UNIVERSITI PUTRA MALAYSIA

NUMERICAL SIMULATION OF TURBINE BLADE HEAT TRANSFER USING TWO-EQUATION TURBULENCE MODELS

ABDUL HAFID M. ELFAGHI

NUMERICAL SIMULATION OF TURBINE BLADE HEAT TRANSFER USING TWO-EQUATION TURBULENCE MODELS

By

ABDUL HAFID M. ELFAGHI

Thesis Submitted in Fulfilment of Requirements for the Degree of Master of Science in Faculty of Engineering
Universiti Putra Malaysia

May 2000

DEDICATED TO:

MY PARENTS,
MY WIFE & MY SONS

Abstract of thesis presented to the Senate of Universiti Putra Malaysia in fulfilment of the requirements for the degree of Master of Science.

NUMERICAL SIMULATION OF TURBINE BLADE HEAT TRANSFER USING TWO-EQUATION TURBULENCE MODELS

By

ABDUL HAFID M. ELFAGHI

May 2000

Chairman: Fuad Abas, Ir.

Faculty: Engineering

The development of high performance gas turbines requires high turbine inlet temperatures that can lead to severe thermal stresses in the turbine blades, particularly in the first stages of the turbine. Therefore, the major objective of gas-turbine designers is to determine the thermal and aero-dynamical characteristics of the turbulent flow in the turbine cascade.

This work is a numerical simulation of fluid flow and heat transfer in the turbine blade using different two-equation turbulence models. The turbulence models used here were based on the eddy viscosity concept, which determined the turbulent viscosity through time-averaged Navier-Stokes differential equations. The most widely accepted turbulence models are the two-equation models, which involves the solution of two transport equations for the turbulent kinetic energy, k, and its rate of dissipation, ε or ω. In the present simulation, four two-equation turbulence models

were used, the standard k-ε model, the modified Chen-Kim k-ε model, RNG model and Wilcox standard $k-\omega$ turbulence model.

A comparison between the turbulence models and their predictions of the heat flux on the blade were carried out. The results were also compared with the available experimental results obtained from a research carried out by Arts et al. (1990) at the von Karman Institute of Fluid Dynamics (VKI).

The simulation was perfonned using the general-purpose computational fluid dynamics code, PHOENICS, which solved the governing fluid flow and heat transfer equations. An H-type, body-fitted-co-ordinate (BFC) grid was used and upstream and downstream periodic conditions were specified. The grid system used was sufficiently fine and the results were grid independent.

All models demonstrated good heat transfer predictions for the pressure side except close to the leading edge. On the suction side, standard model over-predicted the heat transfer, whereas Chen-Kim, RNG and $k-\omega$ models captured the overall behaviour quite well. Unlike $k-\omega$ model, all $k-\varepsilon$ models generated very high turbulence levels in the stagnation point regions, which gave rise to the heat transfer rates close to the leading edge.

Abstrak tesis yang dikemukakan kepada Senat Universiti Putra Malaysia sebagai memenuhi keperluan ijazah Master Sains.

PENYELAKUAN BERANGKA PEMINDAHAN HABA BAGI BILAK TURBIN DENGAN MENGGUNA MODEL TURBULEN DUA-PERSAMAAN

Oleh

ABDUL HAFID M. ELFAGHI

Mei 2000

Pengerusi : Fuad Abas, Ir.

Fakulti : Kejuruteraan

Pembangunan turbin gas berprestasi tinggi memerlukan suhu masukan turbin tinggi yang boleh mewujudkan tegasan terma yang teruk di bilah turbin, terutamanya dalam turbin tahap pertama. Oleh itu, objektif utama bagi pereka bentuk turbin gas ialah untuk menentukan ciri-ciri terma dan aerodinamik bagi aliran turbulen dalam lata turbin.

Kerja penyelidikan ini adalah mengenai penyelakuan berangka aliran bendalir dan pemindahan haba di bilah turbin menggunakan model-model turbulen dua-persamaan yang berbeza. Model turbulen yang digunakan di sini berasaskan konsep keliakatan pusar, yang menentukan keliakatan turbulen dengan menggunakan persamaan kebezaan Navier-Stokes masa-terpurata. Model-model turbulen yang paling diterima pakai ialah model-model dua-persamaan, yang melibatkan penyelesaian dua persamaan angkut bagi tenaga kinetik turbulen, k, dan kadar lesapannya, ε atau ω. Dalam penyelakuan sekarang, empat model turbulen dua-

persamaan telah digunakan, iaitu model k-ε piawai, model k-ε Chen-Kim terubahsuai, model RNG dan model turbulen k-ω piawai Wilcox.

