINAL REPORT

W. TABAKOFF

OCTOBER 1976

This document has been approved for public release and sale; its distribution is unlimited. The findings in this report are not to be construed as an official Department of the Army position, unless so designated by other authorized documents.

See 1473 in back

DDC RECEIVED DEC 20 1976 C

This work was sponsored by the U.S. Army Research Office - Durham under Contract Number DAAG-29-69-C-0016.

REPORT NO. 76-50

INTERNAL AERODYNAMICS, HEAT TRANSFER AND HIGH TEMPERATURE MATERIALS IN AIR-BREATHING ENGINES

FINAL REPORT

W. Tabakoff

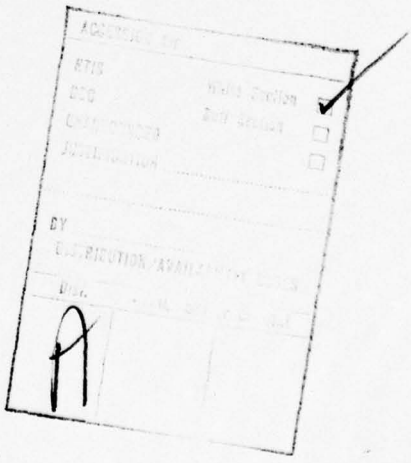
October 1976

This work was sponsored by the U.S. Army Research Office - Durham under Contract Number DAAG-29-69-C-0016.

DISTRIBUTION STATEMENT A
Approved for public release;
Distribution Unlimited

TABLE OF CONTENTS

	Page
ACKNOWLEDGEMENTS	iii
ABSTRACT	1
I. PARTICULATE FLOWS IN PROPULSION SYSTEMS	2
II. CASCADE AERODYNAMICS	34
III. JET MIXING FLOW	40
IV. HIGH TEMPERATURE MATERIALS	45
V. GAS TURBINE BLADES AND COMBUSTION CHAMBERS HEAT TRANSFER	50
APPENDIX A	64
APPENDIX B	66



University of Cincinnati

Cincinnati, Ohio 45221

OFFICE OF UNIVERSITY DEAN FOR
GRADUATE EDUCATION & RESEARCH

October 15, 1976

U.S. Army Research Office
P.O. Box 12211
Research Triangle Park, N.C. 27709

Gentlemen:

I would like to take this opportunity to express for the University of Cincinnati our deep appreciation to the U.S. Army Research Office for the excellent cooperation which was developed between our personnel and the scientific and contracting representatives of the U.S. Army Research Office under the research contract entitled "Internal Aerodynamics, Heat Transfer and High Temperature Materials in Air-Breathing Engines."

The funds provided under this award greatly assisted the University in developing our propulsion research laboratory. In addition to the direct financial assistance, the willingness of the Army to assist us in the acquisition of government surplus equipment has also enabled us to enhance our experimental propulsion capabilities. Perhaps even more importantly, it has given us the opportunity to train additional graduate students for careers in this most important area.

We believe that we have accomplished the original goal of this project, namely, to create a center of excellence in the area of propulsion research. The work performed under this project has apparently found significant application in industry. It is our sincere hope that the results of this work will continue to provide assistance to the researchers working in the field. The results of this award have also been noted not only within our own country but beyond our borders as well. Scientists from five continents have visited our laboratory during the period of this research to learn firsthand more details of our work.

Again, my sincerest thanks for the opportunity that you have provided to the University of Cincinnati to enhance our research capabilities in the propulsion area. This award has been an excellent example of the beneficial relationship which can be established between the Federal Government and a University.

Cordially,

Albert C. Yates
University Dean for Graduate
Education and Research

Key Words:

Engines
particles
aerodynamics
jets
high temperature materials
turbine blades
gas turbines
combustion chambers
heat transfer

ACKNOWLEDGEMENTS

The author is very grateful and wishes to extend sincere appreciation to Director James Murray and Dr. Robert R. Singleton of the Army Research Office - Durham, N.C. for their continued interest, encouragement and suggestions during the course of this work.

As with any experimental investigation, a large number of people have given their time and thoughts to the solution of many of the problems which have arisen. In this respect sincere appreciation is extended to the Propulsion Laboratory personnel; Mr. J. Cupito was of invaluable assistance in design and construction of the test facilities; and to Mr. F. Busher and D. Reedy, special thanks for maintaining the facilities. The author also wishes to express his gratitude to Mrs. K. Weast, J. Donnelly and D. Bollmer for secretarial assistance and accomplishing the enormous task of typing the technical reports and papers, and preparing manuscripts for journal publications.

FOREWORD

This is the final report covering the technical work performed by the Department of Aerospace Engineering and Applied Mechanics under Contract No. DAAG-69-C-0016 (U.S. Research Office Durham). The contract was initiated under Project Themis No. DAHC04-69-C-0016, entitled "Internal Aerodynamics, Heat Transfer and High Temperature Materials in Air-Breathing Engines." The work was supervised by Dr. Widen Tabakoff, Professor of the Department of Aerospace Engineering and Applied Mechanics. The report covers work conducted from September 20, 1968 to June 30, 1976.

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 are given with the first page of the corresponding publication.

The scientific and technical personnel participating in this research project are listed in Appendix A, and the students obtaining degrees in Appendix B.

ABSTRACT

A brief review of the Project Themis Research Program in Air Breathing Propulsion at the University of Cincinnati is presented. The review covers the period from September 1968 to June 1976. The research presented herein is concerned with improving the performance of air-breathing propulsion systems. Theoretical and experimental studies of basic phenomena upon which the performance is dependent, were carried out. These studies covered the following areas:

1. Particulate Flows in Propulsion Systems
2. Cascade Aerodynamics
3. Jet Mixing Flow
4. High Temperature Materials
5. Gas Turbine Blades and Combustion Chambers Heat Transfer.

I. PARTICULATE FLOWS IN PROPULSION SYSTEMS

The study of gas particle flow in turbomachines is of great interest. As a result of the development of high energy propellants for propulsion systems, the products of combustion may partially consist of finely divided particles. For example, solid propellant rockets are sometimes used as starting devices for propulsion systems employing turbines. Air-breathing engines operating in desert areas, where the inlet air flow may contain sand particles, provide another example.

Most of these types of flows involve changes in the gas velocity and temperature. Gas particle interaction, through viscous drag and heat transfer produces corresponding changes in the properties of particles. These particles are swept through the turbine by the gas flow, lagging behind the gas in velocity and temperature. Both these effects cause a deterioration in the performance of turbomachines. Furthermore, these particles may cause blade erosion, and consequently, failure. Even if the erosion is not severe enough to cause rupture, changes in blade geometry are of concern, due to their influence on performance.

The study of the trajectories and velocities of solid particles suspended in a fluid flow through cascades and blade rows is necessary to investigate the erosion damage sustained by the blades. In general, the trajectories and velocities of the particles depend upon the shape of the cascade, the particle and flow inlet conditions, the flow angle of attack, the particle material density, the mean diameter and the location of the initial particle collision. The particles are more likely to follow the fluid streamlines when their material density is of the same order of magnitude as that of the fluid and when their mean diameter is small. The pressure distribution on the blade surfaces changes when solid particles are introduced into the gas stream. Consequently, the performance of the turbine or compressor in an air-breathing engine will change as well. Knowing the effect of the particles on the blade pressure distribution will be very useful for the design purposes.

The research work performed in the above mentioned area is reported in the following abstracts and references.

Hamed, A. and Tabakoff, W., "Momentum Integral Theory for Gas Particle Flow," Project Themis Report No. 69-3, September 1969. (AD 696442)

Abstract

A theoretical treatment of the boundary layer for the two-dimensional gas particle flow is presented. The momentum integral equations of the boundary layer with incompressible gas phase are derived, and a numerical calculation is carried out for the case of a flat plate.

Hussein, M. and Tabakoff, W., "The Properties of a Gas Particle Flow in Cascade," Project Themis Report 69-4, September 1969. (AD 697164)

Abstract

An approximate method is developed for calculating the flow properties of gas-particle mixture flowing over turbine blades in a cascade. Momentum and heat transfer between the gas and particles, as well as compressibility effects were considered. An investigation was made of the effect of particle concentration, mean diameter, particle material density, particle inlet velocity and temperature on the gas particle flow properties on a turbine blade.

Tabakoff, W. and Hussein, M., "An Experimental Study of the Effect of Solid Particles on the Pressure at Blade Surfaces in Cascade," Project Themis Report No. 70-8, March 1970. (AD 703896)

Abstract

Data representing the effect of solid particle flow on the pressure distribution over blade surfaces in cascade are presented. Various inlet mass flow concentrations were tested. Different particle sizes were used, namely 50, 300 and 1000 microns in order to determine their effect on the pressure distribution.

Tabakoff, W. and Hussein, M., "Experimental Investigation of the Trajectories and Velocities of Solid Particles Entrained by Fluid Flows in Cascade Nozzles," Project Themis Report No. 70-12, August 1970. (AD 711121)

Abstract

An experimental investigation was made to determine the trajectories and velocities of solid particles suspended in a fluid flowing through a cascade nozzle.

Hamed, A. and Tabakoff, W., "The Boundary Layer of Particulate Flow in Cascade Nozzles," Project Themis Report No. 70-16, November 1970. (AD 715721)

Abstract

The boundary layer of gas-particle flows in field of changing pressures is analyzed. The analysis, which utilizes the momentum integral method, can be used to study the boundary layer of any particulate flow with incompressible gas-phase provided the flow properties outside the boundary layer are known. Numerical integration of the two momentum integral equations was carried out for the case of particulate flow in cascades. For this example, the results indicate that introduction of particles leads to an increase in the gas boundary layer thickness and that the gas boundary layer characteristics are usually more sensitive to particle concentration than any other particulate flow parameter. However, the particle boundary layer characteristics are more sensitive to mean particle diameter than to particle concentration. Other parameters such as inlet particle gas velocity ratio and particle solid material density have more pronounced effects on the particle boundary than on that of the gas.

Hussein, M. and Tabakoff, W., "Gas-Particle Suspension Properties on a Blade-to-Blade Stream Surface of a Cascade Nozzle," Project Themis Report No. 70-17, December 1970. (AD 715971)

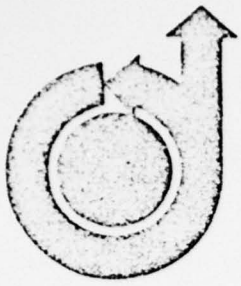
Abstract

The gas-particle flow properties, namely flow pressure, gas and particles velocities and temperatures were calculated theoretically in a blade to blade stream surface of a cascade nozzle. The effects of the particle concentration, mean diameter and material density on the flow properties at any point in the cascade nozzle were investigated.

Tabakoff, W., Hamed, A. and Hussein, M.F., "Experimental Investigation of Gas Particle Flow in a Simulated Axial Flow Compressor Stage," Project Themis Report No. 71-21, July 1971. (AD 728489)

Abstract

An experimental study of the effect of particle size and fluid velocity on the trajectories and velocities of solid particles entrained by a fluid, and passing through a simulated axial flow compressor stage, is carried out. Conclusions concerning areas of blade surface subjected to erosion damage due to particle collisions are drawn. Pressure measurements of the particulate gas flow on the blade surfaces were obtained for various values of particle concentration for two different positions of the second cascade row. A subsonic cascade wind tunnel was used for testing the particulate flow through the simulated compressor stage.



AIAA Paper
No. 70-712

PROPERTIES AND PARTICLE TRAJECTORIES OF GAS–PARTICLE FLOWS IN CASCADES

by
W. TABAKOFF and M.F. HUSSEIN
University of Cincinnati
Cincinnati, Ohio

Abstract

An approximate method for calculating the flow properties of gas-particle mixture flowing over blades in a cascade is developed. The momentum and heat transfer between the gas and particles as well as compressibility effects are taken into consideration. The effect of particle concentration, mean diameter, particle material density, particle inlet velocity and temperature on the gas particle flow properties are investigated. The flow pressure distribution on the blade surface and the particle velocities and trajectories in the nozzle are determined experimentally. The experiments are performed in a wind tunnel and in a water table. The solid particle sizes range from 5 to 1000 microns and the particle concentration range from 0.03 to 0.2.

AIAA 6th Propulsion Joint Specialist Conference

SAN DIEGO, CALIFORNIA/JUNE 15-19, 1970

First publication rights reserved by American Institute of Aeronautics and Astronautics. 1290 Avenue of the Americas, New York, N.Y. 10019. Abstracts may be published without permission if credit is given to author and to AIAA. (Price: AIAA Member $1.00. Nonmember $1.50)

Reprinted from JOURNAL OF AIRCRAFT, Vol. 8, No. 1, January, 1971, pp. 60-62
Copyright, 1971, by the American Institute of Aeronautics and Astronautics, and reprinted of the copyright owner

Trajectories of Particles Suspended in Fluid Flow Through Cascades

W. TABAKOFF* AND M. F. HUSSEIN†
University of Cincinnati, Cincinnati, Ohio

Introduction

THE study of the trajectories and velocities of solid particles suspended in a fluid flow passing through a cascade nozzle is of importance to the investigation of erosion damage sustained by the blades. In general, the trajectories and velocities of the particles depend upon the slope of the cascade nozzle, particle and flow inlet conditions, particle material density, mean diameter, angle of attack and initial place of collision. The particles are more likely to follow the fluid streamlines when their material density is of the same order of magnitude as that of the fluid and when their mean diameter is small.

In this experimental investigation, a cascade row of turbine blades was mounted in a cascade tunnel to produce the desired gas-particle flow. A high-speed camera was used to photograph the flow. The film analysis provided the data for the particle velocities and the particle paths through the cascade nozzle.

Test Facilities and Conditions

A special subsonic cascade wind tunnel, designed for gas-particle studies, was used for this series of tests. Reference 1 gives a detailed description of the tunnel and the airfoil used in the cascade nozzle.

A high speed motion picture camera with a maximum speed of 6000 frames-sec at 220-v input was used to photograph the particulate flow. Solid particles of 1000-μ mean diameter and 1.1-g/cm^3 density were used. The range of particle diameters available was limited by the fact that if particles of smaller diameters were used, it was impossible to determine their trajectories from a film analysis. A direct lighting technique was used in the experiments, and the

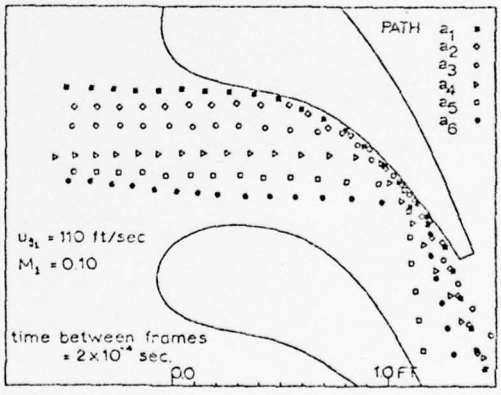


Fig. 1 Trajectories of particles entrained by gas flow (path a).

Presented as Paper 70-712 at the AIAA 6th Propulsion Joint Specialist Conference, San Diego, Calif., June 15-19, 1970; submitted July 13, 1970. This work was sponsored under Project Themis Contract Number DAHCO 4-69C-0016, U.S. Army Research Office—Durham.

* Professor, Department of Aerospace Engineering. Associate Fellow AIAA.

† Graduate Research Assistant, Department of Aerospace Engineering. Student Member AIAA.

Reprinted from JOURNAL OF AIRCRAFT, Vol. 8, No. 2, February, 1971, pp. 121-123
Copyright, 1971, by the American Institute of Aeronautics and Astronautics, and reprinted by permission of the copyright owner

Measurements of Particulated Gas Flow Pressure on Cascade Nozzles

W. TABAKOFF* AND M. F. HUSSEIN†
University of Cincinnati, Cincinnati, Ohio

Cascade Tunnel and Model Description

A SPECIAL subsonic cascade tunnel was built which incorporates a device for injection of solid particles (Fig. 1). The cascade dimensions and the pressure probe locations are shown in Fig. 2. Further test facility information may be obtained from Ref. 1.

Instrumentation

Manometer readings were recorded by camera after a steady state was reached. The primary mass flow was measured by an orifice meter located ahead of the settling chamber. The primary flow temperature was measured with a standard thermocouple located in the settling chamber. An electronic counter was employed to record the time of par-

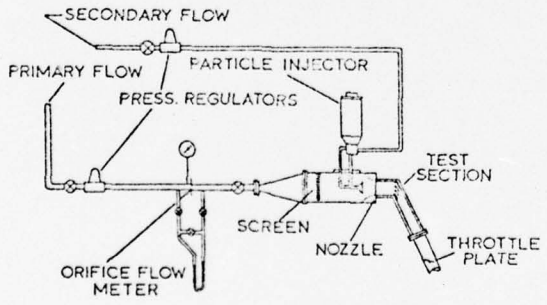


Fig. 1 Cascade tunnel schematic.

Presented as Paper 70-712 at the AIAA 6th Propulsion Joint Specialist Conference, San Diego, Calif., June 15–19, 1970; submitted July 13, 1970; revision received October 13, 1970. This work was sponsored under Project Themis Contract No. DAHC-04-69C-0016, U.S. Army Research Office—Durham.

* Professor, Department of Aerospace Engineering. Associate Fellow AIAA.

† Graduate Research Assistant, Department of Aerospace Engineering. Student Member AIAA.

TENTH

NATIONAL CONFERENCE

ON

ENVIRONMENTAL EFFECTS

ON

AIRCRAFT AND PROPULSION

SYSTEMS

PAPER
No. 71-5

SIMULATION OF ENVIRONMENTAL SOLID-PARTICLES TRAJECTORIES AND VELOCITIES THROUGH AN AXIAL FLOW COMPRESSOR STAGE, AND THE PRESSURE DISTRIBUTION ON BLADES

Abstract

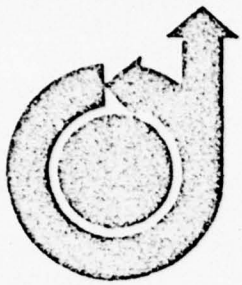
An experimental investigation of the trajectories and velocities of solid particles suspended in a fluid passing through an axial flow compressor stage was performed. Such an investigation is of importance to the study of erosion damage sustained by the blade. Two test facilities were used for this study: a subsonic cascade wind tunnel for compressible flow and a water table for incompressible flow. From the test technique it may be concluded that the present existing theoretical analysis for particle trajectories through a compressor stage is not valid. The wind tunnel test simulation is much better than the water table and may be used for predicting particle trajectories.

