



Journal of Engineering for Gas Turbines and Power

Published Quarterly by ASME

VOLUME 126 • NUMBER 3 • JULY 2004

TECHNICAL PAPERS

Advanced Energy Systems

- 445 A Simulation Model for Hot Spring Thermal Energy Conversion Plant With Working Fluid of Binary Mixtures
Ou Bai, Masatoshi Nakamura, Yasuyuki Ikegami, and Haruo Uehara

Gas Turbines: Combustion and Fuels

- 455 A Novel Approach to the Optimization of Reaction Rate Parameters for Methane Combustion Using Multi-Objective Genetic Algorithms (2003-GT-38018)
L. Elliott, D. B. Ingham, A. G. Kyne, N. S. Mera, M. Pourkashanian, and C. W. Wilson
- 465 Emissions Performance of the Parker Macrolaminate Premixer Tested Under Simulated Engine Conditions (2003-GT-38010)
Adel Mansour and Michael A. Benjamin

Gas Turbines: Controls, Diagnostics & Instrumentation

- 472 Implementation of a Multivariable Control (2000-GT-039)
Louis J. Larkin and James Philpott

Gas Turbines: Cycle Innovations

- 475 One-Disk Nutating-Engine Performance for Unmanned Aerial Vehicles (2001-GT-466)
T. Korakianitis, L. Meyer, M. Boruta, and H. E. McCormick
- 482 Alternative Multinutating Disk Engine Configurations for Diverse Applications (2001-GT-0572)
T. Korakianitis, L. Meyer, M. Boruta, and H. E. McCormick
- 489 The Employment of Hydrogenerated Fuels From Natural Gas Reforming: Gas Turbine and Combustion Analysis (2002-GT-30414)
Fabio Bozza, Maria Cristina Cameretti, and Raffaele Tuccillo
- 498 Analytical Blade Row Cooling Model for Innovative Gas Turbine Cycle Evaluations Supported by Semi-Empirical Air-Cooled Blade Data (2002-GT-30519)
Leonardo Torbidoni and Aristide F. Massardo
- 507 Aspects of Cooled Gas Turbine Modeling for the Semi-Closed O₂/CO₂ Cycle With CO₂ Capture (2003-GT-38067)
Kristin Jordal, Olav Bollard, and Ake Klang

Gas Turbines: Cycle Innovations, and Combustion and Fuels

- 516 Simplified Versus Detailed Solid Oxide Fuel Cell Reactor Models and Influence on the Simulation of the Design Point Performance of Hybrid Systems (2002-GT-30653)
Loredana Magistri, Rocco Bozzo, Paola Costamagna, and Aristide F. Massardo

(Contents continued on inside back cover)

This journal is printed on acid-free paper, which exceeds the ANSI Z39.48-1992 specification for permanence of paper and library materials. ©™
♻️ 85% recycled content, including 10% post-consumer fibers.

Editor

LEE S. LANGSTON (2006)

Assistant to the Editor

LIZ LANGSTON

Associate Editors

Fuels and Combustion Technologies

S. GOLLAHALLI (2004)

Internal Combustion Engines

D. ASSANIS (2005)

Nuclear

R. DUFFY (2004)

International Gas Turbine Institute

IGTI Review Chair

E. BENVENUTI (2002)

H. R. SIMMONS (2003)

A. J. STRAZISAR (2004)

Combustion and Fuels

P. MALTE (2006)

Structures and Dynamics

N. ARAKERE (2007)

M. MIGNOLET (2005)

BOARD ON COMMUNICATIONS

Chair and Vice-President

OZDEN OCHOA

OFFICERS OF THE ASME

President, HARRY ARMEN

Executive Director, V. R. CARTER

Treasurer, R. E. NICKELL

PUBLISHING STAFF

Managing Director, Engineering

THOMAS G. LOUGHLIN

Director, Technical Publishing

PHILIP DI VIETRO

Production Coordinator

JUDITH SIERANT

Production Assistant

MARISOL ANDINO

Transactions of the ASME, Journal of Engineering for Gas Turbines and Power (ISSN 0742-4795) is published quarterly (Jan., April, July, Oct.) by The American Society of Mechanical Engineers, Three Park Avenue, New York, NY 10016. Periodicals postage paid at New York, NY and additional mailing offices. POSTMASTER: Send address changes to Transactions of the ASME, Journal of Engineering for Gas Turbines and Power, c/o THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS, 22 Law Drive, Box 2300, Fairfield, NJ 07007-2300.

CHANGES OF ADDRESS must be received at Society headquarters seven weeks before they are to be effective. Please send old label and new address.

STATEMENT from By-Laws. The Society shall not be responsible for statements or opinions advanced in papers or ... printed in its publications (B7.1, par. 3).

COPYRIGHT © 2004 by the American Society of Mechanical Engineers. For authorization to photocopy material for internal or personal use under circumstances not falling within the fair use provisions of the Copyright Act, contact the Copyright Clearance Center (CCC), 222 Rosewood Drive, Danvers, MA 01923. Tel: 978-750-8400. www.copyright.com.

INDEXED by Applied Mechanics Reviews and Engineering Information, Inc. Canadian Goods & Services Tax Registration #126148048

Gas Turbines: Electric Power

- 524 Siemens Westinghouse Advanced Turbine Systems Program Final Summary (2002-GT-30654)
Ihor S. Kiakunchak, Greg R. Gaul, Gerry McQuiggan, and Leslie R. Southall

Gas Turbines: Industrial and Cogeneration

- 531 A Feasibility Study of Existing Gas Turbines for Recuperated, Intercooled, and Reheat Cycle (2002-GT-30558)
R. Bhargava, M. Bianchi, A. Peretto, and P. R. Spina
- 545 Inlet Fogging of Gas Turbine Engines—Part I: Fog Droplet Thermodynamics, Heat Transfer, and Practical Considerations (2002-GT-30562)
Mustapha Chaker, Cyrus B. Meher-Homji, and Thomas Mee III
- 559 Inlet Fogging of Gas Turbine Engines—Part II: Fog Droplet Sizing Analysis, Nozzle Types, Measurement, and Testing (2002-GT-30563)
Mustapha Chaker, Cyrus B. Meher-Homji, and Thomas Mee III
- 571 Inlet Fogging of Gas Turbine Engines—Part III: Fog Behavior in Inlet Ducts, Computational Fluid Dynamics Analysis, and Wind Tunnel Experiments (2002-GT-30564)
Mustapha Chaker, Cyrus B. Meher-Homji, and Thomas Mee III
- 581 Technical and Tariff Scenarios Effect on Microturbine Trigenerative Applications (2003-GT-38275)
Stefano Campanari and Ennio Macchi

Gas Turbines: Structures and Dynamics

- 590 High-Temperature Fatigue Properties of Single Crystal Superalloys in Air and Hydrogen (2001-GT-585)
N. K. Arakere
- 604 Rotor Balancing Without Trial Weights (2002-GT-30633)
A. El-Shafei, A. S. El-Kabbany, and A. A. Younan
- 610 Nonparametric Modeling of Random Uncertainties for Dynamic Response of Mistuned Bladed Disks
E. Capiez-Lernout and C. Soize
- 619 A Pareto-Based Genetic Algorithm Search Approach to Handle Damped Natural Frequency Constraints in Turbo Generator Rotor System Design
Anders Angantyr and Jan Olov Aidanpaa

Gas Turbines: Turbomachinery

- 626 Lateral Forces From Single Gland Rotor Labyrinth Seals in Turbines (2002-GT-30335)
Bum Ho Song and Seung Jin Song

Internal Combustion Engines

- 635 Computations of Air/Fuel Preparation Process in a Port-Injected Spark-Ignition Engine During Cold-Starting Phase
Yangbing Zeng and Chia-fon F. Lee
- 645 A Characteristic Parameter to Estimate the Optimum Counterweight Mass of Symmetric In-Line Engines
R. Stanley and D. Taraza
- 656 Computational Modeling of Natural Gas Injection in a Large Bore Engine
Gi-Heon Kim, Allan Kirkpatrick, and Charles Mitchell
- 665 Strategies for Reduced NO_x Emissions in Pilot-Ignited Natural Gas Engines
S. R. Krishnan, K. K. Srinivasan, S. Singh, S. R. Bell, K. C. Midkiff, W. Gong, S. B. Fiveland, and M. Willi

Power

- 672 Large Eddy Simulation of the Anti-Erosion Characteristics of the Ribbed-Bend in Gas-Solid Flows
J. R. Fan, K. Luo, X. Y. Zhang, and K. C. Cen

TECHNICAL BRIEF

- 680 Analytical Investigations of Rotating Disks With and Without Incorporating Rotating Heat Pipes
J. Ling, Y. Cao, R. Rivir, and C. MacArthur

The ASME Journal of Engineering for Gas Turbines and Power is abstracted and indexed in the following:

AESIS (Australia's Geoscience, Minerals, & Petroleum Database), Applied Science & Technology Index, AMR Abstracts Database, Aquatic Sciences and Fisheries Abstracts, Civil Engineering Abstracts, Compendex (The electronic equivalent of Engineering Index), Computer & Information Systems Abstracts, Corrosion Abstracts, Current Contents, Engineered Materials Abstracts, Engineering Index, Enviroline (The electronic equivalent of Environment Abstracts), Environment Abstracts, Environmental Science and Pollution Management, Fluidex, INSPEC, Mechanical & Transportation Engineering Abstracts, Mechanical Engineering Abstracts, METADEX (The electronic equivalent of Metals Abstracts and Alloys Index), Pollution Abstracts, Referativnyi Zhurnal, Science Citation Index, SciSearch (The electronic equivalent of Science Citation Index), Shock and Vibration Digest

Ou Bai¹

e-mail: baiou@cntl.ee.saga-u.ac.jp
Department of Advanced Systems Control
Engineering,
Graduate School of Science and Engineering,
Saga University,
Honjomachi 1,
Saga 840-8502, Japan
and Institute of Ocean Energy,
Saga University,
Honjomachi 1,
Saga 840-8502, Japan

Masatoshi Nakamura

Department of Advanced Systems Control
Engineering,
Graduate School of Science and Engineering,
Saga University,
Honjomachi 1,
Saga 840-8502, Japan

Yasuyuki Ikegami

Haruo Uehara

Institute of Ocean Energy,
Saga University,
Honjomachi 1,
Saga 840-8502, Japan

A Simulation Model for Hot Spring Thermal Energy Conversion Plant With Working Fluid of Binary Mixtures

Hot spring thermal energy conversion (STEC) is a system that converts heat energy into electricity using the temperature difference between hot spring water and sea/river water. This paper describes dynamic model construction for the transient performance of STEC plant, which uses a recently developed power cycle with binary mixtures as working fluid. The mathematical models were constructed based on thermodynamics and structural features of the power cycle for representing the timely dependent state variables of the working fluid. Confidence in the accuracy of the developed models has been established by comparison of the simulation results with those obtained experimentally in a pilot STEC plant. [DOI: 10.1115/1.1760526]

1 Introduction

Hot spring thermal energy conversion (STEC) is to convert thermal energy into electric power using the temperature difference between hot spring water (about 70°C to 80°C) and sea/river water (20°C to 30°C), [1]. STEC system, which is different from the conventional geothermal power plant using the steam directly, uses the heat transmitted from hot spring to ammonia/water mixtures through heat transfer, where the hot spring water is the wastewater of spring hotel. Similar to ocean thermal energy conversion (OTEC), [2], STEC is an effective method of power generation, which has a small impact on the environment and can be utilized semi-permanently. However, because of the small temperature difference between spring water and sea/river water, the thermal efficiency of the STEC (about 7% to 8%) is much lower than that of the thermal power generation using fossils or nuclear energy (about 50% to 60%), [3]. In order to improve the thermal efficiency of STEC plant with a low plant construction cost, a new power cycle using ammonia and water mixture as working fluid has been developed in our recent study, and the thermal efficiency employing the new power cycle was demonstrated to be higher than the conventional ones experimentally, [4]. As the heat resource temperature of spring water and sea/river water has a large effect on the power generation of STEC plant, for an efficiency and stable operation of the STEC plant, a comprehensive understanding of the characteristics of the plant is required to achieve a steady operation control, [5]. For example, there may be 5°C temperature change in a hot spring daily, [6]. This temperature change in the heat resource might result a low performance of the STEC plant (power output decrease to 50%) and cause the STEC plant

unstable. Furthermore, before setting up a new plant, sufficient simulations for evaluating the operating performance of the plant are also necessary for the plant design, [7]. The studies concerning about thermal energy conversion using temperature difference are mainly in promoting the performance of the power plant under a steady operation, e.g., the cost or size design of OTEC/STEC system, [8]. Although the model construction for a simple cycle using pure substance as working fluid has been proposed, [9,10], the models for the transient performance of STEC plant using binary mixtures as working fluid have not been reported. Therefore, the development of dynamic models for the transient performance of the new power cycle using binary mixtures becomes necessary because of the complexity of the cycle structure and the different physical properties of the binary mixtures.

This paper concerns with the dynamic model construction and simulation for STEC plant with binary mixtures of ammonia and water as the working fluid, in which the temperature, pressure, mass, and composition of the working fluid in each component are all time-dependent. For the purpose of design and control of STEC plant, the dynamic models were developed based on the thermodynamic theory, and a simulation work was implemented using the developed models. Confidence in the accuracy of the developed models has been established by comparison of the simulation results with those obtained experimentally in a pilot STEC experimental plant in Saga University, Saga, Japan. As a result, the developed computer simulation can be used to actual practice effectively, such as stable control in a steady operation, optimal determination of the plant specifications for a higher thermal efficiency and evaluation of the economic prospects of STEC plant.

2 Power Cycle of STEC Plant

2.1 Power Cycle of Electric Generation Using Temperature Difference. The main components in a STEC plant include

¹To whom correspondence should be addressed.

Contributed by the Power Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the F&C Division March 2002; final revision received September 2003. Associate Editor: D. Y. S. Lou.

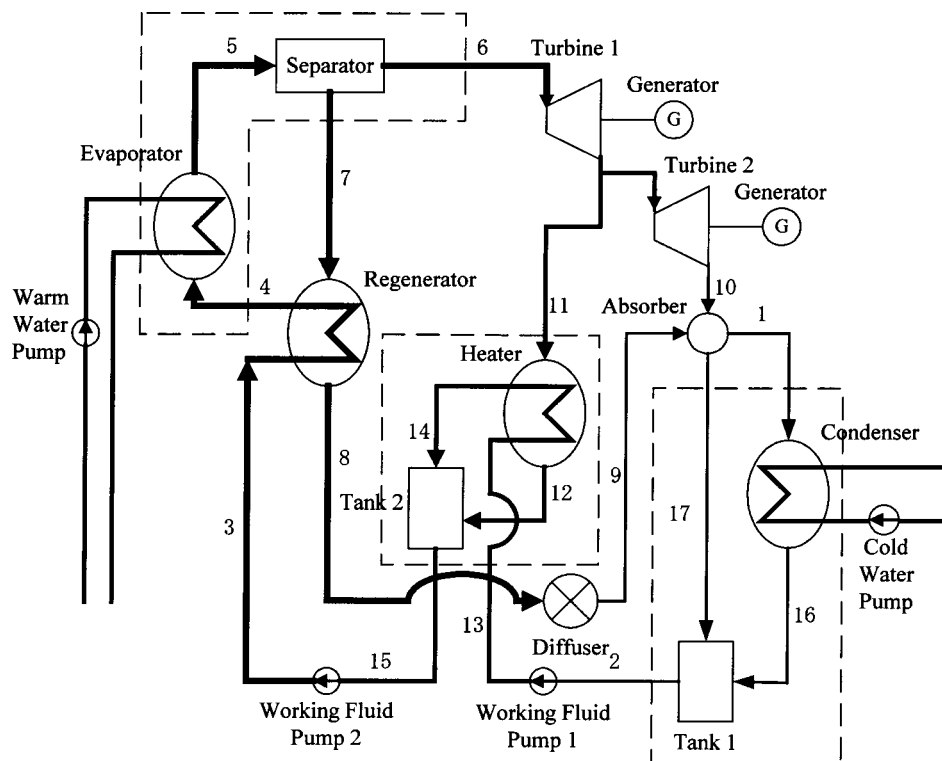


Fig. 1 Diagram of power cycle of the pilot STEC plant. The thick, middle, and thin solid lines stands for the high, middle, and low pressure parts in the power cycle, respectively.

evaporator, condenser, turbine and pump. These components are connected with each other using pipes to construct a simple cycle called Rankine cycle. The liquid state of a pure-substance working fluid, such as ammonia or freon, is transported by the pump to the evaporator, where the working fluid is evaporated into the vapor state by the hot spring water. The vapor of the working fluid passes through the turbine, where it drives the turbine and the connected electrical generator to generate electricity. The vapor from the turbine is cooled into liquid state by the cool sea/river water in the condenser, then, the liquid state of the working fluid is transported to the evaporator again. However, because of the small temperature difference between spring and sea/river water, the thermal efficiency of an STEC plant is very low about 7%–8% owing to the small difference of the evaporation and condensation temperatures of the working fluid.

A. A. Kalina [11] developed a new power cycle using working fluid of binary mixtures instead of a pure substance. In the case of STEC plant, the thermal efficiency increases to 8%–10% theoretically. However, because of the use of binary mixtures as working fluid, the heat transfer performance of the condenser decreases, [4]. Then, the construction cost of a STEC plant will increase if a larger heat transfer area is required in the condenser.

This paper concerns with the model construction for an STEC plant using a regenerating and reheating cycle, where the binary mixtures of ammonia and water are used as the working fluid. In order to solve the problem of Kalina cycle, the outlet of the turbine is divided into two segments. The vapor from segment 1 is extracted so that the burden of the condenser is released. The thermal efficiency of the newly developed cycle was demonstrated 2% higher than that of Kalina cycle experimentally, [4,12]. A pilot STEC plant has been built up in Imari, Saga, Japan, for the development of the technology on the thermal energy conversion. In this study, the experimental data was acquired from the pilot STEC plant.

2.2 Power Cycle With Working Fluid of Binary Mixtures.

The power cycle used in the pilot STEC plant is a closed cycle, since the closed power cycle has a higher thermal efficiency than that of the open power cycle for the STEC plant. The diagram of the developed power cycle for the pilot STEC plant is illustrated in Fig. 1. The overall system consists of 13 essential components; regenerator, evaporator, separator, absorber, condenser, tank 1, heater, tank 2, turbine, electric generator, diffuser, pump 1, and pump 2. The components are connected by pipes, in which the working fluid flows from one component to the succeeding component. In the STEC plant, binary mixtures of ammonia and water were used as the working fluid. The working fluid circulation pump 2 sends the working fluid to the regenerator, where the cool working fluid is warmed by the warm working fluid through heat exchanger. Then the working fluid enters the evaporator, where a part of it is vaporized by the hot spring water and enters the separator, where it is separated into a saturation vapor and a saturation liquid. The vapor enters the turbine 1. A part of the vapor is extracted and enters the heater. Another part of vapor enters the turbine 2. The saturation liquid is cooled by the cool working fluid through the regenerator and passes the diffuser. The outlet vapor from turbine 2 and the liquid from the diffuser are mixed. The mixed vapor and liquid enter the absorber and condenser, where they are cooled by sea/river water, and at the same time the liquid absorbs the vapor. The working fluid circulation pump 1 sends the condensed working fluid to the heater, where the working fluid is warmed by the extracted vapor. The working fluid is returned to the regenerator and the cycle continuous.

Since binary mixtures of ammonia and water are used as the working fluid, the physical properties of the working fluid are more complex than those of the pure substance. The compositions of ammonia and water in the different components of the STEC plant are different and time-dependent. Further, in the saturated phase, the liquid-vapor equilibrium of the working fluid should be

considered. Therefore, enthalpy h and specific volume v are the functions of temperature T , pressure P , and composition Z as

$$\begin{cases} h = f_1(T, P, Z) \\ v = f_2(T, P, Z) \end{cases} \quad (1)$$

and inversely, temperature can be calculated from pressure P , composition Z , and enthalpy h .

$$T = f_3(P, Z, h) \quad (2)$$

In the saturation phase, the vapor and liquid compositions (Z_V, Z_L) of the working fluid are related to the temperature and pressure as

$$(Z_V, Z_L) = f_4(T, P). \quad (3)$$

In this simulation study, the physical properties of the working fluid were calculated using a developed computer program of PROPATH (a PROgram PAcage for Thermo physical properties of fluids), which supplies the thermal physical values of the binary mixtures of ammonia and water [13].

3 Model Construction for STEC Plant Components

3.1 General Descriptions of Model Construction. The mathematic models are constructed for representing the dynamic characteristics of the power cycle of the STEC plant. Although the power cycle of the STEC plant is rather complicated, under the purpose to represent the plant dynamic characteristics sufficiently the model structure is developed as simple as possible. Based on the actual structure of the components in the STEC plant, the models are constructed by employing thermodynamics theory.

The mathematic models for STEC plant are developed under the following basic considerations.

B1: The models are constructed in lumped parameter forms in order to avoid complicated model structures.

B2: According to the principle of the STEC plant, the whole system is divided into seven components, regenerator, evaporator and separator, absorber, condenser and tank 1, heater and tank 2, turbine, diffuser.

B3: The whole power cycle is divided into high-pressure, middle-pressure, and low-pressure parts, where the pressure difference in each part is neglected. The high-pressure part is from the outlet of pump 2 to the inlets of turbine 1 and diffuser, which is shown by the thick line in Fig. 1. The middle-pressure part is from the outlets of turbine 1 and pump 1 to the inlet of pump 2. The low-pressure part is from the outlets of turbine 2 and diffuser to the inlet of pump 1.

B4: The temperature increase of the working fluid resulted from the pressure raise in pumps, and the loss in pipes are neglected.

B5: The transient performance of the STEC plant can be represented by the state variables of temperature, pressure, mass, and composition of the working fluid in each STEC component.

In STEC plant, the temperature, pressure, mass, and composition of the working fluid are all time-dependent. Especially, the liquid and vapor compositions of the working fluid in the saturated phase are changed with the temperature and pressure. Therefore, the temperature, pressure, mass, and composition of the working fluid should be modeled simultaneously. In this study, the state variables of single and two-phase fluid heat, pressure, mass and composition of the working fluid are represented by first-order differential equations.

The model construction for the seven components of the power cycle is described in the following section. According to the characteristic construction of each component, appropriate assumptions are introduced. Then, mathematical models are constructed from the basic thermodynamic theory. Finally, the solution for calculating the variables in the models is described.

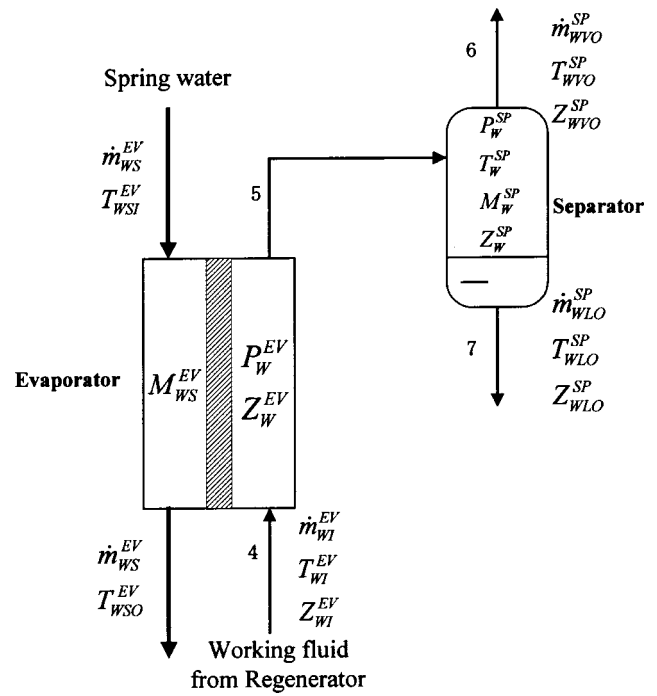


Fig. 2 Model structure of evaporator and separator

3.2 Dynamic Models for Evaporator, Condenser, Heater, and Absorber

Evaporator and Separator Model. The model for the evaporator was constructed by merging the evaporator and the separator into an entity as one of the seven main components for modeling (B2). The model structure is shown in Fig. 2. A plate-type heat exchanger is used in the evaporator. In the evaporator, a part of the working fluid is evaporated so that the working fluid is in the liquid and vapor phases in the heat exchanger. However, the calculation of the heat transfer for the two-phase working fluid is very complex. In order to keep the model simple, the working fluid is assumed in the liquid phase from the inlet of the evaporator to the liquid surface of the separator. Therefore, the heat transfer coefficient from the spring water to the working fluid keeps the same along the area of the heat exchanger, where the overheat effect is neglected. Further, the volume change of the liquid with temperature and pressure is too small than that of the vapor so that the volume change of the liquid is not considered. The temperature and pressure (B3) are assumed to be same from the outlet of the evaporator to the liquid and vapor outlets of the separator,

$$T_w^{SP} = T_{wlo}^{SP} = T_{wvo}^{SP},$$

$$P_w^{EV} = P_w^{SP}.$$

Further, it is assumed that the separator separates the liquid and vapor completely.

The first-order differential equation describing the heat balance in the evaporator and separator in the lumped parameter form (B1) is written as

$$\frac{d}{dt} (M_w^{SP} h_w^{SP}) = \dot{m}_{wI}^{EV} h_{wI}^{EV} + Q^{EV} - \dot{m}_{wvo}^{SP} h_{wvo}^{SP} - \dot{m}_{wlo}^{SP} h_{wlo}^{SP} \quad (4)$$

where M_w^{SP} (kg) is the mass of the working fluid (ammonia and water in liquid and vapor phases) in the evaporator and separator, h_w^{SP} (J/kg) is the corresponding enthalpy, which can be calculated

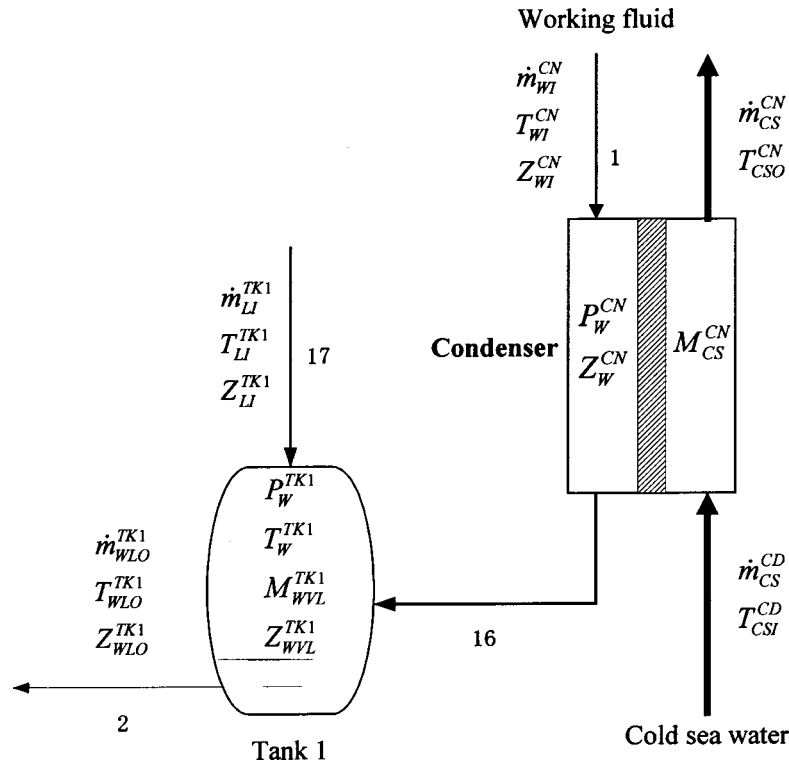


Fig. 3 Model structure of condenser and tank 1

by Eq. (1) with a certain pressure, temperature and composition. An approximation is used here that in a small time interval, the changes of volume and pressure are small so that they can be neglected. Then, the left side of the above equation is considered as the enthalpy instead of internal energy.

The heat transfer rate Q^{EV} (J/s) between the spring water and the working fluid is calculated by

$$Q^{EV} = U^{EV} A^{EV} \Delta T_m^{EV} \quad (5)$$

where the logarithmic mean temperature difference ΔT_m^{EV} ($^{\circ}\text{C}$) is

$$\Delta T_m^{EV} = \frac{(T_{WSI}^{EV} - T_{WO}^{EV}) - (T_{WSO}^{EV} - T_{WI}^{EV})}{\ln \frac{T_{WSI}^{EV} - T_{WO}^{EV}}{T_{WSO}^{EV} - T_{WI}^{EV}}},$$

and the heat transfer coefficient U^{EV} ($\text{W}/\text{m}^2 \text{ } ^{\circ}\text{C}$) is approximated by [14]

$$U^{EV} = U_s^{EV} (\dot{m}_{WS}^{EV} / \dot{m}_s^{EV})^{0.5},$$

where U_s^{EV} and $\dot{m}_s^{EV}$ are the standard heat transfer coefficient and the standard mass flow rate, respectively, which are determined from the experimental study.

The mass balance in the evaporator and separator is described by

$$\frac{d}{dt} (M_w^{SP}) = \dot{m}_{WI}^{EV} - \dot{m}_{WVO}^{SP} - \dot{m}_{WLO}^{SP}. \quad (6)$$

Since binary mixtures of ammonia and water are used as the working fluid, the composition of the working fluid is time-dependent (B5), which is calculated by

$$\frac{d}{dt} (Z_w^{SP}) = \frac{1}{M_w^{SP}} (\dot{m}_{WI}^{EV} Z_{WI}^{EV} - \dot{m}_{WVO}^{SP} Z_{WVO}^{SP} - \dot{m}_{WLO}^{SP} Z_{WLO}^{SP}). \quad (7)$$

The vapor in the separator is considered as ideal gas. According to the ideal gas law, the pressure change can be calculated from the following differential equation.

$$\frac{d}{dt} (P_w^{SP} V_{wV}^{SP} - n_{wV}^{SP} R_0 T_w^{SP}) = 0 \quad (8)$$

where V_{wV}^{SP} (m^3) is the volume of the vapor in the separator, and $V_{wV}^{SP} = M_{wV}^{SP} v_{wV}^{SP}$, v_{wV}^{SP} (m^3/kg) the specific volume of the vapor, and R_0 ($\text{J}/\text{kmol } ^{\circ}\text{C}$) the universal gas constant.

The mole amount of the vapor in the separator n_{wV}^{SP} (kmol) is calculated by

$$n_{wV}^{SP} = M_{wV}^{SP} Z_{wVO}^{SP} / N_{NH3} + M_{wV}^{SP} (1 - Z_{wV}^{SP}) / N_{H2O},$$

where N_{NH3} and N_{H2O} are the molecular weights of ammonia and water, respectively.