Perbandingan di antara model-model turbulen dan ramalan bagi fluks haba tersebut ke atas bilah telah dijalankan. Keputusan juga dibandingkan dengan keputusan ujikaji yang sedia ada yang didapati dari satu penyelidikan yang telah dijalankan oleh Arts et. al. (1990) di von Karman Institute of Fluid Dynamics (VKI).

Satu penyelakuan telah dijalankan dengan mengguna kod dinamik bendalir pengiraan tujuan umum, PHEONICS, yang menyelesaikan persamaan aliran bendalir dan pemindahan haba menakluk. Grid koordinat terpadan jasad (BFC) jenis H telah digunakan dan keadaan berkala hulu dan hilir telah ditentukan. Sistem grid yang telah digunakan adalah cukup halus tetapi keputusannya tidak bergantung kepada grid.

Semua model menunjukkan ramalan pemindahan haba yang baik bagi bahagian bertekanan kecuali dekat dengan pinggir depan. Pada bahagian sedutan model piawai meramalkan pemindahan haba lebih tinggi, tetapi model-model Chen-Kim, RNG dan k-ω menggambarkan kelakuan keseluruhan dengan tepat. Tidak seperti model k-ω, semua model k-ε menghasilkan tahap turbulen yang sangat tinggi di kawasan titik genangan, dan menghasilkan kadar pemindahan haba dekat pinggir depan.

ACKNOWLEDGEMENTS

First of all, I would like to express my greatest thanks and gratitude to Allah (S. W. T.) the most gracious and merciful for giving me the ability to carry out this work

I would like to thank the chairman of the supervisory committee, Ir. Fuad Abas for his excellent supervision and valuable advice during my study and carrying out of this work.

I gratefully acknowledge Dr. Mohammed Musadaq El-Awad, Mechanical Engineering Department, Universiti Tenaga National (UNITEN) for his advice, guidance, and supervision. I would also like to thank Associate Prof. Shah Nor Basri, and Dr. Mohd. Sapuan Salit for their supervision.

I am most grateful to my parents and my wife for their unfailing moral support. Their words of encouragement were very beneficial.

This work was carried out at the Department of Aerospace, Universiti Putra Malaysia, which is gratefully acknowledged. I wish also to thank all technicians at the department.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS AND NOMENCLATURES

q	Heat flux (kW/m^2)
υ	Kinematic viscosity (m^2/s)
υ_t	Turbulent viscosity
c	Constant
ℓ_m	Mixing length of the turbulence
U	Mean velocity, $-$ is dropped for convenient
c_p	Specific heat (J/kg K)
γ	Specific heat ratio
β	Thermal expansion coefficient (1/k)
ρ	Density (kg/m^3)
e	Internal energy
f	Body force
u_i	Velocity (m/s)
ε	Dissipation of k (J/kg s)
ω	Specific dissipation (s^{-1})
μ	Dynamic viscosity (kg/m s)
g	Gravity (m/s^2)
$_{lam}$	Laminar part
$\bar{x}$	Reynolds average
x'	Reynolds fluctuating part
L	Square cavity length (m)

p	Static pressure (Pa)
T	Static temperature
ΔT_w	Temperature difference between the walls (k)
t	Time (s)
k	Turbulence kinetic energy (J/kg)
$_{turb}$	Turbulent part
$c_{\varepsilon 1}, c_{\varepsilon 2}, c_\mu, \sigma_k, \sigma_\varepsilon$	Turbulence model constants
$\frac{Dk}{Dt}$	Particle rate of increase of turbulence kinetic energy
$\frac{\partial k}{\partial t}$	Time rate of change turbulence kinetic energy
D_k	Internal diffusive transport
p_k	Production by shear stress
CFD	Computational Fluid Dynamics
Gr	Grashof number
Pr	Prandtl number
Re	Reynolds number
Ra	Rayleigh number
RNG	Renormalization Group
VKI	von Karman Institute for Fluid Dynamics

CHAPTER I

INTRODUCTION

Turbine blade heat transfer is an important engineering problem characterised by complex flow field and high turbulence level. Fluid flow and heat transfer are primary concerns in modern gas turbine design, where highly efficient performance is sought. The combustion chamber and the turbine blades that follow the combustor experience tremendous thermal loads, which directly affect the overall performance and durability of the engine.