BY

W. TABAKOFF, A. HAMED AND M.F. HUSSEIN

UNIVERSITY OF CINCINNATI, CINCINNATI, OHIO

MAY 18-20, 1971 TRENTON, NEW JERSEY



AIAA Paper
No. 71-82

Abstract

A theoretical method was developed for predicting the pressure distribution over a blade in cascades for both incompressible and compressible flow with particles. Experimental results were obtained from a cascade wind tunnel equipped with a solid particle injecting system. Good agreement was noted between the theoretical and experimental pressure distribution for both incompressible and compressible flow conditions. This agreement indicates that the pressure distribution over a blade surface in cascade nozzles increases with the introduction of solid particles in the flow. The increase in pressure due to the particles gives rise to forces on the blades indicating that this information is of importance to the designer.

THE COMPARISON BETWEEN THEORETICAL AND EXPERIMENTAL PRESSURE DISTRIBUTION ON A BLADE IN CASCADE NOZZLE FOR A PARTICULATE GAS FLOW

by
W. TABAKOFF and M. F. HUSSEIN
University of Cincinnati
Cincinnati, Ohio

AIAA 9th Aerospace Sciences Meeting

NEW YORK, NEW YORK / JANUARY 25-27, 1971

First publication rights reserved by American Institute of Aeronautics and Astronautics. 1290 Avenue of the Americas, New York, N. Y. 10019. Abstracts may be published without permission if credit is given to author and to AIAA. (Price: AIAA Member $1.50. Nonmember $2.00).

Note: This paper available at AIAA New York office for six months; thereafter, photoprint copies are available at photocopy prices from Technical Information Service, 750 3rd Ave., New York, N. Y. 10017

Reprinted from AIAA JOURNAL, Vol. 9, No. 8, August 1971, pp. 1514-1519
Copyright, 1971, by the American Institute of Aeronautics and Astronautics, and reprinted by permission of the copyright owner

Effect of Suspended Solid Particles on the Properties in Cascade Flow

W. TABAKOFF* AND M. F. HUSSEIN†
University of Cincinnati, Cincinnati, Ohio

An approximate method for calculating the flow properties of gas-particle mixture flowing over blades in a cascade is developed. The momentum and heat transfer between the gas and particles as well as compressibility effects are taken into consideration. The effect of particle concentration, mean diameter, particle material density, particle inlet velocity and temperature on the gas particle flow properties are investigated.

Nomenclature

A = cross-sectional area of stream tube (ft²)
A' = nondimensional cross-sectional area of stream tube
α = the ratio of the mass flow rate of particles to the total mass flow rate of the gas and particle mixture
A_p = mean particle projected area (ft²)
A_s = mean particle surface area (ft²)
C_d = particles drag coefficient
C_{pp} = specific heat of solid particles material (Btu/lb°R)
C_{pg} = gas specific heat at constant pressure (Btu/lb°R)
δs = distance along airfoil contour between two successive points (ft)
dt = time in which the particle travels a distance δ (sec)
d_p = particle mean diameter (ft)
F = coefficient
f = correction factor to Stokes drag formula
g = gravity constant
h = coefficient of heat transfer between the particles and the gas (Btu/hr ft²°R)
J = mechanical equivalent of heat
k_g = coefficient of conductivity for the gas (Btu/hr ft°R)
m = mass of one particle (lb)
Nu = Nusselt number
P = gas particle suspension pressure (lb/ft²)
P' = nondimensional gas particle suspension pressure
p = pressure of gas-only flow (lb/ft²)
p' = nondimensional gas-only flow pressure
Pr = Prandtl number
R_g = gas constant
Re = Reynolds number
ρ = the gas-only flow density (lb/ft³)
ρ' = the gas-only flow nondimensional density
ρ_g = the gas density in gas particle flow (lb/ft³)
ρ_p = the particles density (lb/ft³)
ρ_p' = the nondimensional particles density
$\bar{\rho}_p$ = solid particle material density (lb/ft³)
s = distance along airfoil contour (ft)
T_0 = total temperature of gas (°R)
T = the gas-only flow temperature (°R)
T' = the nondimensional temperature of gas-only flow
T_g = the gas temperature in gas particle flow (°R)
T_g' = the nondimensional gas temperature in gas particle flow
T_p = particle temperature (°R)
T_p' = nondimensional particle temperature
t = time (sec)
u = gas-only flow velocity, fps
u' = gas-only flow nondimensional velocity
u_g' = nondimensional gas velocity in gas particle flow
u_g = gas velocity in gas particle flow, fps
u_p = particle velocity, fps
u_p' = nondimensional particle velocity
W = mass flow rate of gas particle mixture or mass flow rate of gas-only flow (lb/sec)
x,y = Cartesian coordinates

Subscripts

g = gas
p = particle
1 = condition at starting point of the stream tubes

Introduction

THE study of gas particle flow over compressor or turbine blades is an area of recent interest. This is, in part, due to the development of high energy propellants for combustion system in which the product of combustion may partially consist of finely divided particles. For example, turbines which are invariably present in nuclear and liquid propellant rockets, are subjected to gas particle flow. Air-breathing engines operating in desert areas, where the inlet airflow may contain sand particles, provide another example.

The flow of gas particle-suspension[1-4] involves gas particle interaction through viscous drag and heat transfer. This interaction causes changes in the properties of gas and particles. These particles are swept through the cascade nozzle by gas flow, lagging behind the gas in velocity and temperature. Both effects lead to the deterioration in the performance of compressor or turbine. Furthermore, the particles may cause blade erosion and consequently failure. Even when the erosion is not severe enough to cause rupture, changes of blade geometry affect cascade performance.

Blade erosion depends upon several factors, some of which are: the properties of the blade material; the total mass of impinging particles; the particle speed and the angle of attack; and the temperature of both the gas and the particles at the blade surface. Before approaching these problems it is necessary to determine the way in which the gas particle properties vary with the distance along the surface of the blade.

Method of Solution

A numerical solution to the problem of gas particle flow over blades in cascade is obtained, utilizing the known solution for isentropic flow without particles over the same blade.

Presented as Paper 70-712 (Pt. I) at the AIAA 6th Propulsion Joint Specialist Conference, San Diego, Calif., June 15-19, 1970; submitted July 13, 1970; revision received January 4, 1971. This work was sponsored under Project Themis Contract Number DAHCO 4-69C-0016, U.S. Army Research Office—Durham.

Index Categories: Multiphase Flows; Nozzle and Channel Flow; Subsonic and Supersonic Airbreathing Propulsion.

* Professor, Department of Aerospace Engineering. Associate Fellow AIAA.

† Graduate Research Assistant, Department of Aerospace Engineering. Student Member AIAA.

Reprinted from JOURNAL OF AIRCRAFT, Vol. 8, No. 9, September, 1971, pp. 736-738
Copyright, 1971, by the American Institute of Aeronautics and Astronautics, and reprinted by permission of the copyright owner

Pressure Distribution on Blades in Cascade Nozzle for Particulate Flow

W. TABAKOFF* AND M. F. HUSSEIN†
University of Cincinnati, Cincinnati, Ohio

Nomenclature

A = cross-sectional area of stream tube (ft²)
α = particle concentration
C_p = specific heat (Btu/lb°R)
d_p = particle mean diameter (ft)
g = gravity constant
J = mechanical equivalent of heat
k_g = coefficient of conductivity for the gas (Btu/hr ft°R)
P = gas particle suspension pressure
p = pressure of gas-only flow
Pr = Prandtl number
R_g = gas constant
Re = Reynolds number
ρ = the gas-only flow density (lb/ft³)
ρ_g = the gas density in gas particle flow (lb/ft³)
ρ_p = the particle density (lb/ft³)
$\bar{\rho}_p$ = solid particle material density (lb/ft³)
T = the gas-only flow temperature (°R)
T_g = the gas temperature in gas particle flow (°R)
u = gas-only flow velocity (fps)
u_g = gas velocity in gas particle flow (fps)
u_p = particle velocity (fps)
W = mass flow rate of gas particle mixture or mass flow rate of gas-only flow in a stream tube (lb/sec)

Superscripts

$'$ = nondimensional parameter

Subscripts

g = gas
p = particle
1 = condition at starting point of the stream tube
0 = total conditions

Introduction

THE pressure distribution on the blade in cascade increases with the introduction of solid particles to the gas flow.[1,2] The forces on the blade are larger thus requiring a stronger blade design. In this work, a theoretical approach to estimate the pressure distribution on the blade surface for a particulate gas flow for incompressible and compressible cases was developed. Gas flow without particles past a cascade was considered first. It was assumed that an experimental pressure distribution on the blade surface is known for given gas flow and inlet conditions. It was assumed that two stream tubes exist in the flowfield around the blade; one at the suction side and one at the pressure side. The gas flow without particles was used to determine the nondimensional area of the stream tube as function of the given pressure distribution and inlet gas conditions of the nonparticulate gas flow. The governing equations for the particulate flow were formulated for incompressible and compressible cases. The resulting differential equations were solved numerically for the pressure distribution and the particulate flow properties. The different parameters in the governing equations were nondimensionalized with respect to values at a starting point 1, which is in the vicinity of the blade leading edge. These parameters are expressed as follows in Eq. (1)

$$T' = T/T_1 \quad p' = p/p_1 \quad u' = u/u_1 \quad \rho' = \rho/\rho_1 \quad P' = P/P_1$$
$$u'_g = u_g/u_{g_1} \quad T'_g = T_g/T_{g_1} \quad \rho'_g = \rho_g/\rho_{g_1} \quad u'_p = u_p/u_{g_1}$$
$$\rho'_p = \rho_p/\rho_{g_1} \quad T'_p = T_p/T_{g_1} \tag{1}$$

$$A' = A/A_1 = 1/u' \quad \text{(Incompressible case)} \tag{2}$$

$$A' = A/A_1 = 1/u'\rho' \quad \text{(Compressible case)} \tag{3}$$

The definition of the particle concentration α is expressed as

$$\alpha = W_p/(W_p + W_g) = W_p/W \tag{4}$$

Incompressible Flow

The continuity and momentum equations for gas-only flow are

$$(1/u)du/ds + (1/A)dA/ds = 0 \tag{5}$$

$$\rho u du/ds = -(g dp/ds) \tag{6}$$

Substituting Eq. (5) into Eq. (6) and putting the resulting equation in nondimensional form by using Eq. (1), one gets an expression for the rate of change of stream tube nondimensional cross-sectional area as a function of the rate of change of the nondimensional gas without particles flow pressure.

$$dA'/ds = (A'/u'^2)(gp_1/\rho u_1^2)(dp'/ds) \tag{7}$$

where u' is given by

$$u' = [(p_0 - p/p_0 - p_1)]^{1/2} \tag{8}$$

The equation for the drag force on the solid particles, the momentum equation of gas and particles, and the continuity equation of gas are

$$u_p du_p/ds = 18\mu_g/\bar{\rho}_p d_p^2(u_g - u_p) \tag{9}$$

$$dP/ds = -(W/gA)[(1 - \alpha)du_g/ds + \alpha du_p/ds] \tag{10}$$

$$\rho_g u_g A = (1 - \alpha)W \tag{11}$$

Differentiation of Eq. (11) and substitution of Eqs. (1) and (7) into the resulting differential equation results in

$$du'_g/ds = -(u'_g/A')(dA'/ds) = -(u'_g/u'^2)(gp_1/\rho u_1^2)(dp'/ds) \tag{12}$$

Substituting Eq. (1) into Eqs. (9) and (10), they take the nondimensional form

$$u'_p du'_p/ds = (18\mu_g/u_{g_1}\bar{\rho}_p d_p^2)(u'_g - u'_p) \tag{13}$$

$$dP'/ds = -\rho_g u_{g_1}^2/gP_1 A'[du'_g/ds + (\alpha/1 - \alpha)du'_p/ds] \tag{14}$$

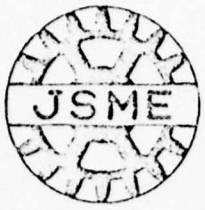
Presented as Paper 71-82 at the 9th Aerospace Sciences Meeting, New York City, January 25–27, 1971, submitted February 1, 1971; revision received May 21, 1971. This work was sponsored under Contract DAHC04-69C-0016, U.S. Army Research Office—Durham,

Index Categories: Multiphase Flows; Subsonic and Supersonic Air Breathing Propulsion.

* Professor, Department of Aerospace Engineering. Associate Fellow AIAA.

† Graduate Research Assistant, Department of Aerospace Engineering.

JSME—14



Tokyo Joint International Gas Turbine Conference and Products Show

Publication

Gas-Particle Suspension Properties in a Cascade Nozzle

W. Tabakoff — Dr., Professor, Department of Aerospace Engineering Univ. of Cincinnati, Cincinnati, Ohio, U.S.A. Memb. ASME

M. F. Hussein — Graduate Research Asst., Department of Aerospace Engineering, Univ. of Cincinnati, Cincinnati, Ohio, U.S.A. Memb. ASME

The gas-particle flow properties, namely flow pressure, gas and particles velocities and temperatures were calculated theoretically in a blade-to-blade stream surface of a cascade nozzle. The effects of the particle concentration, mean diameter and material density on the flow properties at any point in the cascade nozzle were investigated. Some experimental results were obtained from cascades wind tunnel equipped with a solid particle injecting system. The increase in pressure due to the introduction of solid particles gives rise to forces on the blades indicating that this information is of importance to the turbomachinery designer.

Contributed by the Japan Society of Mechanical Engineers for presentation at Tokyo Joint International Gas Turbine Conference and Products Show co-sponcered by the Japan Society of Mechanical Engineers and the Gas Turbine Division of the American Society of Mechanical Engineers, Tokyo, Japan, Oct. 4-7, 1971. Manuscript received at JSME, May 31, 1971.

THE JAPAN SOCIETY OF MECHANICAL ENGINEERS, 4-1-4, AKASAKA, MINATO-KU, TOKYO, JAPAN

Hussein, M.F. and Tabakoff, W., "Calculation of Particle Trajectories in a Stationary Two Dimensional Cascade," Project Themis Report 72-27, June 1972. (AD 764267)

Abstract

The trajectories of solid particles entrained by fluid flow in turbine and compressor stators were calculated. The impact and rebound phenomena of the solid particles was investigated experimentally and then considered in the solution of the equations of motion of the solid particles. The effect of the particle mean diameter, material density, and initial particle and gas velocities on the dynamic behavior of the solid particles through the cascade channel was also investigated. In addition, this study yielded information concerning blade erosion damage.

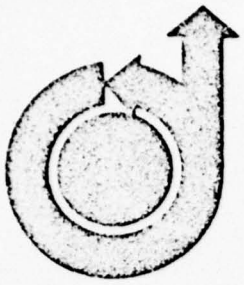
Hussein, M.F. and Tabakoff, W., "Calculation of the Three Dimensional Particle Trajectories in a Turbine Stage," Project Themis Report No. 72-33, September 1972. (AD 753364)

Abstract

The equations of motion, in three dimensions, of solid particles entrained by a gas flow through the stationary and rotating cascades of a turbine are derived. The gas velocity components and density at all the mesh points of a square grid constructed in the flow channels are computed assuming a compressible flow. Formulas to determine the proper drag on the particles for a wide range of Reynolds numbers are given. A gas particle flow tunnel is used to investigate experimentally the phenomenon of particle impact with the turbine blades or casing and their rebound from these walls. Formulas for the restitution ratio due to collision and the rebound to incidence angle ratio are derived. This information is used in the equations of motion of the solid particles.

The dynamic behavior of the solid particles in the turbine stage, namely their absolute and relative trajectories, absolute nondimensional velocity history in the channel, and their velocity diagrams as compared to that of the gas, is investigated. The effect of different flow parameters, mainly, the particle mean diameter, material density, and particle and gas initial velocities on the dynamic characteristics of the solid particles are studied.

Observations concerning the erosion damage suffered by the turbine stator and rotor blades as well as the turbine casing due to the solid particle impingements are presented.



AIAA Paper
No. 72-87

Abstract

The boundary layer of gas-particle flows in fields of changing pressures is analyzed. The analysis, which utilizes the momentum integral method, can be used to study the boundary layer of any particulate flow with an incompressible gas-phase, provided the flow properties outside the boundary layer are known. Numerical integration of the two momentum equations was carried out for the case of particulate flow in cascades. For this example, the results indicate that the introduction of particles leads to an increase in the gas boundary layer thickness. In addition, it was found that the gas boundary layer characteristics are more sensitive to particle concentration than any other particulate flow parameter. However, the particle boundary layer characteristics are more sensitive to mean particle diameter than to particle concentration. Other parameters such as inlet particle gas velocity ratio and particle solid material density have more pronounced effects on the particle boundary layer than on that of the gas.

ANALYSIS OF CASCADE PARTICLE-GAS BOUNDARY LAYER FLOWS WITH PRESSURE GRADIENT

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

AIAA 10th Aerospace Sciences Meeting

SAN DIEGO, CALIFORNIA/JANUARY 17-19, 1972

First publication rights reserved by American Institute of Aeronautics and Astronautics. 1290 Avenue of the Americas, New York, N. Y. 10019. Abstracts may be published without permission if credit is given to author and to AIAA. (Price: AIAA Member $1.50. Nonmember $2.00).

Note: This paper available at AIAA New York office for six months; thereafter, photoprint copies are available at photocopy prices from AIAA Library, 750 3rd Avenue, New York, New York 10017

72-GT-57

an ASME publication

The Society shall not be responsible for statements or opinions advanced in papers or in discussion at meetings of the Society or of its Divisions or Sections, or printed in its publications. Discussion is printed only if the paper is published in an ASME journal or Proceedings.
Released for general publication upon presentation.
Full credit should be given to ASME, the Professional Division, and the author (s).

$3.00 PER COPY
$1.00 TO ASME MEMBERS

Experimental Investigation of Gas-Particle Flow Trajectories and Velocities in an Axial Flow Turbine Stage[1]

W. TABAKOFF

Professor
Mem. ASME

A. HAMED

Graduate Research Assistant

M. F. HUSSEIN

Graduate Research Assistant

Department of Aerospace Engineering,
University of Cincinnati,
Cincinnati, Ohio

This paper describes the results from an investigation of the gas-particle flow trajectories, velocities and pressure distribution in an axial flow turbine stage. A gas-particle flow cascade tunnel and high-speed photographic techniques were used to conduct the experimental investigation. The pressure distribution on the blade surface was measured and compared with the theoretical analysis, the results exhibiting good agreement between the developed theory and experiment.

[1]This work was sponsored under Contract Number DAHCO4-69C-0016, U.S. Army Research Office-Durham.

Contributed by the Gas Turbine Division of the American Society of Mechanical Engineers for presentation at the Gas Turbine and Fluids Engineering Conference & Products Show, San Francisco, Calif., March 26-30, 1972. Manuscript received at ASME Headquarters, December 14, 1971.

Copies will be available until January 1, 1973.