According to the composition balance of the working fluid in the saturated phase, [15], the liquid and vapor compositions are obtained by Eq. (3), and the mass of the vapor M_{wV}^{SP} is calculated by

$$M_{wV}^{SP} = \frac{Z_w^{SP} - Z_{WLO}^{SP}}{Z_{WVO}^{SP} - Z_{WLO}^{SP}} M_w^{SP}.$$

Condenser and Tank 1 Model. Similar to evaporator and separator, the model for condenser and tank 1 was constructed. The model structure of condenser and tank 1 is illustrated in Fig. 3. The working fluid is assumed in the vapor phase from the inlet of the condenser to the liquid surface of the tank 1. The temperature and pressure are assumed to be same from the outlet of the condenser to the liquid outlet of the tank 1,

$$T_w^{TK1} = T_{WLO}^{TK1},$$

$$P_w^{CN} = P_w^{TK1}.$$

The differential equations representing the energy, mass, composition of the working fluid and pressure in the condenser and tank 1 are given by

$$\frac{d}{dt}(M_w^{TK1} h_w^{TK1}) = \dot{m}_{WI}^{CN} h_{WI}^{CN} + \dot{m}_{LI}^{TK1} h_{LI}^{TK1} - Q^{CN} - \dot{m}_{WLO}^{TK1} h_{WLO}^{TK1} \quad (9)$$

$$\frac{d}{dt}(M_w^{TK1}) = \dot{m}_{WI}^{CN} + \dot{m}_{LI}^{TK1} - \dot{m}_{WLO}^{TK1} \quad (10)$$

$$\frac{d}{dt}(Z_w^{TK1}) = \frac{1}{M_w^{TK1}} (\dot{m}_{WI}^{CN} Z_{WI}^{CN} + \dot{m}_{LI}^{TK1} Z_{LI}^{TK1} - \dot{m}_{WLO}^{TK1} Z_{WLO}^{TK1}) \quad (11)$$

$$\frac{d}{dt}(P_w^{TK1} V_w^{TK1} - n_w^{TK1} R_0 T_w^{TK1}) = 0. \quad (12)$$

The heat transfer rate is calculated similar to the case of the evaporator and separator as

$$Q^{CN} = U^{CN} A^{CN} \Delta T_m^{CN} \quad (13)$$

where

$$\Delta T_m^{CN} = \frac{(T_{WI}^{CN} - T_{CSO}^{CN}) - (T_{WO}^{CN} - T_{CSI}^{CN})}{\ln \frac{T_{WI}^{CN} - T_{CSO}^{CN}}{T_{WO}^{CN} - T_{CSI}^{CN}}},$$

and

$$U^{CN} = U_s^{CN} (\dot{m}_{CS}^{CN} / \dot{m}_s^{CN})^{0.5}.$$

Heater and Tank 2 Model. The model for the heater was constructed by merging the heater and the tank 2 into an entity. The model structure of heat and tank 2 is illustrated in Fig. 4. As the heat exchange is only between the high temperature and low temperature working fluids, which finally flow into the tank 2. The heat transfer effect of the heat exchanger is neglected. The temperature is assumed to be same from the outlet of the heater to the liquid outlet of the tank 2,

$$T_w^{TK2} = T_{WLO}^{TK2}.$$

Similar to evaporator and separator, the model for heater and tank 2 was constructed. The differential equations representing heat, mass, composition of the working fluid, and pressure in the heater and tank 2 are given as

$$\frac{d}{dt}(M_w^{TK2} h_w^{TK2}) = \dot{m}_{LI}^{RH} h_{LI}^{RH} + \dot{m}_{VI}^{RH} h_{VI}^{RH} - \dot{m}_{WLO}^{TK2} h_{WLO}^{TK2} \quad (14)$$

$$\frac{d}{dt}(M_w^{TK2}) = \dot{m}_{LI}^{RH} + \dot{m}_{VI}^{RH} - \dot{m}_{WLO}^{TK2} \quad (15)$$

$$\frac{d}{dt}(Z_w^{TK2}) = \frac{1}{M_w^{TK2}} (\dot{m}_{LI}^{RH} Z_{LI}^{RH} + \dot{m}_{VI}^{RH} Z_{VI}^{RH} - \dot{m}_{WLO}^{TK2} Z_{WLO}^{TK2}) \quad (16)$$

$$\frac{d}{dt}(P_w^{TK2} V_w^{TK2} - n_w^{TK2} R_0 T_w^{TK2}) = 0. \quad (17)$$

Absorber Model. The model structure of absorber is illustrated in Fig. 5. Neglecting the pressure loss in the condenser, the pressure in the absorber, which is a component in the low-pressure part of the power cycle (B3), is considered the same as that of the condenser and tank 1

$$P_w^{AB} = P_w^{TK1}.$$

The differential equations representing the energy, mass and composition of the working fluid in the absorber are given by

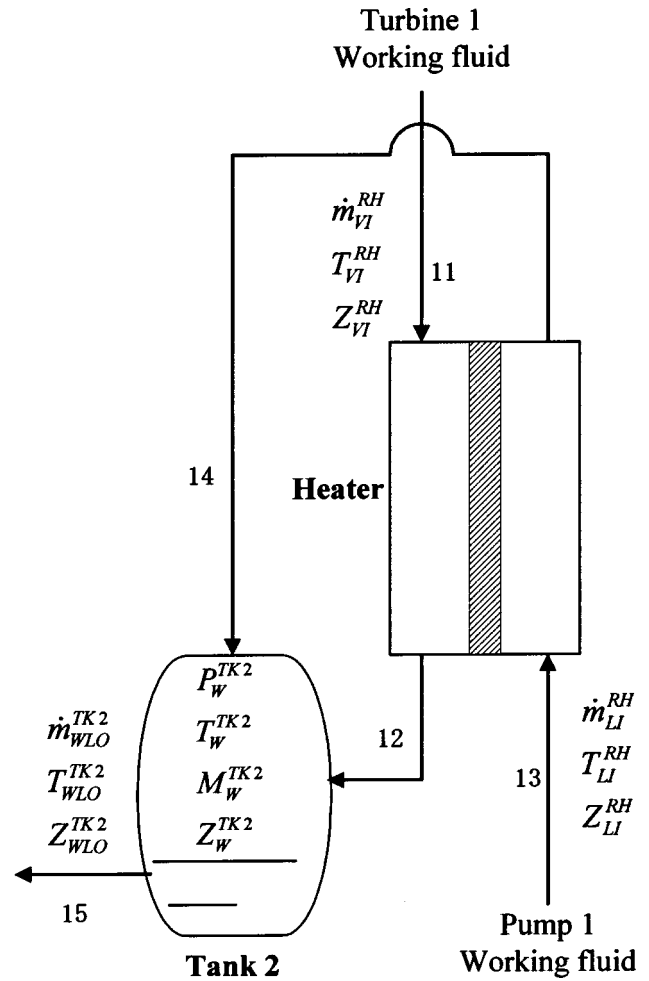


Fig. 4 Model structure of heater and tank 2

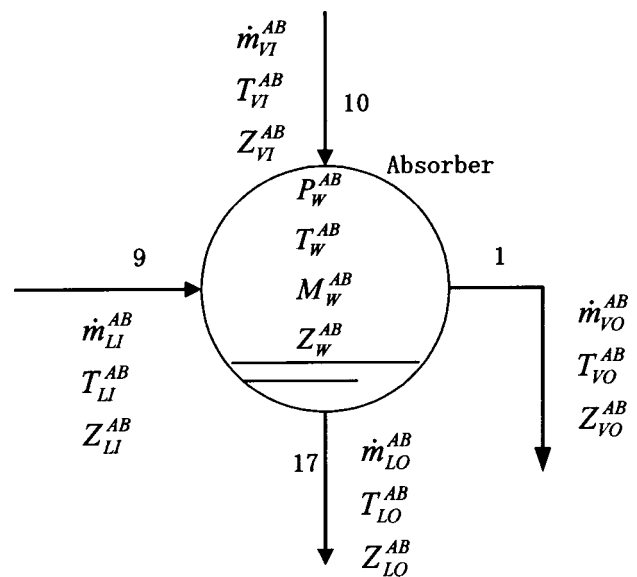


Fig. 5 Model structure of absorber

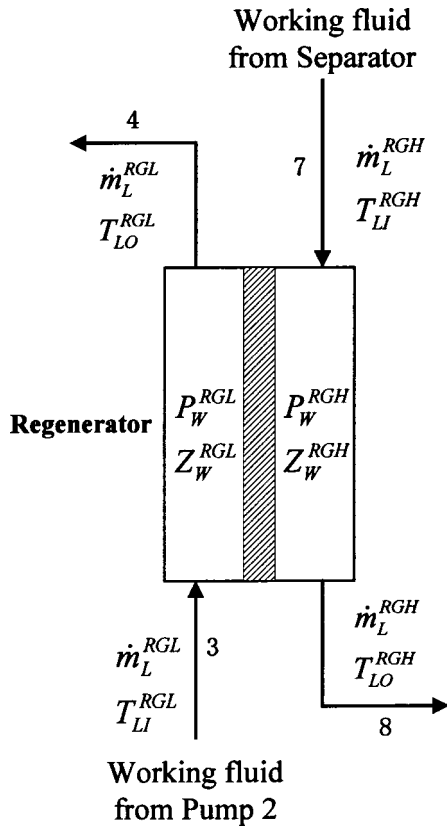


Fig. 6 Model structure of regenerator

$$\frac{d}{dt}(M_W^{AB}h_W^{AB}) = \dot{m}_{LI}^{AB}h_{LI}^{AB} + \dot{m}_{VI}^{AB}h_{VI}^{AB} - \dot{m}_{LO}^{AB}h_{LO}^{AB} - \dot{m}_{VO}^{AB}h_{VO}^{AB} \quad (18)$$

$$\frac{d}{dt}(M_W^{AB}) = \dot{m}_{LI}^{AB} + \dot{m}_{VO}^{AB} - \dot{m}_{LO}^{AB} - \dot{m}_{VO}^{AB} \quad (19)$$

$$\frac{d}{dt}(Z_W^{AB}) = \frac{1}{M_W^{AB}}(\dot{m}_{LI}^{AB}Z_{LI}^{AB} + \dot{m}_{VO}^{AB}Z_{VO}^{AB} - \dot{m}_{LO}^{AB}Z_{LO}^{AB} - \dot{m}_{VO}^{AB}Z_{VO}^{AB}). \quad (20)$$

As the volume of the working fluid is constant,

$$\frac{d}{dt}(M_W^{AB}v_W^{AB}) = 0,$$

where v_W^{AB} is the specific volume of the working fluid in saturated phase. The outlet mass flow rates can be obtained by

$$\left\{ \begin{aligned} \frac{d}{dt}\dot{m}_{LO}^{AB} &= \frac{d}{dt} \left\{ M_W^{AB} \frac{Z_{VO}^{AB} - Z_W^{AB}}{Z_{VO}^{AB} - Z_{LO}^{AB}} \right\} \\ \frac{d}{dt}\dot{m}_{VO}^{AB} &= \frac{d}{dt} \left\{ M_W^{AB} \frac{Z_W^{AB} - Z_{LO}^{AB}}{Z_{VO}^{AB} - Z_{LO}^{AB}} \right\} \end{aligned} \right. \quad (21)$$

3.3 Thermostatic Models for Regenerator, Turbine, and Diffuser

Regenerator Model. As the mass of the working fluid in the regenerator is comparatively smaller than that of the other components of the STEC plant according to the design of the regenerator in the STEC plant, a static model for the heat exchanger was used to construct the regenerator model, [16]. The model structure of the regenerator is illustrated in Fig. 6. Plate type heat exchanger is used in the regenerator. For simplicity, the residues

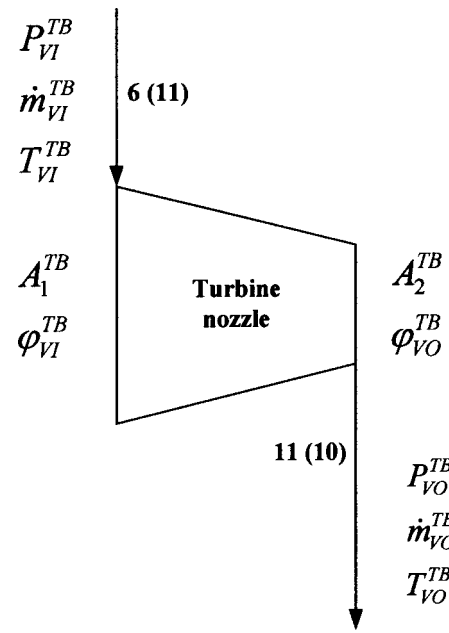


Fig. 7 Model structure of turbine

on the heat exchanger wall are neglected, and the working fluids on the both sides of the heat exchanger are considered in the liquid state.

When the inlet temperature of the working fluid from the pump 2 is T_{LI}^{RGL} ($^{\circ}\text{C}$), the temperature of the working fluid extracted from separator T_{LI}^{RGH} , and the corresponding mass flow rates $\dot{m}_L^{RGL}$ and $\dot{m}_L^{RGH}$ (kg/s), respectively, the outlet temperatures of the regenerator T_{LO}^{RGL} , T_{LO}^{RGH} are calculated by

$$T_{LO}^{RGL} = \frac{\mu T_{LI}^{RGL} W_L^{RGL} + (1 - e^{-U^{RG} \mu A^{RG}}) T_{LI}^{RGH}}{\mu W_L^{RGL} + (1 - e^{-U^{RG} \mu A^{RG}})} \quad (22)$$

$$T_{LO}^{RGH} = \frac{\mu W_L^{RGH} e^{-U^{RG} \mu A^{RG}} T_{LI}^{RGH} + (1 - e^{-U^{RG} \mu A^{RG}}) T_{LI}^{RGL}}{\mu W_L^{RGH} + (1 - e^{-U^{RG} \mu A^{RG}}) (W_L^{RGH}/W_L^{RGL})} \quad (23)$$

where A^{RG} (m^2) is the heat transfer area, and $\mu = 1/W_L^{RGH} - 1/W_L^{RGL}$.

The water equivalent W_L^{RGL} , W_L^{RGH} ($\text{J}/^{\circ}\text{C}$) are

$$\left\{ \begin{aligned} W_L^{RGH} &= \dot{m}_L^{RGH} c_W^{RGH} \\ W_L^{RGL} &= \dot{m}_L^{RGL} c_W^{RGL} \end{aligned} \right.$$

where the specific heat c_W^{RGH} , c_W^{RGL} ($\text{J}/\text{kg}^{\circ}\text{C}$) of the working fluid in liquid state are constant, and obtained using PROPATH.

In the STEC plant, the changes of the mass flow rates $\dot{m}_L^{RGL}$ and $\dot{m}_L^{RGH}$ in the regenerator are small so that their effects on the heat transfer coefficient U^{RG} ($\text{W}/\text{m}^2^{\circ}\text{C}$) are neglected. The heat transfer coefficient along the heat exchanger wall is treated as a constant, which is calculated according to the experimental value.

Turbine Model. The model construction of the turbine is shown in Fig. 7. As the mass of the working fluid in the turbine is smaller than that of the other components in the power cycle, the thermostatic model for the turbine is used, [17]. And it is assumed that the degree of reaction is zero, that is to say, there is no enthalpy drop in the rotor blade of the turbine.

The isentropic enthalpy difference between the inlet and outlet of the turbine is given by

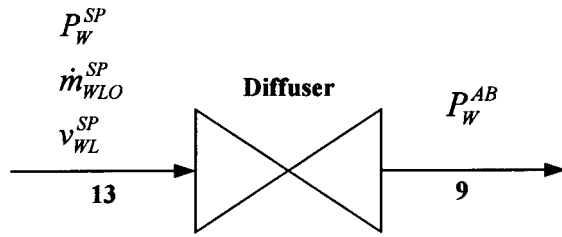


Fig. 8 Model structure of diffuser

$$h_{VI}^{TB} - h_{VOS}^{TB} = P_{VI}^{TB} v_{VI}^{TB} \frac{K_0}{K_0 - 1} \left[1 - \left(\frac{P_{VO}^{TB}}{P_{VI}^{TB}} \right)^{K_0 - 1/K_0} \right] \quad (24)$$

$$h_{VI}^{TB} - h_{VO}^{TB} = \eta_{oi}^{TB} (h_{VI}^{TB} - h_{VOS}^{TB}) \quad (25)$$

where K_0 is the heat isolation index, h_{VOS}^{TB} is the imaginary isentropic enthalpy at outlet, and η_{oi}^{TB} is the relative internal efficiency of the turbine.

The nozzle coefficient η shows the relationship between the velocities of the working fluid at the inlet of the nozzle (φ_{VI}^{TB}) (m/s) and outlet of the nozzle (φ_{VO}^{TB}) as

$$\eta = \frac{(\varphi_{VO}^{TB})^2}{(\varphi_{VI}^{TB})^2 + 2(h_{VI}^{TB} - h_{VOS}^{TB})} \quad (26)$$

The mass flow rate $\dot{m}_{VI}^{TB}$ at the inlet of the turbine can be solved from

$$\dot{m}_{VI}^{TB} = A_1^{TB} \varphi_{VI}^{TB} / v_{VI}^{TB} = A_2^{TB} \varphi_{VO}^{TB} / v_{VO}^{TB} \quad (27)$$

The power rate generated from turbine is written as

$$G^{TB} = \dot{m}_{VO}^{TB} (h_{VI}^{TB} - h_{VO}^{TB}) \quad (28)$$

Diffuser Model. The model construction of the diffuser is shown in Fig. 8. The mass flow rate of the working fluid is determined by, [18],

$$\dot{m}_{WLO}^{SP} = k \sqrt{\frac{P_W^{SP} - P_W^{AB}}{v_{WL}^{SP}}} \quad (29)$$

where k is a coefficient proportional to the ratio of opening area of diffuser.

3.4 Numerical Solution Procedure. There are totally fifteen state variables in the models for representing the states of the working fluid in the STEC components (B5). The 15 variables are described by 15 first-order differential equations, Eqs. (4), (6)–(8) for modeling the evaporator and separator, Eqs. (9)–(12) for modeling the condenser and tank 1, Eqs. (14)–(17) for modeling the heater and tank 2, Eqs. (18)–(20) for modeling the absorber. Those 15 equations should be solved simultaneously, however, it is almost impossible for a purpose of fast calculation. In this study, the following method was adopted.

In general, states of the STEC plant components are described below by a set of first-order differential equations. In terms of the state variables $(X_1, X_2, \dots, X_n)^T$ these first-order differential equations may be written in general form as

$$\begin{cases} \dot{X}_1 = f_1(X_1, X_2, \dots, X_n, U, t) \\ \dot{X}_2 = f_2(X_1, X_2, \dots, X_n, U, t) \\ \vdots \\ \dot{X}_n = f_n(X_1, X_2, \dots, X_n, U, t) \end{cases} \quad (30)$$

where $\dot{X} = dX/dt$. To keep the numerical solution simple, the first-order Euler method was used in the current study.

Table 1 STEC plant specifications

Working fluid	Ammonia and water
Overall composition (kg/kg)	0.90
Heat transfer area (m ²)	
Evaporator A^{EV}	40.6
Condenser A^{CN}	40.6
Regenerator A^{RG}	5.22
Standard mass flow rate (kg/s)	
Hot spring water $\dot{m}_s^{EV}$	25
Cool water $\dot{m}_s^{CN}$	25
Overall heat transfer coefficient (W/m ² °C)	
Evaporator U_s^{EV}	3000
Condenser U_s^{CN}	1000
Regenerator U_s^{RG}	1800
Turbine nozzle sectional area (m ²)	
Turbine inlet A_1^{TB}	0.0012
Turbine outlet A_2^{TB}	0.00056
Turbine nozzle coefficient η	0.18

The Euler method employs a Taylor series expansion truncated after the first derivative. For this case, a sufficiently small time interval Δt is required and the above equation can be written by the following equation set:

$$\begin{cases} X_1(i+1) = X_1(i) + (\Delta t) f_1(X_1(i), X_2(i), \dots, X_n(i), U(i)) \\ X_2(i+1) = X_2(i) + (\Delta t) f_2(X_1(i), X_2(i), \dots, X_n(i), U(i)) \\ \vdots \\ X_n(i+1) = X_n(i) + (\Delta t) f_n(X_1(i), X_2(i), \dots, X_n(i), U(i)) \end{cases} \quad (31)$$

where $(X_1(i), X_2(i), \dots, X_n(i))^T$ are state variables at time $i\Delta t$, and $(X_1(i+1), X_2(i+1), \dots, X_n(i+1))^T$ are those at time $(i+1)\Delta t$. From Eq. (30), one can solve for the system at time $(i+1)\Delta t$ using known state values at time $i\Delta t$. For the differential equations Eq. (30) to be stable, the time step Δt must not be too large. Of course, if the value is too small, the computer processing time becomes excessive and the roundoff error increases. The optimum time-step size can be determined by internal halving. That is, the system is first run with an arbitrary time step, the time step then halved and the system is run again. If the two results do not match, the interval halving procedure is repeated until the two successive run results agree to a present desired accuracy of 1%. Through this procedure, it has been determined that the step size 5 s is adequate and has been taken as the standard time step for the simulation.

4 Model Verification and Results

4.1 Simulation Conditions of Pilot STEC Plant. Based on the developed models, the numerical simulation for the pilot STEC plant has been implemented. Under the assumption that there is no loss in the pipes between the components of the STEC plant (B4), e.g., the temperature and mass flow rate at the inlet of a component are the same as those at the outlet of the preceding component of the STEC plant. One loop of the numerical calculation begins with the regenerator, then the evaporator and separator, turbine, diffuser, absorber, condenser, and tank 1, and ends at the heater and tank 2 (B2). Then, the loop is repeated until the end of simulation. In the simulation, the mass flow rates of the working fluid at pump 1 and pump 2, the hot and the cool water, and the ratio of opening area of diffuser were loaded as the manipulation inputs. The temperatures of the hot and the cool water were loaded as the outside conditions.

Before the simulation, the necessary specifications of the STEC plant were set, which is given in Table 1. As a starting condition it was assumed that the internal working fluid was stabilized. The thermodynamic state of the working fluid was assumed in the saturated phase in the separator and tank 1. The necessary initial conditions were set as given in Table 2.

Table 2 Initial values for the simulation of STEC plant

Turbine		
Mass flow into turbine $\dot{m}_{VI}^{TB}$ (kg/s)	0.3	
Condenser and tank 1		
Pressure P_W^{TK1} (Pa)	660,000	
Mass M_W^{TK1} (kg)	140.0	
Temperature T_W^{TK1} (°C)	15.0	
Composition Z_W^{TK1} (kg/kg)	0.92	
Heater and tank 2		
Pressure P_W^{TK2} (Pa)	785,000	
Mass M_W^{TK2} (kg)	100.0	
Temperature T_W^{TK2} (°C)	20.0	
Composition Z_W^{TK2} (kg/kg)	0.90	
Evaporator and separator		
Pressure P_W^{SP} (Pa)	850,000	
Mass M_W^{SP} (kg)	100.0	
Temperature T_W^{SP} (°C)	24.6	
Composition Z_W^{SP} (kg/kg)	0.90	
Absorber		
Mass M_W^{AB} (kg)	60.0	
Temperature T_W^{AB} (°C)	16.8	
Composition Z_W^{AB} (kg/kg)	0.92	

4.2 Model Verification Under Changeable Spring Temperature. The model verification was implemented under the changeable temperature of the spring water at the inlet of the evaporator. The temperature of the spring water T_{WSI}^{EV} versus time is shown in Fig. 9(a). The experimental data was obtained from the pilot STEC plant in Imari, Japan. At the beginning of the experiment, the whole plant was in a stable state. Although the whole plant is complicated and includes several components (B2), some crucial information for comparison is illustrated in Fig. 9, (b) and (c) temperature T_W^{SP} and pressure P_W^{SP} of the working fluid in the evaporator and separator, which were calculated using the models for evaporator and separator, (d) and (e) temperature T_W^{TK1} and pressure P_W^{TK1} of the working fluid in the condenser and tank 1, which were calculated using the models for condenser and tank 1, (f) the overall power output G^{TB} from turbine. As there was no extraction in turbine at the current experiment, the results of heater and tank 2 was not given. As the STEC plant has a closed type power cycle, the consistence of the given results also suggests the accuracy of the models in the other components. From the illustrated results, the close agreement with the computer model results is an encouraging partial confirmation of the ability of the developed models to simulate a real system.

From the actual data showing the pressure in the evaporator and separator in Fig. 9(c), the large pressure change suggests that the pressure in each STEC component cannot be regarded as time invariant. If the pressure were taken into account as a constant, the transient performance of the STEC power output G^{TB} would be also unchangeable from Eq. (26) so that the transient performance of STEC plant could not be represented sufficiently. Therefore, compared with the previous study relating to the modeling of power cycle, in which the pressure change was neglected [7], the modeling under the consideration that the temperature, pressure, mass and composition of the work fluid are time dependent (B5) in the current study enables a higher accuracy to represent the transient performance of the STEC plant.

5 Simulation Study and Discussions

5.1 Simulation Results

Temperature Step Response of Spring Water. As the temperature of the spring water changes daily, a simulation study was performed to show the transient property of the STEC plant. The

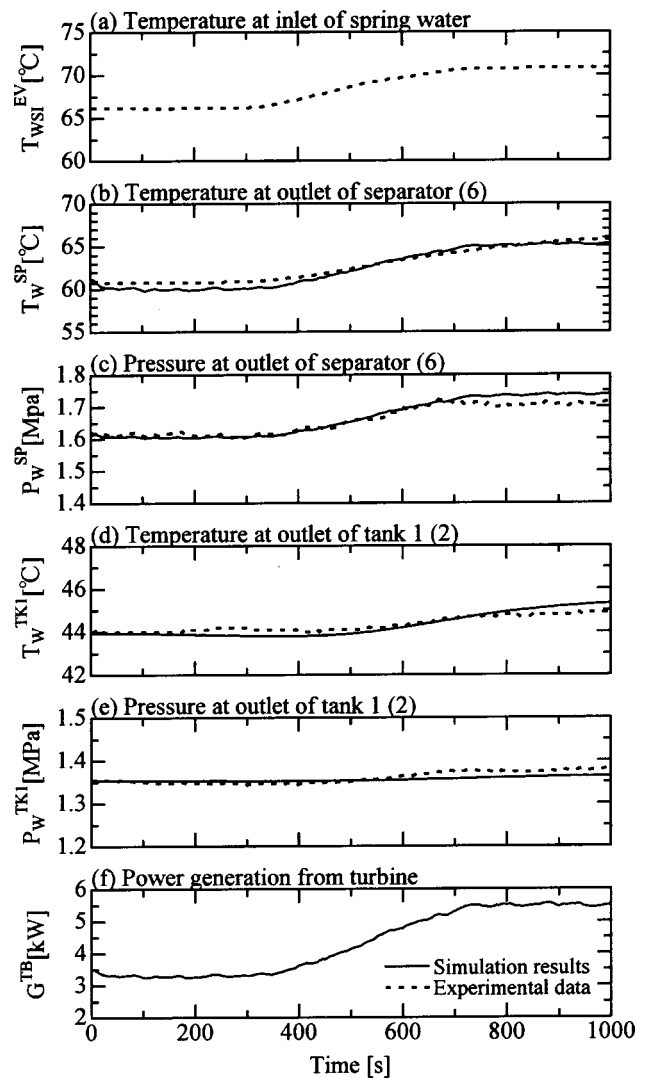


Fig. 9 Model simulation and evaluation under changeable spring water temperature. The experimental data depicted by the dotted lines were acquired from a pilot STEC plant. The consistence between the simulation results and the experimental data demonstrates that the developed models can represent the real STEC plant sufficiently.

result of the temperature step response of the spring water is illustrated in Fig. 10. The STEC plant was assumed in the stable state at the beginning of the simulation, where the same conditions of the plant state in the above verification study were used. At 500 s, the temperature T_{WSI}^{EV} of the spring water changed from 65°C to 70°C as shown in Fig. 10(a). The temperature T_W^{SP} of the working fluid in the evaporator and separator increased slowly, and after 1000 s, the temperature tended to stable about 60°C as illustrated in Fig. 10(b). The pressure P_W^{SP} increased along with the temperature, and tended to 1.575 MPa as shown in Fig. 10(c). With a short delay about 100 s after the change of the spring water, the temperature T_W^{TK1} and pressure P_W^{TK1} of the working fluid in condenser and tank 1 also increased as shown in Fig. 10(d,e). As the result, the power rate G^{TB} increased slightly in about 100 s, then, dropped to 3.8 kW. Although it is expected that the power output from turbine will increase along with a higher temperature of spring water, the simulation result shows that the power output dropped with a higher temperature of spring water. The reason is that the burden of the condenser also increases and cannot cool the vapor sufficiently so that the pressure at the outlet

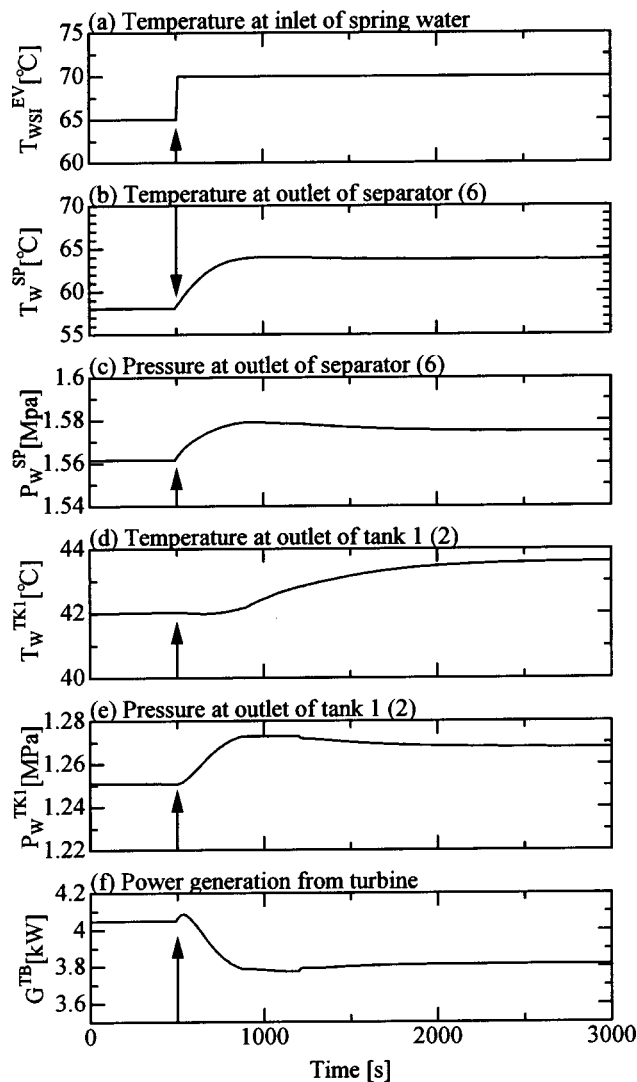


Fig. 10 Model simulation of step response of spring water temperature. The power rate G^{TB} increased slightly in about 100 s, then, dropped to 3.8 kW. The reason is that the burden of the condenser also increases and cannot cool the vapor sufficiently so that the pressure at the outlet of turbine (e.g., the pressure in condenser) increases. The illustrated characteristics are useful for developing an appropriate plant controller to achieve a stable plant operation.

of turbine increases. In order to obtain a stable operation, the following possible adjustment is described.