The high performance gas turbines require a high turbine inlet temperature that can lead to severe mechanical problems limiting the life of the components. Determining the thermal loads and the aerodynamic performances of the turbine is a major objective of designers. Methods for accurate prediction of heat transfer in turbines are important in order to get a good estimate of the thermal loads.

Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest.

Computational fluid dynamics tools have now been developed to the point where they can be reliably used for the design, evaluation, and analysis of high-performance gas turbines, such as those used in modern aeronautical propulsion systems.

CFD allows predicting of fluid flow, heat and mass transfer, aerodynamic, chemical reaction and related phenomena. By solving the fundamental equations governing fluid flow processes, CFD provides information on important flow characteristics such as pressure loss, flow distribution and mixing rates. CFD analysis complements traditional testing and experimentation, providing added insight and confidence in designs. The results are better designs, lower risk and faster time to the market place for the product or process.

Today, CFD is used in a variety of industries, for applications ranging from automotive component design, process plant troubleshooting and combustion system optimisation.

Turbine Blade Heat Transfer

The efficiency and specific thrust of gas turbine engines increase with higher gas temperature after combustion. A typical modern aeroengines experience turbine entry temperatures of up to 1700 K. In practice, there are no metals for turbine components that will withstand uncooled operation at these elevated gas temperatures. At such temperatures, the turbine blade must be safeguarded against the hot gases and cooling is one of the major methods employed (Guo et al., 1998).

These make the study of heat loads in the turbine cascade very important and the methods to predict the thermal loads in the design phase of the turbine would be very valuable engineering tools.

Turbulence Modelling

Turbulence is the most complicated kind of fluid motion, making even its precise definition difficult. According to Hinze (1975), "Turbulent fluid motion is an irregular condition of flow for which the various quantities show a random variation with time and space co-ordinates so that statistically distinct average values can be discerned"

Engineers are not concerned with all the details of the turbulent motion, but rather with its effects on the gross properties of the flow. There is no need to solve for the instantaneous variables if averaged variables are all that are required. A turbulence model is a set of differential equations solved in addition to the averaged Navier-Stokes and other conservation equations, which may express approximately relationships between unknown terms in the latter equations.

The two-equation turbulence model has been and will likely continue to be among the most widely used turbulence model for flow predictions in engineering applications.

The Objective of This Project

The aim of this work is to numerically simulate the turbine blade heat transfer using the commercially available general purpose CFD code PHOENICS. Different two-equation turbulence models are chosen for modelling the turbulence flow between the turbine blades. The predicted results are compared with the measurement results obtained from the transonic high loaded turbine cascade carried out at von Karman Institute of Fluid Dynamics (VKI). Flow and heat transfer experimental measurements at many different operating conditions are available in Arts et al. (1990). Comparisons between the turbulence models used are carried out. The flowchart for this work is shown in Figure 1.

Outline of the Thesis

This thesis consists of six chapters, each developing on from the previous chapter. The following chapter presents a literature review of work that has already been completed in the area of turbine blade heat transfer and the turbulence modelling and simulation. Chapter III focuses on the theoretical background, relevant to the understanding of the computational fluid dynamics (CFD), forming of the governing equations and the different turbulence models used. A brief background on the structure of the general-purpose code used, PHOENICS is discussed.

Chapter IV gives the numerical simulation methodology, starting with two test cases which are selected to test the ability of PHOENICS to simulate the main characteristics of the flow, heat transfer, buoyancy effects, turbulence, and flow separation. This chapter gives the problem definitions, grid system, initial and boundary conditions, turbulence models used, and grid independence and convergence tests.

The results obtained by the different turbulence models are presented and compared with the available experimental data in chapter V. The sixth and final chapter of the thesis brings together the conclusions and suggestions for further work that could be carried out in the future.