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

ELEVENTH

NATIONAL CONFERENCE

ON

ENVIRONMENTAL EFFECTS

ON

AIRCRAFT AND PROPULSION

SYSTEMS

PAPER
No. 74-2

ANALYSIS OF THE DYNAMIC BEHAVIOR OF SOLID PARTICLES IN COMPRESSORS

Abstract

The three-dimensional equations of motion of solid particles entrained by compressible gas flow in a rotating cascade are solved for the case of particles moving through a compressor guide vane followed by a compressor stage. The solution considers the change in particles dynamic characteristics due to impacts with surrounding walls. The dynamic behavior of the particles for different particulate flow parameters are investigated. Observations concerning compressor blades erosion damage are also discussed.

M.F. HUSSEIN, W. TABAKOFF AND A. HAMED

UNIVERSITY OF CINCINNATI, CINCINNATI, OHIO

MAY 21-22, 1974

TRENTON, NEW JERSEY

Sonderdruck aus „**Zeitschrift für Flugwissenschaften 20**" (*Heft 10, 1972*)
Verlag Friedr. Vieweg + Sohn GmbH, Braunschweig

*Widen Tabakoff**) *and Awatef Hamed***), *Cincinnati, Ohio/USA*

DK 53.529.5 : 532.526

The Boundary Layer of Particulate Gas Flow†)

Übersicht: Die Arbeit befaßt sich mit einer theoretischen Untersuchung der laminaren Grenzschicht für die ebene Gas-Partikel-Strömung. Die Impulsintegralgleichungen bei inkompressibler Gasphase und bei Vorhandensein eines Druckgradienten werden abgeleitet. Es wird der Fall einer Mischströmung von Gas und Partikeln längs einer ebenen Platte betrachtet, wobei beide Phasen dieselbe Geschwindigkeit in der Außenströmung haben. Die aus einer numerischen Berechnung gewonnenen Ergebnisse werden in Kurven dargestellt, in denen die Grenzschichtdicke des Gases, die Schubspannung an der Wand, der Formparameter für das Geschwindigkeitsprofil des Gases und die Verdrängungsdicke der Partikel wiedergegeben sind. Alle diese Parameter sind jeweils für verschiedene Partikelkonzentrationen in der Außenströmung, verschiedene Werte der Partikeldurchmesser und verschiedene Werte der Dichte der festen Teilchen dargestellt.

Summary: A theoretical analysis of the laminar boundary layer for the two-dimensional gas particle flow is presented. The momentum integral equations of the boundary layer with incompressible gas phase and in the presence of pressure gradient are derived. The case of a gas particle mixture flow over a flat plate with both phases having the same velocity in the free stream is presented. A numerical calculation is carried out and the results are presented in curves representing the gas boundary layer thickness, the shear stress at the wall, the gas velocity profile shape factor, and the particle displacement thickness. All these parameters are presented for various particle concentrations in the free stream, different values of particle diameters and different values of solid particle material densities.

Résumé: L'étude porte sur une analyse théorique de la couche limite laminaire pour l'écoulement gaz-particules bidimensionnel. On déduit les équations de l'intégrale d'impulsion de la couche limite en cas d'une phase de gaz incompressible et en présence d'un gradient de pression. On présente le cas d'un écoulement mixte se composant de gaz et de particules le long d'une plaque plane, les deux phases ayant la même vitesse dans l'écoulement libre. Les résultats obtenus d'un calcul numérique sont démontrés en courbes représentant l'épaisseur de couche limite du gaz, la tension de cisaillement à la paroi, le facteur de forme pour le profil de vitesse du gaz et l'épaisseur des particules. Tous ces paramètres sont déterminés pour diverses concentrations des particules dans l'écoulement libre, pour diverses valeurs des diamètres des particules et pour diverses valeurs de la densité des particules solides.

1. Introduction

Gas solid flow systems are of considerable importance in the present rapidly expanding technology. Typical examples are the two-phase flow in thrust nozzles using propellants with metal additives, pneumatic conveying systems, nuclear reactors with gas solid feeding, two-phase flow due to ablation cooling, and many others.

The concepts of continuum mechanics can be extended to particulate flow. The effect of a solid boundary on a gas particle mixture flowing parallel to it is confined to the thin region near the wall. The concept of a viscous flow boundary layer can be extended to a gas particle flow. Although most boundary layers with particle suspension are turbulent, the case of a laminar boundary layer of a gas solid suspension is studied with the purpose of developing a basic understanding of the interaction of gas solid flow with the wall. Due to the complexity of the boundary layers of particulate flows, some of the studies done on such problems [1] obtained the first few terms of asymptotic solutions as $U/F\,x \to 0$ which corresponds to very small particle sizes or very large distances from the leading edge, while others [2] obtained also the first few terms of the asymptotic solution as $U/F\,x \to \infty$ which corresponds to very large particle sizes or very short distances from the leading edge. In other cases [3] the gas phase boundary layer characteristics were assumed to be unaffected by the presence of the particle to obtain an estimate of the particle phase boundary layer thickness very close to the leading edge. Similar approaches were used for unsteady problems of infinite rotating disk [4] and both impulse and oscillating motions of infinite flat plate [5, 6]. In the present analysis the momentum integral method known for the gas only flow is used for the gas particle flows. The two resulting ordinary differential equations are coupled. Here the momentum integral equations for the flat plate were simultaneously inte-

*) Professor, Department of Aerospace Engineering, University of Cincinnati.
**) Graduate Research Assistant, Department of Aerospace Engineering, University of Cincinnati.
†) This research work was supported by the U. S. Army Research Office, Durham, under Project Themis Contract Number DAHC 04-69-C-0016.

INTERNATIONAL ASTRONAUTICAL FEDERATION (I.A.F.)

I.A.F. PAPER - PROPULSION II SESSION

PERFORMANCE AND FLOW PROPERTIES CHANGE THROUGH A ROCKET TURBINE BY PRESENCE OF SOLID PARTICLES*

W. Tabakoff**, W. Hosny***, and A. Hamed***

University of Cincinnati, Ohio, U.S.A.

ABSTRACT

A theoretical method was developed for predicting the pressure distribution over a blade in cascade for a compressible flow with solid particles. Experimental results were obtained from a cascade wind tunnel equipped with a solid particle injection system. Good agreement was noted between the theoretical and experimental pressure distribution. The change in pressure due to the particles gives reduction in the force on the blades.

The results of the experimental trajectories investigation can provide a good approximate prediction of the trajectories and velocities of solid particles suspended in a fluid and passing through an axial flow turbine stage. The technique presented allows the determination of the blade areas which are subjected to erosion damage.

The presence of solid particles in the rocket turbine gas flow changes the turbine performance. The overall turbine efficiency decreases as a result of the introduction of solid particles. The performance experiment was performed on an oxidizer pump drive turbine for an M-1 rocket engine.

I.A.F 23RD INTERNATIONAL ASTRONAUTICAL CONGRESS

VIENNA, AUSTRIA

October 8-15, 1972

---

*This work was sponsored under Project Themis Contract Number DAHC04-69C-0016 U.S. Army Research Office - Durham.

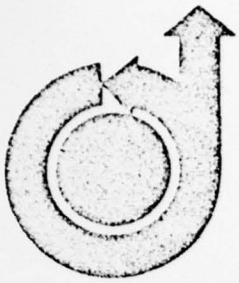
**Professor, Department of Aerospace Engineering, University of Cincinnati,

***Graduate Research Assistants, Department of Aerospace Engineering, University of Cincinnati.

Hussein, M.F. and Tabakoff, W., "Calculation of the Three Dimensional Particle Trajectories in a One and One-Half Stage of a Compressor," Project Themis Report No. 73-34, February 1973. (AD 757164)

Abstract

The equations of motion, in three dimensions, for solid particles moving in a compressible gas flow in a rotating cascade are solved for the case of the particles moving through a compressor guide vane followed by a compressor stage. The solution considers the particle impact and rebound with both the blade walls and the compressor casing. The three dimensional absolute paths of the particles, their trajectories relative to the compressor rotor, their velocity histories, and the combined particle velocity diagrams in the compressor cascades are given. The effects of flow parameters on the dynamic behavior of the solid particles throughout the compressor's one and one-half stage are investigated. Parameters considered included particle mean diameter and material density, initial particle and gas velocities at the guide vane inlet, and compressor rotor rotational speed. Observations concerning the erosion damage to the blades of the compressor cascades are also discussed.



AIAA Paper
No. 73-140

Abstract

The equations that govern the three dimensional motion of solid particles suspended by a compressible gas flow through a rotating cascade of a turbomachine are formulated. These equations are solved for the case of flow through a turbine stage. The solution takes into account the loss in particle momentum due to their collision with the turbine blades or casing. The dynamic characteristics of the solid particles; namely, their absolute trajectories, paths relative to the turbine rotor, velocity distributions, and the combined stage velocity diagrams, are calculated. The effects of changing the particles mean diameter, material density, and initial particle and gas velocities at the stator inlet on the dynamic characteristics of the solid particles are investigated. The results obtained from this study indicate the locations on the turbine blades subjected to severe erosion damage.

THREE DIMENSIONAL DYNAMIC CHARACTERISTICS OF SOLID PARTICLES SUSPENDED BY POLLUTED AIR FLOW IN A TURBINE STAGE

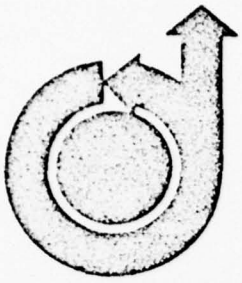
by
M. FATHY HUSSEIN and W. TABAKOFF
University of Cincinnati
Cincinnati, Ohio

AIAA 11th Aerospace Sciences Meeting

WASHINGTON, D.C. / JANUARY 10-12, 1973

First publication rights reserved by American Institute of Aeronautics and Astronautics. 1290 Avenue of the Americas, New York, N. Y. 10019. Abstracts may be published without permission if credit is given to author and to AIAA. (Price: AIAA Member $1.50. Nonmember $2.00).

Note: This paper available at AIAA New York office for six months; thereafter, photoprint copies are available at photocopy prices from AIAA Library, 750 3rd Avenue, New York, New York 10017



AIAA Paper
No. 73-687

ANALYSIS OF NONEQUILIBRIUM PARTICULATE FLOW

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

Abstract

The equations of conservation of angular momentum of the two phases of a particulate flow were derived, and the stress tensor was determined. It was found to consist of anti-symmetric elements in addition to the pressure and symmetric shear stresses in nonparticulate flows. An implicit second order finite difference scheme was used to determine numerically the particulate flow properties throughout the flow field from frozen to equilibrium regimes. The numerical study investigated the impulsive motion of an infinite flat plate in an incompressible viscous gas with suspended solid particles. The effect of some parameters on the resulting particulate flow was obtained.

AIAA 6th Fluid and Plasma Dynamics Conference

PALM SPRINGS, CALIFORNIA / JULY 16-18, 1973

First publication rights reserved by American Institute of Aeronautics and Astronautics. 1290 Avenue of the Americas, New York, N. Y. 10019. Abstracts may be published without permission if credit is given to author and to AIAA. (Price: AIAA Member $1.50. Nonmember $2.00).

Note: This paper available at AIAA New York office for six months; thereafter, photoprint copies are available at photocopy prices from AIAA Library, 750 3rd Avenue, New York, New York 10017

Reprinted from JOURNAL OF AIRCRAFT, Vol. 10, No. 7, July, 1973, pp. 434-440
Copyright, 1973, by the American Institute of Aeronautics and Astronautics and reprinted by permission of the copyright owner

Dynamic Behavior of Solid Particles Suspended by Polluted Flow in a Turbine Stage

M. Fathy Hussein* and W. Tabakoff†
University of Cincinnati, Cincinnati, Ohio

The equations that govern the three dimensional motion of solid particles suspended by a compressible gas flow through a rotating cascade of a turbomachine are formulated. These equations are solved for the case of flow through a turbine stage. The solution takes into account the loss in particle momentum due to their collision with the turbine blades or casing. The dynamic characteristics of the solid particles; namely, their absolute trajectories, paths relative to the turbine rotor, velocity distributions, and the combined stage velocity diagrams, are calculated. The effects of changing the particles mean diameter, material density, and initial particle and gas velocities at the stator inlet on the dynamic characteristics of the solid particles are investigated. The results obtained from this study indicate the locations on the turbine blades subjected to severe erosion damage.

Nomenclature

$\bar{a}$ = particle absolute acceleration
B = frame fixed in blades
β = angle between particle relative velocity and tangent to surface
C = absolute velocity
C' = absolute velocity component in the x,θ plane
Cp_g = specific heat of gas at constant pressure
Ψ = modified stream function
D = drag force on spherical particles
d_p = particle mean diameter
G = coefficient inversely proportional with the particle characteristic time
$g(Re)$ = Reynolds number dependent function
h = normal stream channel thickness (blade height)
λ = angle between meridional streamline and engine axis
m_p = mass of one particle
μ_g = gas viscosity
p = pressure at a point
$\bar{P}$ = force on a spherical particle due to pressure gradient
R = blade mean radius
Re = Reynolds number
r = radius from axis of rotation
ρ_g = gas density
ρ_p = particle material density
s = blade angular spacing
T = temperature
U = blade speed at the mean radius
u = relative velocity component in the x-direction
V = relative velocity
V' = relative velocity component in the x,θ plane
v = relative velocity component in the tangential direction
W = weight flow rate of mixture per channel
w = relative velocity component in the radial direction
ω = blade angular velocity
x,θ,z = coordinate axes in the meridional, tangential and radial directions, fixed in B or components of the relative particle position vector
X,Y,Z = axes fixed in engine or components of the absolute particle position vector
$\dot{x},\dot{\theta},\dot{z}$ = particle velocity components measured in B
$\ddot{x},\ddot{\theta},\ddot{z}$ = particle acceleration components measured in B
$\dot{X},\dot{Y},\dot{Z}$ = particle absolute velocity measured in E
$\ddot{X},\ddot{Y},\ddot{Z}$ = particle absolute acceleration measured in E
y = distance measured along the tangential direction

Subscripts

a = absolute trajectory
g = gas
i = initial conditions at particles entrance
in = conditions at the boundary AH
n = normal to blade surface
p = particle
t = tangent to blade surface
1 = before collision
2 = after collision

Introduction

ROCKETS, aircraft engines, and industrial gas turbines operating in desert areas and in places where the atmosphere is polluted by small solid particles can be examples of machines operating under gas particle two phase flow conditions. The solid particles mixed with the inlet air or combustion gases, due to the difference in their inertia, will be driven away from the streamlines of the gas and impact with the surrounding walls of the engine. The presence of solid particles in the flow constitutes a cause for severe erosion damage especially to the rotating parts of the engine, where the particle velocities and frequency of impacts, as well as the flow temperature, are higher. Hence, turbines, axial or centrifugal, of rockets or gas turbines are parts of the engine critically subjected to particles erosion. The centrifugal forces acting on the particles tend to force them to move radially, and hence, impact with the turbine casing, causing it to also suffer from erosion damage. In order to understand and predict the erosion phenomenon of rotating turbomachines, it is important to study the dynamic characteristics or behavior of the solid particles entrained by the gas flow through the stages of turbines. By the dynamic behavior, it is meant, the absolute and relative position of the particles everywhere in the channels, their velocity history and velocity diagrams as well as a description of the collision and rebound mechanisms of the solid particles from the walls. The effect of different flow parameters such as particle mean diameter, material density and initial particle and gas velocities on the particle dynamic behavior are investigated. Results of this study are then used to make observations concerning the areas of the blades or casing that are subjected to erosion damage.

The three dimensional equations of motion of solid particles moving in a compressible gas stream in a rotating cas-

Received January 11, 1973; also presented as Paper 73-140 at the AIAA 11th Aerospace Sciences Meeting, Washington, D.C., January 10-12, 1973; revision received April 16, 1973. This work was sponsored under Contract Number DAHC04-69-C-0016, U.S. Army Research Office, Durham.

Index categories: Subsonic and Supersonic Air-Breathing Propulsion; Multiphase Flows.

*Postdoctoral Fellow, Department of Aerospace Engineering, University of Cincinnati. Associate Member AIAA.

†Professor, Department of Aerospace Engineering, University of Cincinnati. Associate Fellow AIAA.

Hamed, A. and Tabakoff, W., "The Solution of the Equations of Motion for Unsteady Viscous Particulate Flows," Report No. 74-42, July 1974.

Abstract

A numerical method is developed to solve the equations that govern the motion of the unsteady two dimensional flow of a suspension consisting of incompressible viscous gas and suspended solid particles. The numerical method is implicit, second order and iterative. The frozen and equilibrium wall friction coefficients are compared with the corresponding analytical determined values for a simple example, and are found to be in agreement. It is concluded that the present numerical method for investigating the nonequilibrium flow regime, gives accurate results not only for nonequilibrium flow conditions, but also predicts frozen and equilibrium flow conditions very accurately.

Ball, R. and Tabakoff, W., "An Experimental Investigation of the Particle Dynamics of Quartz Sand Impacting 6Al-4V Titanium and 410 Stainless Steel in an Erosive Environment," Report No. 74-43, October 1974.

Abstract

The impact and rebound characteristics of high speed erosive quartz particles have been experimentally determined. The impact parameters were found to be statistical in nature and the statistical distributions were obtained. The alloys used in this investigation were 410 stainless steel and 6Al-4V titanium.

Reprinted from AIAA JOURNAL, Vol. 12, No. 5, May 1974, pp 581-582
Copyright, 1974, by the American Institute of Aeronautics and Astronautics, and reprinted by permission of the copyright owner

Some Effects Caused by Solid Particles in Flows

A. HAMED* AND W. TABAKOFF†
University of Cincinnati, Cincinnati, Ohio

Theme

THE equations of conservation of angular momentum of the two phases of a particulate flow are derived, and the stress tensor determined. It was found to consist of antisymmetric elements in addition to the pressure and symmetric shear stresses in nonparticulate flows. The particulate flow properties are determined numerically throughout the flowfield from frozen to equilibrium regimes.

Contents

A study of the behavior of particulate flows near the solid boundaries is necessary to determine the effect of the presence of solid particles on the skin friction and heat transfer. In the present analysis, the particles are continuously distributed throughout the flowfield and each of the two phases has its own mean properties. The particles translational velocity is generally different in magnitude and direction from that of the gas. They also have a rotational velocity which is not equal to the gas rotational speed. The particles motion, however, is basically due to the gas motion.