Flow Rate Step Response of Spring Water. One of possible ways is to adjust the mass flow rate of spring water pump. In order to develop an appropriate controller of spring water pump, a simulation study of the flow rate step response of spring water was implemented and the result was illustrated in Fig. 11. At 500 s, the mass flow rate $\dot{m}_{wsi}^{EV}$ of the spring water pump decreased from 30 kg/s to 25 kg/s as shown in Fig. 11(a). The temperature T_w^{SP} of the working fluid in the evaporator and separator decreased slowly, and after 1500 s, the temperature tended to stable about 54°C as illustrated in Fig. 11(b). The pressure P_w^{SP} decreased along with the temperature, and tended to 1.553 MPa as shown in Fig. 11(c). With a short delay about 100 s after the change of the spring water, the temperature T_w^{TK1} and pressure P_w^{TK1} of the working fluid in condenser and tank 1 also decreased as shown in Fig. 11(d,e). The power rate G^{TB} decreased slightly in about 100 s,

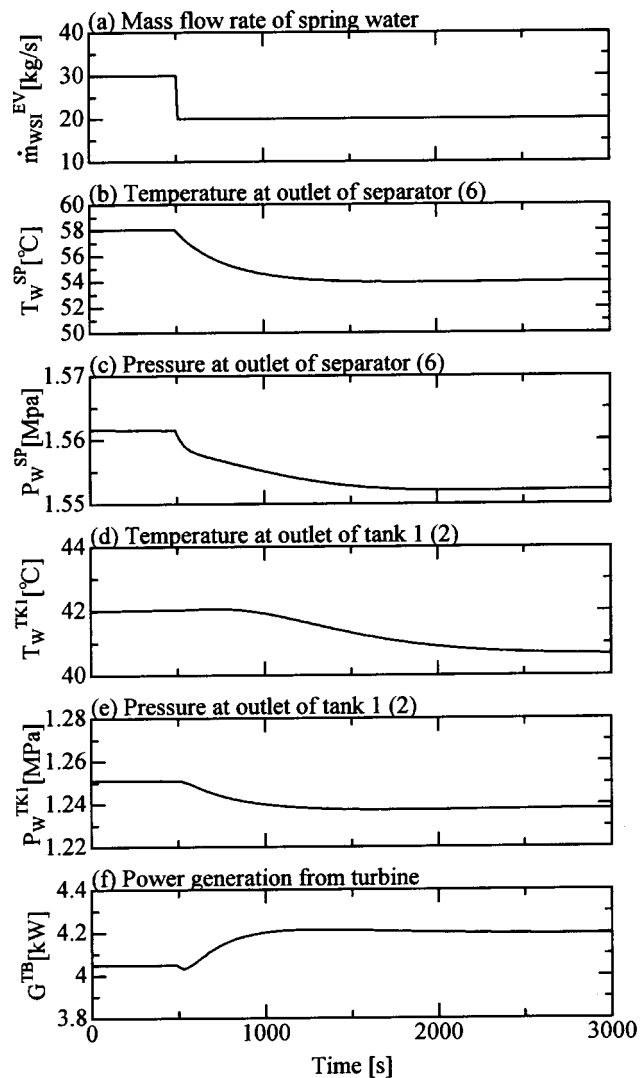


Fig. 11 Model simulation of step response of mass flow rate of spring water. The power rate G^{TB} decreased slightly in about 100 s, then, increased to 4.2 kW. From this simulation, an appropriate gain can be selected in order to develop an appropriate controller for achieving a stable power generation from turbine under the temperature change of spring water.

then, increased to 4.2 kW. From this simulation, an appropriate gain can be selected in order to develop an appropriate controller for achieving a stable power generation from turbine under the temperature change of spring water.

5.2 Modeling Power Cycle With Working Fluid of Binary Mixtures. The STEC is a totally new method of power generation, which has a small impact on the environment. In order to improve the thermal efficient of the STEC plant, a new power cycle with binary mixtures as the working fluid has been developed in our previous study. In this study, the developed dynamic models can appropriately represent the transient characteristics of the STEC plant using the new power cycle, in which the temperature, pressure, mass and composition of the working fluid are all time dependent. The accuracy of the developed models has been demonstrated by comparing the simulation result with the actual experimental data acquired from the pilot STEC plant.

In the current study, the models for the transient performance of STEC plant were constructed in the lumped parameter forms (B1) instead of distributed ones. From the demonstrated results, these simple models guarantee a rather good accuracy compared with

the actual data. Seven main components of STEC plant (B2) have been modeled, and the transient performance of STEC plant has been represented sufficiently. Nevertheless, the dynamic properties of the other components (e.g., pumps) in STEC plant should be also represented for achieving a more precise modeling of STEC plant, which might be one of the future studies. In this study, the pressure different in each part of the power cycle in STEC plant (B3) was neglected since the STEC plant is a compact plant so that the approximation will not reduce the model accuracy for representing the plant dynamics, which has been demonstrated in the simulation study. Similarly, the neglect of temperature change in pumps and pipe loss (B4) is also appropriate for the STEC plant. As the binary mixtures are used as the working fluid in STEC plant, the temperature, pressure, mass and composition are all time-dependent (B5) and represented by the first-order differential equations. The demonstrated simulation results demonstrate that the constructed models represent the state variables sufficiently for the analysis of the transient performance of the STEC plant.

5.3 Prospects in Achieving a Steady Operation With High Efficiency. As the temperature of the spring water has a great effect on the power generation of the STEC plant and the temperature of the spring water is time variant, it is difficult to achieve a stable operation of the STEC plant. Since the developed simulation system enables the transient analysis of power cycle using the binary mixtures with a reasonable accuracy, the dynamic characteristics of the STEC plant can be obtained, which is useful for developing an appropriate plant controller. As shown in Fig. 10 and Fig. 11, the step responses of the STEC plant have been achieved using the developed simulation system so that an adequate controller can be constructed by selecting the gain and time constant from the simulation results. Therefore, a steady operation of the STEC plant with a higher thermal efficiency becomes possible under the developed models, which can represent the dynamic characteristics of the STEC plant sufficiently.

6 Conclusion

The simulation model presented here is a new attempt, which may act as a beginning to a thorough understanding of the behavior of the working fluid of binary mixtures in the power cycle. This presentation demonstrates that the developed simulation system is quite possible, and offers a far less expensive method to understand the transients of STEC power plant than a totally hardware approach.

Acknowledgments

The authors would like to thank Dr. S. Goto, Department of Advanced Systems Control Engineering, Graduate School of Science and Engineering, Saga University, Japan, Dr. S. Jitsuhara and Mr. K. Fukumiya, Xenosys Inc., Japan, for their valuable suggestions and experimental guidance.

Nomenclature

A	= heat transfer area (m^2)
c	= specific heat ($\text{J/kg } ^\circ\text{C}$)
E	= internal energy (J)
h	= specific enthalpy (J/kg)
$\dot{m}$	= mass flow rate (Kg/s)
M	= mass (kg)
n	= amount of moles (kmol)
N	= molecular weight (kg/kmol)
K_0	= heat insulation index;
P	= pressure (MPa)
Q	= heat transfer rate (W)
R_0	= universal gas constant ($\text{J/kmol } ^\circ\text{C}$)
T	= temperature ($^\circ\text{C}$)
U	= overall heat transfer coefficient ($\text{w/m}^2\text{ } ^\circ\text{C}$)

v	= specific volume (m^3/kg)
V	= volume (m^3)
W	= water equivalent (J/kg)
Z	= composition (kg/kg)
η	= turbine nozzle coefficient (%)
η_{oi}	= relative internal efficient of turbine (%)
φ	= velocity (m^3/s)

Superscripts

AB	= absorber
CN	= condenser
EV	= evaporator
RH	= heater
RG	= regenerator
SP	= separator
TB	= turbine
$TK1$	= tank 1
$TK2$	= tank 2

Subscripts

CS	= cool water
I	= inlet
L	= liquid
O	= outlet
V	= vapor
W	= working fluid
WS	= spring water

References

- [1] Vadus, J. R., 1999, "Ocean Technology and Ocean Energy in the 21st Century," *Proc. the International OTEC/DOWA Conference '99*, Imari, Japan, Saga University, Saga, Japan, pp. 10–22.
- [2] Heydt, G. T., 1993, "An Assessment of Ocean Thermal Energy Conversion as an Advanced Electric Generation Methodology," *Proc. IEEE*, **81**(3), pp. 409–418.
- [3] Uehara, H., Kusuda, H., Monde, M. et al., 1980, "Performance Analysis of Ocean Thermal Energy Conversion," *Therm. Nucl. Power*, **31**(5), pp. 549–559 (in Japanese).
- [4] Uehara, H., Ikegami, Y., Mitsumori, T., Sasaki K., and Nogami R., 1999, "The Experimental Research on Ocean Thermal Energy Conversion Using Uehara Cycle," *Proc. the International OTEC/DOWA Conference '99*, Imari, Japan, Saga University, Saga, Japan, pp. 132–135.
- [5] Nakamura, M., Ikegami, Y., and Uehara, H., 1988, "Computer Simulation for OTEC System Design: Pump Control of Flow Rate," *Trans. Jpn. Soc. Refrig. Air Cond. Eng.*, **5**(2), pp. 247–253 (in Japanese).
- [6] Janik, C. J. et al., 1998–1999, "Physical, Chemical and Isotopic Data for Samples From the Anderson Spring area, Lake County, CA, 1998–1999," at <http://geopubs.wr.usgs.gov/open-file/of99-585/>.
- [7] Uehara, H., and Ikegami, Y., 1990, "Optimization of a Closed Cycle OTEC System," *ASME J. Sol. Energy Eng.*, **112**, pp. 247–256.
- [8] Uehara, H., and Ikegami, Y., 1990, "Optimization of a Closed Cycle OTEC System," *ASME J. Sol. Energy Eng.*, **112**(4), pp. 247–256.
- [9] Nakamura, M., Jitsuhara, S., Isogai, H., and Uehara, H., 1991, "Computer Simulation Developments for OTEC Plant Design and Control," *Trans. Soc. Instrument Control Eng.*, **27**(1), pp. 107–114 (in Japanese).
- [10] Uehara, H., Ikegami, Y., and Nishida, T., 1998, "Performance Analysis of OTEC System Using a Cycle With Absorption and Extraction Processes," *Trans. Jpn. Soc. Mech. Eng., Ser. B*, **64**(624), pp. 384–389.
- [11] Kalina, A. I., 1987, "Regeneration of the Working Fluid and Generation of Energy," Japanese Patent, Sho62-39660.
- [12] Uehara, H., Ikegami, Y., and Nishida, T., 1995, "OTEC System Using a New Cycle With Absorption and Extraction Processes," *Physical Chemistry of Aqueous System*, White et al., eds., Begell House, Washington, DC, pp. 862–869.
- [13] Itoi, R. et al., 2001, "A Program Package for Thermal-Physical Properties of Fluids," Kyushu University, Fukuoka, Japan, <http://propath.mech.kyushu-u.ac.jp/>.
- [14] Nakaoka, T., and Uehara, H., 1988, "Performance Test of Shell-and-Plate Evaporator for OTEC," *Exp. Therm. Fluid Sci.*, **1**(3), pp. 283–291.
- [15] Jones, J. B., and Hawkins, G. A. 1986, *Engineering Thermodynamics*, Jone Wiley and Sons, New York, pp. 661–707.
- [16] Tanishita, I., 1998, *Basic Thermodynamics*, Shokabo Inc., Tokyo, pp. 319–322 (in Japanese).
- [17] Yasui, S. (1977). *Turbo Mechanics I*, Jikkyo Inc., Tokyo, pp. 53–54 (in Japanese).
- [18] Koury, R. N. N., Machado, L., and Ismail, K. A. R., 2001, "Numerical Simulation of a Variable Speed Refrigeration System," *Int. J. Refrig.*, **24**, pp. 192–200.

A Novel Approach to the Optimization of Reaction Rate Parameters for Methane Combustion Using Multi-Objective Genetic Algorithms

L. Elliott
D. B. Ingham

Department of Applied Mathematics,
University of Leeds,
Leeds LS2 9JT, UK

A. G. Kyne¹
e-mail: fueagk@sun.leeds.ac.uk

N. S. Mera
M. Pourkashanian

Energy & Resources Research Institute,
University of Leeds,
Leeds LS2 9JT, UK

C. W. Wilson
Department of Mechanical Engineering,
University of Sheffield,
Mappin Street,
Sheffield S1 3JD, UK

This study uses a multi-objective genetic algorithm to determine new reaction rate parameters (A 's, β 's and E_a 's in the non-Arrhenius expressions) for the combustion of a methane/air mixture. The multi-objective structure of the genetic algorithm employed allows for the incorporation of both perfectly stirred reactor and laminar premixed flame data in the inversion process, thus enabling a greater confidence in the predictive capabilities of the reaction mechanisms obtained. Various inversion procedures based on reduced sets of data are investigated and tested on methane/air combustion in order to generate efficient inversion schemes for future investigations concerning complex hydrocarbon fuels. The inversion algorithms developed are first tested on numerically simulated data. In addition, the increased flexibility offered by this novel multi-objective GA has now, for the first time, allowed experimental data to be incorporated into our reaction mechanism development. A GA optimized methane-air reaction mechanism is presented which offers a remarkable improvement over a previously validated starting mechanism in modeling the flame structure in a stoichiometric methane-air premixed flame (<http://www.personal.leeds.ac.uk/~fuensm/project/mech.html>). In addition, the mechanism outperforms the predictions of more detailed schemes and is still capable of modeling combustion phenomena that were not part of the optimization process. Therefore, the results of this study demonstrate that the genetic algorithm inversion process promises the ability to assess combustion behavior for fuels where the reaction rate coefficients are not known with any confidence and, subsequently, accurately predict emission characteristics, stable species concentrations and flame characterization. Such predictive capabilities will be of paramount importance within the gas turbine industry. [DOI: 10.1115/1.1760531]

Introduction

Combustion may be modelled using a system of chemical reactions for which the rates of each reaction are known. Databases holding measurements of the reaction rate parameters (A 's, β 's and E_a 's) are available and these provide the logarithm of the reaction rate as a function of $1/T$, where T is the temperature. However, there is a considerable amount of uncertainty in the rate data which causes problems to occur when calculating concentration of species for the combustion products away from their equilibrium values.

The Gas Research Institute (GRI) initiated an extensive research program for the optimization of the chemical scheme with regard to natural gas. Precise details of the optimization scheme in that program are discussed in detail by Bowman et al. [1]. At temperatures above about 1000 K, combustion of simple fuels, such as hydrogen, can be calculated using an eight-step reaction mechanism for which the reaction rate data is known with confidence, see Dixon-Lewis et al. [2]. However, complex hydrocarbon fuels, such as kerosene, require more than 1000 reaction steps with over 200 species, see Dagaut et al. [3]. Hence as all the

reaction rate data is not well known there is a high degree of uncertainty in the results obtained using these large detailed reaction mechanisms.

We note that various methods have been proposed in order to find a set of reaction rate parameters that gives the best fit to a given set of experimental data. Indeed, once the problem is reformulated as an optimization problem, the object function may be minimized/maximized by various methods. However, in the case of complex hydrocarbon fuels, the object function is usually highly structured, having multiple ridges and valleys and exhibiting multiple local optima. For object functions with such a complex structure, traditional gradient based algorithms, see for example Rabitz et al. [4], are likely to fail. Optimization methods based upon linearization of the objective function, see Milstein [5] or Bock [6], fall into the same category. A different method, the *solution mapping method*, was proposed by Frenklach et al. [7] and is based on approximating the surface of the object function by *polynomial response surfaces* generated by means of a relatively small number of computer experiments. Although efficient from the point of view of computational time, the method of *solution mapping* has a major drawback since the optimization is based not on the real values of the object function but on a *polynomial response surface* approximating the real landscape of the object function. For a highly structured surface with multiple valleys and hills, the errors in approximating the object function are large and these errors are likely to affect the results of the optimization procedure. On the other hand, genetic algorithms are particularly suitable for optimizing object functions with complex, highly structured landscapes; see, for example, Michalewicz [8].

¹To whom correspondence should be addressed.

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Atlanta, GA, June 16–19, 2003, Paper No. 2003-GT-38018. Manuscript received by IGTI, October 2002, final revision, March 2003. Associate Editor: H. R. Simmons.

A powerful inversion technique, based on a single objective binary encoded genetic algorithm (GA), was developed by Harris et al. [9] in order to determine the reaction rate parameters for the combustion of a hydrogen/air mixture in a perfectly stirred reactor (PSR). This technique was further developed by Elliott et al. [10] by incorporating constraints associated with the experimental data listed in the National Institute of Standards and Technology (NIST) database. The original binary encoding was also replaced by floating point encoding. An extension of this single-objective genetic algorithm procedure to methane/air combustion was also considered in Elliott et al. [11]. A multi-objective genetic algorithm which incorporates both perfectly stirred reactor and laminar premixed flames data in the inversion process was developed in Elliott et al. [12] for hydrogen/air combustion.

It is the purpose of this paper to extend to the case of methane/air mixture combustion such a procedure that takes into account measurements from both laminar premixed flames and perfectly stirred reactors. Inversion procedures based on a reduced set of available measurements are also considered. In the previous studies by Harris et al. [9] and Elliott et al. [10–12] the inversion calculations were performed on sets of numerically simulated data. In this study we consider both numerically simulated data and experimental data in order to test the robustness and stability of the inversion procedure proposed, thus enabling a greater confidence in the predictive capabilities of the inversion scheme, before proceeding to complex hydro/carbon fuels.

Description of the Problem

Reaction Rate Parameters. In a combustion process, the net chemical production or destruction rate of each species results from a competition between all the chemical reactions involving that species. In this study it is assumed that each reaction proceeds according to the law of mass action and the forward rate coefficients are in the three parameter functional Arrhenius form, namely

$$\text{Rate: } k_{fi} = A_i T^{\beta_i} \exp\left(-\frac{E_{a_i}}{RT}\right) \quad (1)$$

for $i = 1, \dots, N_R$, where R is the universal gas constant and there are N_R competing reactions occurring simultaneously. The rate Eq. (1) contain the three parameters A_i , β_i , and E_{a_i} for the i th reaction. These parameters are of paramount importance in modeling the combustion processes since small changes in the reaction rates may produce large deviations in the output species concentrations. In both perfectly stirred reactors, as well as in laminar premixed flames, if the physical parameters required are specified, and the reaction rates (1) are given for every reaction involved in the combustion process, then the species concentrations of the combustion products may be calculated. It is the possibility of the determination of these parameters for each reaction, based upon outlet species concentrations, which is investigated in this paper.

Various software packages may be used for the direct calculations to determine the output species concentrations if the reaction rates are known. For these direct calculations we have used the PSR code, see Glarborg et al. [13], and the PREMIX code, see Kee et al. [14], which are described in the next sections.

The Perfectly Stirred Reactor Calculations (PSR) and the Laminar Premixed Flames (PREMIX) Calculations. The perfectly stirred reactor calculations are performed using the PSR FORTRAN computer program that predicts the steady-state temperature and composition of the species in a perfectly stirred reactor, see Glarborg et al. [13] while the laminar premixed flame structure calculations were performed using the PREMIX code, see Kee et al. [14].

We use X_k to denote the mole fraction of the k th species, where $k = 1, \dots, K$ and K represents the total number of species. The

temperature and composition which exit the reactor are assumed to be the same as those in the reactor since the mixing in the reactor chamber is intense.

If PSR calculations are undertaken for N_S^{PSR} different sets of reactor conditions then the output data which is used in the GA search procedure will consist of a set of KN_S^{PSR} mole concentrations, $(X_{jk}, j = 1, \dots, N_S^{\text{PSR}}, k = 1, \dots, K)$, where X_{jk} represents the mole concentration of the k th species in the j th set of reactor conditions.

The laminar premixed flame structure calculations were performed using the PREMIX code for burner stabilized flames with a known mass flow rate. The code is capable of solving the energy equation in determining a temperature profile, however, throughout this investigation a fixed temperature profile was preferred. The reasons for fixing the temperature were two fold. First, in many flames there can be significant heat losses to the external environment, which are of unknown and questionable origin and thus troublesome to model. Secondly, the CPU time involved in arriving at the final solution was decreased significantly.

If all the physical operating conditions are specified then, for a given set of reaction rates (1), the profiles of the species concentration and the burning velocity as functions of the distance from the burner's surface are calculated. If PREMIX calculations are performed for N_S^{PREMIX} different sets of operating conditions then the output data of the code which is used by the GA in the matching process consists of a set of species concentration profiles $(Y_{jk}(x), j = 1, \dots, N_S^{\text{PREMIX}}, k = 1, \dots, K)$ and a set of burning velocity profiles $(V_j(x), j = 1, \dots, N_S^{\text{PREMIX}})$, where Y_{jk} is the profile of the mole concentration above the burner for the k th species in the j th set of operating conditions and V_j is the burning velocity profile for the j th set of operating conditions.

Both PSR and PREMIX programs are not stand-alone programs, they are designed to be run in conjunction with pre-processors from the CHEMKIN library, see Kee et al. [15], which handles the chemical reaction mechanism and the thermodynamic and transport properties.

It is the purpose of this paper to determine the reaction rate coefficients A 's, β 's and E_a 's in Eq. (1), given species concentrations X_{jk} in a perfectly stirred reactor and/or profiles of species concentration Y_{jk} and the burning velocity profiles V_j in laminar premixed flames.

Reformulation of the Problem as an Optimization Problem

An inverse solution procedure attempts, by calculating new reaction rate parameters that lie between predefined boundaries, to recover the profiles of the species (to within any preassigned experimental uncertainty) resulting from numerous sets of operating conditions. The inversion process aims to determine the unknown reaction rate parameters $((A_i, b_i, E_{a_i}), i = 1, \dots, N_R)$ that provide the best fit to a set of given data. Thus, first a set of output concentration measurements of species is simulated (or measured). If numerical simulations are performed for N_S^{PSR} different sets of PSR operating conditions and/or N_S^{PREMIX} different sets of PREMIX operating conditions then the data will consist of a set of KN_S^{PSR} concentration measurements of species for PSR calculations and $(K+1)N_S^{\text{PREMIX}}$ profiles of species concentrations and burning velocities for PREMIX calculations.

Genetic algorithms based inversion procedures seek for the set of reaction rate parameters that gives the best fit to these measurements. This can be achieved by looking for the maximum of either of the functions

$$f_{\text{PSR}}((A_i, b_i, E_{a_i})_{i=1, N_R}) = \left\{ \varepsilon + \sum_{j=1}^{N_S} \sum_{k=1}^K W_k \frac{|X_{jk}^{\text{calc}} - X_{jk}^{\text{orig}}|}{X_{jk}^{\text{orig}}} \right\}^{-1} \quad (2)$$

$$f_{\text{PREMIX}}((A_i, b_i, E_i)_{i=1, N_R}) = \left\{ \varepsilon + \sum_{j=1}^{N_S} \left(\frac{\|V_j^{\text{calc}} - V_j^{\text{orig}}\|_{L^2}}{\|V_j^{\text{orig}}\|_{L^2}} + \sum_{k=1}^K W_k \frac{\|Y_{jk}^{\text{calc}} - Y_{jk}^{\text{orig}}\|_{L^2}}{\|Y_{jk}^{\text{orig}}\|_{L^2}} \right) \right\}^{-1} \quad (3)$$

where

- X_{jk}^{calc} , Y_{jk}^{calc} , and V_j^{calc} represent the mole concentrations of the k th species in the j th set of operating conditions and the burning velocity in the j th set of operating conditions using the set of reaction rate parameters $((A_i, \beta_i E_{a_i}), i = 1, \dots, N_R)$,
- X_{jk}^{orig} , Y_{jk}^{orig} , and V_j^{orig} are the corresponding original values which were measured or simulated using the exact values of the reaction rate parameters,
- $\|\cdot\|_{L^2}$ represents the L^2 norm of a function which is numerically calculated using a trapezoidal rule,
- W_k are different weights (lying in the range between 0 and 1) that can be applied to each species depending on the importance of the species. Initially all these weights are set to be equal to 1.0. These weights are set to zero for those species for which measurements or numerically simulated data are not available.
- ε is a constant added to the fitness function in order to avoid numerical overflow. We note that the maximum value of these object functions is $1/\varepsilon$, which indicates a perfect fit to the data. A small value, namely, $\varepsilon = 10^{-8}$ was used in order to increase the selective pressure applied by the GA. However, similar results are obtained with various other values.

The Multi-Objective Genetic Algorithm Inversion Technique

There is a vast variety of genetic algorithms approaches to multi-objective optimization problems. GAs are in general a very efficient optimization technique for problems with multiple, often conflicting, object functions, see Van Veldhuizen and Lamont [16]. However, most GA approaches require that all the object functions are evaluated for every individual generated during the optimization process. For the problem considered in this paper, the object function f_{PREMIX} requires N_S^{PREMIX} executions of the PREMIX code, which has a long run time. On the other hand, PSR calculations are much faster. Therefore, in this paper we use multi-objective GA developed in Elliott et al. [12] that avoids PREMIX evaluations at every generation. The algorithm is based on alternating the fitness functions that are used for selecting the survivors of the populations generated during the GA procedure. Under the assumptions that the optimization problem considered has n objective functions, the multi-objective GA employed consists of the following steps:

- Construct n fitness functions $f_1, f_2, \dots, f_n$ and assign a probability for each of these functions $p_1, p_2, \dots, p_n$.
- Construct an initial population.
- Select a fitness function f_i with the probability p_i using a roulette wheel selection.
- Using the fitness function selected at step (iii) construct a new population.
 - Use the fitness function f_i to select a mating pool.
 - Apply the genetic operators (crossover and mutation) to create a population of offsprings.
 - Use the fitness function f_i to construct a new population by merging the old population with the offsprings population.
- Repeat steps (iii)–(iv) until a stopping condition is satisfied, i.e., convergence is achieved.

We note that the GA employed is an extension for multi-objective optimization of the floating point encoded GA developed in Michalewicz [8] and used for reaction rates retrieval by Elliott et al. [10]. The genetic operators and the GA parameters were taken to be as follows:

- population size $n_{\text{pop}} = 50$
- number of offspring $n_{\text{child}} = 60$
- uniform arithmetic crossover, crossover probability $p_c = 0.65$
- tournament selection, tournament size $k = 2$, tournament probability $p_t = 0.8$
- non-uniform mutation, mutation probability $p_m = 0.5$
- elitism, elitism parameter $n_e = 2$

We note that the non-uniform mutation operator, see Michalewicz [8], is an operator that enables fine local tuning by searching the space uniformly during the first generations and locally at later stages. This enables a high level of exploration during the first few generations when the GA is searching for promising regions in the parameter space, while at later stages the solutions found are refined by analysing the effects on the fitness function of small perturbations applied to the reaction rate parameters.

The new sets of rate constants generated every iteration are constrained to lie between predefined boundaries that represent the uncertainty associated with the experimental findings listed in the National Institute of Standards and Technology (NIST) database, see Elliott et al. [10].

Numerical Results

The first task for the genetic algorithm was to recover the species concentration profiles predicted by a previously validated hydrogen/air reaction mechanism for a wide range of reactor conditions. Thus, numerical computations were undertaken on a set of numerically simulated data. The reaction scheme used to generate the data is a reduced version of GRI-mechanism 1.2, with 24 species and 103 reactions (see Kazakov and Frenklach <http://www.me.berkeley.edu/drm/>), the same as that previously used by Elliott et al. [11]. The inversion procedure developed is also tested on experimental data for which numerical results are presented in Sec. 5.3.

Inversion Procedures Using Numerically Simulated Data

The test problem considered is the combustion of a methane/nitrogen/oxygen mixture at a constant atmospheric pressure, $p = 1$ atm. A set of $N_S^{\text{PSR}} = 20$ different operating conditions have been considered for PSR runs corresponding to various changes to the fixed reactor temperature (energy equation is not solved) $T = 1500$ K, 1750 K, 2000 K, 2250 K, and 2500 K and equivalence ratios $\Phi = 0.5, 1.0, 1.5$, and 2.0. The reactor has a volume $V = 50.0$ cm³ and the nominal residence time is $\tau = 3.0E-2$ s. We assume that there is no heat loss from the reactor, i.e., $Q = 0$, and the temperature profile is specified, rather than solving the energy equation.

For PREMIX calculations a set of $N_S^{\text{PREMIX}} = 4$ operating conditions, corresponding to various values of the equivalence ratios, namely $\Phi = 0.5, 1.0, 1.5$, and 2.0 for a steady-state burner stabilized premixed laminar flames with specified temperature, pressure $p = 1$ atm and mass flow rate 9.89E-3 (g/cm²/sec).

The reaction mechanisms developed in the previous GA formulations for methane combustion, see Elliott et al. [11], were generated using complete sets of data, i.e., the concentrations of all the output species were assumed to be known for a set of operating conditions. However, in many practical situations, measurements are available only for the most important species. Therefore it is the purpose of this section to test and compare the GA inversion procedure for different sets of available data. Three reaction mechanisms are generated using three sets of data that simulate complete or incomplete sets of measurements as follows:

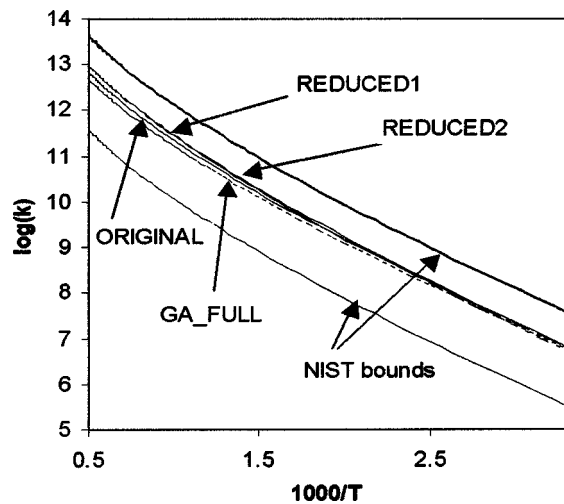


Fig. 1 The reaction rate k for the reaction $O+H_2 \rightleftharpoons H+OH$ given by the inversion procedures employed in various GA formulations in comparison with the original reaction rate and the NIST constraints imposed on the reaction rate

- GA_FULL—is generated using multi-objective optimization based on PSR and PREMIX data only for all the 24 species.
- REDUCED1—is generated using multi-objective optimization based on PSR and PREMIX data only for the major species, namely, CO_2 , CO , CH_4 , OH , O_2 , H_2O , and N_2 .
- REDUCED2—is generated using multi-objective optimization based on PSR and PREMIX data for the major species CO_2 , CO , OH , O_2 , H_2O , and N_2 . One extra measurement is given by the total additive concentration of the species CH_4 , CH_3 , CH_2 , $CH_2(S)$, C_2H_3 , C_2H_4 , C_2H_5 , and C_2H_6 .

We note that incorporation of measurements for only some of the species in the inversion process is done by setting the appropriate values (1.0 or 0.0) for the weights W_k in Eqs. (2) and (3).

It is the purpose of this section to investigate the convergence and the efficiency of the multi-objective genetic algorithm proposed, GA_FULL, and to compare it with the reduced inversion schemes REDUCED1 and REDUCED2.

Figure 1 presents the reaction rate retrieved for one of the reactions considered, namely $O+H_2 \rightleftharpoons H+OH$, by the three inversion formulations considered in comparison with the original reaction rates and the NIST constraints on the reaction rate, constructed as described in Elliott et al. [10]. We note that the reaction rates produced by the mechanisms REDUCED1 and REDUCED2 are graphically indistinguishable. It can be seen that all three formulations are efficient in retrieving the reaction rate. However, a more efficient way to compare the reaction mechanisms generated is by analyzing the predictions that they provide for the various chemical processes since for example very small variations in the reaction rates may produce large deviations in the predicted species concentrations.