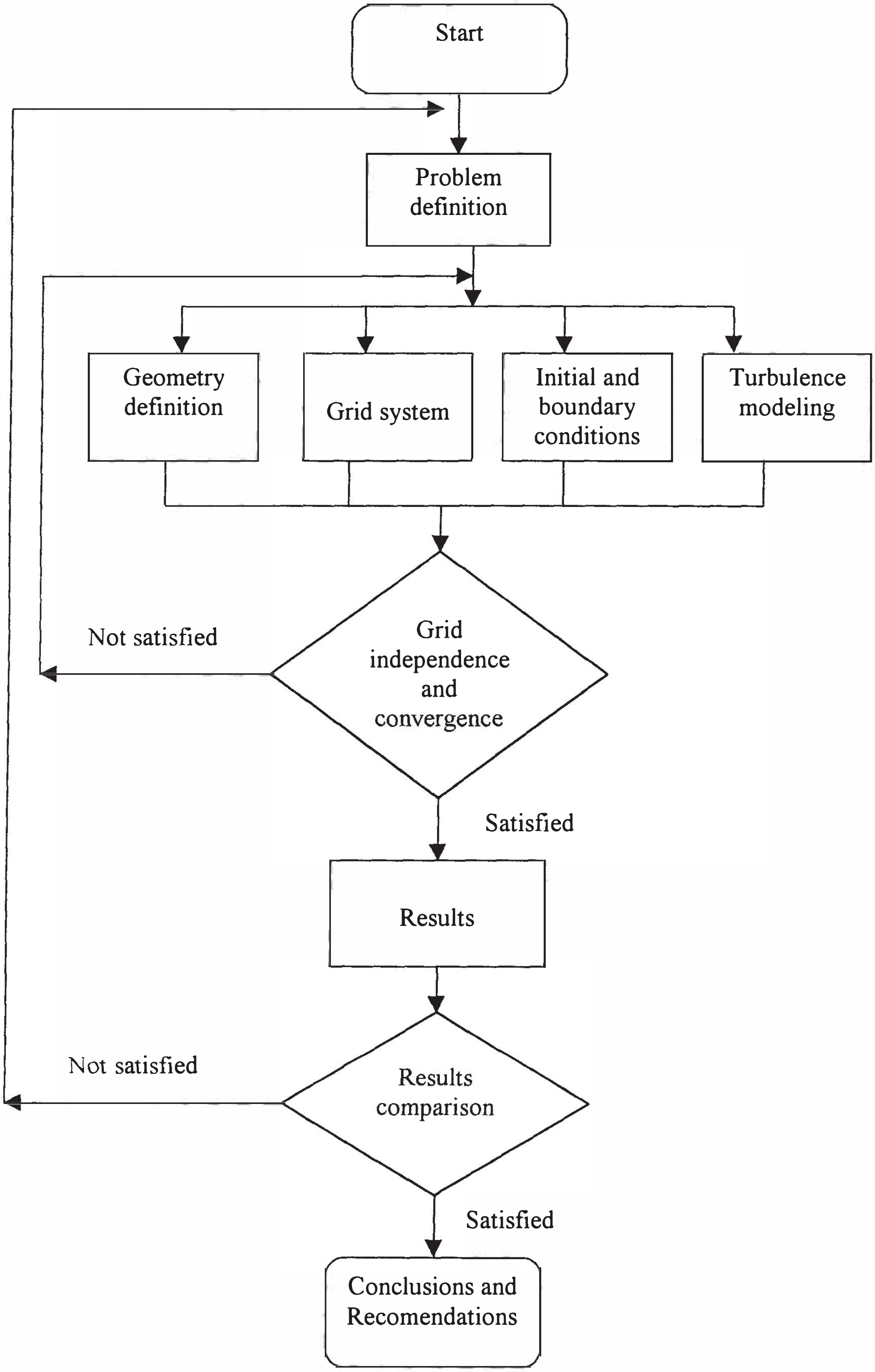

Figure 1: Research Flowchart

CHAPTER II

LITERATURE REVIEW

There has been a considerable growth in the development and application of Computational Fluid Dynamics to all aspects of fluid dynamics. In design and development, CFD programs are now considered to be standard numerical tools, widely utilised within industry. As a consequence there is a considerable demand for specialists in the subject, to apply and develop CFD methods throughout engineering companies and research organisations.

The heat loads on the blades of a gas turbine are very important. A good design from a thermal point of view might allow a higher inlet temperature, less cooling and a lighter design, thus increasing the performance efficiency of the machine. For the prediction of flow and heat transfer to the turbine cascade, often blade-to-blade and general elliptic calculation procedures are applied. These were the major interests of many researchers (Larsson, 1997, Slimani and Kulisa, 1996, Chen et al., 1994 and Nicholson et al., 1982).

PHOENICS is a commercially available CFD code that widely used since 1981 for the simulation of fluid-flow, heat-transfer and chemical-reaction processes in engineering equipment and the environment. In the present work, numerical simulation of turbine blade heat transfer was carried out using PHOENICS, which is available at Department of Aerospace, Universiti Putra Malaysia. The code has been used previously for various applications (Jal, 1996, Myszko and Knowles, 1996,