If the rate of change of the angular momentum of the suspension in the volume V is set equal to the total moment of the surface forces about an axis in the k-direction passing through a point 0 within this volume we obtain

$$\int (1-\chi)\rho\varepsilon_{kij} r_i (dv_j/dt)\,dV + \int \chi\rho_p \varepsilon_{kij} r_i (dv_{p_j}/dt)\,dV + \int \chi\rho_p r_j^2 (d\omega_k/dt)\,dV = \int \varepsilon_{kij}[\partial(r_i\sigma_{ij})/\partial r_i]\,dV \qquad (1)$$

where ρ and ρ_p are the fluid and solid particle material densities, χ is the volume occupied by the solid particles per unit volume of mixture, r_j is the radius of gyration of a solid particle, v and v_p are the gas and particle velocities, ω is the solid particle rotational velocity, and r is the position vector, whereas the subscripts i, j, k refer to components in these directions.

If the volume, V, is reduced to zero such that the configuration, which is made up of the boundary of the volume and the fixed point 0, retains the same shape, the last term on the left-hand side and the first term on the right-hand side of Eq. (1), approach zero in the same order as V, while the rest of the terms approach zero as $V^{4/3}$. Thus

$$\chi\rho_p r_j^2 (d\omega_k/dt) = \varepsilon_{kij}\sigma_{ij} \qquad (2)$$

The stress tensor is written below as the sum of three terms. The first term is spherically symmetric and represents uniform compression, the second is a symmetrical tensor, and the third is antisymmetric

$$\sigma_{ij} = -p\delta_{ij} + \tau_{ij} + \tau_{ij}^a \qquad (3)$$

where p is the pressure and

$$\tau_{ij} = 2\mu\left[\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right] \qquad (4)$$

If the expression for the stress tensor given by Eq. (3) is substituted into Eq. (2), the contribution of the spherically symmetric and symmetrical tensors to the right-hand side of Eq. (2) is zero and, hence, the following relation is obtained:

$$\tau_{ij}^a = (\chi\rho_p/2) r_j^2 (d\omega_k/dt); \qquad i \neq j \neq k \qquad (5)$$

Thus, the stress tensor in suspension flows consists of an antisymmetric shear stress, (τ_{ij}^a), which is equal to half the rate of change of the local angular momentum of the solid particles per unit volume, in addition to the familiar pressure and symmetric shear stress in nonparticulate fluid flows.

Some numerically determined, incompressible particulate flow properties due to the impulsive motion of an infinite flat plate[1] are presented. The effect of the initial particle concentration, χ_o and the particle to gas density ratio, ρ_p/ρ, on the flow properties was investigated. The friction coefficient is plotted against time for different particle to gas density ratios in Fig. 1. The nondimensional skin-friction and antisymmetric stress tensor are defined as

$$C_f^* = C_f (Re\, t^*)^{1/2}$$

$$C_{xy^a}^* = \frac{2\tau_{xy}^a (Re\, t^*)^{1/2}}{\rho U^2 \chi_o}$$

The Reynolds number, Re, is based on the plate velocity, U, and the relaxation time of particle translation, τ_t

$$Re = U^2\tau_t/\nu$$

The time is normalized with respect to τ_t

$$t^* = t/\tau_t$$

where

$$\tau_t = (d^2/18\nu)(\rho_p/\rho)$$

In the last equation, d is the solid particle diameter and ν is the kinematic viscosity.

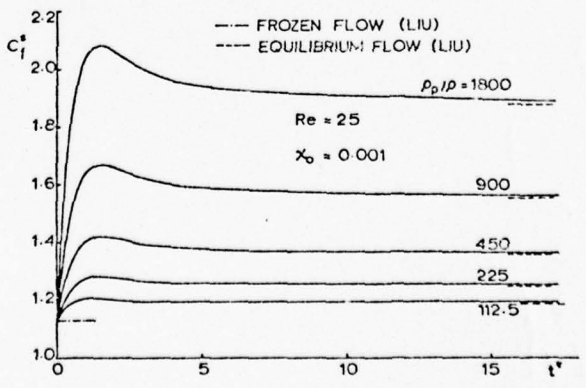


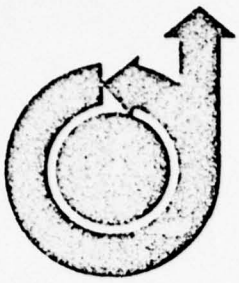
Fig. 1 Effect of particle to gas density ratio on the friction coefficient at the wall.

Presented as Paper 73-687 at the AIAA 6th Fluid and Plasma Dynamics Conference, Palm Springs, Calif., July 16–18, 1973; synoptic received August 15, 1973; revision received November 26, 1973. Full paper available from AIAA Library, 750 Third Avenue, New York, N.Y., 10017. Price: Microfiche, $1.00; hard copy, $5.00. **Order must be accompanied by remittance.** This work is sponsored under Contract DAHC04-69-C-0016, U.S. Army Research Office—Durham.

Index category: Multiphase Flows.

* Assistant Professor, Department of Aerospace Engineering. Associate Member AIAA.

† Professor, Department of Aerospace Engineering. Associate Fellow AIAA.



AIAA Paper
No. 74-561

A NUMERICAL METHOD FOR THE SOLUTION OF PARTICULATE FLOW EQUATIONS

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

Abstract

A numerical method is developed to solve the equations governing the motion of a gas-solid particle suspension. The values of the skin friction coefficient at the plate surface at zero time and at time equal to fifteen times the characteristic time of particle translational relaxation are compared to the skin friction coefficients of the frozen and equilibrium flow regimes determined analytically. The two values are found to be in agreement. It is concluded that the present numerical method for investigating the nonequilibrium flow regime, gives accurate results not only for nonequilibrium flow conditions, but also predicts frozen and equilibrium flow conditions very accurately.

AIAA 7th Fluid and Plasma Dynamics Conference

PALO ALTO, CALIFORNIA / JUNE 17-19, 1974

First publication rights reserved by American Institute of Aeronautics and Astronautics. 1290 Avenue of the Americas, New York, N. Y. 10019. Abstracts may be published without permission if credit is given to author and to AIAA. (Price: AIAA Member $1.50. Nonmember $2.00).

Note: This paper available at AIAA New York office for six months; thereafter, photoprint copies are available at photocopy prices from AIAA Library, 750 3rd Avenue, New York, New York 10017

Computers and Fluids, Vol. 2, pp. 1–15. Pergamon Press, 1974. Printed in Great Britain

COMPUTATION AND PLOTTING OF SOLID PARTICLE FLOW IN ROTATING CASCADES*

M. FATHY HUSSEIN and W. TABAKOFF
University of Cincinnati, Cincinnati, Ohio 45221, U.S.A.

(*Received* 3 *January, revised* 29 *November* 1973)

Abstract—A numerical technique to solve the three dimensional equations of motion of solid particles suspended by a compressible fluid flow in a rotating cascade of a turbomachine is presented. The solution of these equations utilizes the numerically computed nonparticulate fluid flow properties through the same cascade channel. It also takes into consideration the experimentally investigated phenomenon of solid particle impingement with the blade surface and turbomachine casing and their subsequent rebound. Computer plottings of the absolute paths of the solid particles, their trajectories relative to the rotor blades, and their velocity distributions in a turbine stator, turbine stage, and a compressor one and one-half stage are given.

NOMENCLATURE

β	angle between particle relative velocity and tangent to surface
B	frame fixed in blades
C	absolute velocity
C_D	drag coefficient on spherical particles
C_{P_g}	specific heat of gas at constant pressure
C_{P_n}	particle normal velocity component relative to the blade at impingement
Ψ	modified stream function
d_p	particle mean diameter
Δx	spacing between adjacent points in the axial direction
$\Delta\theta$	spacing between adjacent points in the tangential direction
Δt	increament of time
E	frame fixed in the engine
η	angle between relative velocity and meridional plane
G	coefficient
h	normal stream channel thickness (blade height)
λ	angle between meridional streamline and engine axis
μ_g	gas viscosity
$\bar{n}_1, \bar{n}_2, \bar{n}_3$	unit vectors in the direction of the coordinate curves x, θ, z
$\bar{N}_1, \bar{N}_2, \bar{N}_3$	unit vectors in the direction of the axes X, Y, Z
R	blade mean radius
Re	Reynolds number
r	radius from axis of rotation
ρ_g	gas density
$\bar{\rho}_g$	particle material density
s	blade angular spacing
T	temperature
u	relative velocity component in the x-direction
V	relative velocity
v	relative velocity component in the tangential direction
W	weight flow rate of mixture per channel
w	relative velocity component in the radial direction

* This work was sponsored under Contract Number DAHC04-69-C-0016, U.S. Army Research Office, Durham.

Astronautica Acta. Vol. 18 (Supplement), pp. 249–257. Pergamon Press 1974. Printed in Great Britain

Performance and Flow Properties Change Through a Rocket Turbine by Presence of Solid Particles[1]

Widen Tabakoff[2], Wishaa M. Hosny[3] and Awatef A. Hamed[4]
University of Cincinnati, Ohio 45221, U.S.A.

(*Received* 25 *September* 1973)

Abstract—**Performance and Flow Properties Change Through a Rocket Turbine by Presence of Solid Particles.** A theoretical method was developed for predicting the pressure distribution over a blade in cascade for a compressible flow with solid particles. Experimental results were obtained from a cascade wind tunnel equipped with a solid particle injection system. Good agreement was noted between the theoretical and experimental pressure distribution. The change in pressure due to the particles gives reduction in the force on the blades.

The results of the experimental trajectories investigation can provide a good approximate prediction of the trajectories and velocities of solid particles suspended in a fluid and passing through an axial flow turbine stage. The technique presented allows the determination of the blade areas which are subjected to erosion damage.

The presence of solid particles in the rocket turbine gas flow changes the turbine performance. The overall turbine efficiency decreases as a result of the introduction of solid particles. The performance experiment was performed on an oxidizer pump drive turbine for an M-1 rocket engine.

Translated abstracts appear at the end of this article

Nomenclature

A	cross-sectional area of stream tube (ft^2);
A'	non-dimensional cross-sectional area of stream tube;
α	the ratio of the mass flow rate of particles to the total mass flow rate of the gas and particle mixture;
b.h.p.	brake horsepower (h.p.);
C_{pp}	specific heat of solid particles material (Btú/lb °R);
C_{pg}	gas specific heat at constant pressure (Btu/lb °R);
δs	distance along airfoil contour between two successive points (ft);
δ	ratio of inlet total pressure to NACA standard S.L. pressure ($P_{t_1}/14.696$);
d_p	particle mean diameter (ft);
g	gravity constant;
θ	ratio of inlet total temperature to NACA standard S.L. temperature ($T_{t_1}/519$);
h	enthalpy (Btu/lb);
J	mechanical equivalent of heat;
k_g	coefficient of conductivity for the gas (Btu/hr ft °R);
N	turbine rotative speed;
N_D	turbine design speed;
P	gas particle suspension pressure;
P'	non-dimensional gas particle suspension pressure;
p	pressure of gas-only flow;
p'	non-dimensional gas-only flow pressure;
Pr	Prandtl number;
R_g	gas constant;
Re	Reynolds number;
ρ	the gas-only flow density (lb/ft^3);
ρ'	the gas-only flow non-dimensional density;
ρ_g	the gas density in gas particle flow (lb/ft^3);
ρ'_g	the non-dimensional gas density in gas particle flow;
ρ_p	the particle density (lb/ft^3);
ρ'_p	the non-dimensional particle density;
$\bar{\rho}_p$	solid particle material density (lb/ft^3);
T	the gas-only flow temperature (°R) or turbine torque;

[1] This work was sponsored by the U.S. Army Research Office—Durham, under Contract Number DAHC04-69C-0016.

[2] Professor, Department of Aerospace Engineering, University of Cincinnati.

[3] Graduate Research Assistant, Department of Aerospace Engineering, University of Cincinnati.

[4] Research Assistant Professor, Department of Aerospace Engineering, University of Cincinnati.

Reprinted from March 1975 Journal of Fluids Engineering

Solid Particle Demixing in a Suspension Flow of Viscous Gas

A. HAMED
Assistant Professor.
Mem. ASME

W. TABAKOFF
Professor. Mem. ASME

Department of Aerospace Engineering, University of Cincinnati, Cincinnati, Ohio

The motion of a medium consisting of an incompressible viscous fluid and suspended solid particles was analyzed from the standpoint of continuum mechanics. It was assumed that the particles' translational and rotational velocities were different from those of the fluid. A numerical scheme was used to determine the non-equilibrium particulate flow properties as well as their equilibrium and frozen values. The results are presented for the case of particulate flow due to the impulsive motion of an infinite flat plate in a suspension. A demixed region, with no particles present, was found to develop near the plate due to particle migration away from the wall. Similar demixed particle regions were noticed in the experimental data of particulate flows in pipes and channels.

Introduction

Systems of fluid flows with suspended solid particles have practical applications in numerous industrial areas. In some of the applications the contact of the particles with the solid boundary should be avoided in order to reduce erosion damage, as in the case of particulate flows in nozzles [1][1] and turbomachines [2], or the transport of solid material in suspended flows [3]. In other applications like filtration and separation of solid particles, the deposition of the particles on the wall is desired [4]. Since the effects of fluid viscosity also prevail near the boundaries, the study of particulate flows with a viscous fluid phase is necessary in order to analyze the solid particle motion at the walls.

For particulate flow systems with a nonviscous fluid phase, the problem is basically the study of the relaxation of translational motion and temperature of the particles from their initial values, to the fluid velocities and temperature. If the fluid phase is viscous, the relaxation of the particle rotation as a solid body, which is initially different from the fluid rotation, should be included in the analysis. A lift force of interaction exists in addition to the drag.

Although many of the fluid-solid particle suspension flows are turbulent, theoretical treatment of such flows is formidable since an extensive amount of experimental data is needed in order to obtain semiempirical hypothesis necessary to treat the problem mathematically. The amount and accuracy of the available experimental data of turbulent particulate flows [1], [3], [5], [6] are insufficient to arrive at a conclusive hypothesis. The study of laminar suspension flows points out the important flow parameters and determines their effect on the behavior of the two phases.

Analysis

Governing Equations. The equations of motion are written from the continuum point of view, for a two dimensional fluid-solid particle medium, treating it as consisting of two continuous homogeneous phases, each having its mean properties [7], [8], [9], [10]. The particles are assumed to be of homogeneous size and physical properties.

The equation of conservation of mass of the fluid is:

$$-\frac{\partial \chi}{\partial t}+\frac{\partial[(1-\chi)u]}{\partial x}+\frac{\partial[(1-\chi)v]}{\partial y}=0 \qquad (1)$$

while conservation of mass of the particle phase is given by:

$$\frac{\partial \chi}{\partial t}+\frac{\partial[\chi u_p]}{\partial x}+\frac{\partial[\chi v_p]}{\partial y}=0 \qquad (2)$$

The equations of conservation of linear momentum of the fluid in the x and y directions are:

$$(1-\chi)\rho\left[\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right]=\frac{\partial}{\partial x}[\sigma_{xx}]+\frac{\partial}{\partial y}[\sigma_{yx}]-R_x \qquad (3)$$

$$(1-\chi)\rho\left[\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right]=\frac{\partial}{\partial x}[\sigma_{xy}]+\frac{\partial}{\partial y}[\sigma_{yy}]-R_y \qquad (4)$$

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

Contributed by the Fluids Engineering Division and presented at the Applied Mechanics/Fluids Engineering Conference, Atlanta, Ga., June 20–22, 1973, of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERING. Manuscript received at ASME Headquarters January 6, 1975.

ANAIS
DO
III CONGRESSO BRASILEIRO
DE ENGENHARIA MECANICA

RIO DE JANEIRO, DEZ. 1975

PAPER NO. B - 8
PP. 367 - 382

PROCEEDINGS
OF THE THIRD
BRAZILIAN CONGRESS
OF
MECHANICAL ENGINEERING

DYNAMICS AND EROSION STUDY OF SOLID PARTICLES IN A CASCADE

M.F. HUSSEIN, W. TABAKOFF and A. HAMED
Department of Aerospace Engineering
University of Cincinnati
Cincinnati, Ohio 45221, USA

1. Introduction

This investigation was undertaken to determine a theoretical approach to calculate the dynamic behavior of solid particles entrained by a gas flow in a two-dimensional stationary cascade of a turbine.

The equations of motion of solid particles moving in a stream of gas were formulated assuming that the only force exerted on the particles is the drag force which causes the particles acceleration. In order to determine the solution of the equations of motion, information about the drag

Reprinted from AIAA JOURNAL, Vol. 13, No. 6, June 1975, pp. 715-716
Copyright, 1975, by the American Institute of Aeronautics and Astronautics, and reprinted by permission of the copyright owner

Numerical Method for Solution of Particulate Flow Equations

A. HAMED* AND W. TABAKOFF†
University of Cincinnati, Cincinnati, Ohio

Theme

THE equations describing the motion of a medium consisting of viscous fluid and suspended solid particles are nonlinear, partial differential equations, and are always coupled even in the case of incompressible fluid phase. Analytical solutions could be obtained for frozen and equilibrium flow regimes of a few problems with simplified models for the forces of bulk interaction between the two phases. If the nonequilibrium particulate flow is to be studied, numerical computational methods have to be used. A numerical method to analyze the unsteady two-dimensional motion of incompressible viscous gas and suspended solid particles is presented. This method is used to study the particulate flow due to the impulsive motion of an infinite flat plate in an otherwise stationary suspension.

Contents

The motion of the fluid and solid particles is governed by the equations of continuity and momentum of the gas, the equations of linear and angular momentum of particles, and the equation of conservation of mass of particles. The equation of angular momentum of the suspension was derived by the authors[1] and was used to determine the stress tensor in the particulate flow. The equations of motion are solved to determine the following particulate flow properties; the fluid velocity, the particle concentration, and the translational and rotational speeds of the solid particles throughout the flowfield. The solid particle translational and rotational velocities are generally different from those of the gas phase, depending on the flow regime. In the frozen or near frozen regime, the difference between the translational velocity of the two phases which is referred to as the slip velocity is very large. The particulate flow is said to be in the near equilibrium and equilibrium regimes when the slip velocity is small and approaching zero. The non-equilibrium flow regime corresponds to the intermediate range of slip velocity.

The equations of motion are solved in nondimensional form, the normalized suspension flow variables are defined as

$$t^* = t/\tau_t, \quad u^* = u/U, \quad u_p^* = u_p/U, \quad \omega^* = [\tau_t/(Re)^{1/2}]\omega$$
$$y^* = [(Re)^{1/2}y/U\tau_t], \quad v^* = (Re)^{1/2}v/u; \quad v_p^* = (Re)^{1/2}v_p/U \tag{1}$$

where $Re = U^2\tau_t/\nu$. The time t is normalized with respect to the characteristic time of particle translational motion τ_t where $\tau_t = (d^2/18\nu)(\rho_p/\rho)$, u and v are the gas velocity components in the x and y directions, u_p and v_p are the particle velocity components in the x and y directions, ω is the solid particle rotational velocity, ρ the gas density, ρ_p the solid particle material density, d the solid particle diameter, and U is a characteristic velocity of the flow.