Table 1 presents the species concentration calculated with the PSR program, using the reaction mechanisms generated in the GA inversion procedure and the original mechanism for $T=2000$ K and the air/fuel ratio given by $\Phi=1.0$. It can be seen that all the major species are accurately predicted by all three mechanisms generated by the inversion procedure investigated. Naturally, the GA_FULL mechanism outperforms the other two mechanisms since it used more information to generate the species' concentration profiles. However, also the reaction mechanisms REDUCED1 and REDUCED2 are accurate in predicting the mole fractions of the major species. Large percentage errors are obtained only for those species which have a very low concentration being of a reduced importance to the combustion process.

Similar results are obtained for various operating conditions, see Figs. 2 and 3 which present the species concentrations predicted by the three reaction mechanisms developed by the GA inversion procedure for the air/fuel ratio given by $\Phi=1.0$ and various temperatures, in comparison with the concentrations predicted by the original GRI mechanism. It can be seen that all three mechanisms are very efficient in predicting the mole fractions of two of the major species, namely O_2 , Fig. 2, and CO_2 , Fig. 3, for various operating conditions. Although not presented here, it is reported that all the major species were predicted with high accu-

Table 1 The mole fractions of the output species as calculated by PSR for $\Phi=1.0$, $T=1500$ K and $p=1$ atm using the reaction mechanisms generated by the inverse GA formulations considered

species	original	GA_FULL		REDUCED1		REDUCED2	
		calculated	% error	calculated	% error	calculated	% error
H_2	5.02E-03	5.03E-03	0.25	5.02E-03	0.00	5.02E-03	0.00
H	9.07E-04	9.12E-04	0.48	9.08E-04	0.04	9.08E-04	0.05
O	4.34E-04	4.35E-04	0.22	4.33E-04	0.13	4.33E-04	0.12
O_2	7.17E-03	7.19E-03	0.17	7.17E-03	0.04	7.17E-03	0.00
OH	3.27E-03	3.27E-03	0.23	3.27E-03	0.07	3.27E-03	0.03
H_2O	1.81E-01	1.81E-01	0.02	1.81E-01	0.00	1.81E-01	0.00
HO_2	1.12E-06	1.19E-06	5.68	1.31E-06	16.48	1.25E-06	11.38
H_2O_2	1.97E-07	1.96E-07	0.86	1.80E-07	8.84	1.76E-07	11.09
CH_2	9.59E-07	1.09E-06	14.12	1.44E-06	50.56	1.48E-06	54.65
$CH_2(S)$	5.49E-08	5.57E-08	1.48	6.68E-08	21.60	6.87E-08	25.08
CH_3	7.34E-06	7.22E-06	1.62	6.24E-06	14.99	6.38E-06	13.08
CH_4	1.47E-05	1.47E-05	0.16	1.47E-05	0.32	1.55E-05	5.19
CO	1.08E-02	1.09E-02	0.14	1.08E-02	0.01	1.08E-02	0.02
CO_2	8.33E-02	8.33E-02	0.02	8.33E-02	0.00	8.33E-02	0.00
HCO	1.19E-07	1.20E-07	0.78	9.57E-08	19.52	9.66E-08	18.79
CH_2O	3.04E-06	3.04E-06	0.14	2.47E-06	18.66	1.45E-06	52.18
CH_3O	5.69E-10	6.58E-10	15.72	5.32E-10	6.46	5.06E-10	11.07
C_2H_2	1.35E-08	1.26E-08	7.15	1.27E-08	6.05	1.33E-08	1.45
C_2H_3	3.37E-10	3.14E-10	6.94	4.04E-10	19.68	4.09E-10	21.30
C_2H_4	7.59E-09	7.33E-09	3.45	4.06E-10	94.65	6.30E-10	91.69
C_2H_5	4.58E-11	4.60E-11	0.47	3.83E-11	16.25	4.12E-11	10.06
C_2H_6	1.77E-10	1.50E-10	15.21	9.87E-11	44.29	1.16E-10	34.64
N_2	7.08E-01	7.08E-01	0.00	7.08E-01	0.00	7.08E-01	0.00
AR	0.00E+00	0.00E+00	0.00	0.00E+00	0.00	0.00E+00	0.00

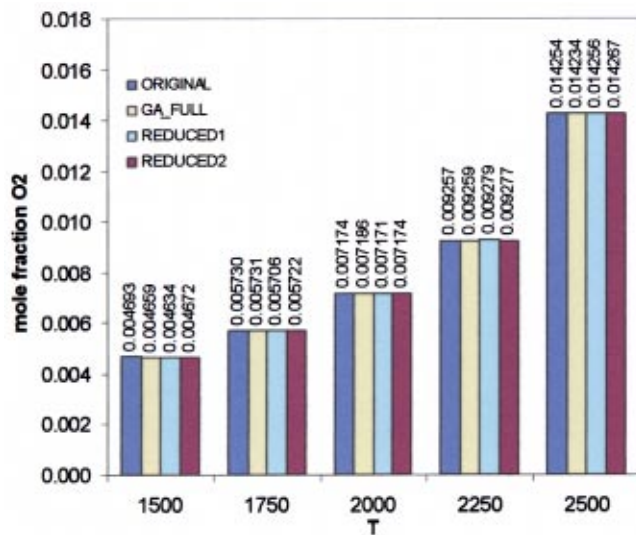


Fig. 2 The mole fractions of O_2 for various temperatures, as calculated by PSR using the reaction mechanisms generated by the inverse GA formulations considered

racy for all the 20 PSR operating conditions considered in the inversion process. Thus we may conclude that the three mechanisms generated, GA_FULL, REDUCED1 and REDUCED2 are efficient in predicting the PSR combustion process for any operating conditions from the range which was used in the inversion process. The predictive capabilities of these mechanisms outside the operating conditions range used in the inversion process are investigated in the next section.

The profiles of the mole fractions in PREMIX calculations are also accurately predicted by all three mechanisms considered, see Figs. 4 and 5 which presents the species profiles above the burner for O_2 (Fig. 4) and CO_2 (Fig. 5) as predicted by the three reaction mechanisms generated by GA. We note that the GA_FULL mechanism is, as expected, slightly more accurate in predicting the species profiles, but also the reduced mechanisms are efficient in predicting the profiles of the mole fractions of species above the burner.

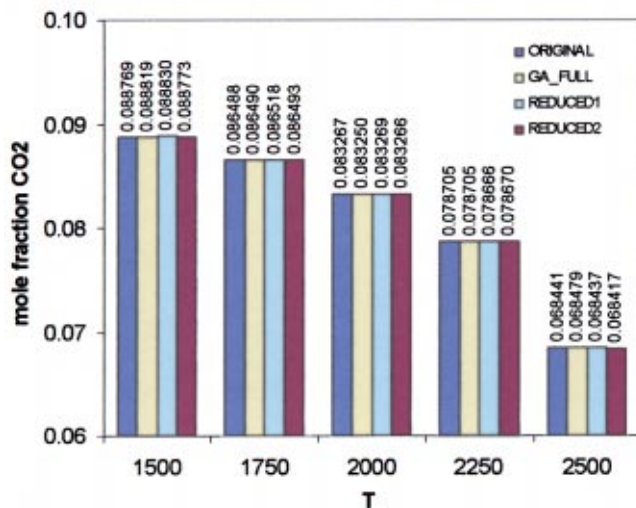


Fig. 3 The mole fractions of CO_2 for various temperatures, as calculated by PSR using the reaction mechanisms generated by the inverse GA formulations considered

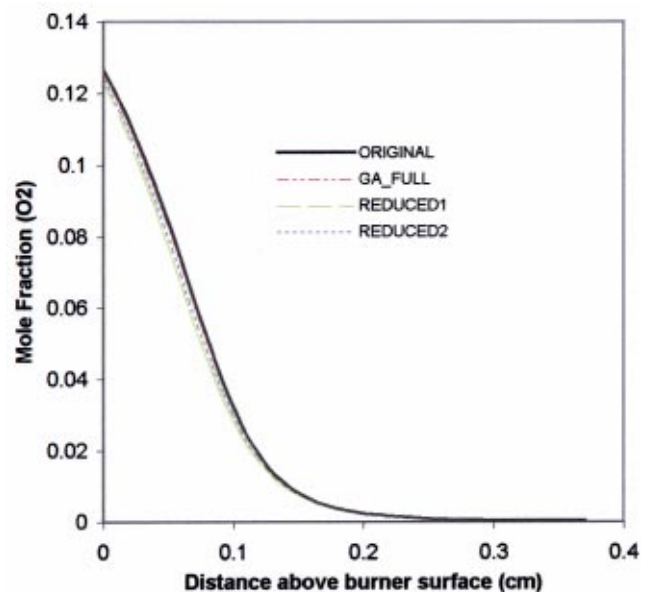


Fig. 4 The profiles of the mole fractions above the burner, for O_2 as calculated by PREMIX using various reaction mechanisms

Thus, even if some species concentration measurements are not available, the reaction mechanisms generated with a reduced set of data produce accurate predictions for both PSR and PREMIX simulations.

In arriving at these GA solutions a Pentium 4, 1.7 GHz workstation took three days for the computations. It is interesting to note that less than 1% of the CPU time is associated with the GA. In fact one generation involves calling either the PSR code 50 (number of individuals) $\times$ 20 (number of PSR conditions) or the PREMIX code 50 (number of individuals) $\times$ 4 (number of PREMIX conditions). A total of 1000 generations were made in arriving at the final solutions with the probability of the PSR rather than the PREMIX code being called = 0.9.

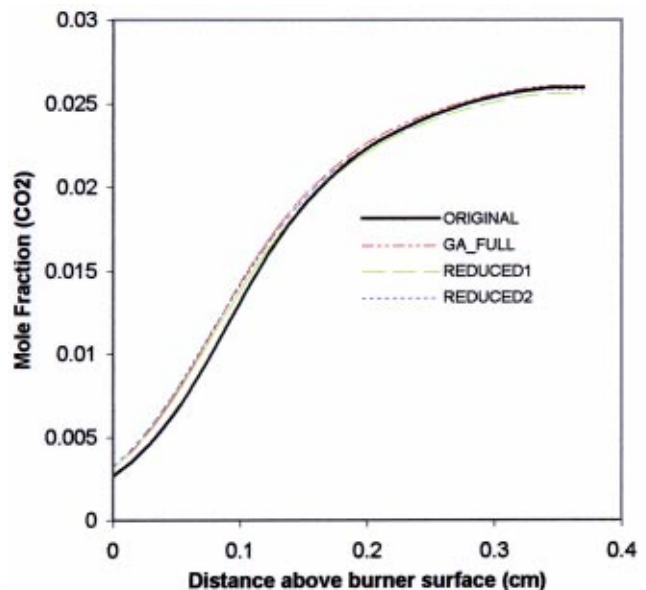


Fig. 5 The profiles of the mole fractions above the burner, for CO_2 , as calculated by PREMIX using various reaction mechanisms

Table 2 The mole fractions of the output species as calculated by PSR at operating conditions which are not within the range covered by the data used in the inversion process, namely, $\Phi=1.0$, $T=2000$ K and $p=2$ atm using the reaction mechanisms generated by the inverse GA formulations considered

species	original	GA_FULL		REDUCED1		REDUCED2	
		calculated	% error	calculated	% error	calculated	% error
H ₂	3.34E-03	3.36E-03	0.45	3.35E-03	0.41	3.34E-03	0.13
H	3.90E-04	3.94E-04	0.93	3.94E-04	0.96	3.92E-04	0.32
O	1.85E-04	1.86E-04	0.55	1.87E-04	0.70	1.85E-04	0.03
O ₂	4.91E-03	4.93E-03	0.31	4.92E-03	0.27	4.91E-03	0.08
OH	2.15E-03	2.16E-03	0.48	2.16E-03	0.56	2.15E-03	0.17
H ₂ O	1.84E-01	1.84E-01	0.02	1.84E-01	0.02	1.84E-01	0.01
HO ₂	1.09E-06	1.11E-06	2.38	1.22E-06	12.27	1.15E-06	5.96
H ₂ O ₂	1.78E-07	1.78E-07	0.15	1.70E-07	4.42	1.66E-07	6.69
CH ₂	7.51E-07	8.51E-07	13.31	1.06E-06	40.60	1.09E-06	45.15
CH ₂ (S)	4.03E-08	4.22E-08	4.57	4.90E-08	21.44	5.07E-08	25.58
CH ₃	7.67E-06	7.74E-06	0.87	6.99E-06	8.86	7.22E-06	5.86
CH ₄	1.43E-05	1.46E-05	1.95	1.44E-05	0.44	1.53E-05	6.85
CO	7.45E-03	7.47E-03	0.24	7.48E-03	0.28	7.46E-03	0.06
CO ₂	8.70E-02	8.70E-02	0.02	8.70E-02	0.03	8.70E-02	0.00
HCO	6.10E-08	6.18E-08	1.31	4.92E-08	19.41	4.96E-08	18.66
CH ₃ O	2.58E-06	2.57E-06	0.49	2.21E-06	14.41	1.20E-06	53.48
CH ₃ O	4.28E-10	5.03E-10	17.57	4.28E-10	0.04	4.08E-10	4.55
C ₂ H ₂	2.92E-08	2.72E-08	6.59	2.54E-08	12.98	2.75E-08	5.71
C ₂ H ₃	3.76E-10	3.59E-10	4.64	4.34E-10	15.39	4.48E-10	19.19
C ₂ H ₄	1.24E-08	1.26E-08	1.14	7.25E-10	94.16	1.18E-09	90.48
C ₂ H ₅	6.81E-11	7.12E-11	4.55	6.58E-11	3.44	7.40E-11	8.59
C ₂ H ₆	4.68E-10	4.11E-10	12.21	2.98E-10	36.43	3.62E-10	22.62
N ₂	7.10E-01	7.10E-01	0.00	7.10E-01	0.00	7.10E-01	0.00
AR	0.00E+00	0.00E+00	0.00	0.00E+00	0.00	0.00E+00	0.00

Validation of the Reaction Mechanisms Generated by Genetic Algorithms. The previous section presented the predictions given by the constructed reaction mechanism for operating conditions which were included in the optimization process. However, the aim of the inversion procedure is to develop reaction mechanisms that are valid over wide ranges of operating conditions. Therefore, next the reaction mechanisms are tested at operating conditions outside the range included in the optimization procedure.

Table 2 presents the species concentration calculated with the PSR program, using the reaction mechanisms generated in the GA inversion procedure and the original mechanism for $T=2000$ K and the air/fuel ratio given by $\Phi=1.0$ at a pressure given by $p=2$ atm while the reaction mechanisms were constructed using numerically simulated data for $p=1$ atm. It can be seen that all the major species are accurately predicted by all three mechanism generated by the inversion procedure investigated. The GA_FULL mechanism outperforms the other two mechanisms but also the reaction mechanisms REDUCED1 and REDUCED2 are accurate in predicting the mole fractions of the major species. Large percentage errors are obtained only for those species which have a very low concentration being of a reduced importance to the combustion process.

Thus, if a reaction mechanism is generated using measurements at some operating conditions then this reaction mechanism accurately predicts all the species concentrations, even when the operating conditions are outside the range of conditions used for the optimization.

Similar conclusions may be obtained for PREMIX predictions, see Fig. 6 which presents the profiles of the mole fraction concentrations of O₂ (Fig. 6(a)) and CO₂ (Fig. 6(b)) by the three mechanisms considered, GA_FULL, REDUCED1 and REDUCED2 for operating conditions which were not part of the optimization process, namely, an air/fuel ratio given by $\Phi=2.5$. It can be seen that the species profiles are accurately predicted by all three reaction mechanisms constructed.

Several other operating conditions have been tested and similar conclusions have been obtained for a wide range of operating conditions. Overall, the GA_FULL reaction mechanism generated by the multi-objective GA proposed in this study was found to be

very efficient in predicting the output species concentrations and burning velocity for both PSR and PREMIX simulations. Accurate predictions were obtained also by using the reaction mechanisms REDUCED1 and REDUCED2 generated using incomplete sets of measurements. Thus we may conclude that the inversion procedure proposed efficiently generates a general reaction mechanism that can be used to predict the combustion products for various combustion problems at different operating conditions. In order to provide additional validation of the results obtained by the inversion procedures presented in this paper, the predicted ignition delay times are computed using all the reaction mechanisms generated in this study. In testing the ignition delay time, the SENKIN code of Sandia National Laboratories, see Lutz et al. [17], was used to predict the time dependent chemical kinetic behavior of a homogeneous gas mixture in a closed system. In addition to predicting the species and temperature histories, the program can also compute the first-order sensitivities with respect to the elementary reaction rate parameters. For the purpose of this paper, the system was considered to be adiabatic under conditions of constant volume and atmospheric pressure, with the initial mole fractions such that the air/fuel ratio is $\Phi=1.0$. Table 3 compares the ignition delay times predicted by the reaction mechanisms generated in this study at five different initial temperatures for an air/fuel ratio given by $\Phi=2.0$. We note that the mechanism based on a complete set of data reproduce the same ignition delay times as the original mechanism and this indicates that the GA does not just generate a mechanism whose validity only extends as far as those operating conditions included in the optimization process.

The accuracy in predicting the ignition delay time is decreased if the inversion procedures are based on incomplete sets of data, see Table 3. However, it is expected that more accurate predictions for the ignition delay time are obtained if measurements for it are included in the optimization process. This can be done by including in the multi-objective genetic algorithm an additional fitness function measuring the goodness of fit to some available measurements for the ignition delay time. The inclusion of such an additional fitness function, and other possible components that incorporate in the multi-objective genetic algorithms other types of measurements available, is considered as future work.

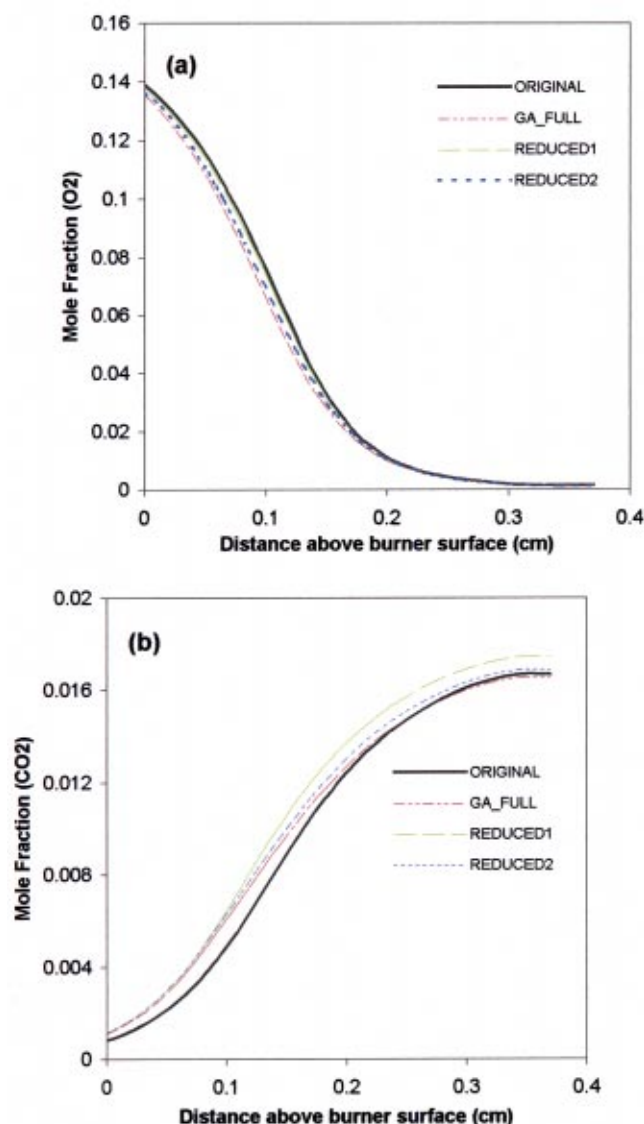


Fig. 6 The profiles of the mole fractions above the burner, for (a) O_2 and (b) CO_2 , as calculated by PREMIX at an air/fuel ratio $\Phi=2.5$ using various reaction mechanisms

Inversion Procedures Using Experimental Data. In the previous sections we investigated the ability of GAs in generating reaction schemes that matched simulated species profiles where at least one solution exists. In many practical situations, reaction mechanisms are validated against experimental measurements where there is not always a set of reaction rate coefficients that produce satisfactory species profile predictions. In many of these cases additional reactions have to be introduced in order to improve the experiment/model correlation. In this section we dem-

onstrate how the GA can be used to identify when additional reactions need to be introduced in order to improve the species profile predictions.

In this section the experimental data of Bernstein et al. [18] are used during the optimization process. The experiments investigated a 20 Torr stoichiometric, 12.5% methane/25% oxygen/62.5% argon, burner stabilised flame with a mixture flow rate of 0.004417 (g/cm²-sec) using the techniques of LIF and REMPI to obtain concentration profiles. The GA inversion procedure is used to improve all species concentration profiles predicted by the original starting mechanism that was used in generating the benchmark profiles of the previous sections (i.e., a reduced version of the GRI 1.2 mechanism consisting of 103 reactions involving 24 species). The procedure is unbiased with each of the six species investigated contributing equally to the fitness function of Eq. (3). In addition to comparing the experimental measurements with the predictions of the original (see Kazakov and Frenklach <http://www.me.berkeley.edu/drm/>) and GA optimised reaction mechanisms, Figs. 7(a)–(f) also compares the predictions of the more detailed GRI 3.0 scheme consisting of 325 reactions involving 57 species. It can be seen from Fig. 7(a) that while the CH_3 predictions of the original mechanism are worse than those generated from the more detailed GRI 3.0, the GA is successful in improving the predictions so that they closely match the experiments of Bernstein et al. [18]. Figure 7(b) displays the concentration profiles of CO and it can be seen that all predictions lie significantly below those found experimentally. Having improved upon the original and GRI 3.0 schemes using the GA, an attempt was made to further improve the CO profile by biasing the GA such that the remaining five species concentrations are independent of changing a reaction rate. This was achieved by assigning $W_{CO}=1$ and $W_{k \neq CO}=0$ in Eq. (3). Although it has not been shown here, it should be noted that the improved CO prediction was obtained at the significant expense of the remaining species' predictions. Based upon the success of the GA in recovering simulated species profiles, it can be concluded that within the framework of the mechanism being optimised, it is not possible to recover this CO profile without using non physical rate constants that lie outside the constraints of the NIST database. Figures 7(c) and (d) again demonstrate the power of the GA inversion procedure with the experimental profiles of Bernstein et al. [18] being accurately matched for the O and OH radicals, respectively. Figure 7(e) illustrates that while there is no improvement over the predictions of GRI 3.0 mechanism, the GA scheme shows an overall improvement to the original mechanism's predictions of the H atom profile. Figure 7(f) shows that both the original and to a lesser extent the GA mechanisms produce improved predictions of HCO over the more detailed GRI 3.0 scheme.

It should be stressed that the predictions of the GRI 3.0 mechanisms have been noticeably improved upon when using the GA inversion procedure on a reaction mechanism with less than a third of the reactions and involving less than half the number of species. Thus, the potential of this powerful technique with regard to optimizing reduced schemes has been clearly demonstrated.

It should be pointed out that in an earlier investigation (Elliott et al. [11]), the reaction mechanisms produced by the GA in matching simulated data could not be applied outside the range of

Table 3 A comparison of the ignition delay time in seconds (s) predicted by the reaction mechanisms generated by various GA inversion procedures considered in this paper for different initial temperatures

	original	GA_FULL	% error	REDUCED1	% error	REDUCED2	% error
1500	1.65E-03	1.24E-03	24.98	3.51E-04	78.68	3.55E-04	78.43
1750	1.92E-04	1.71E-04	10.64	5.42E-05	71.74	5.43E-05	71.70
2000	4.57E-05	4.36E-05	4.70	1.73E-05	62.25	1.92E-05	58.01
2250	1.63E-05	1.60E-05	2.02	8.31E-06	49.07	9.49E-06	41.81
2500	7.56E-06	7.50E-06	0.80	4.76E-06	36.97	5.36E-06	29.07

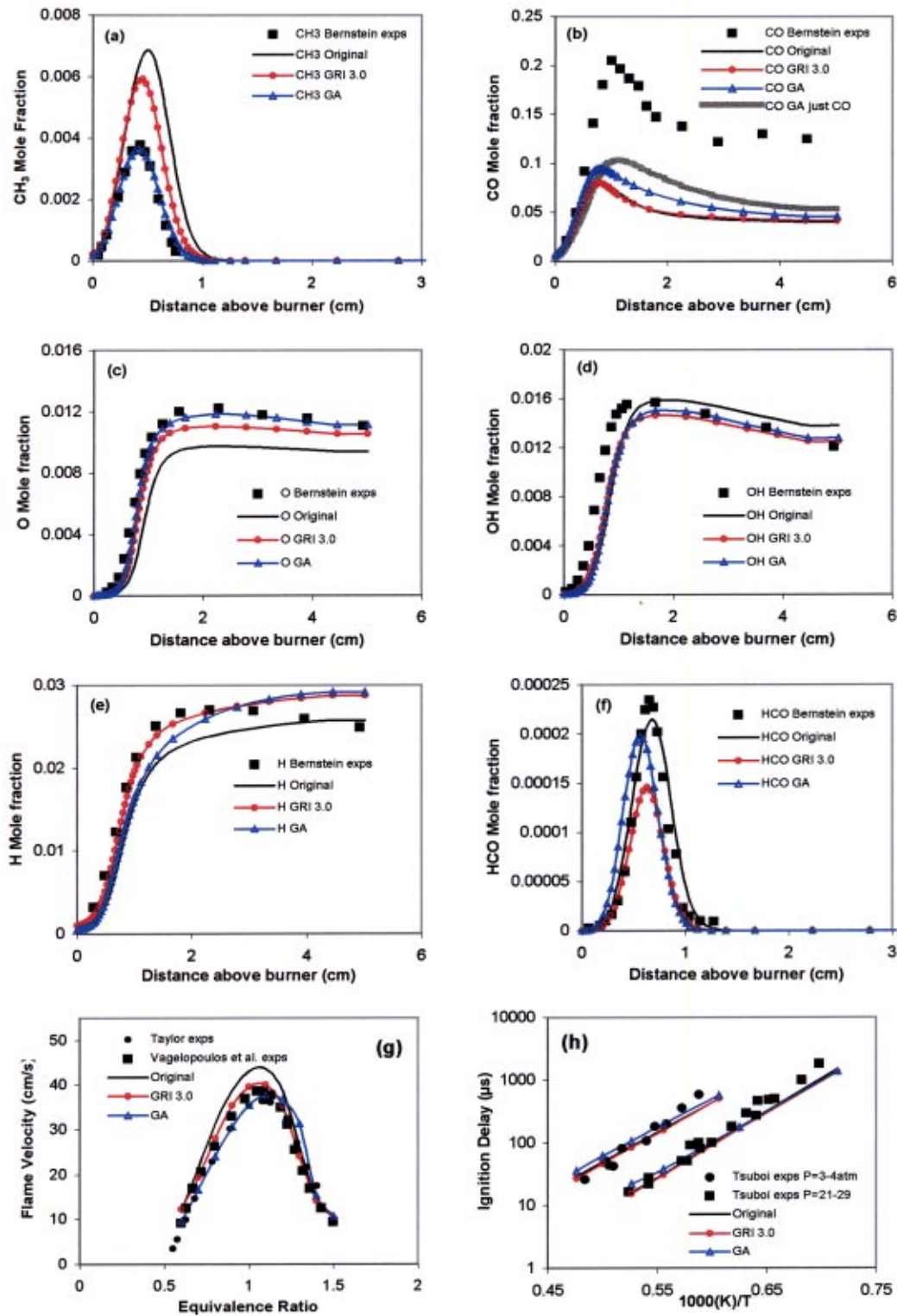


Fig. 7 Mole fraction concentration profiles of (a) CH₃, (b) CO, (c) O, (d) OH, (e) H, and (f) HCO as a function of distance from burner surface for a premixed, stoichiometric, 12.5% methane/25% oxygen/62.5% argon flame at 20 Torr. ■, experimental data of Bernstein et al. [18]. (g) Flame velocity as a function of equivalence ratio for a methane/air mixture at 1 atm and $T_u = 298$ K. ●, experimental data of Taylor [20]; ■, experimental data of Vagelopoulos et al. [19]. (h) Ignition delay times in a 0.2% methane/2% oxygen/97.8% argon mixture. ●, experimental data of Tsuboi and Wagner [21] for pressures = 3–4 atm, $T = 16,500$ –2050 K; ■, experimental data of Tsuboi and Wagner [21] for pressure = 21–29 atm, $T = 1400$ –2000 K.

operating conditions used during the optimization process. This was a result of the GA search space lying between artificially imposed boundaries that often led to reaction mechanisms containing unrealistic rate coefficients. To compensate for this the NIST chemical kinetic database was used to provide physically realistic bounds for the search space and as a consequence the applicability of the GA generated mechanisms now exceeds those used in the optimization process. Figures 7(g) and (h) examine how the mechanism generated by the GA in order to fit the laminar flame species profiles of Bernstein et al. [18] performs at reproducing other combustion phenomena. Figure 7(g) demonstrates how the GA generated mechanism is still able to reasonably reproduce the experimental flame velocity measurements of Vagelopoulos et al. [19] and Taylor [20] for a methane/air mixture at 1 atm and $T_u = 298$ K. While there exists a discrepancy at equivalence ratios between 1.2 and 1.35 it is thought that future work aimed at incorporating the freely propagating flame speed data within the GA optimization procedure will resolve this issue. In addition, an improvement to the GA curve of Fig. 7(g) is to be expected had there been burner stabilized experimental data available at richer conditions rather than just the one stoichiometric value. Figure 7(h) demonstrates the success of the GA optimized reaction mechanism in simulating the ignition delay experiments of Tsuboi and Wagner [21] consisting of 0.2%-methane 2%-oxygen 97.8%-argon mixtures for pressure = 21–29 atm, $T = 1400$ – 2000 K and pressure = 3–4 atm, $T = 1650$ – 2050 K. The mechanisms used to generate any of the data in this study are available at (<http://www.personal.leeds.ac.uk/~fuensm/project/mech.html>).

Conclusions

In this study we have presented a multi-objective GA inversion procedure that can be used to generate new reaction rate coefficients that successfully predict the profiles of the species that are experimentally measured or produced by a previously validated methane/air reaction scheme.

The new reaction rate coefficients generated by GAs were found to successfully match species concentrations in PSR simulations and burning velocity and species profiles in PREMIX simulations, over a wide range of operating conditions, despite their origin being based solely on measurements for a restricted range of operating conditions.

The success of the GA in matching those simulated species' profiles whose mole fractions were above 2×10^{-6} suggests that during reaction mechanism validation the GA inversion procedure may be used as an efficient tool for identifying incomplete reaction mechanisms. In addition, the performance of the GA optimised scheme over the more detailed GRI 3.0 mechanism in matching the experimental profiles of Bernstein et al. [18] has illustrated the potential for improving reduced reaction schemes that may also be applied to combustion phenomena outside those used during the optimization process.

Overall, the numerical results obtained in this study for both experimentally and numerically simulated data ensures a greater confidence in the predictive capabilities of the GA inversion procedure proposed when extended to higher-order hydrocarbon fuels.

Acknowledgments

The authors would like to thank QinetiQ, the EPSRC, the Corporate Research Program from UK MOD and the UK Department of Trade and Industry CARAD Program for the funding and permission to publish this work.