Before solving the governing equations, a transformation of the independent variables is used. Self-similar solutions cannot be obtained in the case of particular flows due to the characteristic relaxation times associated with the translational and rotational motions of the particles. A transformation that leads to a self-similar solution in nonparticulate flows is used to eliminate the discontinuity that would otherwise exist in the initial gas velocity profile in the physical plane. The transformation also makes it unnecessary to add more mesh points in the direction normal to the flow as the thickness of the momentum boundary layer increases. For the example considered here, the following transformation is used:

$$\eta = y^*/2(t^*)^{1/2} \tag{2}$$

The forces and torque of bulk interaction between the two phases depend generally on the range of some parameters of the particulate flow under consideration. The drag force and torque due to the difference between the translational and rotational velocities of the particles and the fluid, as well as the lift force due to the translational motion of the particles in the shear flow are considered. The expressions used in the present example correspond to dilute suspensions and to low slip Reynolds number, ignoring the gravity forces. The slip Reynolds number is based on the slip velocity between the particles and the gas and the solid particle diameter. The numerical method presented here can be used however for any other range of flow parameters if the corresponding expression of the forces and torque of bulk interaction are used.

The equations that govern the unsteady two-dimensional motion of the suspension in the transformed plane (η, t^*) are

$$\frac{\partial[(1-\chi)v^*]}{\partial\eta} + \frac{\partial[\chi v_p^*]}{\partial\eta} = 0 \tag{3}$$

$$\frac{\partial\chi}{\partial t^*} - \frac{\eta}{2t^*}\frac{\partial\chi}{\partial\eta} + \frac{1}{2(t^*)^{1/2}}\left(v_p^*\frac{\partial\chi}{\partial\eta} + \chi\frac{\partial v_p^*}{\partial\eta}\right) = 0 \tag{4}$$

$$(1-\chi)\left[\frac{\partial u^*}{\partial t^*} + \left(\frac{v^*}{2(t^*)^{1/2}} - \frac{\eta}{2t^*}\right)\frac{\partial u^*}{\partial\eta}\right] = \frac{1}{4t^*}[1+1.5\chi]\frac{\partial^2 u^*}{\partial\eta^2} + \frac{1.5}{4t^*}\frac{\partial\chi}{\partial\eta}\frac{\partial u^*}{\partial\eta} + \frac{3}{2(t^*)^{1/2}}\frac{\partial(\chi\omega^*)}{\partial\eta} - \frac{\rho_p}{\rho}\chi G(u^*-u_p^*) + \frac{3.23}{2\pi}\chi\left(-\frac{1}{(Re\,t^*)^{1/2}}\frac{\rho_F}{\rho}\frac{\partial u^*}{\partial\eta}\right)^{1/2}\times (v^*-v_p^*) \tag{5}$$

$$\frac{\partial u_p^*}{\partial t^*} + \left(\frac{v_p^*}{2(t^*)^{1/2}} - \frac{\eta}{2t^*}\right)\frac{\partial u_p^*}{\partial\eta} = G(u^*-u_p^*) - \frac{3.23}{\pi}\left(\frac{\rho}{\rho_p}\right)^{1/2}Re^{-1/4}\left(-\frac{1}{4(t^*)^{1/2}}\frac{\partial u^*}{\partial\eta}\right)^{1/2}(v^*-v_p^*) \tag{6}$$

$$\frac{\partial v_p^*}{\partial t^*} + \left(\frac{v_p^*}{2(t^*)^{1/2}} - \frac{\eta}{2t^*}\right)\frac{\partial v_p^*}{\partial\eta} = G(v^*-v_p^*) + \frac{3.23}{\pi}\left(\frac{\rho}{\rho_p}\right)^{1/2}Re^{3/4}\left(-\frac{1}{4(t^*)^{1/2}}\frac{\partial u^*}{\partial\eta}\right)^{1/2}(u^*-u_p^*) \tag{7}$$

Presented as Paper 74-651 at the AIAA 7th Fluid and Plasma Dynamics Conference, Palo Alto, California, June 17-19, 1974; submitted June 18, 1974; synoptic received September 12, 1974; revision received November 11, 1974. Full report on which Paper 74-561 was based is available from National Technical Information Service, Springfield, Va., 22151, as N75-12227 at the standard price (available upon request). This work was sponsored under Contract DAHCO4-69-C-0016, U.S. Army Research Office – Durham.

Index category: Multiphase Flows.

* Assistant Professor, Department of Aerospace Engineering. Member AIAA.

† Professor, Department of Aerospace Engineering. Associate Fellow AIAA.

Hussein, M.F. and Tabakoff, W., "Computer Program for Calculations of Particle Trajectories Through a Rotating Cascade," Report No. 76-47, May 1976.

Abstract

This report gives a listing of the computer program used to calculate the three dimensional absolute paths of particles through a rotating cascade, their trajectories relative to the rotor and their velocity histories. The program considers the impact and rebound of particles with the blade walls or casing. The program is written for axial flow machines and may be modified to handle other types. The program procedures, description of the program input and output as well as program subroutines and their functions are discussed.

Reprinted from the January 1976
Journal of Engineering for Power

W. Tabakoff
Professor.

W. Hosny
Graduate Research Assistant.

A. Hamed
Assistant Professor.

Dept. of Aerospace Engrg.,
University of Cincinnati,
Cincinnati, Oh.

Effect of Solid Particles on Turbine Performance

A theoretical method was developed for predicting the pressure distribution over a blade in cascade for a compressible flow with solid particles. Experimental results were obtained from a cascade wind tunnel equipped with a solid particle injection system. Good agreement was noted between the theoretical and experimental pressure distribution. The change in pressure due to the particles gives reduction in the force on the blades. The presence of solid particles in air-breathing engine gas flow changes the turbine performance. The overall turbine efficiency decreases as a result of the introduction of solid particles. The performance experiment was performed on a two-stage velocity-compounded turbine.

Introduction

The problem of a gas-flow mixed with solid particles in axial flow turbines has great importance in industrial, naval, and aeronautical applications. For example, the operation of engines in desert areas or solid particle polluted atmospheres. In some combustion processes, solid particles are formed as by-products of the combustion. This is especially true for the more advanced, powerful fuels. The presence of solid particles can cause erosion damage of the turbine blades and alter the pressure distribution on them. The erosion damage on turbine blades from a particulated flow may be very critical. This damage is especially critical in turbine nozzles where the temperature of the gas-flow and the solid particles are high, along with high particle velocities which greatly increase the erosion damage. Air flow with suspended sand particles would also affect the compressor and the turbine performances and consequently the engine performance.

In this work the following two basic problems are presented:

1 A theoretical analysis which is used to estimate the pressure distribution on the blades for a gas particle suspension flowing through a turbine cascade.

2 An experimental study to show the effect of particulated flow on a jet engine turbine performance.

Pressure Distribution on the Blades for Gas Particle Flow

In references [1, 2][1] a discussion and analysis of the flow properties and pressure distribution of gas particle suspension over blade surfaces of a cascade nozzle were presented. In these references the gas flow was considered inviscid except for the drag it exerts on the particles.

The theoretical approach to the solution of this problem is to consider gas flow without particles past a cascade to determine the pressure distribution over the blade and inlet flow conditions. Next, it was assumed that two stream tubes exist in the flow field around the blade; one at the suction side and one at the pressure side. The gas flow without particles was used to determine the nondimensional area of the stream tube as a function of the given pressure distribution and inlet gas conditions of the nonparticulate gas flow. The governing equations of the particulate compressible gas flow were formulated. These equations were solved numerically for the pressure distribution. The different parameters in the governing equations were nondimensionalized with respect to values at some starting point which is in the vicinity of the blade leading edge.

These parameters are respectively, nondimensional temperature (T'), pressure (p'), velocity (u'), and density (ρ') of the gas without particles flow. The nondimensional parameters for the gas-particle flow are: particulate flow pressure (P'), gas velocity (u'_g), temperature (T'_g) and density (ρ'_g) and nondimensional particle velocity (u'_p), temperature (T'_p) and density (ρ'_p) for gas particle suspension.

$$\left.\begin{aligned} T' &= \frac{T}{T_1} \;;\; p' = \frac{p}{p_1} \;;\; u' = \frac{u}{u_1} \;;\; \rho' = \frac{\rho}{\rho_1} \;; \\ P' &= \frac{P}{P_1} \;;\; u'_g = \frac{u_g}{u_{g_1}} \;;\; T'_g = \frac{T_g}{T_{g_1}} \;;\; \rho'_g = \frac{\rho_g}{\rho_{g_1}} \;; \\ u'_p &= \frac{u_p}{u_{g_1}} \;;\; \rho'_p = \frac{\rho_p}{\rho_{g_1}} \;;\; T'_p = \frac{T_p}{T_{g_1}} \end{aligned}\right\} \quad (1)$$

The nondimensional area distribution of the stream tube of the gas-flow on the airfoil surface can be calculated from the continuity equation. For the compressible flow

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

Contributed by the Gas Turbine Conference Division and presented at the Gas Turbine Conference, Houston, Texas, March 2-6, 1975, of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS. Manuscript received at ASME Headquarters December 2, 1974. Paper No. 75-GT-41.

221 - 240

PRACE INSTYTUTU MASZYN PRZEPŁYWOWYCH
1976 Zeszyt 70 - 72

Published in Polish Academy of Sciences, 1976.

WIDEN TABAKOFF, AWATEF HAMED

U.S.A.*

Dynamics and Erosion Study of Solid Particles in a Cascade**

Badanie dynamiki cząstek stałych i ich erozyjne działanie w palisadzie

Streszczenie

Oblicza się trajektorie i prędkości cząstek stałych niesionych przez przepływ ściśliwy w palisadzie turbiny i sprężarki. Badano eksperymentalnie i uwzględniono przy rozwiązywaniu równań ruchu cząstek mechanizm zderzenia i odbicia cząstek stałych od ścianki łopatki. Badano wpływ średniej średnicy cząstek stałych, gęstości materiału, z którego są zbudowane oraz początkowych prędkości cząstek i gazu na ich charakterystyki dynamiczne w palisadzie. Ponadto przeprowadzono badania doświadczalne mające na celu znalezienie wpływu kąta padania, prędkości cząstek i ich rozmiarów na erozję łopatek. Przedyskutowano obserwacje dotyczące erozji łopatek.

Исследование динамики твердых частиц и эрозии в решетке

Резюме

Проводится расчет траекторий и скоростей твердых частиц в сжимаемом потоке, протекающем через турбинную или компрессорную решетку профилей. Экспериментально исследовался механизм ударов и отражений твердых частиц от стенки лопатки, который принимался во внимание при решениях уравнений движения частиц. Исследовалось влияние среднего диаметра твердых частиц, плотности материала, из которого они состоят, а также начальных значений скорости частиц и газа на их динамические характеристики в решетке. Кроме того, проводились экспериментальные исследования с целью определения влияния угла падения, скорости частиц и их размеров на эрозию лопаток. Обсуждаются результаты наблюдений эрозии лопаток.

II. CASCADE AERODYNAMICS

The design of turbomachine components has depended, to the present day, on the data developed during the design of similar previous components. The designed model is usually tested, and adjustments are made until the design conditions are achieved. Such methods of trial and error, when applied to the design of advanced engines, are costly. The development of new theoretical techniques that could be directly applied to achieve the optimum design is therefore very important. Any significant improvement, on the present level of the engine performance, can only come with a thorough understanding of the three dimensional flow in turbomachines.

Secondary flow is a principal phenomena associated with the three dimensional flow in turbomachine compressors and turbines. It can be defined as the difference between the actual flow, and the flow which would occur on two dimensional axisymmetric and meridional stream surfaces. Of the many factors that contribute to the establishment of secondary flow, end wall boundary layer is the most important. The interaction of the hub and casing slow moving boundary layer flow with the main flow, which is turning through the blades, results in the secondary flow. This interaction is caused by the blade to blade pressure gradient, the radial pressure gradient, and the centrifugal forces. The blade end clearance and the relative motion between the blade end and the annulus walls are additional factors that contribute to the establishment of the secondary flow. In the work reported here, the secondary flow resulting from the inlet end wall boundary layers was investigated. The losses resulting from the viscous dissipation of the secondary flow velocities and the flow separation near the corner between the end wall and the blade suction surface were also included in this work.

Addition research work in cascade aerodynamics is reported in the following abstracts and references.

Sheng, James, "Sanders' Type Improved Shell Theory for General Orthogonal Coordinates," Project Themis Report No. 69-6, October 1969. (AD 617818)

Abstract

To achieve higher performance of the jet engines, the recent trend is to replace the solid cross section of the compressor blades and turbine buckets by thin shells. There is inconsistency in Love's first-approximation theory for these shells. J.L. Sanders, Jr. made remedies to these defects based on the lines-of-curvature coordinates. However, for blades and buckets, this kind of coordinate system is not appropriate for use and should be avoided. In analogous manner, equilibrium equations based on any arbitrary orthogonal curvilinear coordinate system for this purpose and for other purposes in general are established.

Wells, W. and Pavri, R.E., "Estimation of Supersonic Compressor Efficiency," Project Themis Report No. 70-15, October 1970. (AD 715005)

Abstract

A semi-empirical determination of the adiabatic efficiency of a supersonic compressor stage is obtained. The method utilizes the normal shock relations, a recently developed efficiency expression for subsonic cascade flow and experimental pressure loss data from supersonic cascades of various geometrical shapes. A correction factor is determined which makes it possible to convert the subsonic stage efficiency to the case of supersonic flow involving shock losses.

Hosny, W. and Tabakoff, W., "An Analysis of Losses and Secondary Flow in Turbine Cascades," Project Themis Report 71-23, December 1971. (AD 736853)

Abstract

A general review of different cascade losses is presented with emphasis on the secondary flow. An analytical analysis utilizing the Squire and Winter secondary flow analysis along with the triangle model for end wall boundary layer proposed by Taylor is detailed in an attempt to show the behavior of stream-lines near the end wall.

Reprinted from JOURNAL OF AIRCRAFT, Vol. 8, No. 4, April, 1971, pp. 282-284
Copyright, 1971, by the American Institute of Aeronautics and Astronautics, and reprinted of the copyright owner

Calculation of Supersonic Compressor Losses

W. R. WELLS* AND W. TABAKOFF†
University of Cincinnati, Cincinnati, Ohio

Nomenclature

A_I/A_{II} = $S/S + \tau$ = dump area ratio
c = chord
f = correction factor
q = $(P_t)_2/(P_t)_1$
h = radial blade length
M = Mach number
P = pressure
S = blade spacing
γ = ratio of specific heats
ϵ = $f_n - f$
θ = blade camber angle
η = stage adiabatic efficiency
σ = c/S = solidity
τ = trailing edge thickness
ξ = cascade stagger angle, measured from axial direction

Subscripts

0 = subsonic conditions
1 = conditions upstream ahead of cascade inlet
2 = condition's downstream of cascade
n = nominal conditions
s = conditions behind shock wave
t = *total conditions*

Received October 26, 1970. This work was sponsored under Project Themis Contract Number DAHC-04-69C-0016, U.S. Army Research Office—Durham.

* Associate Professor, Department of Aerospace Engineering. Member AIAA.

† Professor, Department of Aerospace Engineering. Associate Fellow AIAA.

I. Introduction

THE theoretical prediction of supersonic flow properties in a curved channel or between adjacent blades of a cascade presents a formidable task because of the complex nature of the interaction of shock waves, the vortex sheets, and the boundary layer.[1,2] A more reliable means of obtaining the performance of these compressors is experimental testing, as described in Refs. 3–7. For design purposes the idea of a simple analytical or semiempirical means of predicting the performance is attractive. Several papers along these lines have appeared recently in the literature, among them is the work of Balzer[8] in which the boundary-layer blockage effect and change in shock position are accounted for. A semiempirical method for predicting the performance of high reaction supersonic compressor blade sections is given by Boxer.[9]

The present analysis is an attempt to supplement the previous analyses with still another semiempirical performance estimation that is believed to be simpler in application and more widely applicable to a large family of cascade geometries. The success of the method is due, in part, to the manner in which the experimental data was used to determine the initial Mach number influence on a key parameter in the efficiency expression. This method extends a successful formulation developed for subsonic compressors by Losey and Tabakoff[10] to the cases of supersonic compressors in which shock losses are present and accounted for. Other effects, such as dump losses and errors in the shock structure model used, are partially accounted for through the use of the experimental data.

II. Mathematical Flow Model

The purpose of this analysis is the development of a simple realistic means to compute the adiabatic efficiency of a super-

Weidenhamer, Gerald H. and Sheng, James, "On the Free Vibration of a Long Helicoidal Membrane Strip," Project Themis Report No. 72-32, September 1972.

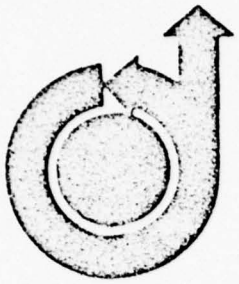
Abstract

The equations of motion for a shell are developed in general orthogonal coordinates using Hamilton's principle. They are modified to yield a set of improved equations which are then transformed to helicoidal shell coordinates. The membrane portion of these equations are solved for the natural frequencies and associated mode shapes for a uniformly twisted membrane boundary valve problem.

Hosny, W. and Tabakoff, W., "Numerical Solution for Three Dimensional Rotational Flow in Cascades," Report No. 76-48, June 1976.

Abstract

A theoretical analysis is established to deal with the inviscid three dimensional rotational flow which results from the secondary vorticity in turning channels. The equations of motion are written and manipulated in a special form to include the secondary vorticity as a variable among the other flow variables. The equations in their forms are not attractable to analytic solutions and therefore numerical solutions are adopted. The solution to the problem in general yields to the full picture of the three dimensional flow together with the different characteristics of the secondary vorticity distribution in the flow field. Since there was no limitation on the flow deflection angle, the channel radius of curvature or the flow inlet shear, the analysis is then recommended to be used to obtain the rotational flow in the channels of highly loaded turbomachinery cascades.



AIAA PAPER
NO. 76-372

AN ANALYSIS OF THE THREE DIMENSIONAL SECONDARY FLOW PROBLEM

by
W. HOSNY and W. TABAKOFF
University of Cincinnati
Cincinnati, Ohio

Abstract

An analysis of the three dimensional rotational flow due to the secondary vorticity will be presented. The equations of motion are written and manipulated in a special form to include the secondary vorticity as a variable among the other flow variables. The equations in their forms are not attractable to analytic solutions and therefore numerical solutions are adopted. The solution to the problem in general yields to the full picture of the three dimensional flow together with the different characteristics of the secondary vorticity distribution in the flow field.

AIAA 9th Fluid and Plasma Dynamics Conference
SAN DIEGO, CALIFORNIA/JULY 14-16, 1976

For permission to copy or republish, contact the American Institute of Aeronautics and Astronautics, 1290 Avenue of the Americas, New York, N.Y. 10019.

Sponsored by—
American Institute of Aeronautics and Astronautics (AIAA)
Society of Automotive Engineers (SAE)

AIAA PAPER NO. 76-617

AN EXPERIMENTAL INVESTIGATION ON LOSS REDUCTION IN SMALL GUIDE VANES

by
W. TABAKOFF and W. HOSNY
University of Cincinnati
Cincinnati, Ohio

Abstract

A new concept for reducing secondary losses in small turbine guide vanes is presented. An experimental investigation was conducted to determine the flow field and the reduction in the losses using this new concept.

AIAA/SAE 12th Propulsion Conference
PALO ALTO, CALIFORNIA/JULY 26-29, 1976

For permission to copy or republish, contact the American Institute of Aeronautics and Astronautics, 1290 Avenue of the Americas, New York, N.Y. 10019.

III. JET MIXING FLOW

Previous analyses have shown that at medium and low flight Mach numbers an exhaust gas jet of relatively small mass and high velocity is an inefficient method of producing thrust. This is due to the high energy losses at the nozzle exit and the poor use of the kinetic energy of the exhaust jet for the production of thrust. Two other important considerations for current engines are good fuel economy and a low noise level. In order to accomplish these objectives of maximizing thrust, lowering fuel consumption and decreasing noise, a mixed flow turbofan engine is the optimal candidate.

A high airflow is necessary to increase thrust while a high jet velocity will increase thrust but decrease propulsive efficiency. The turbofan engine produces a compromise between maximum airflow and maximum jet exhaust velocity. Theoretical studies have shown that the thrust of a turbofan engine can be increased by mixing the hot primary jet with the colder secondary airflow. Many investigators have confirmed these theories.

Therefore it is important to optimize the amount of mixing in a turbofan nozzle. In order to optimize this mixing, it is necessary to better understand the mechanics of turbulent jet mixing. Although an actual turbofan engine may have turbine exit swirl and an augmentor flame holder in the flow path, a simplified analysis, which is both reliable and accurate, is necessary before the more complex problem can be solved. To satisfy this need for a tenable turbulent mixing theory, we have made analytical studies. The theory proposed seems to provide a fairly simple, yet promising, approach to the turbulent mixing problem. The theory lacked an experimental investigation of the predicted results. Therefore, additional studies were performed to experimentally verify the accuracy of the theoretical analysis of turbulent jet mixing between two concentric air streams in a constant area duct. An eccentric mixing configuration was used as an additional experimental study. Non-isoenergetic turbulent jet mixing experiments were conducted, where the measured temperatures were compared with the mixing theory. These two non-isoenergetic mixing studies were believed to have resulted in a better understanding of turbulent mixing in a turbofan engine.

The research work performed in this area is presented in the following abstracts and references.

Khanna, K.K. and Tabakoff, W., "A Study of Non-Isoenergetic Turbulent Jet Mixing Between Compressible Subsonic Streams in Axi-Symmetric Constant Area Duct," Project Themis Report No. 69-1, August 1969. (AD-696439)

Abstract

The problem of turbulent mixing between two compressible streams in a constant area duct is analyzed. The velocity and temperature profiles have been worked in the main region of the mixing chamber for axi-symmetric case. Turbulent free jet experimental data is used, to obtain expressions for velocity and temperature as a function of the distance from the inlet to the mixing chamber. It is found that energy diffuses more rapidly than momentum for ducted mixing. The results of the theoretical analysis indicate increasing tendency towards segregation of the streams for decreasing initial velocity difference between the mixing streams. On the basis of the mixing chamber length required to achieve adequate degree of mixing, it is concluded that a corrugated rim primary nozzle is needed to accelerate the mixing rate by increasing the contact surface between the streams.

Tabakoff, W. and Hosny, W.M., "Theoretical and Experimental Investigations on the Mixing of Isoenergetic Confined Co-Axial Jets," Project Themis Report No. 70-10, June 1970. (AD 710284)

Abstract

A survey of experimental results for turbulent mixing between two compressible streams in a constant area duct is presented. The experimental results are for area ratios 3 and 7.16, for velocity ratios ranging from 0.4 to 2.8, and with velocity magnitudes varying from 200 ft/sec. to 950 ft/sec. A new constant of turbulence is recommended for the theoretical analysis of the confined jet mixing which uses the free jet mixing concepts. The regions in which this analysis could be used are delineated.

Tabakoff, W. and Hosny, W.M., "Theoretical and Experimental Jet Mixing of an Eccentric Primary Jet in a Constant Area Duct," Project Themis Report No. 70-11, July 1970. (AD 712333)

Abstract

The problem of turbulent mixing between two compressible streams in a constant duct area and with eccentric primary flow jet is analyzed. A theoretical analysis is presented for the

velocity profiles. It was found that these profiles with or without eccentricity are almost the same if they are referred to the axis of the primary flow jet. An experimental investigation for the case of eccentricity ratio (e/R) equal 0.25, given area ratios 3 and 7.16 and with a velocity ratio ranging from 1.2 to 2.9 was performed.

The theoretical analysis shows good agreement with the experimental results, especially for cases with high area ratios and with low velocity magnitudes.

Ghia, K.N., "Analytical Investigation of Confined Turbulent Mixing of Jets," Project Themis Report 72-24, June 1972.

Abstract

The turbulent mixing of confined axisymmetric incompressible jets was studied, using the turbulent boundary layer equations in terms of transformed von Mises variables, with semi-empirical formulations for the turbulent transport coefficients. The solution was obtained by the method of finite differences. Analytical predictions of the jet mixing flow fields were obtained for several configurations and covered a reasonably wide range of values of the problem parameters. The velocity ratio U_1/U_2 ranged from 0.438 to 5.0, the radius ratio R_1/R from 0.167 to 0.5 and the by-pass ratio λ varied from 1.626 to 16.37. The average flow velocity U_{avg} ranged from about 52 ft/sec to approximately 380 ft/sec.

Correlation of the analytical solutions with the corresponding available experimental data leads to the following principal results. For confined turbulent jets, the mixing becomes faster as the velocity ratio U_1/U_2 deviates from unity, or as the radius ratio R_1/R reduces below 0.5, or as the average flow velocity U_{avg} decreases. The initial mixing region manifests a rise in the local static pressure and an accompanying deceleration of the flow. The magnitude of this pressure rise decreases as the amount of mixing becomes smaller. A turbulent mixing model which satisfactorily predicts mean flow profiles for a reasonably broad spectrum of flow conditions has been formulated. The behavior of the correlating parameters appearing in this model with respect to the by-pass ratio λ has been determined and presented graphically. Analysis of additional data may lead to further refinements of this behavior.

Reprinted from JOURNAL OF AIRCRAFT, Vol. 9, No. 10, October, 1972, pp. 689-690
Copyright, 1972, by the American Institute of Aeronautics and Astronautics, and reprinted by permission of the copyright owner

Theoretical and Experimental Jet Mixing of an Eccentric Primary Jet in a Constant Area Duct

WIDEN TABAKOFF* AND WISHA M. HOSNY†
University of Cincinnati, Cincinnati, Ohio

Theme

THE problem of turbulent mixing between two incompressible streams in a constant area duct with an eccentric primary flow jet is analyzed. A theoretical analysis is presented for the velocity profiles. An experimental investigation was performed for an eccentricity ratio (e/R) of 0.25, area ratios of 3 and 7.16, and a velocity ratio ranging from 1.2 to 2.9.

The theoretical analysis shows good agreement with the experimental results, especially for the cases with high area ratios and with low velocity magnitudes.

Contents

In many industrial applications it is necessary to deal with a jet expanding eccentrically into a confined stream of fluid. Typical examples are: jet engines, by-pass ducts, arrangements for reduction of aerodynamic noise level and many others.

The theoretical analysis of confined coaxial jet mixing is presented by Tabakoff and Khanna.[1] In their analysis they assumed that velocity profiles in the main region are similar to those of a freejet for which the universal function of the nondimensional excess velocity is

$$\Delta U/\Delta U_m = (1 - \xi^{1.5})^2 \tag{1}$$

where $\xi = Y/r$; r = freejet radius; Y = distance from the axis in the transverse direction; ΔU = excess velocity at Y; and ΔU_m = excess velocity at axis.

In the case of coaxial mixing, the velocity profile is taken to be the portion of the nondimensional velocity profile which lies between $Y = R$ and $Y = -R$, where R is the radius of the duct. For eccentric mixing, the velocity profiles are assumed to be the eccentric portion of the freejet. These profiles are not symmetric with the mixing duct axis, however, they are symmetric with the primary flow axis. Figure 1 shows the physical model with the corresponding notation.

In the case of a freejet, it was concluded that, owing to the similarity in velocity profiles, l/r will remain constant (l is the mixing length). Moreover, $dr/dx = \text{const} = C$ or $r = Cx$. Now, $l = (r)\,(\text{const}) = (Cx)(\text{const})$; from this last relation it is clear that the mixing length depends on C. Since the mixing length is the main parameter which specifies the degree of turbulence, the constant C is named the "turbulence constant." This constant was found to be equal to, 0.2 in the case of free jet mixing and 0.7 for confined jet mixing.[3] For confined eccentric mixing, the turbulence constant C is taken to be equal to 0.7.

Received January 12, 1972; synoptic received April 14, 1972. Full paper available from National Technical Information Service, Springfield, Va., 22151, as N 71-10010 at the standard price (available upon request). This work was sponsored under Contract Number DAHC04-69C-0016, U.S. Army Research Office—Durham.

Index category: Jets, Wakes, and Viscid Flow Interactions.

* Professor, Department of Aerospace Engineering. Associate Fellow AIAA.

† Graduate Research Assistant, Department of Aerospace Engineering.

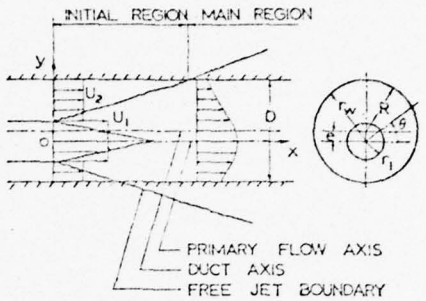


Fig. 1 Confined jet mixing with eccentricity.

The fundamental equations governing the flow of isoenergetic eccentric jet mixing are developed as follows. The continuity equation for one-dimensional flow for a control volume beginning at an arbitrary cross section at a distance x, and ending at the section where a uniform flow is attained, may be written as follows:

$$\int_0^A \rho U dA = \rho_3 U_3 A_3 \tag{2}$$

where the subscript, 3, refers to the uniform flow properties after complete mixing.

In order to integrate Eq. (2), using the universal excess velocity profile for turbulent freejets, as shown by Abramovich,[2] one subtracts the quantity

$$\rho_3 U_2 A_3 = \int_0^A \rho U_2 dA$$

from both sides of Eq. (2), and using polar coordinates the following equation is obtained

$$\Delta U_3 \rho_3 \pi R^2 = \int_0^{2\pi} \int_0^{y_w} \rho \Delta U dy d\theta \tag{3}$$

where y_w is the wall radius measured from the primary flow axis.

Dividing each side of Eq. (3) by the quantity $\Delta U_m \pi R^2$, introducing $\xi = y/r$; $\xi_k = R/r$; $\xi_w = y_w/r$, and substituting Eq. (1) into Eq. (3), for the incompressible case, the following expression is obtained:

$$\frac{\Delta U_3}{\Delta U_m} = \frac{1}{\pi \xi_k^2} \int_0^{2\pi} \left(\frac{\xi_w^2}{2} + \frac{\xi_w^5}{5} - \frac{2\xi_w^{3.5}}{3.5} \right) d\theta \tag{4}$$

Upon substitution of $\xi_w \simeq \xi_k\,(1 + \bar{e}\cos\theta)$, where $\bar{e} = e/R$ one obtains

$$\Delta U_3/\Delta U_m = A_1(\xi_k) + \bar{e}^2 B_1(\xi_k, \bar{e}) \tag{5}$$

where $B_1(\xi_k, \bar{e}) = 0.5 - 2.5\xi_k^{1.5} + \xi_k^3(2 + 0.75\bar{e}^2)$ and $A_1(\xi_k) = 1 + 0.4\xi_k^3 - 1.143\xi_k^{1.5}$.

Blasenak, J.H. and Tabakoff, W., "Study of Non-Isoenergetic Turbulent Jet Mixing in a Constant Area Duct," Report No. 76-49, September 1976.

Abstract

A study of non-isoenergetic turbulent jet mixing between two streams has been conducted. Using a previously derived theoretical analysis for ducted mixing, an experimental investigation was performed to verify this theory and to determine the non-isoenergetic turbulent jet mixing characteristics in a constant area duct. Temperature profiles were measured at several axial locations in the duct for both a concentric and an eccentric configuration. It was determined that the theoretical and experimental temperature profiles agreed fairly well for both cases, although the concentric case showed better agreement than the eccentric case. It was also determined that a new constant of turbulence in the initial region was needed for non-isoenergetic mixing, mixing is generally more rapid than the theory predicted, the initial temperature difference between the two streams did not have much effect on the rate of mixing and a higher area ratio produced better agreement between the theory and the experimental data. It was concluded that the theory was good for a fairly simplified analysis.

IV. HIGH TEMPERATURE MATERIALS

The high temperature requirements of future advanced propulsion systems necessitates the careful consideration of materials to be used in turbine blades, nozzles, and other high temperature components. The absolute upper limit of operating temperatures of metal components of an engine is dictated by the solidus temperatures of the alloys used. Through the phenomenon of dispersion hardening the design engineer can develop alloys that have good strength and creep resistance at appreciable fractions of the solidus temperature. In addition to strength at higher temperatures, suitable jet engine alloys must be resistant to attack by oxidation and corrosion from the engine gases. Superalloys based on nickel with major amounts of chromium and cobalt, and lesser amounts of titanium, aluminum, and molybdenum give workable alloys of good strength at high temperatures, and good corrosion and oxidation resistance. Through constant efforts the operating temperature of alloys has been slowly increased over the years. Columbium metal is a potential high-temperature construction material for gas-turbine buckets, because it retains a useful degree of strength at higher temperatures where conventional creep resisting alloys prove to be inadequate. The major obstacle in using columbium in pure or unprotected form at high temperatures is its poor oxidation resistance in oxygen-bearing environments. Alloying with other elements has already proved a partial rectification of this disadvantage and although considerable knowledge about the oxidation behavior of these alloys has been accumulated, both from basic and engineering studies, reaction mechanism(s) are not clearly understood. Principal hinderance in interpreting the alloy oxidation is the insufficient information on the oxidation of pure columbium because of complexity of the oxidation rate-temperature relationship.

The main purpose of this study was to investigate in detail the oxidation behavior of Cb-10 at .%W alloy. To supplement the understanding of the oxidation mode of Cb-10 at .%W alloy, attempt has also been made to resolve the unusual oxidation rate-temperature dependence of the columbium metal. The experimental work was essentially confined to oxidation over a temperature range of 900-1200°C in dry oxygen at 100-760 torr.

The research work performed in this area is reported in the following abstracts and references.

Sikka, V.K. and Rosa, C.J., "Determination of Oxygen Diffusion Coefficients in Tungsten Oxide," Project Themis Report No. 70-13, August 1970. (AD 715006)

Abstract

"Interruption Kinetic Technique" experiments carried out on tungsten oxide films formed on tungsten yield the following experssion for the diffusion coefficient of oxygen anion vacancies:

$$D = 6.83 \times 10^{-2} \exp(-29{,}889/RT)$$

for the temperature range 568° - 908°C.

Calculations are made for the concentration of oxygen vacancies, for their free energy of formation and for ionic conductivity in tungsten oxide. Deviations from stoichiometry in the oxide are plotted against temperature on the available W-O phase diagram.

Sikka, V.K. and Rosa, C.J., "High Temperature Oxidation of Columbium and CB-10 At .%W Alloy," Project Themis Report No. 70-14, September 1970. (AD 715007)

Abstract

Oxidation kinetics of Cb and Cb-10 at .%W alloy over the temperature 900-1200°C, and oxygen pressures of 100-760 torr were determined using highly sensitive Cahn Electrobalance. The products of oxidation were examined metallographically and by x-ray diffraction methods.

Activation energy for the parabolic rate constant has confirmed a diffusion controlled growth process for the oxidation of Cb-10 at .%W alloy. The overall oxidation kinetics of Cb-10 at .% alloy in the temperature range 1100-1200°C is best described by the paralinear model.

Possible explanations for the abnormal oxidation rate-temperature relationship for the oxidation of Cb and Cb-10 at .%W alloy have been discussed.

Reprinted from JOURNAL OF APPLIED PHYSICS, Vol. 41, No. 6, 2439–2442, May 1970
Copyright 1970 by the American Institute of Physics
Printed in U. S. A.

Theoretical Approach to a Technique for Measuring Thermal Conductivity at High Temperature

JOSEPH VARDI* AND ROBERT LEMLICH

Department of Chemical and Nuclear Engineering, University of Cincinnati, Cincinnati, Ohio 45221

(Received 22 August 1969; in final form 12 January 1970)

A theoretical solution by finite difference methods is presented for the steady-state temperature field within a solid uniform right circular cylinder which is heated by a uniform source along its curved surface and which simultaneously loses heat by fourth-power radiation from all its surfaces to much cooler surroundings. The results of the solution are more accurate than the semi-empirical relationship of Hoch *et al.*, and so can be applied with greater reliability to their new metal-envelope induction technique for determining thermal conductivities of nonmetals at high temperatures.

INTRODUCTION

Several years ago, Hoch *et al.*[1] introduced an experimentally simple method for indirectly measuring the thermal conductivity of certain materials at high temperature. In this method, a solid right circular cylinder of the material is placed coaxially within an induction coil and heated electromagnetically in vacuum. At steady state, the radial temperature profile at one of the flat faces of the cylinder is measured with an optical pyrometer. Then, combining these measurements with the emissivity and the dimensions of the cylinder, the thermal conductivity is calculated.

The method has been used for Mo,[1,2] Va,[1] and graphite.[3] Recently, the method was extended to materials such as Al_2O_3 by covering the curved surface of the cylinder with a metal envelope.[4]

For either technique, the calculation of k is based on a highly simplified thermal model.[1] It assumes that the curved surface is isothermal, and that the temperature at the flat surfaces varies quadratically with radial distance. These assumptions have been shown to be inaccurate when applied to the original technique of direct induction.[5] The demonstration was by means of a theoretical solution of the electric field, followed by a theoretical solution for the thermal field. However, based on these more accurate results, a simple and more reliable method for calculating k was presented.[5,6]

The objective of the present paper is to examine the validity of Hoch's assumptions as applied to the later envelope induction technique and to improve on them as necessary. This examination will be conducted by employing more realistic assumptions, and then solving for the temperature field numerically via finite differences.