Nomenclature

- A_i = pre-exponential factor for the i th reaction
 B_i = non-Arrhenius parameter for the i th reaction
 Ea_i = activation energy for the i th reaction

- k_{fi} = forward reaction rate for the i th reaction
 T = temperature
 R = universal gas constant
 X_{jk} = mole fraction of the k th species in the j th set of PSR conditions
 K = total number of species
 N_S^{PSR} = total number of PSR conditions
 N_S^{PREMIX} = total number of PREMIX conditions
 Y_{jk} = profile of the mole concentration above the burner for the k th species in the j th set of PREMIX operating conditions
 V_j = burning velocity profile for the j th set of PREMIX operating conditions
 f_{PSR} = PSR fitness function
 f_{PREMIX} = PREMIX fitness function
 $\|\cdot\|_{L^2}$ = L^2 norm of a function calculated using the trapezoidal rule
 W_k = weighting factor applies to the k th species
 ε = constant added to fitness functions to avoid numerical overflow
 n_{pop} = population size
 n_{child} = number of offspring
 p_c = uniform arithmetic cross over probability
 p_t = tournament selection probability
 p_m = nonuniform mutation probability
 n_e = elitism parameter
 p = operating pressure
 V = reactor volume
 Φ = equivalence ratio
 τ = residence time
 Q = heat loss from reactor

Superscripts

- calc = calculated values
 orig = original values

References

- [1] Bowman, C. T., Hanson, R. K., Gardiner, W. C., Lissianski, V., Frenklach, M., Goldenberg, M., and Smith, G. P., 1997, "GRI/MECH 2.11. An Optimised Detailed Chemical Reaction Mechanism for Methane Combustion and NO Formation and Reburning," GRI Technical Report 97/0020.
- [2] Dixon-Lewis, G., Goldsworthy, F. A., and Greenberg, J. B., 1975, "Flame Structure and Flame Reaction Kinetics. IX: Calculation of Properties of Multi-Radical Premixed Flames," *Proc. R. Soc. London, Ser. A*, **346**, pp. 261–275.
- [3] Dagaut, P., Reuillon, M., Boetner, J.-C., and Cathonnet, M., 1994, "Kerosene Combustion at Pressures up to 40 atm: Experimental Study and Detailed Chemical Kinetic Modelling," *Proceedings of the Combustion Institute*, **25**, pp. 919–926.
- [4] Rabitz, H., Kramer, M., and Dacol, D., 1983, "Sensitivity Analysis in Chemical Kinetics," *Annu. Rev. Phys. Chem.*, **34**, pp. 419–430.
- [5] Milstein, J., 1981, "The Inverse Problem: Estimation of Kinetic Parameters," *Modelling of Chemical Reaction Systems*, K. H. Ezzert, P. Deuflhard, and W. Jager, eds., Springer, Berlin.
- [6] Bock, H. G., 1981, "Numerical Treatment of Inverse Problems in Chemical Reaction Kinetics," *Modelling of Chemical Reaction Systems*, K. H. Ezzert, P. Deuflhard, and W. Jager, eds., Springer, Berlin.
- [7] Frenklach, M., Wang, H., and Rabinowitz, J., 1992, "Optimization and Analysis of Large Chemical Kinetic Mechanisms Using the Solution Mapping Method—Combustion of Methane," *Prog. Energy Combust. Sci.*, **18**, pp. 47–73.
- [8] Michalewicz, Z., 1996, *Genetic Algorithms/Data Structures/Evolution Programs*, 3rd Ed., Springer, Berlin.
- [9] Harris, S. D., Elliott, L., Ingham, D. B., Pourkashanian, M., and Wilson, C. W., 2000, "The Optimisation of Reaction Rate Parameters for Chemical Kinetic Modelling of Combustion Using Genetic Algorithms," *Comput. Methods Appl. Mech. Eng.*, **190**, pp. 1065–1090.
- [10] Elliott, L., Ingham, D. B., Kyne, A. G., Mera, N. S., Pourkashanian, M., and Wilson, C. W., 2002, "Incorporation of Physical Bounds on Rate Parameters for Reaction Mechanism Optimisation Using Genetic Algorithms," *Combust. Sci. Technol.*, in press.
- [11] Elliott, L., Ingham, D. B., Kyne, A. G., Mera, N. S., Pourkashanian, M., and Wilson, C. W., 2002, "The Optimisation of Reaction Rate Parameters for Chemical Kinetic Modelling Using Genetic Algorithms," *ASME Paper No. GT-2002-30092*.
- [12] Elliott, L., Ingham, D. B., Kyne, A. G., Mera, N. S., Pourkashanian, M., and Wilson, C. W., 2002, "Multi-Objective Genetic Algorithms Optimization for

Calculating the Reaction Rate Coefficients for Hydrogen Combustion,” Ind. Eng. Chem. Res., in press.

- [13] Glarborg, P., Kee, R. J., Grcar, J. F., and Miller, J. A., 1988, “PSR: A FORTRAN Program for Modelling Well-Stirred Reactors,” Sandia National Laboratories Report No. SAND86-8209.
- [14] Kee, R. J., Grcar, J. F., Smooke, M. D., and Miller, J. A., 1985, “A Fortran Program for Modelling Steady One-Dimensional Premixed Flames,” Sandia Report No. SAND85-8240.
- [15] Kee, R. J., Miller, J. A., and Jefferson, T. H., 1980, “CHEMKIN: A General-Purpose, Problem-Independent, Transport Table, FORTRAN Chemical Kinetics Code Package,” Sandia Report No. SAND80-8003.
- [16] Van Velhuizen, D., and Lamont, G., 2000, “Multiobjective Evolutionary Algorithms: Analyzing the State-of-the-Art,” *Evol. Comput.*, **8**(2), pp. 125–147.
- [17] Lutz, A. E., Kee, R. J., and Miller, J. A., 1987, “SENKIN: A FORTRAN Program for Predicting Homogeneous Gas Phase Chemical Kinetics With Sensitivity Analysis,” Sandia Report No. SAND87-8248.
- [18] Bernstein, J. S., Fein, A., Choi, J. B., Cool, T. A., Sausa, R. C., Howard, S. L., Locke, R. J., and Miziolek, A. W., 1993, “Laser-Based Flame Species Profile Measurements—A Comparison With Flame Model Predictions,” *Combust. Flame*, **92**, pp. 85–105.
- [19] Vagelopoulos, C. M., Egolfopoulos, F. N., and Law, C. K., 1994, “Further Considerations on the Determination of Laminar Flame Speeds With the Counterflow Twin-Flame Technique,” *Twenty-fifth Symposium (International) on Combustion*, The Combustion Institute, Pittsburgh, PA, pp. 1341–1347.
- [20] Taylor, S. C., 1991, “Burning Velocity and the Influence of Flame Stretch,” Ph.D. thesis, University of Leeds.
- [21] Tsuboi, T., and Wagner, H. G., 1974, “Homogeneous Thermal Oxidation of Methane in Reflected Shock Waves,” *Fifteenth Symposium (International) on Combustion*, The Combustion Institute, Pittsburgh, PA, pp. 883–890.

Emissions Performance of the Parker Macrolaminate Premixer Tested Under Simulated Engine Conditions

Adel Mansour

Michael A. Benjamin

Parker Hannifin Corporation,
Gas Turbine Fuel Systems Division,
9200 Tyler Boulevard,
Mentor, OH 44060

Single-injector high-pressure rig evaluation of the prototype Parker macrolaminate dual fuel premixer (previously tested at NETL, Mansour et al., 2001) with pressure swirl macrolaminate atomizers was conducted under simulated engine operating conditions running on No. 2 diesel fuel (DF2). Emissions, oscillations and lean blowout (LBO) performance on liquid fuel at high, part and no load operating points (pressures of 160, 100, 120 psig, and inlet temperatures of 690, 570, 590°F, respectively) and various pressure drops ($\Delta P/P$) and air fuel ratio conditions were investigated. The results indicate that the Parker premixer design has the potential to reduce the DF2 NO_x emission to below 15 ppmv, 15% O_2 . At simulated high load conditions with a nominal flame temperature (T_{PZ}) of 2700°F, the NO_x and CO emissions are approximately 10 and 2.5 ppmv at 15% O_2 , respectively. These NO_x results have not been corrected for fuel bound nitrogen (FBN). From the studies of Lee (2000), small amounts of FBN in the liquid fuel generally are completely converted over to fuel NO_x under lean premixed conditions. The fuel tested has a nominal 60 ppmw of FBN which converts to an estimated fuel NO_x of 4 ppmv at 15% O_2 . These results compare extremely favorable to existing commercially available premixer technologies tested under similar rig operating conditions. More importantly, the NO_x yield for the Parker Macrolaminate premixer appears to be independent of operating conditions (from high to no load and various pressure drop conditions). Variations in combustor pressure, inlet temperature (T_2) and residence time (τ) or pressure drop ($\Delta P/P$) does not seem to have an effect on the formation of NO_x . According to Leonard and Stegmaier (1993), insensitivity of NO_x formation to operating conditions is a good indication of high degree of premixing. Additionally, the premixer NO_x data is only 1 to 2 ppmv higher than the jet stirred reactor (JSR) results (ran at $T_2=661^\circ\text{F}$, $P_{CD}=1\text{ atm}$ and $T_{PZ}=2762^\circ\text{F}$ with similar DF2) of Lee et al. (2001) further confirming the quality of premixing achieved. Combustion driven oscillations was not investigated by tuning the rig so that oscillations would not be a factor. [DOI: 10.1115/1.1762907]

Introduction

The power generation industry is faced with increasingly stringent emissions requirements for ozone precursors, such as nitrogen oxides and carbon monoxide. To achieve lower pollutant emissions, gas turbine manufacturers have adopted lean premixed (LP) combustion as a standard technique, particularly for natural gas applications. LP combustion achieves low levels of pollutant emissions without additional hardware for steam injection or selective catalytic reduction (SCR). By premixing the fuel and air, localized regions of near stoichiometric fuel air mixtures are avoided and a subsequent reduction in thermal NO_x can be realized, [1,2].

Lean premixed combustion systems for both liquid and gaseous fuel applications must meet a number of design constraints, including low-pressure drop, minimum size, and resistance to flashback and autoignition. Operating experience has shown that both static and dynamic combustion instabilities are possible and need to be addressed. Static instabilities occur when operational upsets, or changes in fuel properties, produce unexpected changes in flame anchoring (e.g., flashback, flame extinction near the lean limit, etc.). Dynamic instabilities occur when minor variations in

the fuel air ratio, or mixing processes, lead to significant changes in the combustor heat release rate. Subsequent coupling between the heat-release rate and the acoustic processes in the combustor can result in large pressure oscillations that can damage mechanical components in the turbine engine.

In addition to the general constraints outlined above, dual fuel premixers must be capable of handling different fuels without affecting combustor operability. Additional issues such as autoignition, vaporization, and mixing must be addressed in the design of a dual fuel premixer. For many years, the importance of LP combustion for dual fuel applications has been well recognized, [3–6]. This paper describes test results from a prototype, dual fuel, premixer designed by the Gas Turbine Fuel System Division of the Parker Hannifin Corporation. Experimental test data are presented that demonstrate the NO_x and CO performance of the Parker macrolaminate premixer.

To achieve lower levels of NO_x emissions, homogeneous fuel air mixture distributions are necessary. Relative to single-point fuel-injection, multipoint fuel injection offers numerous advantages, such as significantly shorter mixing length and time scales. These shorter mixing scales can result in shorter premixer lengths and a significantly lower propensity for flashback and autoignition. Macrolamination is a novel manufacturing technique that allows complex internal flow channels to be formed by fabricating the fuel injectors in layers. The fuel delivery channel, the spin slots and the swirl chamber are chemically etched into a substrate.

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Atlanta, GA, June 16–19, 2003, Paper No. 2003-GT-38010. Manuscript received by IGTI, October 2002, final revision, March 2003. Associate Editor: H. R. Simmons.

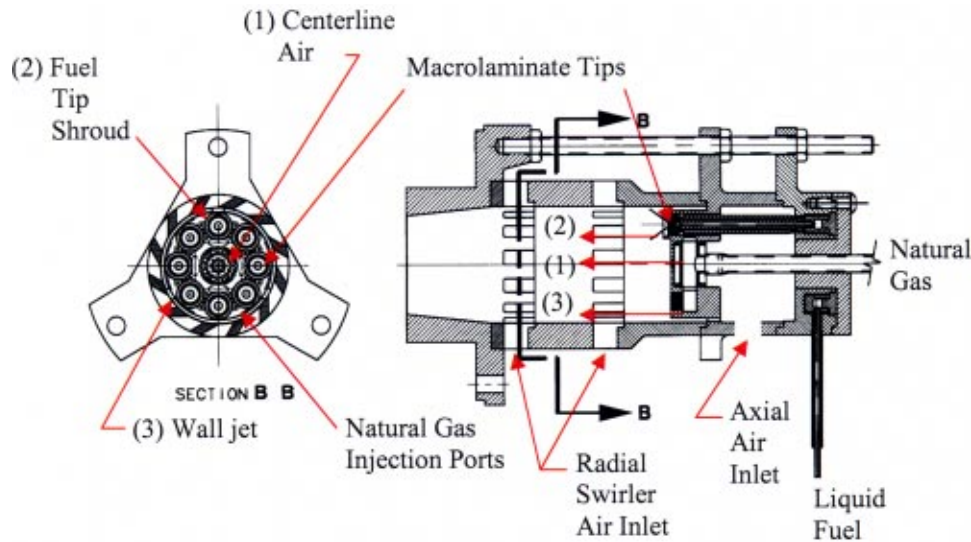


Fig. 1 Cross section of the Parker macrolaminate premixer

Using this macrolamination technology (US Patent 5435884), multi-point fuel injection can be achieved without the associated hardware complexities of conventional designs. Therefore, macrolamination technology is an ideal fit for fabrication and design of dual fuel premixers for LP combustion applications. The dual fuel premixer developed by Parker has been tested at the National Energy Technology Laboratory (NETL) as part of a cooperative research and development agreement (CRADA). This paper extends the results of the NETL tests with an expanded set of premixer operating parameters covering typical midsize (4 to 10 MW) engine-like operating conditions.

The dynamic stability of a dual fuel premixer is also of interest for gas turbine applications. As previously mentioned, dynamic instabilities can occur when temporal fluctuations in the heat release rate couple with the resonant acoustics in the combustor. When this type of thermo-acoustic instability occurs, large oscillations in the pressure can occur. These pressure oscillations can have amplitudes on the order of ten percent of the mean combustor pressure and frequencies approaching the kilohertz range. Clearly, this type of cyclic loading can be detrimental to components in a gas turbine.

The fuel transport time, τ , plays a critical role in the phase between the combustor pressure, $P(t)$ and any subsequent variation in the heat-release rate, $Q(t + \tau)$. A number of recent studies, [7–10], show the importance of altering these time scales to solve dynamics problems. These papers show both experimentally and theoretically how fuel transport time scales can be changed to produce stable combustion. This idea, however, is not new. Putnam [11] has shown that by gathering data as a function of flow rate, it is possible to map the stable and oscillating regions in a given combustor. These plots can be generated by plotting the observed RMS pressure levels against the product of the fuel transport time, τ , and the frequency, f , of the pressure oscillations. This time lag model suggests that the combustion oscillations are confined to a fairly narrow range of $\tau * f$. This approach of correlating combustion oscillations for a given combustor is widely accepted for natural gas applications.

In this paper, the prototype dual fuel premixer is described, as well as the experimental combustor. In the results section, the quantitative effects of equivalence ratio on the performance of the premixer are discussed. The effects of these variables on NO_x and CO emissions are reported.

Description of Experimental Apparatus

Premixer Layout. Figure 1 shows a cross section of the premixer. The liquid fuel is injected via eight macrolaminate injection tips (see Fig. 1). These injection tips produce fine hollow cone fuel sprays. Gaseous fuel injection is achieved through eight radial spokes located between each of the macrolaminate injection tips (see Fig. 1). Each of these radial spokes contains five holes through which gaseous fuel is injected along the axis of the premixer.

The injector configuration tested includes three main axial air inlets (centerline, wall, and fuel tip shroud air) in the backplane, a single radial injection location with two sizes of radial swirlers located just downstream of the atomizer tips, radial gas injection in the backplane and liquid injection with eight pressure swirl macrolaminate atomizers close to the backplane in a circular pattern (see Fig. 1). The injector is designed with consideration for anticoking. Thermal heat shields are employed in the liquid fuel manifold and the eight fuel lances. Wall wetting with liquid fuel that leads to external fouling and high emissions is minimized through the use of fuel tip shroud and wall air and the proper location and strength of the radial swirl air.

The airflow to the premixer is introduced through a number of different paths. These flow paths are described in the following paragraphs:

Air is admitted axially, along the centerline of the premixer, through several injection holes in the upstream end of the premixer (see Fig. 1; flow path 1). In addition to preventing large-scale recirculation zones, this axial flow entrains the fuel droplets and directs them toward the centerline. This entrainment process reduces droplet deposition at the wall.

Eight annular air passages surround the macrolaminate injection tips (see Fig. 1, flow path 2). These air jets have the effect of propelling the fuel spray in the axial direction. The additional momentum from these air jets forces the fuel air mixture to penetrate the swirling air introduced further downstream.

Air is also injected axially along the wall of the premixer (see Fig. 1, flow path 3). This curtain of air surrounds the fuel injectors and acts as an aerodynamic barrier to minimize droplet impingement on the walls. This wall jet also injects sufficient energy into the boundary layer to delay separation.

Two radial inflow swirlers are positioned to provide sufficient swirl for flame stabilization and yet minimize the impingement of

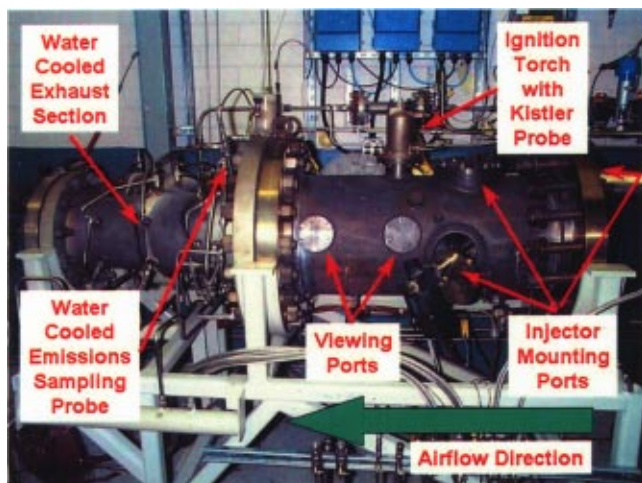


Fig. 2 Schematic of the single injector, high-pressure combustion rig

fuel droplets on the premixer wall. The initial swirl strength, or swirl number, is maintained relatively low to minimize droplet impingement on the walls. A number of investigation have been performed with both single and dual swirler arrangements and it was determined that the NO_x performance was superior with one swirler only as opposed to two swirlers, [10]. This is owing to the fact that relatively high swirl numbers usually lead to excessive wetting of the premixer walls and, thus, to nonhomogenous fuel air mixture at the exit of the premixer. It should be noted though that the one-swirler arrangement resulted in a swirl number equal to 0.6, which resulted in good static stability of the flame.

Low Emissions Experimental Combustor. The dual viewing port, 8" augmented backside cooled (ABC), can type liner was used with an overall liner effective area (AE) of 1.44 in². The ABC liner does not have any film cooling air and is cooled predominantly by cool-side heat transfer. A boroscope-color CCD camera combination was used to visualize the flame structure in the primary zone, emissions probes were placed downstream of the combustor exit plane for emissions sampling and a dynamic pressure transducer was introduced at the torch drain port for oscillations monitoring. An extension tube was added downstream of the water injection cooling section of the rig as an acoustic damping volume to mitigate combustion oscillation frequencies of the order of 300 Hz that are typically seen in the combustor rig of this kind.

The temperature of the gases exiting the sample probes is about 420 K (300°F). The gas sample then passes through a heated gas sampling line before being cooled in a chiller to condense the water vapor. The dry gas sample is analyzed for concentrations of NO_x , CO, unburned hydrocarbons, CO_2 , and O_2 at each operating point. In this paper, only the NO_x and CO emissions are reported. However, it should be noted that the measured levels of unburned hydrocarbons are negligible for the range of operating conditions investigated.

Ignition/light-off is achieved with the use of a torch located near the primary zone of the combustor (see Fig. 2). None of the cases reported in this paper utilize the torch to assist flame stabilization once ignition is established.

The combustor pressure is maintained with a backpressure control valve and the flow in the exhaust neck region is not choked. This design allows operation over a wide range of nozzle velocities at a constant combustor pressure. The inlet air is nonvitiated and the temperature is controlled using an electric air preheater and is also independent of the combustor operating pressure. Therefore, the inlet air temperature, airflow rate and combustor operating pressure can be selected arbitrarily, if necessary. The

Table 1 Typical diesel fuel properties

Carbon (wt%)	86.6
Hydrogen (wt%)	13.4%
Nitrogen	60 ppmw
Specific gravity (60/60)	0.843
API gravity	36.28
50% boiling point	533 K

combustor operating pressure ranged from 780–1200 kPa (7.8–12 atm) allowing the rig to operate over a wide range of air and fuel flows.

The No. 2 diesel fuel used in these tests is a commercially available, low sulfur diesel with 0.05% sulfur content by mass. The typical properties of the DF2 used are listed in Table 1. The nitrogen content in the fuel is about 60 ppm by weight, as measured by an independent laboratory. The contribution of this fuel bound nitrogen to the measured NO_x emissions is estimated to be less than 4 ppmv at 15% oxygen. It should be noted that this is approximately 40% of the measured NO_x levels for the best pre-mixer configuration tested.

Successful purging of the liquid system during shutdown and fuel transfer was accomplished with the use of forward purging with high pressure (21 atm) nitrogen.

Instrumentation. The oscillating combustor pressure is monitored using a fast response Kistler dynamic pressure transducer. The pressure transducer is installed with an infinite coil to minimize the effects of reflections; similar to the technique described in Mahan and Karchmer [12]. The root mean square (RMS) value of this dynamic pressure signal is recorded at each operating condition, along with the dominant frequency in the pressure signal. The strength of the combustor oscillations is expressed as a percentage of the mean combustor pressure.

NO_x levels are measured using a Rosemount chemiluminescent analyzer. CO measurements are taken with a nondispersive infrared analyzer manufactured by Rosemount. All emissions are reported on a dry basis and have been corrected to 15% O_2 .

Test Conditions

The prototype premixer is a modular design that incorporates different swirl vanes and spacers. By changing components in the premixer nozzle, the effects of these different nozzle features are investigated. For this particular study two premixer geometries were studied. These correspond to case A and B shown in Tables 2 and 3. Other geometries have been studied in previous experiments (see [10]). The overall premixer length (defined as the total duct length extending from the liquid fuel injection location to the exit of the premixing duct) was maintained at 9.5 cm. For this study only one radial inflow swirler was used. The total effective area of the premixer was adjusted by increasing or decreasing the swirl vane thickness. The thick vane swirler resulted in a total premixer effective area of AE is 1.03 in² (Premixer A). The thin vane swirler resulted in a total premixer effective area of AE is 1.47 in² (Premixer B). This resulted in effective swirl numbers of 0.6 and 0.8, respectively.

The airflow and the equivalence ratio are varied. Clearly, increasing the airflow decreases the residence time, but it should also be noted that the premixer configuration also dramatically

Table 2 Premixer configurations tested

	Premixer Length (cm)	Vane Thickness Swirler 1 (cm)	Swirl Number (cm)
Premixer A	9.5	0.510	0.6
Premixer B	9.5	0.381	0.8

Table 3 Effect of nozzle configuration on relative air splits

	Approximate Air Split Ratios				
	Y_1			Y_2	Y_3
	Fuel Tip Shroud	Annular Wall Jet	Center-Line Air	Swirler 1	Swirler 2
Case A	0.16	0.20	0.14	0.50	0.00
Case B	0.11	0.13	0.10	0.66	0.00

alters the fuel residence time. Since the fuel is injected from the base of the premixer, any changes to the nozzle length, or air splits (see Table 3) result in changes to the residence time.

The fuel transport time is approximated as a function of the average air velocity at the premixer exit, the critical nozzle dimensions and the approximate air splits for the different configurations. The fuel transport time is estimated as

$$\tau = \tau_1 + \tau_2 \quad (1)$$

Therefore, the fuel transport time is simply a sum of the bulk transport times between air injection points along the nozzle axis. The time required to transport a parcel of fuel air mixture between the fuel injection plane and the first swirler is denoted as τ_1 . Similarly, τ_2 is the time required to advect the parcel of fuel air mixture between the first swirler and the exit of the premixer. Therefore, the bulk fuel transport time can be approximated using the nozzle configuration and approximate air splits as shown in Eq. (2), where L_1 is the distance between the axial air injection plane and first swirler, and L_2 is the distance between the first swirler and the exit of the premixer; y_1 is the fraction of air flowing through the base plane; and y_2 is the fraction of air flowing through the swirler. U is the bulk velocity at the exit of the premixer.

$$\tau = \left(\frac{L_1}{y_1 U} \right) + \left(\frac{L_2}{(y_1 + y_2) U} \right) \quad (2)$$

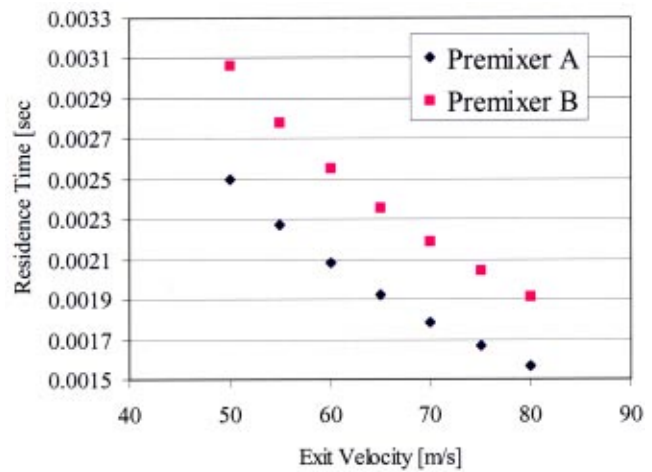
Equation (2) shows that the fuel-transport time, τ , is equal to the length of the section divided by the average, or bulk air velocity. The velocity, U , is calculated based on the measured airflows and exit area of the nozzle. This velocity represents the average (plug flow) value at the nozzle exit. Obviously, this approximation does not consider the details of the flow field inside the premixer. Each individual section would see a range of transport times that are distributed around the average transport time approximated by Eq. (2). Nonetheless, the notion of an average transport time is very useful for engineering purposes. Figure 3 shows the range of transport times obtained using the above equation for Premixer A and B as a function of average exit velocities. Note that Premixer A has a shorter residence time than Premixer B. The residence time is found to be in the range of 1.5 to 3 ms.

Results

Investigation of the emissions, oscillations and lean blow out performance for the Parker premixer was carried out under simulated engine operating conditions for two injector configurations (Premixer A and Premixer B as described in the previous section).

Premixer A (AE=1.03 in², SN=0.6). As shown in Figs. 4 and 5, at simulated high load conditions with a T_{PZ} of 2700°F, the NO_x and CO emissions are approximately 10 and 2.5 ppmv at 15% O₂, respectively. The results indicate that low CO is possible even under low NO_x conditions showing good emissions turn-down capability of the Parker premixer when used in conjunction with an ABC liner. These results compare extremely favorably to current liquid LP injector designs.

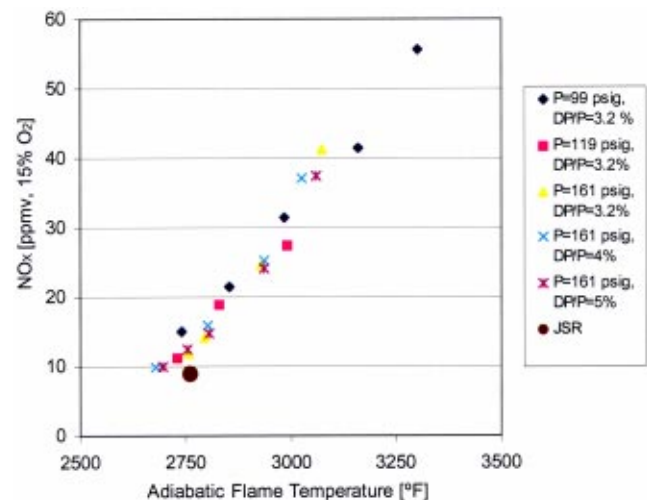
In general, the Parker design (Premixer A) has a factor of two to five lower NO_x yield than current commercially available pre-

**Fig. 3 Range of fuel transport times**

mixer technologies tested under similar rig like conditions. More importantly, the NO_x yield for Premixer A appears to be independent of operating. Variations in pressure (P_{CD}), inlet temperature (T_2) and residence time (τ) or pressure drop ($\Delta P/P$) do not seem to have an effect on the formation of NO_x. According to Leonard and Stegmaier [13], insensitivity of NO_x formation to operating conditions is a good indication of high degree of premixing. Additionally, the Premixer A NO_x data is only 1 to 2 ppmv higher than the jet stirred reactor (JSR) results (ran at $T_2=661^\circ\text{F}$, $P_{CD}=1$ atm and $T_{PZ}=2762^\circ\text{F}$ with similar DF2) of Lee et al., [14] further confirming the quality of premixing achieved.

From the studies of Lee [15], small amounts of fuel bound nitrogen (FBN) in the liquid fuel are generally completely converted over to fuel NO_x under lean premixed conditions. For the above cases mentioned (Premixer A and JSR), the DF2 was supplied by the same source leading to a high likelihood of similar amounts of FBN (estimated to be 60 ppmw) and, thus, fuel NO_x (estimated to be 4 ppmv, 15% O₂). Therefore, the above NO_x results are most possibly not significantly affected by differences in fuel bound nitrogen.

Good acoustic stability was also achieved with no significant oscillations (<0.5 psid rms). This is the case for all conditions tested shown in Fig. 4.

**Fig. 4 NO_x emission profiles for the Parker Premixer A under various simulated engine operating conditions and the University of Washington jet stirred reactor (JSR)**

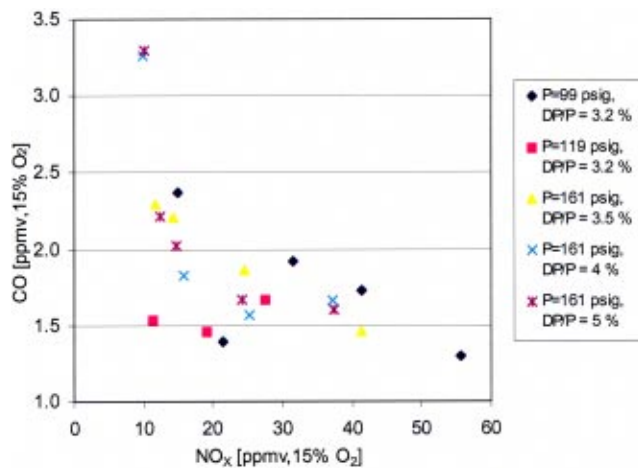


Fig. 5 Emissions performance for the Parker Premixer A under various simulated engine operating conditions

Still video images of the flame in the primary zone are shown in Figs. 6 and 7 for simulated high and part load conditions, respectively. As can be seen in Fig. 6, at high load conditions, the flame is light bluish with a compact luminous core. The luminous core is reduced, as the throughput is increased by the increase in pressure drop. The flame images coupled with the low emissions indicate a high degree of premixing is possible with the Premixer A design.