NOMENCLATURE

- a radius of cylinder
- k thermal conductivity
- L half-length of cylinder
- q'' rate of heat input, per unit surface
- r radial distance
- t absolute temperature
- t_a absolute temperature along the circular edge ($r=a$, $z=L$)
- t_c absolute temperature at the center of the flat face ($r=0$, $z=L$)
- t_m absolute temperature along the center of the circular face ($r=a$, $z=0$)
- z axial distance
- σ total emissivity
- ϵ Stefan–Boltzmann radiation constant

THEORY

The cylinder is shown in Fig. 1. By virtue of angular symmetry, Fourier's conduction equation for a material of isotropic constant k at steady state with no *volumetric* heat sources can be written as follows:

$$(\partial^2 t/\partial r^2)+r^{-1}(\partial t/\partial r)+(\partial^2 t/\partial z^2)=0. \tag{1}$$

Equation (1) is the governing differential equation.

Symmetry also gives Eqs. (2), (3) as two of the boundary conditions.

$$\partial t(0, z)/\partial r=0 \tag{2}$$

$$\partial t(r, 0)/\partial z=0. \tag{3}$$

For convenience, attention can now be restricted to the first quadrant in Fig. 1.

The cylinder is assumed to be much hotter than its surroundings, so that thermal radiation from the surroundings to the cylinder is negligible compared to that from the cylinder. A conduction–radiation balance at a differential area element in the flat surface yields Eq. (4) as the third boundary condition.

$$\partial t(r, L)/\partial z=-(\epsilon\sigma/k)t^4(r, L) \tag{4}$$

Due to the vacuum, there is assumed to be no conduction between the cylinder and its surroundings.

The heat which is furnished by the metal envelope

Rosa, C.J. and Chen, G.C., "High Temperature Oxidation of Columbium-10 at .% Chromium Alloy," Project Themis Report No. 71-19, February 1971.

Abstract

Oxidation kinetics of Cb-10 at .% Cr alloy were studied at three temperature levels: 900, 1000 and 1100°C under oxygen pressures ranging from 100-760 torr. At 1000 and 1100°C the oxidation kinetics are lower than for pure columbium. This improvement of oxidation resistance of the alloy is mainly attributed to the modifying effects of Cr cations in α-Cb_2-O_5 by stabilizing the compact, inner, oxide layer. The reaction mechanism is essentially inward oxygen diffusion across the compact inner scale. At 900°C the oxidation kinetics of the alloy are higher when compared with pure columbium and follow a linear rate. This decrease in oxidation resistance is related to polymorphic transformation in Cb_2O_5 which leads to fracture of the highly distorted α-Cb_2-O_5. The reaction mechanism at 900°C could be associated with oxygen absorption. X-ray diffraction patterns, electron microprobe analyses and the observed morphologies of the oxide scale support the proposed oxidation mechanisms of the alloy.

Vieth, D.L. and Pool, M.J., "Design and Construction of a High Temperature Liquid Metal Solution Calorimeter," Project Themis Report No. 72-30, October 1972.

Abstract

The design consideration and calibration of a new single well liquid metal solution calorimeter for use at high temperature is described. The calorimeter has an operational temperature range of 500°C to 1300°C in a vacuum and was designed primarily to operate 1200°C using liquid uranium as a solvent. Calibration of the instrument indicates that the energy equivalent ranged from 0.8100 joules/μV (.1950 cal/μV) to 0.6950 joules/μV (0.160 cal/μV). Calibration was based on the heat content, between room temperature and the calorimeter temperature, of weighed samples of pure uranium. The partial heat of solution of vanadium, chromium, iron, and nickel in uranium at 1150°C were determined.

Birla, N.C. and Hoch, M., "The Solutioning and Aging Reactions and Their Effect on Hardness and Microstructure in Some Columbium Base Alloys," Project Themis Report No. 72-31, September 1972.

Abstract

A study was made to investigate the solutioning and aging reactions and their effects on microstructure and hardness in twenty-four experimental columbium-base alloys, involving Cb-C, Cb-Si-C, Cb-15Hf-C, Cb-15Hf-C-Si, Cb-33Hf-C and Cb-33Hf-C-Si systems. After selected heat treatments, the microstructural changes were determined by electron microscopy and the phases were extracted and identified by x-ray diffraction. This study involved more than 72 extractions and their analysis by x-ray diffraction, more than 135 solution treatments in an induction furnace, about 75 heat treatments for aging in a Brew furnace and more than 550 samples for polishing, metallography and microhardness measurements.

The aging reactions in the Cb-15Hf-C, Cb-15Hf-C-Si, and Cb-33Hf-C-Si alloys are characterized by precipitation of the complex monocarbide (Cb, Hf) (C, O, N) phase.

Replacement of some of the carbon by silicon in the Cb-15Hf-1C alloys raises the overaging temperature from 800°C to 1000°C and leads to strength retention for longer times than the alloys which contain carbon only. However, the alloys in Cb-33Hf-C system, did not show precipitation hardening due to the precipitation of coarse Cb_2C particles during aging process.

V. GAS TURBINE BLADES AND COMBUSTION CHAMBERS HEAT TRANSFER

The efficiency of gas-turbine engines is highly dependent on the turbine inlet temperature of the working fluid. If the present achievable inlet gas temperature is doubled, this means theoretically, that a turbine can double its power per pound of working fluid. As a consequence, the turbine's bulk and weight can be reduced. Because of today's metallurgical limitations of temperature with respect to the blade load force, production engines are limited to 2200°F. With this temperature limitation on the blades, the only way to increase the turbine inlet gas temperature is to use some method to cool the turbine blades. The requirement for this cooling is likely to remain for a long time to come, even with the new materials now being developed for turbine blade use.

The reported basic research program was to investigate the heat transfer phenomena associated with internally cooled blades, namely by jets impinging and transpiration. A jet impinging on a surface an be utilized to achieve high performance heat transfer configurations. The surface may be cooled by the jet flow to obtain high local heat-transfer directly under the jet, while good overall heat transfer can also be obtained when the surface area to be cooled is appreciably larger than the jet. The transpiration cooling uses cooler fluid which is injected through the porous wall into the boundary layer of the blades to protect the external hot surfaces. Such type of cooling is well known to have high cooling efficiency and excellent shielding from severe thermal environment. Critical temperatures and thermal environments are encountered in combustion chambers and over gas turbine blades. Transpiration cooling perhaps finds its most important applications in keeping these components at an acceptable temperature. To insure efficient performance over a reasonable lifetime makes the problem of blade and combustion chambers cooling absolutely necessary.

The research work performed in the above mentioned areas is reported in the following abstracts and references.

Clevenger, W. and Tabakoff, W., "Investigation of the Heat Transfer Characteristics of a Two-Dimensional Jet Impinging on a Semi-Cylinder," Project Themis Report No. 69-5, October 1969. (AD 697165)

Abstract

Boundary layer methods are used to find the heat transfer rate caused by a two-dimensional jet impinging on the inside surface of a semi-cylinder. Because of a wider pressure distribution on the semi-cylinder, the solution predicts a delayed transition from a laminar to a turbulent boundary layer. This delayed transition causes the average heat transfer from the semi-cylinder to be less than the average heat transfer from the flat plate. Experimental data supports the theoretically predicted heat transfer rate from the stagnation area of the semi-cylindrical plate and indicates heat transfer rates that are less than those predicted by theory in the other regions of the flow field.

Tsuei, Y.G. and Allis, E.W., "Laminar Jet Mixing with Initial and Boundary Effects," Project Themis Report No. 70-9, April 1970. (AD 704983)

Abstract

The objectives of this report are to investigate the flow field and the wall shear stress for laminar tangential injection. The effects of the boundary and initial velocity profiles under zero pressure gradient are considered. A linearized approximation with a coefficient chosen from the parameters of the boundary layer has been used to analyze the slot injection. It is found that wall shear stress is almost independent of the ratio of jet velocity to free stream velocity and the ratio of slot height to initial outside boundary layer thickness up to a downstream distance equal to 0.036 Re_L times the slot height. It is also found that the effectiveness of the jet mixing is greater for large ratios of slot height to initial outside boundary layer thickness than for small ratios. The asymptotic solution of the velocity distribution far downstream is compared with the Blasius profile and good agreement is found

The results can be used for film cooling to estimate the spacing between the injection slots for effective protection.

Ravuri, R. and Tabakoff, W., "Heat Transfer Characteristics of a Row of Air Jets Impinging on the Inside of a Semi-Circular Cylinder," Project Themis Report No. 71-18, January 1971. (AD 717111)

Abstract

Experimental results of average heat transfer coefficients for a row of air jets impinging on the inside of a semi-cylinder are presented. The variation of the average Nusselt Number with the wide range of Reynolds Numbers are found. Two semi-cylinder concave surfaces with different diameters, four different impinging jet configurations and variable impinging heights were investigated. The effect of introducing solid particles into the cooling air on the heat transfer performance was studied.

Tabakoff, W., Pavri, R. and Clevenger, W., "Heat Transfer by a Multiple Array of Round Jets Impinging Perpendicular to a Concave Surface," Project Themis Report No. 71-20, June 1971. (AD 728489)

Abstract

An experimental investigation was undertaken to study jet impingement cooling of a semi-cylindrical concave surface in a semi-enclosed environment. The investigation showed that it was possible to obtain a correlating formula for the various parameters involved. The results are compared with those of Kercher and Tabakoff who carried out a similar investigation using a flat plate.

Pavri, R. and Tabakoff, W., "An Analytical Solution of Wall Temperature Distribution for Transpiration and Local Mass Injection Over a Flat Plate," Project Themis Report No. 72-25, March 1972.

Abstract

This analysis describes an analytical solution of the non-similar laminar boundary layer with continuous or local injection and variable wall temperature. The solution is for a flat plate with zero pressure gradient, however, it can be modified for variable pressure. The method consists of transforming the partial differential equations for momentum and enthalpy and then solving the transformed equations by assuming polynomial velocity and temperature profiles.

The analysis is presented in two parts. The first part presents the solution for continuous injection (transpiration cooling) while the second part involves discontinuous boundary conditions due to local injection.

Pavri, R.E. and Tabakoff, W., "An Analytical Solution of the Compressible Laminar Momentum and Thermal Boundary Layers with Pressure Gradient and Continuous Mass Injection," Project Themis Report No. 72-29, September 1972. (AD 751804)

Abstract

The study describes an analytical solution of the nonsimilar laminar boundary layer with pressure gradient, variable wall temperature and continuous injection. The method consists of transforming the partial differential equations for momentum and enthalpy and then solving the transformed equations by assuming polynomial stream function and enthalpy profiles. Solutions obtained show very good agreement with exact numerical results.

The solutions are obtained for flows over wedges as well as at the two-dimensional stagnation point and over curved surfaces of a two-dimensional body in crossflow. The results of the study show that the boundary layer is very strongly effected by the injection mass flow rate.

W. TABAKOFF
Professor. Mem. ASME

W. CLEVENGER
Instructor.

Department of Aerospace Engineering, University of Cincinnati, Cincinnati, Ohio

Gas Turbine Blade Heat Transfer Augmentation by Impingement of Air Jets Having Various Configurations[1]

An experimental investigation of heat transfer characteristics for various configurations of air jets impinging on the leading edge inner surface of the blade wall is presented. Three configurations were investigated, namely a slot jet, a round jet row and an array of round jets. The effect on the heat transfer coefficient of injecting solid particles into the air flow is considered. The study treats an important class of turbine blade cooling for which small cooling mass flow rates are of interest. The experimental facility and procedures are described in detail. A theoretical technique is introduced for predicting the heat transfer in the case of the slot jet configuration. The results are compared to experimental data.

Introduction

THE efficiency of gas-turbine engines is highly dependent on the turbine inlet temperature of the working fluid. In today's operating gas turbine the inlet gas temperature is below the adiabatic flame temperature of hydrocarbon fuels. If the present inlet gas temperature of 2,000 deg F is doubled, theoretically the turbine can double its power per pound of working fluid. As a consequence, the turbine's bulk and weight can be reduced. Because of today's metallurgical limitations of temperature with respect to the blade load force, production engines are limited to 2,300 deg F. With this temperature limitation on the blades, the only way to increase the turbine inlet gas temperature is to use some method to cool the turbine blade. In order to design and develop an effective cooling means, a knowledge of the blade heat transfer is essential.

The purpose of this project was to compare the effectiveness of three different systems of air jets impinging on the inside surface of a half-circular cylinder. Such a configuration would be similar to that of a turbine blade cooled by gas jets impinging on its in-

[1] This work was sponsored under Project Themis Contract Number DAHC04-69C-0016, Army Research Office, Durham.

Contributed by the Gas Turbine Division and presented at the Gas Turbine Conference and Products Show, Houston, Texas, March 28–April 1, 1971, of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS. Manuscript received at ASME Headquarters, December 21, 1970. Paper No. 71-GT-9.

Nomenclature

A_p = surface area of the heater plates, in.^{2}

B = constant used to correctly position the velocity decay of a turbulent wall jet, in.

C_0 = constant used in the velocity decay equation of a turbulent wall jet

C_p = specific heat at constant pressure, Btu/lbm deg R

D = round jet diameter, in.

H = height of the nozzle exit above the plate, in.

h = local heat transfer coefficient, Btu/hr ft^2 deg F

$\bar{h}$ = average height transfer coefficient, Btu/hr ft^2 deg F

k = thermal conductivity of air, Btu/hr ft deg F

L = length of the semicylinder model and the length of the slot jets, in.

m = exponent used with the Prandtl Number $m = 1/3$

$\dot{m}$ = total air mass flow from all jets, lbm/hr

$N = \dfrac{n+1}{2}$

n = exponent used in the velocity and temperature profiles of the wall jet

Nu = Nusselt number based on plate length $\mathrm{Nu} = \bar{h}L/k$

$\mathrm{Nu}_{.5}$ = local Nusselt Number based on the jet half width at half depth $\mathrm{Nu}_{.5} = h\zeta/k$

$P(x^*)$ = pressure at any point x^* along the plate, psia

$P(0)$ = pressure at the point $x^* = 0$, the stagnation pressure; psia

P_{amb} = ambient pressure, psia

Pr = Prandtl number $\mathrm{Pr} = \mu C_p/k$

Q_G = gross power input to the plate, Btu/hr

Q_L = power losses through the insulated box supporting the model, Btu/hr

q_w'' = theoretical heat transfer rate from the plate, Btu/hr ft^2

Re = Reynolds number $\mathrm{Re} = \dot{m}/\mu L$

St = Stanton number $\mathrm{St} = h/C_p U_\delta \rho$

s = spacing between the center line of the round jets, in.

T_w = plate wall temperature, deg F

(Continued on next page)

72-HT-57

an ASME publication

The Society shall not be responsible for statements or opinions advanced in papers or in discussion at meetings of the Society or of its Divisions or Sections, or printed in its publications. *Discussion is printed only if the paper is published in an ASME journal or Proceedings.*
Released for general publication upon presentation.
Full credit should be given to ASME, the Professional Division, and the author (s).

$3.00 PER COPY
$1.00 TO ASME MEMBERS

An Analytical Solution of Wall-Temperature Distribution for Transpiration and Local Mass Injection over a Flat Plate

R. PAVRI

Graduate Research Assistant

W. TABAKOFF

Department of Aerospace Engineering,
University of Cincinnati,
Cincinnati, Ohio
Mem. ASME

This analysis describes an analytical solution of the nonsimilar laminar boundary layer with continuous or local injection and variable wall temperature. The solution is for a flat plate with zero pressure gradient, however, it can be modified for variable pressure. The method consists of transforming the partial differential equations for momentum and enthalpy and then solving the transformed equations by assuming polynomial velocity and temperature profiles. The analysis is presented in two parts. The first part presents the solution for continuous injection (transpiration cooling;) the second part involves discontinuous boundary conditions due to local injection.

Contributed by the Heat Transfer Division of The American Society of Mechanical Engineers for presentation at the AIChE-ASME Heat Transfer Conference, Denver, Colo., August 6-9, 1972. Manuscript received at ASME Headquarters, May 4, 1972.

Copies will be available until June 1, 1973.

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

DEUTSCHE GESELLSCHAFT FÜR LUFT - UND RAUMFAHRT E.V. (DGLR)

OPTIMUM HEAT TRANSFER CHARACTERISTICS OF SEMI-CIRCULAR SURFACES COOLED BY AIR IMPINGEMENT FROM AIRJET ARRAYS AND ROW OF AIR JET NOZZLES*

by

W. TABAKOFF and R. STEENECK
Department of Aerospace Engineering
University of Cincinnati
Cincinnati, Ohio U.S.A.

The trend in today's gas turbine engine design is toward higher turbine inlet temperature. As a consequence the turbine's bulk and weight can be reduced. Because of today's metalurgical limitation of temperature with respect to the blade load forces, the only way to increase the inlet gas temperature is to use some method to cool the turbine blade. The authors investigated different systems of impinging air jets cooling the inside surface of a half-circular cylinder. Such a configuration will be similar to that of a turbine blade cooled by gas jets impinging on the leading edge inside surface. This paper is a continuation of the above mentioned work, the purpose of which is to derive general equations or scaling factors for optimum cooling of semi-cylinder surfaces of any diameter by air impingement from air jet arrays and nozzle rows. In order to obtain experimental data 5 inch, 2.5 inch, 1.25 inch and 0.5 inch diameter semi-cylindrical models were investigated.

DGLR - JAHRESTAGUNG 1972

Berlin, Germany

October 3-6, 1972

PAPER NO. 72-61

*This work was sponsored under Contract Number DAHC04-69C-0016, U.S. Army Research Office, Durham.

Ravuri, R. and Tabakoff, W., "A Numerical Solution for the Heat Transfer Between an Axi-Symmetric Air Jet and a Heated Plate," Report No. 73-38, September 1973.

Abstract

Due to their characteristic high rate of heat transfer, impinging air jets are being used extensively in industry. A numerical scheme is devised to predict the heat transfer between an axi-symmetric air jet and a heated plate, neglecting the effect of curvature, compressibility, and turbulence. The momentum and continuity equations are transformed to vorticity and stream function equations and are then solved by iterative successive substitution techniques to determine the flow field and subsequently the temperature distribution in the fluid. An attempt is made to predict the average heat transfer coefficient of a row of axi-symmetric air jets impinging on a heated plate by assuming that each jet cools a fraction of the plate without any interference from the neighboring jets.

Nilson, R.H. and Tsuei, Y.G., "A Numerical Method for Boundary Layer Equations," Report No. 74-41, March 1974.