The part load images shown in Fig. 7 indicate a much more luminous core without much increase in the NO_x output. The increased luminosity is most likely due to the reduced atomizer performance (e.g., less dispersion and larger droplet sizes). This is as expected since the macrolaminate atomizers employed in the Parker premixer is of the pressure swirl type which degrades in performance as the fuel flow is reduced (small turndown). Nevertheless, the NO_x emissions remain comparable to the high load cases indicating that perhaps a two-dimensional chromatic flame image may not be sufficient to understand the emissions characteristics of a lean premixed flame. (Work is underway to develop both sampling and temperature probes to investigate the primary zone under rig conditions.)

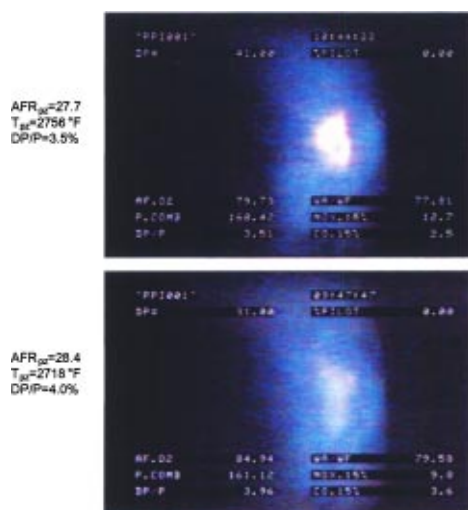


Fig. 6 Simulated high load conditions ($P_{CD}=162$ psig, $T_2=681^\circ\text{F}$) with Premixer A

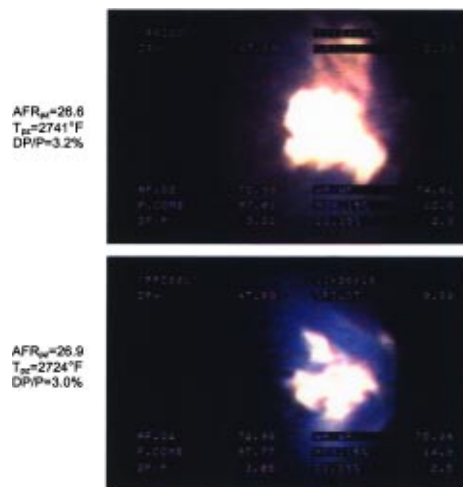


Fig. 7 Simulated part load conditions ($P_{CD}=99$ psig, $T_2=574^\circ\text{F}$) with Premixer A

Premixer B ($AE=1.47$ in², $SN=0.8$). Emissions performances for Premixer B are shown in Figs. 8 and 9. Similar to Premixer A, the NO_x yield for part and no load conditions are at about the same level and are comparable to the values for Premixer A, again indicating a high degree of premixing. On the other hand, the high load data indicate higher yields of NO_x at a given T_{PZ} and an extension of the LBO. These characteristics coupled with the increased swirl number or swirl strength (from 0.6 to 0.8 for Premixer A to Premixer B, respectively) suggest wall wetting where the droplets are centrifuged to the wall forming a film on the wall with possibly increased combustion in the diffusion mode yielding higher NO_x and extending the lean blow-out limit. Evidence of wall wetting is also suggested in Figs. 10 through 12 where the luminous part of the flame is diametrically larger, almost spanning the diameter of the injector discharge. Furthermore, small amount of liquid fuel and gum was observed in the crevices between the injector sections after the injector was pulled confirming the likelihood of wall wetting. It is important to note that no gumming or liquid fuel was evident post Premixer A testing.

Premixer B was also run at a different simulated high load

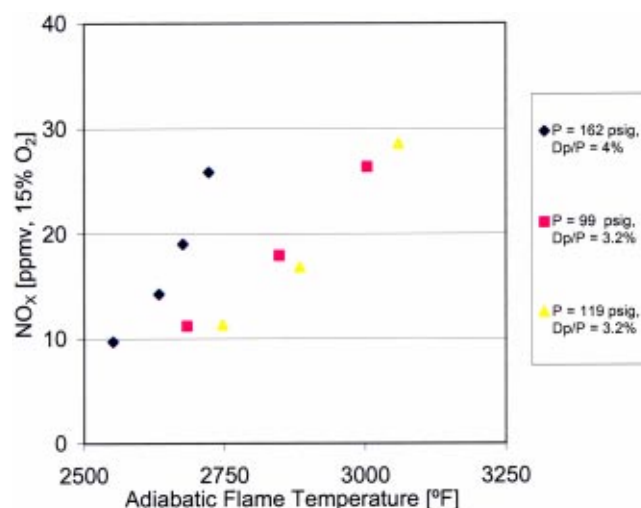


Fig. 8 NO_x emission profiles for the Parker Premixer B under various simulated engine operating conditions

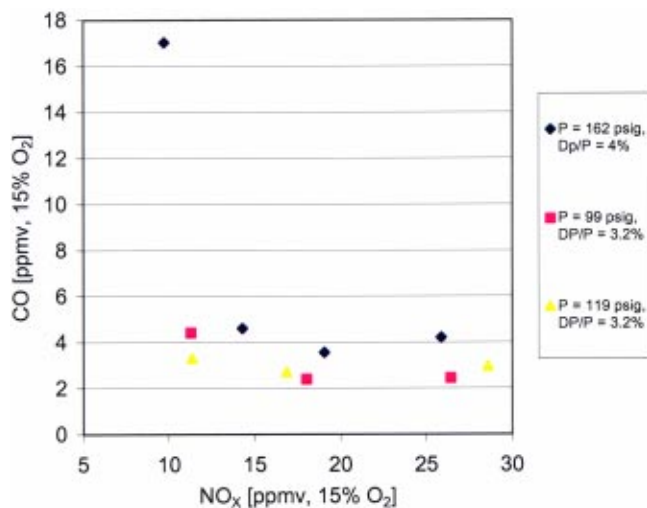


Fig. 9 Emissions performance for the Parker Premixer B under various simulated engine operating conditions

conditions as shown in Fig. 8. As can be seen, the NO_x yield is low and is essentially at the same level for the part and no load data.

Conclusions

A prototype lean premixed prevaporized fuel injector has been designed and successfully rig tested to demonstrate performance at 690°F inlet air temperature and 1200 kPa. After testing a number of nozzle configurations, NO_x and CO emissions of less than 15 and 2.5 ppmv at 15% O₂, respectively, are observed for operation on DF2. These results are encouraging for a prototype dual fuel premixer. It is important to note that reported results are obtained without any pilot fuel to stabilize the flame.

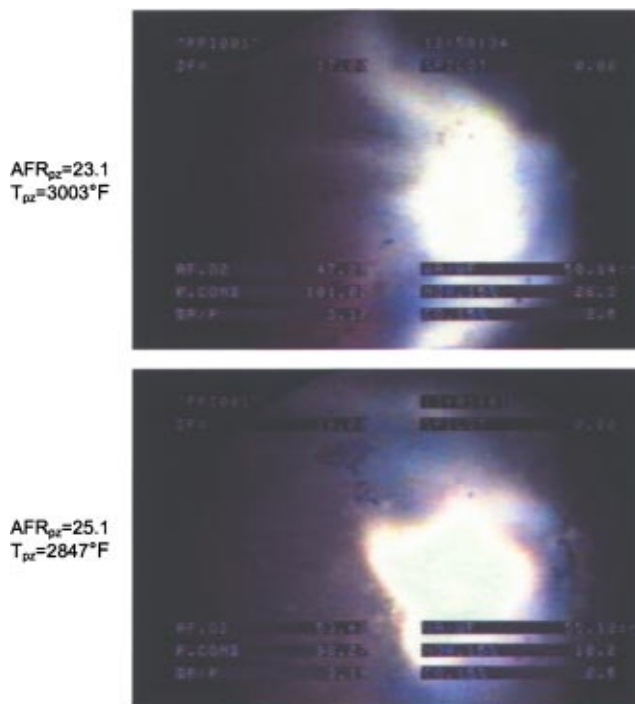


Fig. 11 Simulated part-load conditions ($P_{CD}=99$ psig, $T_2=574^\circ\text{F}$, $\Delta P/P=3.2\%$) with Premixer B

Initial single injector liquid fuel results of the Parker premixer with pressure swirl macrolaminated atomizer tested under simulated engine operating conditions indicate significant potential for very low NO_x performance (e.g., two to five times less than current lean premix injector technologies).

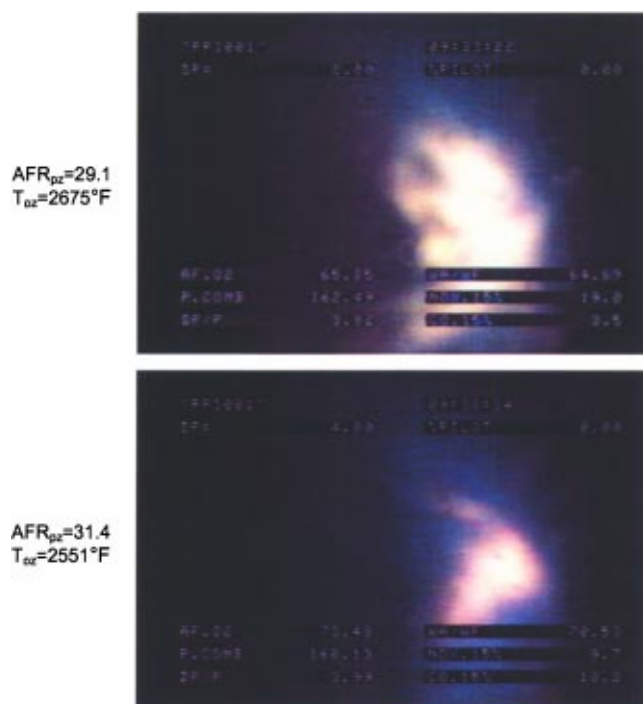


Fig. 10 Simulated high load conditions ($P_{CD}=162$ psig, $T_2=681^\circ\text{F}$, $\Delta P/P=4.0\%$) with Premixer B

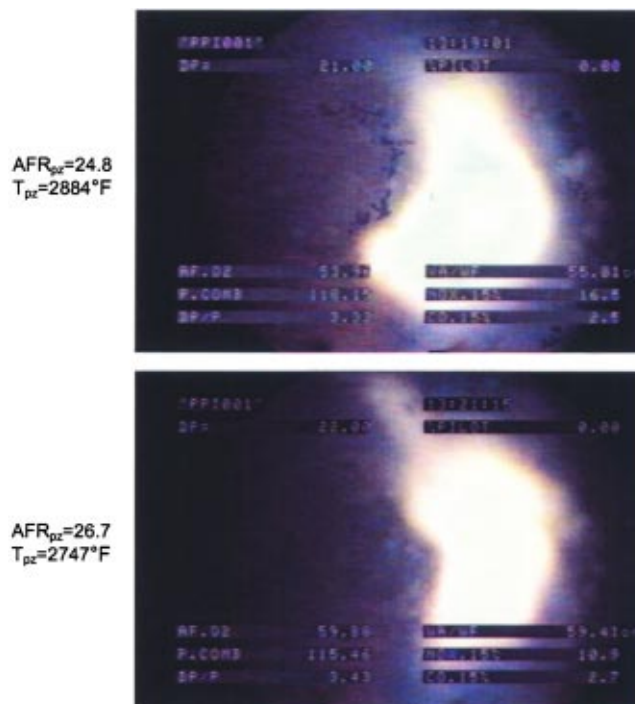


Fig. 12 Simulated no-load conditions ($P_{CD}=119$ psig, $T_2=595^\circ\text{F}$, $\Delta P/P=3.2\%$) with Premixer B

References

- [1] Appelton, J. P., and Heywood, J. B., 1972, "The Effects of Imperfect Fuel-Air Mixing in a Burner on NO Formation From Nitrogen in the Air and the Fuel," *14th International Symposium on Combustion*, The Combustion Institute, Pittsburgh, PA, pp. 777–786.
- [2] Lyons, V. J., 1982, "Fuel/Air Non-Uniformity Effects on Nitric Oxide Emissions," *AIAA J.*, **20**, pp. 660–665.
- [3] Cowell, L. H., Rajput, A., and Rawlins, D. C., 1996, "Development of a Dual-Fuel Injection System for Lean Premixed Industrial Gas Turbines," ASME Paper No. 96-GT-195.
- [4] Smith, K. O., and Cowell, L. H., 1989, "Experimental Evaluation of Liquid-Fueled, Lean-Premixed Gas Turbine Combustor," ASME Paper No. 89-GT-264.
- [5] McVey, J. B., and Kennedy, J. B., 1980, "Lean Stability Augmentation Study," *J. Energy*, **4**.
- [6] Snyder, S. S., Rosfjord, T. J., McVey, J. B., and Chiappetta, L. M., 1994, "Comparison of Fuel/Air Mixing and NO_x Emissions for Tangential Entry Nozzle," ASME Paper No. 94-GT-283.
- [7] Richards, G. A., and Janus, M. C., 1998, "Characterization of Oscillations During Premix Gas Turbine Combustion," ASME J. Eng. Gas Turbines Power, **120**, pp. 294–302.
- [8] Straub, D. L., and Richards, G. A., 1998, "Effect of Fuel Nozzle Configuration on Premix Combustion Dynamics," ASME Paper No. 98-GT-492.
- [9] Lieuwen, T., and Zinn, B. T., 1998, "Theoretical Investigation of Combustion Instability Mechanisms in Lean Premixed Gas Turbines," AIAA Paper No. 98-0641.
- [10] Mansour, A., Benjamin, M., Straub, D. L., and Richards, G. A., 2001, "Application of Macrolamination Technology to Lean, Premixed Combustion," ASME J. Eng. Gas Turbines Power, **123**, pp. 796–803.
- [11] Putnam, A. A., 1971, *Combustion Driven Oscillations in Industry*, Elsevier, New York.
- [12] Mahan, J. R., and Karchmer, A., 1991, "Combustion Core Noise," *Aeroacoustics of Flight Vehicles: Theory and Practice, Volume 1: Noise Sources*, NASA Langley Research Center, AIAA Technical Library.
- [13] Leonard, G., and Stegmaier, J., 1993, "Development of an Aeroderivative Gas Turbine Dry Low Emissions Combustion System," ASME Paper No. 93-GT-288.
- [14] Lee, J. C. Y., Malte, P. C., and Benjamin, M. A., 2001, "Low NO_x Combustion for Liquid Fuels: Atmospheric Pressure Experiments Using a Staged Prevaporizer-Premixer," ASME Paper No. 2001-GT-0081.
- [15] Lee, J. C. Y., 2000, "Reduction of NO_x Emission for Lean Prevaporized-Premixed Combustors," Ph.D. dissertation, University of Washington, Seattle, WA.

Implementation of a Multivariable Control

Louis J. Larkin

James Philpott

Propulsion Systems Analysis & Integration,
Pratt & Whitney Engineering,
West Palm Beach, FL

A multivariable control (MVC) was designed and implemented for the Joint Technology Demonstrator Engine (JTDE) XTE65-2. The engine control system utilized an existing MC68000 processor that used fixed point ADA for its programming language and was limited in throughput and memory. A canonical formulation for the MVC compensator was used to minimize memory and calculation load on the processor. Use of this formulation resulted in a number of numerical difficulties. This paper relates the issues associated with the implementation of a MVC in this environment, and some approaches to solve these difficulties.

[DOI: 10.1115/1.1362668]

Introduction

Multivariable Controls (MVC's) have been studied for use in aircraft gas turbine engines since the mid 1970s. The typical twin spool, afterburning turbofan engine used for most military applications exhibits coupling between a number of its process variables and the effectors used to control the engine system. The process coupling often limits the bandwidth performance of the control system. Bandwidth limiting can have the negative impacts of reduced thrust response and sloppy control of engine match point. The lack of crisp match point control can, in turn lead to matching the engine away from its optimum performance point to avoid the potential for stalls, thereby adversely impacting engine performance. Hence the desire has existed for some time to design a practical MVC to overcome the system coupling effects and provide enhanced engine performance and operability.

A multivariable control (MVC) regulator was designed for the Joint Technology Demonstrator Engine (JTDE) XTE65-2 to evaluate the potential improvements over conventional regulators. The JTDE program is focused on developing and demonstrating technologies in support of the integrated high performance turbine engine technology (IHPTET) initiative. The MVC was designed specifically with the intent of achieving very tight control over the engine's compression system operating lines (Larkin, 1994 [1]). By achieving tight transient and steady state control over the operating lines, the engine could be upmatched to achieve improved performance. The design point linear model of the JTDE engine is provided as an appendix.

Implementation issues arose due to the limited processor capabilities of the full authority digital engine control (FADEC) used to control the engine. Due to budget constraints, the program decided to use an existing FADEC to control the engine. The FADEC chosen was an existing YF119 flight test FADEC, which had minimal spare memory and throughput, and used 16 bit fixed point arithmetic. These features presented some challenges in the implementation of the MVC. This paper defines approaches used to implement the MVC in this environment.

The Observer Canonical Form. The MVC was implemented as a discrete-time state-space system

$$\dot{\mathbf{x}}(k) = \mathbf{A}\mathbf{x}(k-1) + \mathbf{B}\mathbf{e}(k) \quad (1)$$

$$\mathbf{u}(k) = \mathbf{C}\mathbf{x}(k) + \mathbf{D}\mathbf{e}(k) \quad (2)$$

Note the "one step ahead" implementation of the state equation. In these equations, $\mathbf{x}$ represents the internal state vector of the regulator, $\mathbf{e}$ is the errors in the controlled variables (the inputs to the regulator), and $\mathbf{u}$ is the vector of control requests, which are sent to the control effectors.

Regulators were designed at discrete operating points and gain scheduled across the system's operating range. In order to minimize the memory and computational requirements for the processor, the MVC was implemented in the "observer canonical" (OC) realization. The transformation to the OC realization is performed using the algorithm defined by Kailath (1980) [2]. The OC realization is very sparse in both the A and C matrices. Table 1 shows a control point design in OC form.

It can be seen that the OC formulation results in a significant reduction in the number of parameters in the A and C matrices. In fact, the C matrix ceases to exist, since the output vector is formed by selecting states 1 and 4 and adding these states to a linear combination of the inputs (errors). Table 2 shows that the OC realization saves almost 50 percent in computation compared to a realization that fully populates the matrices.

XTE65 Control Issues. Initial MVC implementation studies using a FORTRAN model of the engine and control showed very favorable results. Implementation of the MVC in the target processor (FADEC), however, yielded unfavorable results. Limit cycles were encountered in initial testing, and steady state errors were evident despite the integral control laws implemented in the regulator. Comparisons of open-loop MVC performance between the design tool (CTRL-C), the FORTRAN simulation and the target processor showed that the MVC operated differently in the target environment than in the nonreal time environments. After

Table 1 Control point design in OC form

A Matrix	0.000000	1.000000	0.000000	0.000000	0.000000
	0.000000	0.000000	1.000000	0.000000	0.000000
	0.513917	-1.907166	2.393248	0.060159	-0.060159
	0.000000	0.000000	0.000000	0.000000	1.000000
	0.130238	-0.261677	0.131439	-0.872850	1.872849
B Matrix	1.488489	-0.217077			
	1.314203	-0.123734			
	1.155458	-0.062402			
	0.445191	-0.571043			
	0.453585	-0.582215			
C Matrix	1.000000	0.000000	0.000000	0.000000	0.000000
	0.000000	0.000000	0.000000	1.000000	0.000000
D Matrix	1.806800	-0.445633			
	0.549238	-0.745230			

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Munich, Germany, May 8-11, 2000; Paper 00-GT-039. Manuscript received by IGTI February 2000; final revision received by ASME Headquarters January 2001. Associate Editor: M. Magnolet.

Table 2 Computational Requirements For 5STATE/2I/2O OC Vs Fully Populated Matrices

Realization	Table look-ups	Additions	Multiplications
Arbitrary	49	42	49
OC	24	22	24

significant troubleshooting efforts, FORTRAN simulation experiments revealed that the effects seen in the target environment were the result of using fixed-point arithmetic. With the fixed point processing used by the XTE65-2 control, elemental operations were performed in 32 bit registers but results were truncated to 16 bits for storage. The loss of precision proved to be especially detrimental to the MVC algorithm in which multiple operations are performed for each state.

Differential State Formulation. In order to reduce the sensitivity to numerical truncation a differential state formulation was implemented. This formulation takes advantage of the fact that in the OC form, the differences between the MVC states are much lower in magnitude than the actual values of the states. A typical example is shown in Fig. 1. In this case, a representative minimum to maximum power transition, the lower magnitudes of the state differences are readily apparent. A comparison of states to state differences for this case (Fig. 2) shows a 2^6 – 2^{10} decrease in magnitude. This implies a 6–10 bit improvement in controller precision can be realized by transforming the MVC to the difference state formulation.

In practice, the improvement in resolution with the difference state formulation is less than the full potential. This is due to the uncertainty in estimating maximum values for the state differences. In the base OC formulation (sparse C matrix with only values of unity) the states have the same steady state values as the control requests. Since control request limits are typically well defined, i.e., maximum flow rate or effector position, setting the scales for the states is straightforward. However, the difference states have a steady state value of zero. In nonsteady operation, difference state magnitudes are determined by both the rate and magnitude of input perturbations. Due to the wide range of operation conditions and uncertainty in estimating worst case perturbations, scaling safety margin is required to protect against fixed-point overflow. For the implementation discussed here, a five bit resolution improvement was realized.

The difference state formulation is derived by observing the relationship between the coefficients of the A matrix. It can be seen from inspection of the OC realization A matrix in Fig. 1 that

$$a_{33} = 1 - a_{31} - a_{32} \quad (3)$$

$$a_{35} = -a_{34} \quad (4)$$

$$a_{53} = -a_{51} - a_{52} \quad (5)$$

$$a_{55} = 1 - a_{54} \quad (6)$$

Substituting these relationships into the appropriate equations yields (ignoring the Bu terms)

$$\begin{aligned} x_3(k) = & a_{31}[x_1(k-1) - x_3(k-1)] + a_{32}[x_2(k-1) - x_3(k-1)] \\ & + x_3(k-1) + a_{34}[x_4(k-1) - x_5(k-1)] \end{aligned} \quad (7)$$

$$\begin{aligned} x_5(k) = & a_{51}[x_1(k-1) - x_3(k-1)] + a_{52}[x_2(k-1) - x_3(k-1)] \\ & + a_{54}[x_4(k-1) - x_5(k-1)] \end{aligned} \quad (8)$$

Taking the next logical step, difference states can be defined as:

$$\Delta_1(k) = x_1(k) - x_3(k) \quad (9)$$

$$\Delta_2(k) = x_2(k) - x_3(k) \quad (10)$$

$$\Delta_4(k) = x_4(k) - x_5(k) \quad (11)$$

then,

$$\Delta_3(k) = a_{31}\Delta_1(k-1) + a_{32}\Delta_2(k-1) + a_{34}\Delta_4(k-1) \quad (12)$$

$$\Delta_5(k) = a_{51}\Delta_1(k-1) + a_{52}\Delta_2(k-1) + a_{54}\Delta_4(k-1) \quad (13)$$

Now,

$$\Delta_1(k) = \Delta_2(k-1) + (b_{11} - b_{31})u_1(k) + (b_{12} - b_{32})u_2(k) - \Delta_3(k) \quad (14)$$

$$\Delta_2(k) = (b_{21} - b_{31})u_1(k) + (b_{22} - b_{32})u_2(k) - \Delta_3(k) \quad (15)$$

$$\Delta_4(k) = (b_{41} - b_{51})u_1(k) + (b_{42} - b_{52})u_2(k) - \Delta_5(k) \quad (16)$$

Since the OC formulation requires actual states not difference states for the outputs, it is still necessary to find states 3 and 5 as:

$$x_3(k) = \Delta_3(k) + x_3(k-1) + b_{31}u_1(k) + b_{32}u_2(k) \quad (17)$$

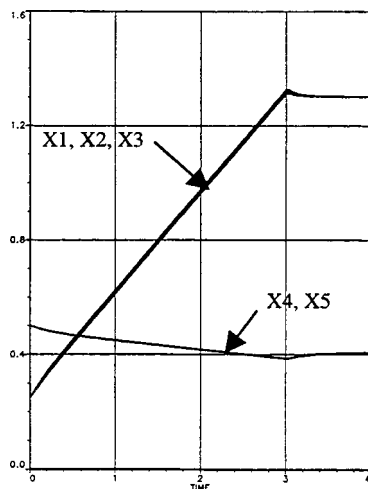


Fig. 1 MVC State excursion

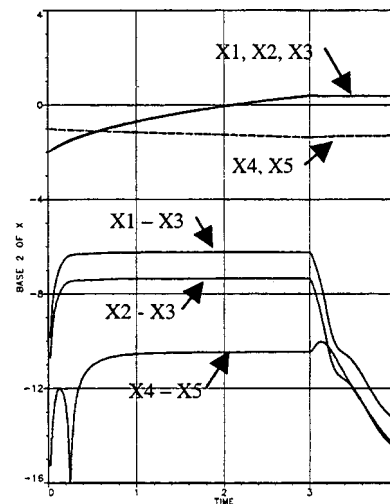


Fig. 2 State differences during transient

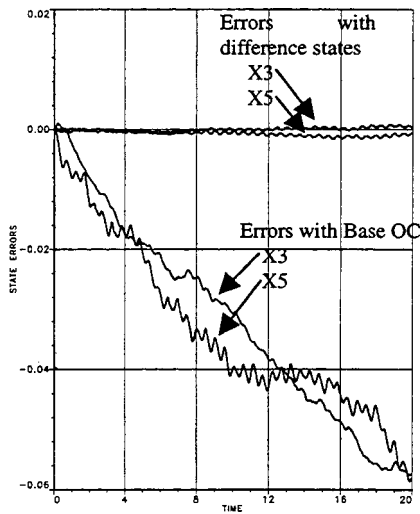


Fig. 3 Benefits of differential state formulation

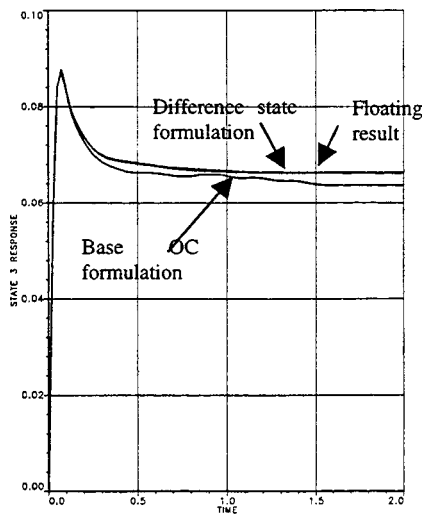


Fig. 4 Closed loop comparison

$$x_5(k) = \Delta_5(k) + x_5(k-1) + b_{51}u_1(k) + b_{52}u_2(k) \quad (18)$$

Now, the outputs are:

$$y_1(k) = \Delta_1(k) + x_3(k) + d_{11}u_1(k) + d_{12}u_2(k) \quad (19a)$$

$$y_2(k) = \Delta_4(k) + x_5(k) + d_{21}u_1(k) + d_{22}u_2(k) \quad (19b)$$

The increased precision gained with the difference formulation results in a significant improvement in MVC performance. An example is given in Fig. 3, comparing open loop MVC behavior for the difference state OC formulation to the base OC formulation. In this test case, a 1-hz disturbance was input into the MVC. The floating-point result was assumed to be correct. All fixed-point parameters (states, gains and inputs) were assumed to be 16 bit and arithmetic operations were assumed to be 32 bit. The tendency of the base OC formulation to accumulate errors due to lack of numerical precision is evident. In contrast, the difference state OC formulation closely matches the intended response.

Further comparisons of MVC performance with loops closed are shown in Fig. 4. In this test case, a 5 percent step in both requests was input into the MVC. The integral component of the MVC eliminates much of the steady state numerical errors in the

base OC formulation but dynamic errors are still present. Again, the difference state OC formulation closely matches the intended response.

Conclusion

The OC formulation provides significant advantages in terms of reducing memory and throughput utilization. The OC formulation has the disadvantage of exaggerating numerical precision issues when implemented in fixed-point controllers. This formulation is also not robust with respect to interpolation between gain-scheduling points. Utilization of a differential-state formulation provided significant improvements in numerical robustness. This formulation is recommended for floating point applications as well. This study, carried out as part of the JTDE program provided significant insight into implementation issues associated with MVC's. The benefits of this research were realized on the JPATS and Joint Strike Fighter programs aircraft programs.

Acknowledgments

Thanks to Mr. Timothy Lewis of the Air Force Research Lab for his support of propulsion controls technology.

Nomenclature

FADEC = full authority digital electronic control
MVC = multivariable control
OC = observer canonical

Appendix: Engine Point Model

States:

N1 — low rotor speed (RPM)
N2 — high rotor speed (RPM)
TMT — metal temperature (R)

Inputs:

Wf — fuel flow rate (Lbm/sec)
A8 — jet area (In²)
CVV — compressor variable vane angle (deg.)

Outputs:

OPR — ratio of burner pressure/inlet
EPR — ratio of jet pressure/inlet
N2 — high rotor speed (RPM)
T₂₅ — fan discharge temp. (R)

A				B	
-4.24407	2.203365	0.123153	0.536729	1.106265	0.57131
-1.29609	-2.51523	-0.08878	0.66324	0.356616	-0.68915
-0.11971	-0.07745	-0.80866	0.243502	0.026791	-0.007
C				D	
1.275155	0.762255	0.015858	0.338071	-0.26695	0.199099
1.831501	0.120614	0.049849	0.249334	-1.08233	0.032707
0	1	0	0	0	0
0.912668	-0.2564	0.007387	0.040278	-0.17135	-0.06513

References

- [1] Larkin, Louis, 1994, "Pressure Based Closed Loop Thrust Control In A Turbofan Engine," US Patent No. 5303545.
- [2] Kailath, Thomas, 1980, *Linear Systems*, Prentice-Hall, New York.

One-Disk Nutating-Engine Performance for Unmanned Aerial Vehicles

T. Korakianitis

James Watt Professor
of Mechanical Engineering,
University of Glasgow,
Glasgow G12 8QQ, UK

L. Meyer

M. Boruta

Kinetic R&D, Inc.,
South Elgin, IL 60177

H. E. McCormick

C-K Engineering, Inc.,
Ballwin, MO 63011

The nutating engine is a new type of internal combustion engine. The engine has unique advantages over conventional piston engines and gas turbines in small power ranges suitable for unmanned aerial vehicles (UAV), and other applications. This publication is the original presentation of the performance potential of the simplest version of the engine, a one-disk engine operating at constant compression ratio, for light airframe propulsion. In its basic configuration the core of the engine is a nutating nonrotating disk, with the center of its hub mounted in the middle of a Z-shaped shaft. The two ends of the shaft rotate, while the disk "nutates," performs a wobbling motion without rotating around its axis. The motion of the disk circumference prescribes a portion of a sphere. A portion of the area of the disk is used for intake and compression, a portion is used to seal against a center casing, and the remaining portion is used for expansion and exhaust. The compressed air is admitted to an external accumulator, and then into an external combustion chamber before it is admitted to the power side of the disk. The external combustion chamber enables the engine to use diesel fuel in small engine sizes, giving it unique capabilities for UAV propulsion. The performance of the one-disk engine configuration for flight Mach numbers from 0 to 1 and altitudes from 0 to 20 km is presented and discussed. The performance with equal compression and expansion volume is compared with the higher-efficiency version with expansion volume higher than compression volume. A companion paper examines multidisk alternative engine configurations and load control schemes. [DOI: 10.1115/1.1496770]

Motivation

There is an ongoing military need to fuel low-power (2–500 kW) small internal combustion engines with heavy diesel, or JP types of fuels. In that power range gas turbine engines have small centrifugal compressors and turbines, and their component efficiencies are comparatively low, penalizing engine power and efficiency. Corresponding piston engines have lower rotational speeds and mass flow rates, and thus larger components than gas turbines, but they have significant difficulties with diesel fuel. Diesel requires a certain time to absorb the required heat input to evaporate after injection (Fig. 1). The small size of lower-power piston engines coupled with heavy-fuel (diesel) evaporation, flame-propagation, and space-and-speed limits introduce inherent problems with conventional diesel-fuel systems at all diesel-engine compression ratios. Thus in the power range of interest both gas turbines and piston engines have distinct drawbacks. In order to power auxiliary power units and small UAVs, among others, military units carry both gasoline and diesel fuel in operations, and there is a need to convert all operations to one JP-type (diesel) fuel. There is also a continuous need for alternative small-engine designs of low fuel consumption (high thermal efficiency) in small power ranges. This need exists for UAV, but also for several other portable-power needs of consumers (small electric generators). This is a niche market that can not be met with conventional turbomachinery or piston-engine technology. There is huge potential of a new high-power-density engine concept in the small power range (2 to 500 kW) that combines the best advantages of piston engines and turbomachinery without compromising efficiency.

The nutating engine has several distinct advantages over competing engine designs in this power range. The basic engine configuration and thermodynamic model were presented in earlier publications (Meyer [1] Korakianitis et al. [2]). However, there is significant design freedom to change engine arrangements, such as operating the engine at altitude, or with fixed but unequal sizes of compression and expansion ratios, or varying compression ratio with fixed intake volume, or with several separate disks operating in series or in parallel that have not been previously presented. These changes provide tremendous design and operational flexibility, so that the U.S. Army is sponsoring an SBIR program to investigate the performance potential of the new engine for UAV applications. The performance potential with a few of these engine variations is examined in this and a companion paper (Korakianitis et al. [3]).

Introduction

Various wobble-disk-type engines have been proposed before (Patten [4], Roseland [5], Wicha [6], Cohen [7], Nasvytis [8], Day and Koerner [9], Kreimeyer [10], Van Meegen [11], Wahl [12,13], Parker [14], Ziegler [15], and Kim [16]), but the disk was rotating as well as the shaft, thus creating friction and sealing problems along the disk perimeter. The most successful disk engine was built and tested by the National Gas Turbine Establishment in the U.K. and Caterpillar in the U.S.A. (Clarke et al. [17]). This engine had different arrangements, with complex gearing, and a rotating seal that resulted in high seal velocities (in the present engine the disk is nutating, not rotating, and the seal velocities are much lower). Clarke et al. [17] concluded that (a) satisfactory seal designs, albeit of high wear rate due to their high velocity, were developed; (b) the engine produced the power predicted by theoretical calculations, which was lower than that of a highly boosted diesel; (c) the cost of production would be higher than that of diesels due to the complexity of the engine; and (d) Caterpillar did not have a financial or technical incentive to produce that engine. The engine described in this paper has some advantages over the

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, New Orleans, LA, June 4–7, 2001; Paper 01-GT-0466. Manuscript received by IGTI, December 2000, final revision, March 2001. Associated Editor: R. Natole.

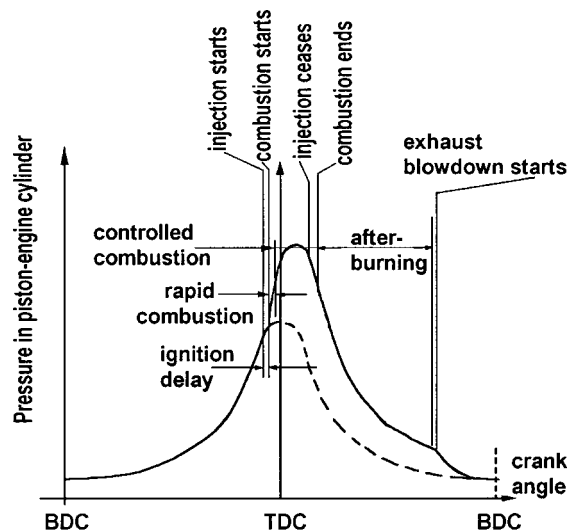


Fig. 1 Ignition delay to vaporize injected fuel in large two-stroke diesels extends into the power stroke in small engines

Caterpillar approach (the disk does not rotate), while technology has advanced to the point where such an engine has potential applications.

Disk Operation and Sealing

There are no analogies between the nutating engine and the Wankel engine. The combustion chamber of the nutating engine does not revolve. The process volumes do not become crevices. The seals wipe against their sealing surfaces at a fraction of the

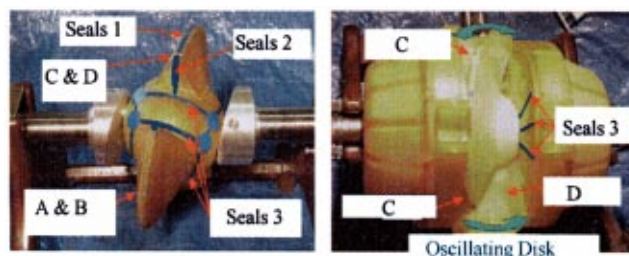


Fig. 3 Arrangements of four seal types

speed suggested by the engine rpm. Depending on the geometric details of the nutating engine, its surface to volume ratio is 1.0 to 1.2 times that of comparable piston engines. Thus any comparison of the nutating engine with the Wankel engine is misleading.

The nutating engine is compared to gas turbines and piston engines in Figs. 2–4. Figure 2 shows a photograph of a stereolithography model of the simplest version of the engine. The photograph shows the shaft and disk (left), as well as them mounted inside a housing (right). Below the photograph are cross-sectional views of the disk identifying how the disk is split by the intake and exhaust port areas of the housing into the intake/compression half of the disk (bottom) and the expansion/exhaust half of the disk (top). The two disk halves may be equal, or the intake-port and exhaust-port dividers may be rotated radially to increase the intake side at the expense of the expansion side, or vice versa. Below the cross-sectional view is a wrap-around view around the intake side (disk areas A and B) and expansion side (disk areas C and D) of the disk. These are separated by the external accumulator and pre-chamber that control the flow between the intake and expansion sides of the engine.

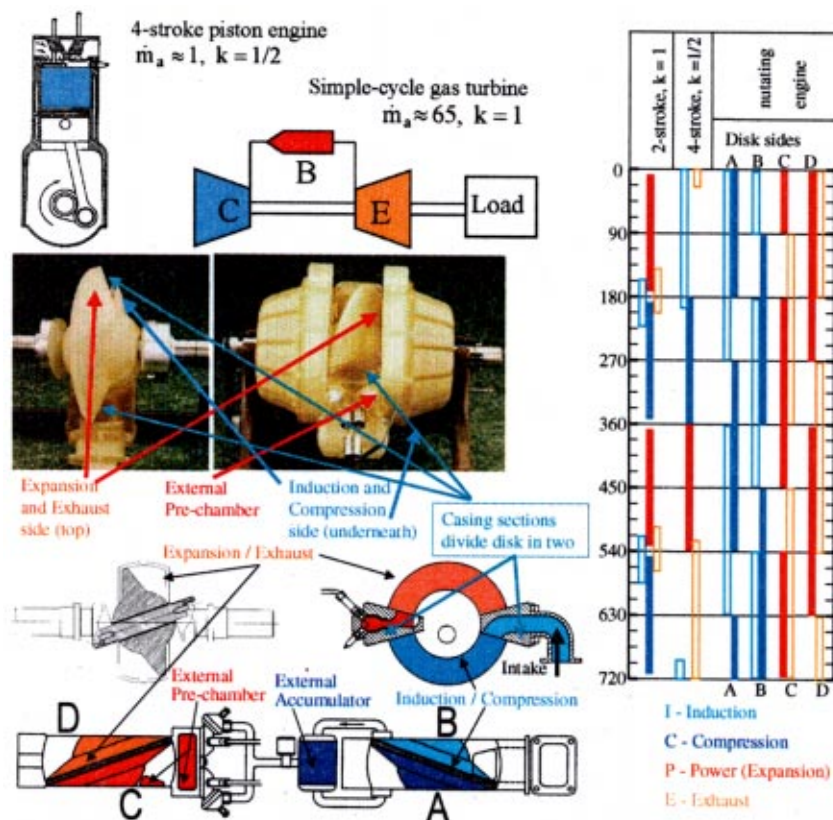


Fig. 2 Comparison of competing engines

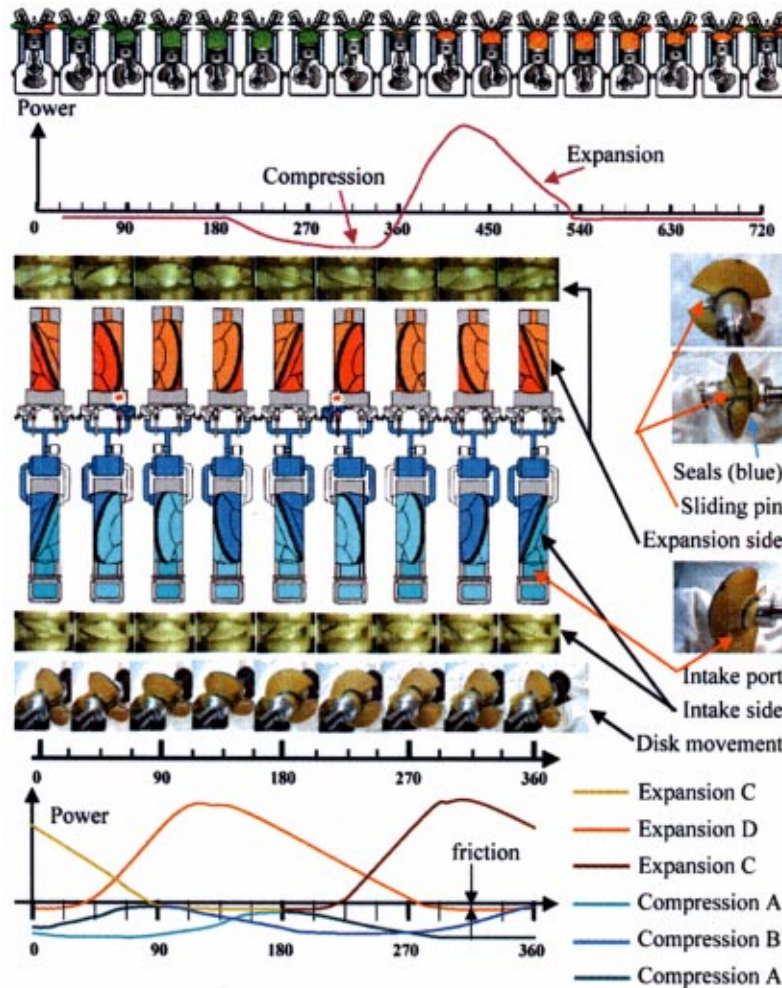


Fig. 4 Nutating-engine intake, compression, power, and exhaust processes

The disk is mounted at an angle to the Z-shaped shaft. As the shaft rotates it internally twists inside the disk, so that the disk performs a nutating (wobbling) motion without rotating in the direction of the shaft. The interior of the engine casing is defined by the volume of revolution prescribed by the two nutating-disk segments. The major advantage of the nutating engine is that each side of the disk is used once per engine revolution. While one side (A) of the intake/compression performs part of the intake process, the other side (B) is compressing the working fluid admitted 90 deg shaft-angle earlier. Because of the nutating motion of the disk and the twisting of the Z-shaft inside it, each process lasts for 270 deg crank angle, and the processes are 90 deg out of phase on each side of the disk. Similarly, 270 deg crank angle power and exhaust processes are concurrently occurring 90 deg out of phase from each other on the other side (C and D) of the disk, resulting in two intake and two power processes per shaft revolution. The right side of Fig. 2 compares the intake, compression, expansion and exhaust processes for two and four-stroke piston engines and the nutating engine.

After compression, the working fluid is collected in the accumulator. At the outlet of the accumulator there is a pressure valve, feeding the compressed air to the combustion chamber. Fuel is admitted and partial combustion occurs in the combustion chamber, which acts as a pre-chamber. Ports admit the working fluid to the expansion/exhaust sides of the disk at the correct crank angles, and the remaining combustion occurs during the beginning of the expansion process.

Figure 3 illustrates the sealing arrangements between the disk (left) and housing (right) as well as the oscillating motion of the nutating disk between the intake and exhaust port dividers. Seals (1) are between the housing and the outside periphery of the disk. Seals (2) are between the flat mid-section of disk and the housing port dividers (cf. Fig. 2). Seals (3) are radial seals mounted on the housing, acting against the rolling surface of the disk as it nutates (wobbles without rotating its perimeter) around the engine shaft. Seals (4) are between the spherical bearing area of the disk and the housing.

Seals (1) are two-piece seal energized by metal springs and working-fluid pressures (alternating). Lubricating oil is supplied to these seals from the center of the disk in a manner similar to that of cooling flows in axial turbine blades, or that in the top compression ring of piston engines). This seal minimizes leakage between the two chambers on each side of the disk (intake compression, A and B; and expansion and exhaust, C and D, Figs. 2 and 3). Disk segment A/B operates as an air compressor. The maximum pressure difference across this seal is 8 to 10 bar (with a slight vacuum on the intake side and the compression-ratio pressure on the compression side). The temperature of the seal and sealing surface of the nutating disk outside-diameter seal is under 500 K. On the intake/compression side the temperature is determined by friction against the housing interface, heat transfer from the hotter expansion/exhaust side of the disk, and heat transfer from the seal to the adjacent metal surfaces. In the expansion/exhaust side of the disk heat is transferred from the hot engine

charge to the adjacent metal surfaces. The heat generated by friction on seals (1) on the expansion exhaust side of the disk is significant due to the higher peak gas pressures energizing this seal. Three possible gas leakage paths may occur through this seal: (a) across the outside diameter surface between the outside diameter of the seal and the seal housing; (b) between side of seal and the disk groove; and (c) across the seal between the two sides of the engine. The outside surface of the seal is optimized for the shape of the interior surface of the housing. Lap joints at the seal end permit the seal to expand and contract without creating a leak path across the end of the seal. A secondary wedge minimizes the clearance between the side of the seal and the disk groove. A special-design energizing spring loads the seal radially to minimize the radial leak path at the lap joint. The area between the two seals (Fig. 3) is energized with gas pressure similarly to that between the top and second compression ring in conventional gasoline or diesel engines. Materials such as plasma spray molybdenum alloys (similar to those used on automotive/diesel engine top compression rings), or a recently developed vapor-deposition Cr N+O materials developed for the top compression ring for abrasion resistance, are suitable for the outer diameter of the seals. The expander spring is made of Inconel X-718, with a special heat-treat cycle to provide high-temperature creep/fatigue resistance in the spring. This is similar to the approach used for the apex seal during the General Motors rotary water-cooled engine project.

The mean speed of these seals is much lower than if the disk was rotating at engine rpm (the disk does not rotate), and therefore the wear rate of these seals is less than that on the rings of piston engines. Theoretical calculations of seal loads and required seal areas similar to those of Clarke et al. [17] show that a "figure of merit" of a seal design defined as a function of engine swept volume, component speed and seal area is a function of the pressure drops across the seals integrated around the thermodynamic cycle. Calculations of seal area required shows that the seals are feasible and technologically attainable.

The nutating-disk engine has some similarities to gas turbines, where the compressor, burner and expander each perform dedicated thermodynamic processes (Fig. 1, top), as opposed to piston engines where the same device (piston/chamber) is used for all processes; and it has some similarities to piston engine thermodynamic processes. It has several secondary advantages over conventional piston engines. Figure 4 compares the engine geometry and power of a four-stroke piston engine (over 720 deg) and of the nutating engine (over 360 deg). Photographs of the intake and expansion side of the disk in the housing, and of the shaft, are shown every 45 deg of crank angle, emphasizing the motion of the disk in the housing. As the compression and expansion power of the nutating engine occur for 270 deg and include a 90 deg overlap between the two sides of the disk, the torque versus shaft crank angle of the nutating engine is much smoother than that of piston engines. The center of mass of the disk remains stationary, resulting in reduced bearing loads and sizes. In addition, all the inertia is evenly distributed around the center of the ring, eliminating counterweights and the need for a flywheel (provided the load has sufficient inertia to maintain engine operation against sudden load transients).

There is considerable flexibility for alternative configurations, some of which are examined in a companion paper (Korakianitis et al. [3]). The compression and expansion sides may have different volumes (Fig. 5). In one variation the intake volume is larger than the exhaust volume, thus "self-supercharging" the engine for high-altitude applications. In the opposite variation the intake volume is smaller than the exhaust volume, permitting more-efficient expansion than piston engines (to near ambient pressure). In a different configuration two disks can be mounted in series but with disks 180 deg out of phase, thus eliminating axial thrust. In yet a different configuration two disks can be mounted in series, one dedicated to intake on both sides, the second to exhaust on both sides, thus reducing thermal stresses in the casing and disks.



Fig. 5 The intake and expansion volume ratios, r_{12} and r_{71} can be designed equal or not equal

Several disks dedicated to compression can be mounted in geometric series on the shaft, but in parallel in terms of fluid flow. These intakes can feed a common accumulator and pre-chamber, which in turn supplies fluid to several expansion disks arranged similarly to the compression disks. Different numbers of disks can be operating at different operating conditions, on ground or at altitude, at part load or full load, to maximize power or thermal efficiency at any operating condition.

Here are the "top 10" advantages of the nutating engine: (1) two power strokes per crank revolution, (2) inherently balanced, (3) fewer moving parts, less friction, higher reliability, (4) reduced seal (perimeter ring) velocities compared to piston rings; permits higher shaft speeds for increased power, or lower seal friction for comparable shaft speed, (5) variable compression ratio (accumulator valve pressure), optimizes off-design performance by changing the compression ratio, (6) expansion volume can be greater than intake volume for increased efficiency (long-range low-altitude missions), (7) compression volume can be greater than expansion volume for "self-supercharging" high-altitude low-range applications, (8) external combustion pre-chamber permits burning diesel in small engine sizes, and allows better arrangements for emission control, (9) external combustion pre-chamber increases length of flame travel, providing fuel flexibility; for example it permits use of diesel fuel in small-size engines, (10) flexibility: two disks in series displaced 180 deg along the shaft axis minimize axial-thrust bearing; one disk may be dedicated to compression, the other to expansion, improving power density and heat-transfer flows.

Thermodynamic Model

The thermodynamic cycle calculations for the nutating engine were presented in an earlier paper (Korakianitis et al. [2]). The model uses nonideal components and processes, and is illustrated in Fig. 6 (not to scale). The working fluid is either air, or a mixture of air and products of combustion, modeled as a semi-perfect gas, using the approach in (Korakianitis and Wilson [18]). The static conditions at engine inlet, assuming it is at altitude D , for $0 \leq D \leq 11000$ m T_{st0} in K and p_{st0} in N/m² are given by (Shepherd [19])

$$T_{st0} = 288.16 - 0.0065 \cdot D \quad (1)$$

$$p_{st0} = 101325 \cdot \left(\frac{T_{st0}}{288.16} \right)^{5.25757}$$

For $D > 11000$ m they are given by

$$T_{st0} = 216.66$$

$$p_{st0} = 22622.50 \cdot \exp\left(\frac{11000 - D}{6339.87}\right) \quad (2)$$

The corresponding stagnation conditions are evaluated from the flight Mach number. The inlet diffuser with a small stagnation pressure loss recovers some of the pressure head, increasing static pressure at engine inlet. The volumetric efficiency is the ratio of swept volume to volume of incoming working fluid at inlet conditions.

$$\dot{m}_a = \eta_v \rho_{il} \dot{V} = k \eta_v \rho_{il} V_{sw} N_s \quad (3)$$

$$\dot{W} = \eta_{th} \dot{m}_a (F/A) L_{hv} = k \eta_{th} \eta_v \rho_{il} V_{sw} N_s (F/A) L_{hv} \quad (4)$$

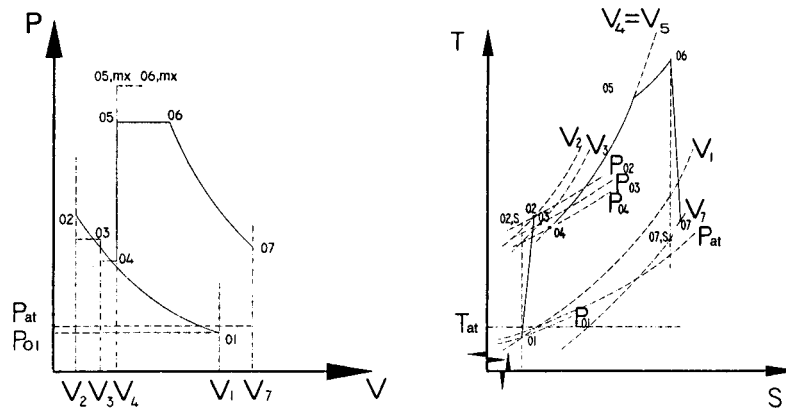


Fig. 6 Computed η_{th} and thrust

The factor k denotes the number of power cycles per swept volume V_{sw} per engine revolution. For two-stroke engines $k=1$; for four-stroke engines $k=1/2$. Equation (4) shows the critical performance factors. Given the type of fuel (L_{hv} and stoichiometric (F/A)), the determining parameters are η_{th} , η_v , ρ_{il} (increased by turbocharging or supercharging), N_s and V_{sw} .

The engine processes are modeled as follows. Every engine revolution two times the mass of working fluid that has been admitted to the swept volume V_{sw} , as affected by the is compressed to the compression ratio and admitted to the accumulator. From the accumulator the mass of air flows into the combustion chamber, where fuel is injected and combustion begins. From there the still burning mixture is admitted to the expansion volume. During the initial phases of expansion we have both burning and expansion. At the end of expansion valves or ports are opened, thus exhausting the products of combustion.

The model accounts for inlet pressure drops $f_{pl,il}$ (e.g., filter), volumetric efficiency η_v , heat losses through the engine surface to the environment and the coolant $f_{ql,cy}$ leading to temperature loss from $T_{06,mx}$ to T_{06} , isentropic compression and expansion efficiencies $\eta_{s,c}$ and $\eta_{s,e}$, pressure drop between the compressor and accumulator $f_{pl,23}$, pressure drop between the accumulator and prechamber $f_{pl,23}$, combustion efficiency η_{bu} , mechanical efficiency η_{me} , and various volume ratios $r_{ij}=V_i/V_j$ (r_{12} , r_{23} , r_{34} , r_{71} , Fig. 6). All possible fuel injection schedules from constant volume to constant pressure are modeled by $0 < \theta_1 < 1$ and $0 < \theta_2 < 1$

$$T' \equiv \frac{T_{mx}}{T_{at}} = \frac{T_{06,mx}}{T_{01}} \quad (5)$$

$$\theta_1 \equiv \frac{T_{06,mx} - T_{05,mx}}{T_{06,mx} - T_{04}} \quad (6)$$

$$f_{ql,cy} \equiv \frac{T_{06,mx} - T_{06}}{T_{06,mx} - T_{at}} \quad (7)$$

$$\theta_2 \equiv \frac{T_{06} - T_{05}}{T_{06} - T_{04}} \quad (8)$$

The model accounts for working fluid leakages from the high-pressure side to the low-pressure side of the disks. Extensions to this model compute inlet-duct pressure recovery at various flight Mach numbers for airborne applications.

In the 2–500 kW range the components of gas turbines will be much smaller than the nutating engine's and therefore of lower efficiency. Piston engines use the swept volume once per engine revolution, but for the nutating engine $k=2$ in Eq. (4). Since the thermodynamic processes of the nutating engine are similar to those of piston engines and gas turbines, its η_{th} is similar to that of comparably sized (physical volume) piston engines as well as

simple-cycle gas turbines. The nutating engine will have additional small pressure drops associated with the flow of working fluid from intake to accumulator to combustion chamber to exhaust. The average speed of the disk seals is much lower than that of comparable piston rings at the same engine rpm (or the nutating engine speed can be increased, increasing mass-flow rate and power in proportion).

UAV Performance Comparisons

The following results are for a 101.6-mm (4-in.) diameter engine, rotating at 22,000 rpm, and designed to give about 8 kW at $A=10$ km and $M=0.5$. If 70% of this power is delivered to a 6-in. diameter propeller or fan (so that the propeller tip Mach number is just subsonic), the resultant thrust is calculated from the flight velocity. The design-point performance of several engines is shown in Fig. 7. Each line shows the on-ground stationary engine (takeoff) thermal efficiency and thrust (engine power times propeller efficiency divided by flight velocity) of several nutating engines as function of compression ratio $r_c=r_{12}$ starting from $r_c=6$ (which is the last clockwise point on each line) and moving counterclockwise with increment $\delta r_c=3$. Line 2 has been calculated with the input parameters shown on the left side of Table 1. The remaining lines have been obtained by varying one input at a time from those of line 2, as indicated on the right column of Table 1. Several effects are illustrated by comparison of lines: lines 1, 2, 3 and 4 the effect of $T_{06,mx}$; lines 2, 5 and 6 the effect of heat losses; lines 2, 7 and 8 the effect of nearly constant volume versus nearly limited pressure cycle; and lines 2, 9, and 10 the effect of decreasing and increasing seal leakage by 2.5%. One of

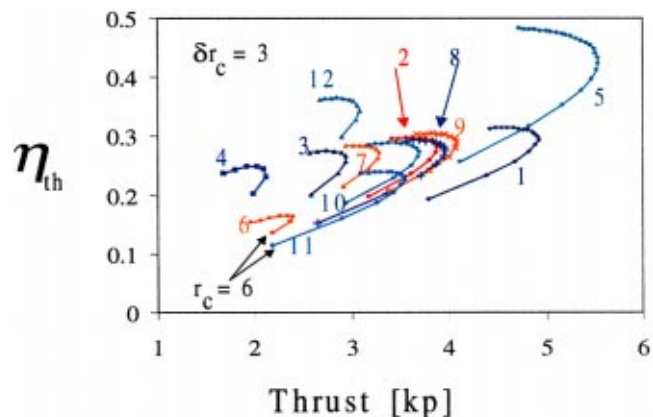


Fig. 7 Illustration of p-V and T-s diagrams of thermodynamic models of the nutating engine (not to scale)

Table 1 Inputs for the performance in Fig. 7

Standard Inputs	Line/Input Variation
$V_{sw} = 0.0595$ lt	2: Standard
$\eta_v = 0.85$	1: $T_{06, mx} = 2600$ K
$T_{06, mx} = 2300$ K	3: $T_{06, mx} = 2000$ K
$f_{pl, il} = 0.05$	4: $T_{06, mx} = 1700$ K
$\eta_{s, c} = \eta_{s, e} = 0.85$	5: $f_{ql, cy} = 0.000$
$f_{ql, cy} = 0.20$	6: $f_{ql, cy} = 0.400$
$\theta_1 = \theta_2 = 0.70$	7: $\theta_1 = \theta_2 = 0.10$
$r_{23} = r_{34} = 0.95$	8: $\theta_1 = \theta_2 = 0.95$
$f_{pl, 23} = f_{pl, 34} = 0.050$	9: leakages = 0%
$r_{71} = 1.00$	10: leakages = 5%
$\eta_{bu} = 0.996$	11: $r_{71} = 0.8230$
$\eta_{me} = 0.95$	12: $r_{71} = 2.1325$
leakage each side = 2.5%	

the unique characteristics of the nutating-disk engine is the option to design unequal compression and expansion volume ratios (Fig. 5). The effect is shown by comparison of lines 2, 11, and 12 in Fig. 7, where the performance effect is also limited by $p_{07} > p_{st0}$. Similarly to the performance of simple-cycle gas turbine engines (Korakianitis and Wilson [18]) each line in Fig. 7 indicates that there is an optimum r_c for power and higher optimum r_c for thermal efficiency. The best performance is obtained by maximizing T_{mx} . Contrary to gas turbines, in a fashion similar to piston engines, T_{mx} can be close to stoichiometric (it is acting only instantaneously in the prechamber, while the average combustion temperatures are lower). Most performance points in Fig. 7 can be considered as the design-point performance of different engines, but as explained in Korakianitis et al. [3], they can also be considered as the off-design performance points of the same engine operating at different fuel-injection schedules or at different compression-ratio settings. Thus lines 1–10 in Fig. 7 also illustrate the on-ground load-control capabilities of the nutating engine by varying the fuel injection schedule (T_{mx} and θ_2) and the compression ratio ($r_c = r_{12}$).

The flight Mach number effect at low altitudes is illustrated in Fig. 8, which shows η_{th} power [kW] and propeller thrust [kp] of a $d = 101.6$ -mm (4-in.) disk nutating engine on ground ($D = 0$), as a function of UAV flight M and engine compression ratio r_c . The power produced is 12 to 15 kW. The mass of the nutating engine with auxiliaries is about 10 kg. A piston engine of comparable compression ratio would deliver 25% of the power and thrust shown, at similar thermal efficiency. Figure 8(d) shows the performance potential of simple-cycle gas turbine engines with centrifugal impellers operating at current maximum component temperatures and efficiencies as a function of pressure ratio (solid lines efficiency, dashed lines power). This figure has been computed assuming maximum (continuous) gas turbine cycle temperature of 1400 K and component efficiencies as a function of size and compression ratio shown in Chap. 3 of (Wilson and Korakianitis [20]). Gas turbines with larger axial components could potentially reach the power levels of the nutating engine, but with 4-in. or smaller-diameter-impellers gas turbines can only produce about a third of the power of the nutating engine, due to reduced component efficiencies.

Figures 9 and 10 show the performance potential of two simple versions of the notating-disk engine suitable to power a UAV via a propeller, as a function of $0 < D < 20$ km and $0 < M < 1$. For both figures $d = 4$ in, the engine is operating at 22,000 rpm, the propeller efficiency is 70%, and the compression ratio $r_{12} = 15$. Other inputs for Figs. 9 and 10 are as shown for lines 2 and 12 in Table I, respectively; the disk is split into equal compression and expansion segments for Fig. 9 (160 deg each), while for Fig. 10 the expansion side is 240 deg and the intake side is 80 deg. Power and thrust can be changed (reduced) from those shown in Figs. 9 and 10 by changing T_{mx} and r_{12} as examined in the companion paper.