Abstract

A general marching procedure for numerical solution of the two-dimensional boundary layer equations is presented. Although the present method traces its origin to that of Patankar and Spalding, major modifications have been undertaken. The method is tested for a variety of laminar flow configurations including wedge flows, incompressible flow on a flat plate at various Eckert numbers, compressible flow on an adiabatic flat plate at various Mach numbers, flow over a cylinder, uniform suction on a flat plate, and Howarth flow. The results show excellent agreement with solutions by other methods, even when as few as fifteen grid lines are used. A separate report entitled "Film Cooling by Oblique Slot Injection" presents results for a variety of injection configurations which show the effects of coolant mass flow rate, injection angle, boundary layer thickness, slot width, and multiple slots.

Reprinted from AIAA JOURNAL, Vol. 12, No. 5, May 1974, pp 727-729
Copyright, 1974, by the American Institute of Aeronautics and Astronautics, and reprinted by permission of the copyright owner

Film Cooling by Oblique Slot Injection

R. H. NILSON* AND Y. G. TSUEI†
University of Cincinnati, Cincinnati, Ohio

Introduction

WALL cooling by boundary-layer injection or transpiration is used in engineering applications.[1] The present investigation concerns the film cooling effectiveness of oblique injection from the wall into a compressible laminar boundary layer through single or multiple slots. Numerical solutions of the boundary-layer equations are obtained by a finite-difference method which has been extensively tested and found to be accurate, versatile, and very stable. Film cooling effectiveness is presented for a wide variety of injection configurations so that the effects of coolant mass flow, injection angle, boundary-layer thickness, slot width, and the presence of upstream cooling slots can be investigated. The results are interesting, and conclusions heretofore unreported are drawn regarding selection of film cooling parameters.

Received November 6, 1973; revision received December 26, 1973. This work was partially supported by U.S. Army Research Office—Durham, under Contract DAHC 04-69-C-0016.

Index category: Boundary Layers and Convective Heat Transfer—Laminar.

* Instructor and Graduate Student, Department of Mechanical Engineering.

† Associate Professor, Department of Mechanical Engineering. Member AIAA.

Tsuei, Y.G. and Nilson, R.H., "Influence of Injection Angle on Film Cooling Effectiveness in Laminar Compressible Flow," Report No. 75-45, May 1975.

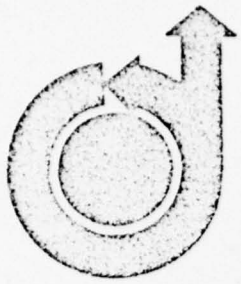
Abstract

Wall cooling effectiveness is investigated for tangential, inclined, and normal injection of coolant through single or multiple wall slots into a laminar compressible boundary layer. Numerical solutions of the boundary layer equations are obtained by a finite difference method which has been extensively tested and found to be accurate, versatile, and stable. A grid control procedure which maintains a constant flow rate between grid lines is found to be well suited to the present injection calculations wherein the boundary layer growth in the slot is as much as hundred-fold and the longitudinal component of the injection velocity is in some cases as large as the free stream velocity. Film cooling effectiveness is reported for a variety of injection configurations so that the effects of injection angle, coolant mass flow rate, Mach number, upstream boundary layer thickness, slot width, and the presence of upstream cooling slots can be investigated.

McFarland, E. and Tabakoff, W., "Study of Impingement Heat Transfer With Rough Surfaces," Report No. 75-46, August 1975.

Abstract

The effects of surface roughness on impingement cooling of a gas turbine blade were determined experimentally. The study was conducted in a heated flow two-dimensional cascade tunnel. Results showed that roughening of the impingement surfaces decreased the cooling effectiveness from that of a smooth surface in an optimal cooling configuration. However, the roughened surfaces increased the cooling effectiveness over that of a smooth surface in a nonoptimal configuration.



AIAA PAPER
75-162

FILM COOLING EFFECTIVENESS FOR COMBUSTION CHAMBERS

by
W. TABAKOFF and R. RAVURI
University of Cincinnati
Cincinnati, Ohio

Abstract

The most effective technique for cooling combustion chamber walls is accomplished by the utilization of a continuous slot jet, which produces a thin sheet of cool air on the wall as a protection against convective heating by the hot gases. An alternative method of cooling is the injection of cold gas through discrete louvers cut into the wall. When the louvers are of small width, the problem becomes similar to film cooling by a row of discrete jets. In this investigation the cooling effectiveness of a row of discrete jets is compared experimentally with that of a continuous slot jet having an equivalent coolant flow area.

AIAA 13th Aerospace Sciences Meeting

PASADENA, CALIF./JANUARY 20-22, 1975

For permission to copy or republish, contact the American Institute of Aeronautics and Astronautics, 1290 Avenue of the Americas, New York, N.Y. 10019.

Reprinted from AIAA JOURNAL, Vol. 13, No. 9, September 1975, pp. 1199-1202
Copyright, 1975, by the American Institute of Aeronautics and Astronautics, and reprinted by permission of the copyright owner

Film Cooling by Oblique Slot Injection in High-Speed Laminar Flow

R. H. Nilson* and Y. G. Tsuei†
University of Cincinnati, Cincinnati, Ohio

Wall-cooling effectiveness is investigated for oblique injection of coolant through single or multiple wall slots into a high-speed laminar compressible boundary layer by numerical solutions of the boundary-layer equations. A grid control procedure which maintains a constant flow rate between grid lines is found to be well suited to the present injection calculations wherein the boundary-layer growth in the slot is as much as a hundred-fold and the logitudinal component of the injection velocity is in some cases as large as the freestream velocity. Film-cooling effectiveness is reported for a variety of injection configurations so that the effects of coolant mass flow rate, injection angle, upstream boundary-layer thickness, slot width, and the presence of upstream cooling slots can be investigated. For the coolant mass flow rates considered, normal injection provides better cooling than tangential injection, particularly when frictional heating effects caused by tangential injection become a dominant consideration. However, the excessive boundary-layer growth which accompanies normal injection may reduce aerodynamic performance, thus making inclined injection a desirable compromise.

Nomenclature

C_p = constant pressure specific heat
L = length of leading edge upstream of injection slot
M_∞ = freestream Mach number
$\dot{m}_c$ = coolant mass flow rate
Pr = Prandtl number (taken as 1.0 in present study)
s = slot width
T = temperature
T_{adw} = adiabatic wall temperature upstream of injection slot
T_c = coolant temperature
T_w = wall temperature
T_∞ = freestream temperature
U_j = longitudinal component of injection velocity
U_∞ = freestream velocity
u = longitudinal velocity component
V_j = transverse component of injection velocity
x,y = coordinates parallel and normal to main flow
α = injection angle as shown in Fig. 3.
δ^* = displacement thickness defined in Eq. (6)
η = cooling effectiveness defined in Eq. (5)
ψ = stream function defined in Eq. (2)
ν = kinematic viscosity
ρ = density
τ_w = shearing stress at wall
ω = dimensionless stream function defined in Eq. (4)

Introduction

WALL cooling or reduction in heat transfer by boundary-layer injection is used in engineering applications.[1] Although injection geometry varies, all configurations are included in either of two categories: 1) parallel injection in which a layer of coolant enters beneath the boundary layer through an offset wall; and 2) transverse injection where coolant is blown up into the boundary layer through a slot or holes in the wall. In either of these cases the angle between the coolant flow and the primary flow may vary from tangential to normal. Most previous investigations are confined either to parallel injection which is tangent to the primary flow or to transverse injection which is normal to the primary flow. The present study considers the film-cooling effectiveness of transverse injection at various injection angles through single or multiple wall slots into a compressible boundary layer. Results have been reported for low-speed laminar flow ($U=50$ m/sec).[2] High-speed laminar flow is reported herein, and work is presently underway to extend the study.

Numerical solutions of the boundary-layer equations are obtained by a finite-difference method which has been extensively tested. Film-cooling effectiveness is presented for a variety of injection configurations so that the effects of freestream Mach number, coolant mass flow, injection angle, boundary-layer thickness, slot width, and the presence of upstream cooling slots can be investigated.

Received October 7, 1974; revision received February 10, 1975. Research partially sponsored by U.S. Army Research Office—Durham, under Contract DAHC 04-69-C-0016.

Index categories: Boundary Layers and Convective Heat Transfer—Laminar; Jets, Wakes, and Viscid-Inviscid Flow Interaction.

*Instructor, Department of Mechanical Engineering.

†Associate Professor, Department of Mechanical Engineering. Member AIAA.

Numerical Method

A finite-difference method[3] is used to solve the following form of the boundary-layer equations

$$\rho u\frac{\partial u}{\partial x}-\frac{\partial \psi}{\partial x}\frac{\partial u}{\partial y}=-\frac{dp}{dx}+\frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right)$$

$$\rho u\frac{\partial T}{\partial x}-\frac{\partial \psi}{\partial x}\frac{\partial T}{\partial y}=\frac{u}{C_p}\frac{dp}{dx}+\frac{\partial}{\partial y}\left(\frac{\mu}{Pr}\frac{\partial T}{\partial y}\right)+\frac{\mu}{C_p}\left(\frac{\partial u}{\partial y}\right)^2 \quad (1)$$

where the stream function satisfies the requirements

$$\frac{\partial \psi}{\partial y}=\rho u \qquad \frac{\partial \psi}{\partial x}=-\left[\rho_0 v_0-\int_0^y \frac{\partial}{\partial x}(\rho u)\,dy\right] \quad (2)$$

The solution method is quite general and allows arbitrary specification of equation of state, viscosity model, pressure distribution, wall temperature or heat flux distribution, as well as arbitrary distribution of both transverse and longitudinal velocity components at the injection slot.

A system of finite-difference equations is derived by double integration of the boundary-layer equations over a small control volume and after linearization the difference equations are reduced to the tri-diagonal forms:

75-WA/HT-106

an ASME publication

The Society shall not be responsible for statements or opinions advanced in papers or in discussion at meetings of the Society or of its Divisions or Sections, or printed in its publications. *Discussion is printed only if the paper is published in an ASME journal or Proceedings.*
Released for general publication upon presentation.
Full credit should be given to ASME, the Technical Division, and the author(s).

$3.00 PER COPY
$1.50 TO ASME MEMBERS

A Numerical Solution for the Heat Transfer Between an Axi-Symmetric Air Jet and a Heated Plate

R. RAVURI

Teledyne CAE,
Toledo, Ohio

W. TABAKOFF

Department of Aerospace Engineering,
University of Cincinnati,
Cincinnati, Ohio
Mem. ASME

Impinging air jets are being used extensively in the industry due to their characteristic high rate of heat transfer. A numerical scheme is devised to predict the heat transfer between an axi-symmetric air jet and a heated plate, neglecting the effect of curvature, compressibility, and turbulence. In this analysis the momentum and continuity equations are transformed to vorticity and stream function equations, respectively, and solved by iterative successive substitution techniques. The flow field is determined which subsequently allows for a calculation of the temperature distribution in the fluid. It is possible to predict the average heat transfer coefficient of a row of axi-symmetric air jets impinging on a heated plate by assuming that each jet cools a fraction of the plate without any interference from the neighboring jets. The heat transfer coefficients are compared with some experimentally determined values.

Contributed by the Heat Transfer Division of The American Society of Mechanical Engineers for presentation at the Winter Annual Meeting, Houston, Texas, November 30–December 4, 1975. Manuscript received at ASME Headquarters August 13, 1975.

Copies will be available until August 1, 1976.

Discussion on this paper will be received at ASME Headquarters until January 5, 1976.

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

75-WA/HT-107

an ASME publication

The Society shall not be responsible for statements or opinions advanced in papers or in discussion at meetings of the Society or of its Divisions or Sections, or printed in its publications. *Discussion is printed only if the paper is published in an ASME journal or Proceedings.*
Released for general publication upon presentation.
Full credit should be given to ASME, the Technical Division, and the author(s).

$3.00 PER COPY
$1.50 TO ASME MEMBERS

Gas Turbine Blade Heat Transfer with Rough Surfaces

W. TABAKOFF

Mem. ASME

E. R. McFARLAND

Department of Aerospace Engineering,
University of Cincinnati,
Cincinnati, Ohio

The effects of surface roughness on impingement cooling of a gas turbine blade were experimentally determined. The study was conducted in a heated flow two-dimensional cascade tunnel. Results showed roughening impingement surfaces decreased the cooling effectiveness from that of a smooth surface in an optimal cooling configuration (H/D = 1). However, the roughened surfaces increased the cooling effectiveness over that of a smooth surface which was not in an optimal configuration (H/D = 2).

Contributed by the Heat Transfer Division of The American Society of Mechanical Engineers for presentation at the Winter Annual Meeting, Houston, Texas, November 30-December 4, 1975. Manuscript received at ASME Headquarters August 13, 1975.

Copies will be available until August 1, 1976.

Discussion on this paper will be received at ASME Headquarters until January 5, 1976.

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

APPENDIX A

The following scientific and technical personnel were participating in this research project.

NAME	TITLE
W. Tabakoff	Professor (Program Manager)
M. Hoch	Professor (Senior Investigator)
R. Lemlich	Professor
J. Sheng	Professor
R.G. Dale, Jr.	Associate Professor
S.D. Antolovich	Associate Professor
A. Hamed	Associate Professor
K.N. Ghia	Associate Professor
T.G. Tsuei	Associate Professor
W. Wells	Associate Professor
C.J. Rosa	Associate Professor
M.J. Pool	Associate Professor
E. Allis	Instructor
R. Earley	Instructor
G. Weidenhamer	Instructor
F. Tepe, Jr.	Instructor
W. Clevenger	Instructor
D.L. Vieth	Instructor
J. Krupowicz	Research Associate
B. Hannah	Jr. Research Associate
A. Beg	Postdoctoral Fellow
G Grant	Graduate Research Assistant
M.F. Hussein	Graduate Research Assistant
P.C. Godha	Graduate Research Assistant
R. Ravuri	Graduate Research Assistant
K. Khanna	Graduate Research Assistant
S. Lin	Graduate Research Assistant
R. Logani	Graduate Research Assistant
W. Hosny	Graduate Research Assistant
L. Lee	Graduate Research Assistant
S. Talabi	Graduate Research Assistant
N.C. Birla	Graduate Research Assistant
H. Hinds	Graduate Research Assistant
T.M. Yi	Graduate Research Assistant
A. Momi	Graduate Research Assistant
V.K. Sikka	Graduate Research Assistant
G.C. Chen	Graduate Research Assistant
P.M. Shete	Graduate Research Assistant
R. Pavri	Graduate Research Assistant
R. Nilson	Graduate Research Assistant
J. Blasenak	Graduate Research Assistant
D. Page	Graduate Research Assistant
J. Sokhey	Graduate Research Assistant
R. Steeneck	Graduate Research Assistant
F. Siller	Graduate Research Assistant
G. Shaffer	Graduate Research Assistant
E. McFarland	Graduate Research Assistant
N. Gat	Graduate Research Assistant
R. Ball	Graduate Research Assistant

NAME	TITLE
J. Kutney	Graduate Research Assistant
E. Anagnostopoulos	Graduate Research Assistant
I. Khalil	Graduate Research Assistant
D. Hudsen	Graduate Research Assistant
G. Cobb	Student Assistant
C. Gebhard	Student Assistant
J. Black	Student Assistant
H. Lebovitz	Student Assistant
J. Livingston	Student Assistant
S. Meyer	Student Assistant
J.E. Lueke	Student Assistant
W. Young	Student Assistant
P. Fritz	Student Assistant
B.C. Hu	Student Assistant
W. Poplarchek	Computer Consultant
D. Reedy	Electronic Technician
J. Cupito	Mechanician
C. Nordman	Accounting Technician II
Y. Hopkins	Intermediate Stenographer
K. Weast	Senior Stenographer
J. Donnelly (Fisher)	Senior Stenographer
D. Bollmer	Senior Typist

APPENDIX B

Students working on the project and obtaining degrees.

Ph.D. Degree Recipients

G. Weidenhamer
W. Clevenger
D.L. Vieth
B. Hannah
A. Hamed
M.F. Hussein
R. Ravuri
W. Hosny
S. Talabi
R. Pavri
R. Nilson
J. Sokhey
E. McFarland
G. Grant
N.C. Birla
J. Krupowicz

B.S. Degree Recipients

G. Cobb
C. Gebhard
J. Black
H. Lebovitz
J. Livingston
S. Meyer
J.E. Lueke
W. Young
P. Fritz
B.C. Hu

M.S. Degree Recipients

R. Nilson
S. Talabi
K. Khanna
J. Blasenak
I. Khalil
R. Earley
M.F. Hussein
S. Lin
E. McFarland
R. Steeneck
F. Siller
W. Hosny
N. Gat
A. Hamed
R. Ball
G. Shaffer
L. Lee
D. Hudsen
V.K. Sikka
J. Sokhey
W. Clevenger
G.C. Chen
D. Page
P.C. Godha

Unclassified
Security Classification

DOCUMENT CONTROL DATA - R & D

(Security classification of title, body of abstract and indexing annotation must be entered when the overall report is classified)

1. ORIGINATING ACTIVITY (Corporate author)	2a. REPORT SECURITY CLASSIFICATION
University of Cincinnati	Unclassified
	2b. GROUP: Na

3. REPORT TITLE

Internal Aerodynamics, Heat Transfer and High Temperature Materials in Air-Breathing Engines

4. DESCRIPTIVE NOTES (Type of report and inclusive dates)

~~Technical Report~~ Final Report, 20 Sep 68 - 30 Jun 76,

5. AUTHOR(S) (First name, middle initial, last name)

W. Tabakoff

6. REPORT DATE	7a. TOTAL NO. OF PAGES	7b. NO. OF REFS
October 1976	66	

8a. CONTRACT OR GRANT NO.	9a. ORIGINATOR'S REPORT NUMBER(S)
DAAG-29-69-C-0016	Report 76-50
b. PROJECT NO.	
c.	9b. OTHER REPORT NO(S) (Any other numbers that may be assigned this report)
d.	ARO T-4.78-E

10. DISTRIBUTION STATEMENT

Distribution of this report is unlimited.

11. SUPPLEMENTARY NOTES	12. SPONSORING MILITARY ACTIVITY
None	U.S. Army Research Office Post Office Box 12211 Research Triangle Park, NC 27709

13. ABSTRACT

The research presented herein is concerned with improving the performance of air-breathing propulsion systems. Theoretical and experimental studies of basic phenomena upon which the performance is dependent, were carried out. These studies covered the following areas: (1) Particulate Flows in Propulsion Systems; (2) Cascade Aerodynamics; (3) Jet Mixing Flow; (4) High Temperature Materials; and (5) Gas Turbine Blades and Combustion Chambers Heat Transfer.

083 810

DD FORM 1 NOV 68 1473 REPLACES DD FORM 1473, 1 JAN 64, WHICH IS OBSOLETE FOR ARMY USE.

Unclassified
Security Classification