Conclusions

The performance potential of the one-disk nutating engine powering a UAV or other similar small aircraft is presented. The thermal efficiency of the nutating engine is similar to that of gas

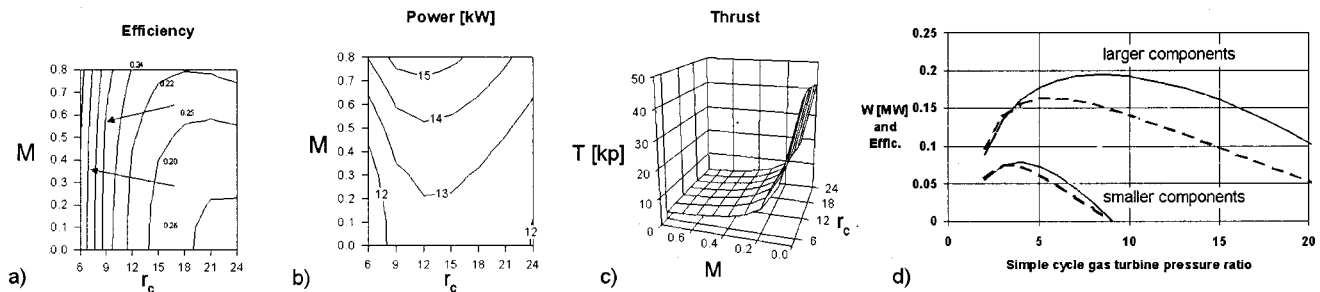


Fig. 8 (a) η_{th} ; (b) power [kW]; (c) propeller thrust [kp] of a $d = 101.6$ mm (4-in.) disk nutating engine; and (d) comparable sea level performance of a 4-in. diameter-impeller gas turbine

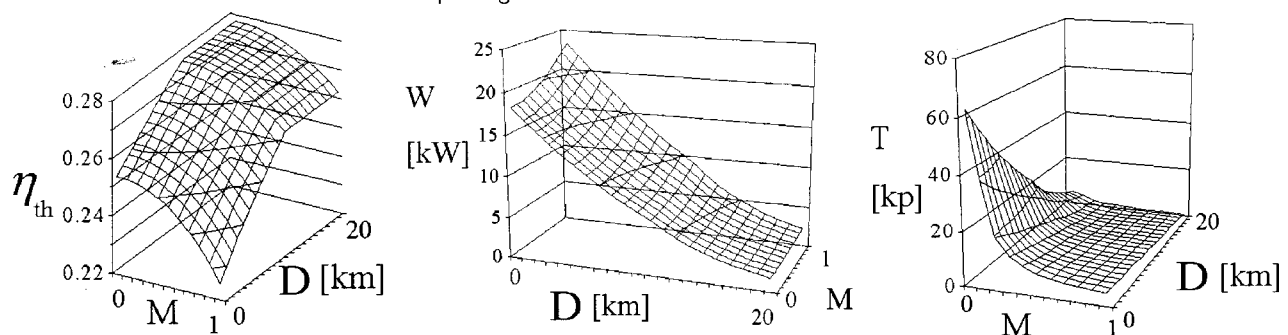


Fig. 9 Computed η_{th} , power and thrust for an engine with $r_c = 15$ and the other inputs of line 2 in Fig. 7 (expansion volume = compression volume), at several operating altitudes and Mach numbers.

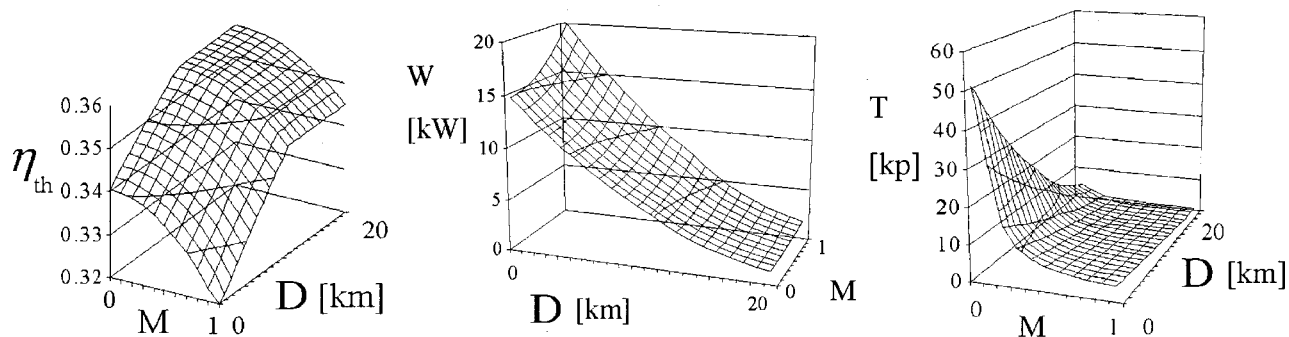


Fig. 10 Computed η_{th} , power and thrust for an engine with $r_c=15$ and the other inputs of line 12 in Fig. 7 (expansion volume > compression volume), at several operating altitudes and Mach numbers

turbines and piston engines of comparable technology and size. The power density (specific power) of gas turbines is higher than that of the nutating engine, except in low power ranges (2–500 kW) where the size of gas turbine components becomes too small thus reducing component efficiencies. In these power ranges and at 2000 to 20,000 shaft rpm the size of the nutating engine disk is between 0.10 to 0.30 m, making the component performance suitable for efficient power generation. The major advantage of nutating engines compared to piston engines of similar weight and volume is the higher power density. It is concluded that the engine has significant advantages over comparable gas turbines and piston engines. In addition, because of its external combustion, it can burn diesel in small engine sizes. The performance depends on maximum cycle temperature (fuel added), compression ratio, design point Mach number and altitude, but the arrangement flexibility of the engine enables it to be changed so that it can be optimized for several off-design conditions. The nutating engine does not require complex gearing, reducing production costs, and it is aimed at applications requiring a technical solution of higher power density in moderate power ranges.

Nomenclature

D	= altitude
f	= heat-loss or pressure-loss factor
(F/A)	= fuel-to-air mass ratio
k	= volume sweeps (power strokes) per shaft rotation ($k=2$ for volume V_{sw} in the nutating engine)
L_{hv}	= lower heating value of the fuel
M	= flight Mach number
$m, \dot{m}$	= mass, mass-flow rate
N_s	= revolutions per second
p	= pressure
r	= compression (volume) ratio, $r_{ij} \equiv V_i/V_j$, $i=1, \dots, 7$
T, T'	= temperature, temperature ratio
V	= volume
$W, \dot{W}$	= work, power
η	= efficiency
θ	= combustion-model factor
ρ	= density

Subscripts

01 to 07	= thermodynamic cycle points (Fig. 7)
a	= air

bu	= burner
c, e	= compression, expansion
f	= fuel
il	= engine inlet
me	= mechanical
mn, mx	= minimum, maximum
pl, ql	= pressure loss, heat loss
s	= isentropic
sw	= swept
th	= thermal
v	= volumetric

References

- [1] Meyer, L., 1993, "Nutating Internal Combustion Engine," U.S. Patent No. 5,251,594.
- [2] Korakianitis, T., Meyer, L., Boruta, M., and McCormick, H. E., 1999, "Introduction and Performance Prediction of a Nutating-Disk Engine," ASME, Journal of Engineering for Gas Turbines and Power, accepted for publication.
- [3] Korakianitis, T., Meyer, L., Boruta, M., and McCormick, H. E., 2001 "Alternative Multi-Nutating-Disk Engine Configurations for Diverse Applications," under review.
- [4] Patten, J., 1882, "Rotary Engine," U.S. Patent No. 263,573.
- [5] Roseland, T. C., 1905, "Rotary Engine," U.S. Patent No. 790,264.
- [6] Wicha, A., 1935, "Gas and Vapor Compressor," U.S. Patent No. 1,986,454.
- [7] Cohen, W. A. 1937, Rotary Engine, U.S. Patent No. 2,069,646.
- [8] Nasvytis, A. L., 1961, "Nutating Disk Motor," U.S. Patent No. 2,992,635.
- [9] Day, D. B., and Koerner, W. J., 1963, Nutating Disk Internal Combustion Engine, U.S. Patent No. 3,102,517.
- [10] Kreimeyer, H., 1970, "Rotary Nutating Power Device," U.S. Patent No. 3,492,974.
- [11] Van Meegen, H. A. B., 1974, "An Improved Motor, Pump or the Like," Australian Patent No. 67,511.
- [12] Wahl, R. H., 1975, "Rotary Nutating Engine," U.S. Patent No. 3,895,610.
- [13] Wahl, R. H., 1976, "Nutating Engine," U.S. Patent No. 3,981,639.
- [14] Parker, 1982, "Swashplate Machines," U.S. Patent No. 3,942,384.
- [15] Ziegler, M., 1982, "Die Erfindung betrifft einen Rotationsskolbenverbrennungsmotor insbesondere für den Automobilbau," German Patent No. 3,942,384.
- [16] Kim, J. D., 1992, "Rotary Wavy Motion Type Engine," U.S. Patent No. 5,138,993.
- [17] Clarke, J. M., Walker, D. F., and Hamilton, P. H., 1972, "A New Class of Rotary Piston Machine Suitable for Compressors, pumps and Internal Combustion Engines," Proceedings of Inst. Mech. Eng., **186**, 62/72.
- [18] Korakianitis, T., and Wilson, D. G., 1994, "Models for Predicting the Performance of Brayton Cycle Engines," ASME J. Eng. Gas Turbines Power, **116**, pp. 381–388.
- [19] Shepherd, D. G., 1972, *Aerospace Propulsion*, Elsevier, New York.
- [20] Wilson, D. G., and Korakianitis, T., 1998, *The Design of High-Efficiency Turbomachinery and Gas Turbines*, 2nd Ed., Prentice-Hall, Englewood Cliffs, NJ.

Alternative Multinutating Disk Engine Configurations for Diverse Applications

T. Korakianitis

James Watt Professor of Mechanical Engineering,
University of Glasgow,
Glasgow G12 8QQ, UK

L. Meyer

M. Boruta

Kinetic R&D, Inc.,
South Elgin, IL 60177

H. E. McCormick

C-K Engineering, Inc.,
Ballwin, MO 63011

The nutating engine is a new type of internal combustion engine with distinct advantages over conventional piston engines and gas turbines in small power ranges. The engine's unique arrangement flexibility allows several alternative disk and shaft configurations, each selected for a different application. Variations in cycle temperature ratio and compression ratio during normal operation enable the engine to effectively become a variable-cycle engine, allowing significant flexibility for maximum efficiency or power or other optimizing function for on-ground stationary or for airborne applications. In its basic configuration the core of the engine is a nutating nonrotating disk, with the center of its hub mounted in the middle of a Z-shaped shaft. The two ends of the shaft rotate, while the disk "nutates," performs a wobbling motion without rotating around its axis. The motion of the disk circumference prescribes a portion of a sphere. A portion of the area of the disk is used for intake and compression, a portion is used to seal against a center casing, and the remaining portion is used for expansion and exhaust. The compressed air is admitted to an external accumulator, and then into an external combustion chamber before it is admitted to the power side of the disk. A companion paper examines the performance potential of the one-disk engine. This paper examines alternative engine configurations. The external combustion chamber enables the engine to operate on a variable compression ratio cycle. In addition, separate disks of unequal size are used for intake and expansion, resulting in distinct and significant power, efficiency, fuel flexibility, and arrangement advantages over conventional piston engines, over gas turbines, and over the basic nutating-engine configuration. The performance of these arrangements is examined for: on-ground power, on-ground efficiency, (auxiliary power, automotive); and for small and light airframe applications for flight Mach numbers from 0 to 1 and altitudes from 0 to 20 km. This publication is the original presentation of the performance potential of several alternative configurations of the basic engine, such as multidisk arrangements, combustion and exhaust disks of different size, and variable-compression ratio (variable cycle) configurations. [DOI: 10.1115/1.1688770]

Introduction

There are continuous demands for higher efficiency and higher power from internal combustion engines at off-design conditions. In the 2–500 kW power range gas turbine engines have small centrifugal compressors and turbines, and their component efficiencies are comparatively low, penalizing engine power and efficiency. Corresponding piston engines have lower mass flow rates and thus slightly larger components than gas turbines, but they are heavier. At off-design and part-load operating conditions the performance of gas turbines deteriorates much faster than that of piston engines. The off-design performance problem is further exacerbated for airborne applications, where the engine performance is directly dependent on inlet air density, and where the flight Mach number varies from zero (static condition at take off) to the maximum Mach number in the flight envelope. Thus in this power range both gas turbines and piston engines have distinct drawbacks. Therefore there is a continuous need for alternative small-engine designs of low fuel consumption (high thermal efficiency) in small power ranges. This need exists for unmanned aerial vehicles (UAV), but also for several other portable power needs of consumers (small electric generators) as well as the military (small underwater vehicles, UAV, etc). This is a niche market that

can not be met with conventional turbomachinery or piston engine technology. There is huge potential of a new high-power-density engine concept in the small power range (2 to 500 kW) that combines the best advantages of piston engines and turbomachinery without compromising efficiency.

The nutating disk engine is a unique combination of the best features of gas turbines and piston engines, and it is well suited to fit the above need. The basic engine configuration and thermodynamic model were presented in earlier publications (Meyer [1] and Korakianitis et al. [2]). There is considerable flexibility for alternative configurations, some of which are examined in a companion paper (Korakianitis et al. [3]). This paper investigates additional configuration variations such as those illustrated in Fig. 1(c) (right) and Fig. 2.

Basic and Alternative Configurations

The simplest form of the engine concept has been presented and compared to gas turbines and piston engines and illustrated in Figs. 1–4, [3], which are included here by reference. The one-disk configuration is illustrated in the stereolithography models of Fig. 1. The disk is mounted at an angle to the shaft. As the shaft rotates the disk performs a nutating motion. The disk is separated into two parts, the intake/compression and power/exhaust sides. One set of seals on the perimeter of the disk wipe past the engine casing around it. A second set of seals wipe against the center of the housing between the compression and expansion sides of the disk. A third set of seals separates the intake and compression

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, New Orleans, LA, June 4–7, 2001; Paper 2001-GT-0572. Manuscript received by IGTI, December 2000, final revision, March 2001. Associate Editor: R. Natole.



Fig. 1 (Left) stereolithography model of basic configuration; (center) one-disk compression-ratio variations; (right) multidisk shaft arrangement and compression-ratio variations

functions of the disk, (and another similar set separates the expansion and exhaust sides). Details of seal arrangements and materials are included in [3].

The interior of the engine casing is defined by the volume of revolution prescribed by the two nutating disk segments. The major advantage of the nutating engine is that each side of the disk is used once per engine revolution. Because of the nutating motion each process (intake, compression, power, exhaust) lasts for 270 deg crank angle, and the processes are 90 deg out of phase on each side of the disk. This results in two intake and two power processes per shaft revolution. After compression, the working fluid is collected in the accumulator. At the outlet of the accumulator there is a pressure valve, feeding the compressed air to the combustion chamber. Fuel is admitted and partial combustion occurs in the combustion chamber, which acts as a pre-chamber. Ports admit the working fluid to the expansion/exhaust sides of the disk at the correct crank angles, and the remaining combustion occurs during the beginning of the expansion process. This feature of the external accumulator and combustion chamber separates the intake and compression function from the expansion and exhaust function, and allows the intake and compression disk or disks to be separate from the expansion and exhaust disk or disks. This is one significant element of the design flexibility of the engine examined in this paper. The other element is that the pressure regulating valve at the outlet of the accumulator can be set to different (lower) pressures than the maximum pressure ratio dictated by the volumetric compression ratio of the inlet-disk geometry. Essen-

tially this makes the engine a variable compression ratio engine (variable cycle), allowing extra dimensions for the optimization of engine performance at off-design operating conditions that are not available to conventional piston engines and gas turbines. These previously unpublished and unique features of the engine are examined in this paper.

Overall the nonrotating nutating disk engine has some similarities to gas turbines, where the compressor, burner, and expander each perform dedicated thermodynamic processes, as opposed to piston engines where the same device (piston/chamber) is used for all processes; and it has some similarities to piston engine thermodynamic processes. It has several secondary advantages over conventional piston engines. For example, the thermodynamic processes last for 270 deg crank angle and overlap for 90 deg, so that the nutating engine has a smoother torque curve versus crank angle. The center of mass of the disk remains stationary, resulting in reduced bearing loads and sizes.

There are no suitable analogies between the nutating engine and the Wankel engine. The combustion chamber of the nutating engine does not revolve. The process volumes do not become crevices. The seals wipe against their sealing surfaces at a fraction of the speed suggested by the engine rpm. Depending on the geometric details of the nutating engine, its surface to volume ratio is 1.0 to 1.2 times that of comparable piston engines. Any comparison with the Wankel engine is misleading.

There is significant design freedom to change engine arrangements, such as operating the engine with fixed but unequal sizes

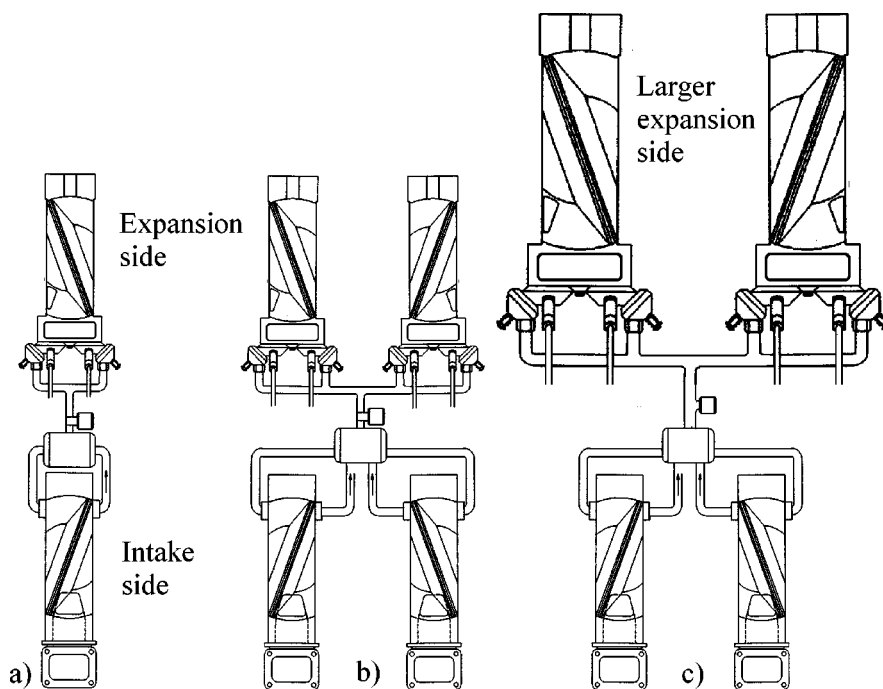


Fig. 2 Disk configurations: (a) one disk; (b) two intake and two expansion disks ($d_e = d_i$); and (c) two intake and two expansion disks ($d_e > d_i$)

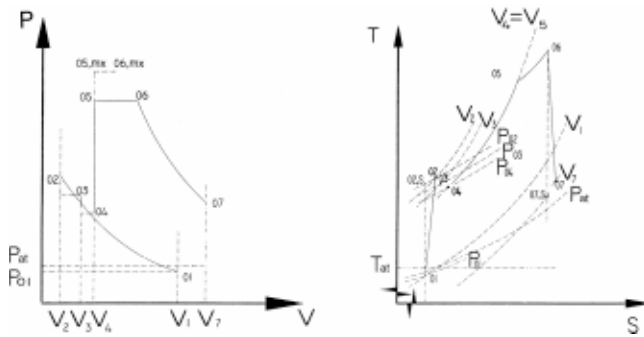


Fig. 3 Illustration of p-V and T-s diagrams of thermodynamic models of the nutating engine (not to scale)

of compression and expansion ratios, or varying compression ratio with fixed engine volumes, or with several separate disks operating in series or in parallel that have not been previously presented. For example, in the one-disk configuration the compression and expansion sides may have different volumes (Fig. 1, center, and Fig. 2(a)). In one variation the intake volume is larger than the exhaust volume, thus “self-supercharging” the engine for high-altitude applications. In the opposite variation the intake volume is smaller than the exhaust volume, permitting more efficient expansion than piston engines (to near ambient pressure).

Significant efficiency advantages can be gained by having the expansion volume bigger than the compression volume to an extent higher than that illustrated in the center of Fig. 2. This is easier to achieve by using several disks of equal or unequal sizes mounted on the same shaft (illustrated in Fig. 1 right with three disks, and in Figs. 2(b) and 2(c) with two disks). Two disks can be mounted in series but with disks 180 deg out of phase (Fig. 2(b)), thus eliminating axial thrust. Each of the two disks can have an intake and exhaust side (each disk similar to the basic configuration), or one disk can be dedicated to compression on both sides, and the other dedicated to expansion on both sides, thus reducing thermal stresses in the casing and disks. In these multidisk configurations the intake and expansion sides and/or disk diameters need not be equal (Fig. 2(c)), so that the concept presented in the center of Fig. 1 can be extended to multidisk configurations.

In yet a different configuration (Fig. 1, right) several intake disks can be mounted in geometric series on the shaft, but in parallel in terms of fluid flow. These intakes can feed a common accumulator and pre-chamber, which in turn supplies fluid to several expansion disks arranged similarly to the compression disks and coupled to them on a common multidisk shaft. Different numbers of intake or exhaust disks can be operating at different altitudes and Mach numbers, at part load or full load. For example if three intake and three exhaust disks were used, then for takeoff and low-altitude operations two intake disks could be feeding

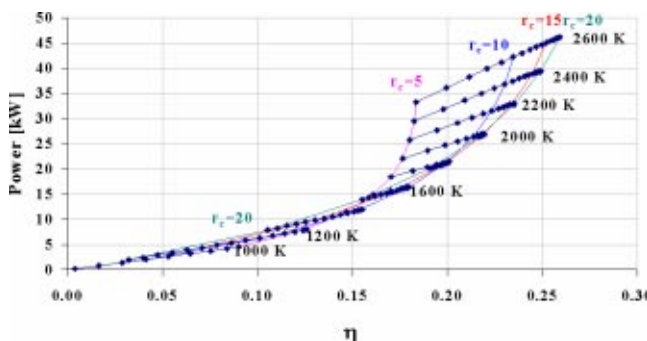


Fig. 4 Sea level static η_{th} and shaft power of a two-disk nutating engine with $d_i=d_e=6$ -in. (Fig. 2(b)) at 3600 rpm

three expansion disks, thus maximizing thermal efficiency by expanding closer to atmospheric pressure. At high altitudes three intake disks could be feeding two expansion disks, providing a “self-supercharging” option for increased power (intake air) at the expense of thermal efficiency. This flexibility allows maximizing power or thermal efficiency at any operating condition. These changes provide tremendous design and operational flexibility. The purpose of this paper is the original presentation of these unique advantages of alternative configurations of the nutating-disk engine that are not apparent in previous publications.

Performance Computations

Engine performance has been computed using the thermodynamic model presented in [2]. The thermodynamic cycle is illustrated in Fig. 3. The working fluid is modeled as a semi-perfect gas before combustion, and a mixture of semi-perfect gases as a function of fuel/air ratio after combustion. The static conditions at engine inlet, assuming it is at altitude D , for $0 \leq D \leq 11000$ m T_{st0} in K and p_{st0} in N/m² are given by ([4])

$$T_{st0} = 288.16 - 0.0065 \cdot D$$

$$p_{st0} = 101325 \cdot \left(\frac{T_{st0}}{288.16} \right)^{5.25757} \quad (1)$$

For $D > 11000$ m they are given by

$$T_{st0} = 216.66$$

$$p_{st0} = 22622.50 \cdot \exp\left(\frac{11000 - D}{6339.87}\right) \quad (2)$$

The model accounts for inlet pressure drops $f_{pl,il}$ (e.g., filter), volumetric efficiency η_v , heat losses through the engine surface to the environment and the coolant $f_{ql,cy}$ leading to temperature loss from $T_{06,mx}$ to T_{06} , isentropic compression and expansion efficiencies $\eta_{s,c}$ and $\eta_{s,e}$, pressure drop between the compressor and accumulator $f_{pl,23}$, pressure drop between the accumulator and pre-chamber $f_{pl,23}$, combustion efficiency η_{bu} , mechanical efficiency η_{me} , and various volume ratios $r_{ij} = V_i/V_j$ (r_{12} , r_{23} , r_{34} , r_{71} , Fig. 3). All possible fuel injection schedules from constant volume to constant pressure are modeled by $0 < \theta_1 < 1$ and $0 < \theta_2 < 1$,

$$T' \equiv \frac{T_{mx}}{T_{at}} = \frac{T_{06,mx}}{T_{01}} \quad (3)$$

$$\theta_1 \equiv \frac{T_{06,mx} - T_{05,mx}}{T_{06,mx} - T_{04}} \quad (4)$$

$$f_{ql,cy} \equiv \frac{T_{06,mx} - T_{06}}{T_{06,mx} - T_{at}} \quad (5)$$

$$\theta_2 \equiv \frac{T_{06} - T_{05}}{T_{06} - T_{04}} \quad (6)$$

The model accounts for working fluid leakages from the high-pressure side to the low-pressure side of the disks. Extensions to this model compute inlet-duct pressure recovery at various flight Mach numbers for airborne applications.

On-Ground Static Performance

Figure 4 shows the sea level static ($D=0$ km, $M=0$) performance of two $d_i=d_e=6$ in. diameter disks mounted in series as shown in Fig. 2(b) operating at 3600 rpm (power increases proportionately with rpm). The results are identical whether one disk is dedicated to compression and the second to expansion, or whether each individual disk is split between compression and expansion functions as in the basic configuration (Fig. 2(a) for each disk). The performance is shown as a function of theoretical

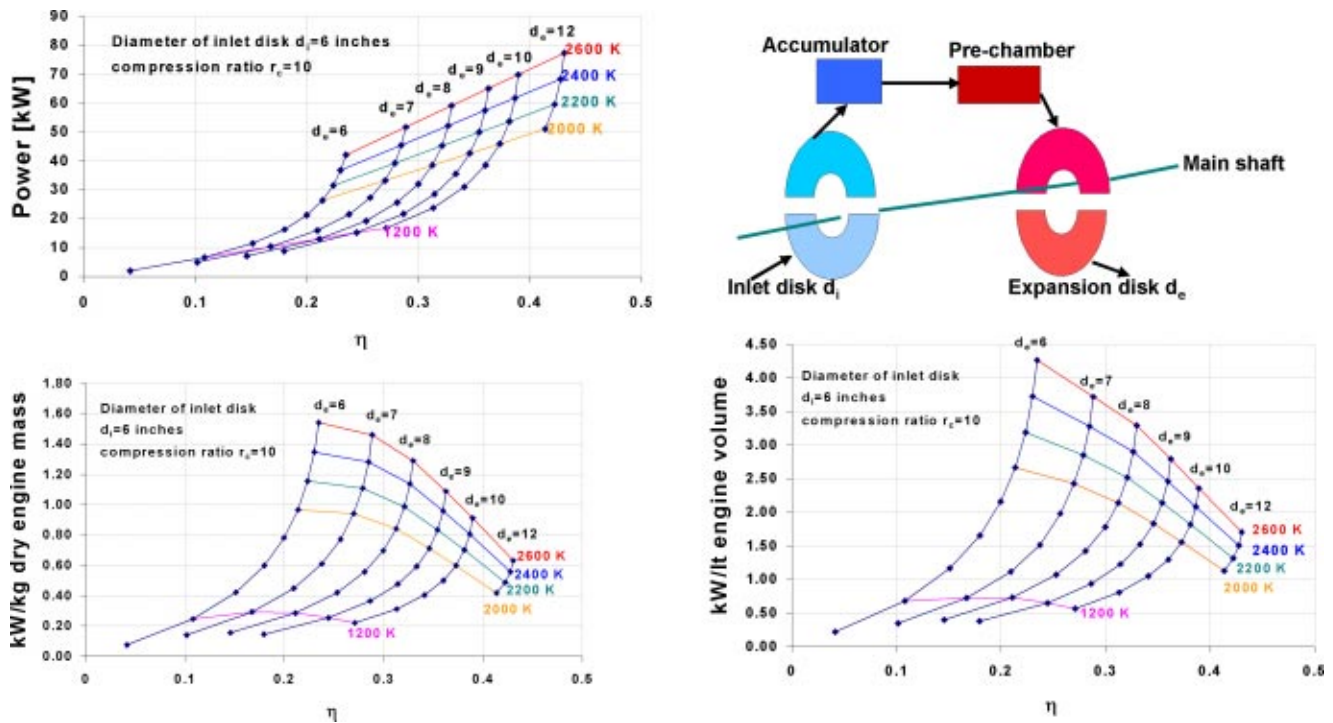


Fig. 5 Shaft power [kW] (top left), shaft power/engine dry mass [kW/kg] (bottom left), and shaft power/engine volume occupied [kW/lt] (bottom right) versus thermal efficiency for a two-disk engine with intake disk diameter $d_i=6$ in. as a function of expansion disk diameter d_e (Fig. 2(c)) compression ratio r_c and maximum theoretical cycle temperature T_{mx}

maximum cycle temperature $T_{mx}=T_{06,mx}=1000$ K, 1200 K, . . . 2600 K, (where the actual maximum temperature T_{06} is much lower than T_{mx} due to heat losses $f_{ql,cy}$), and compression ratios $r_c=5,10,15,20$. The “horizontal” lines indicate values of theoretical T_{mx} . The tick marks on each line represent the compression ratios starting from a value of 5 in increments of 1. Other inputs for Fig. 4 are

$$\begin{aligned} T_{at} &= 288 \text{ K} & p_{at} &= 101.3 \text{ kPa} & f_{p,il} &= 0.02 & \eta_{s,c} &= 0.85 \\ \eta_{s,e} &= 0.85 & f_{ql,cy} &= 0.150 & \theta_1 &= 0.50 & \theta_2 &= 0.50 \\ r_{23} &= 0.97 & f_{p,l,23} &= 0.200 & r_{34} &= 0.97 & f_{p,l,23} &= 0.200 \\ r_{71} &= 1.00 & \eta_{bu} &= 0.996 & \eta_{me} &= 0.98 & & 3600 \text{ rpm} \end{aligned}$$

and leakage flows 2% across each seal.

In previous similar “carpet plots,” [2,3,5], each point on the graph represented the design point performance of a different engine. Figure 4 is more general in that each point on the graph can also represent the off-design performance of a nutating engine. For example let’s consider an engine designed to operate at $T_{mx}=2600$ K and $r_c=20$ at 3600 rpm so that it delivers about 47 kW with $\eta_{th}=0.26$ (top-right point in Fig. 4), and assume we wanted to couple it with a propeller load that needs 12.5 kW at 3000 rpm. Since (to a first approximation) power is proportional to rpm 12.5 kW at 3000 rpm corresponds to 15 kW at 3600 rpm on Fig. 4. This power can be obtained at several compression ratios, for example: (a) $r_c=5$ at 1600 K attaining $\eta_{th}=0.16$; (b) $r_c=10$ at 1550 K attaining $\eta_{th}=0.17$; (c) $r_c=15$ at 1590 K attaining $\eta_{th}=0.165$; and (d) $r_c=20$ at 1570 K attaining $\eta_{th}=0.162$. Clearly in this case the optimum efficiency is obtained at $r_c=10$. The control scheme could dictate the fuel injection such as to correspond to $T_{mx}=1550$ K and the pressure valve at accumulator outlet could reduce the accumulator pressure to correspond to $r_c=10$. When the pressure during compression exceeds this value then the working fluid escapes from the compression chamber to the accumulator without absorbing any further shaft work. This feature allows the engine to operate on a variable compression

ratio cycle. At different powers and speeds higher or lower r_c are used to attain the maximum efficiency. Thus figures like Fig. 4 can be used to design the engine control scheme (fuel injection schedule, ignition timing, and compression ratio) such as to maximize performance at each off-design condition.

An alternative engine configuration is discussed in Fig. 5. This involves a two-disk engine, where the smaller (intake) disk has diameter $d_i=6$ in. and the larger (expansion) disk has diameter $d_e=6, 7, 8, 9, 10, 12$ in. This configuration allows greater flexibility in the value of r_{71} than the one shown in the center of Fig. 1. This is because the volume ratio r_{71} varies by the cube of the disk-diameter ratio, so that with two disks when $d_e/d_i=12/6=2$ then $r_{71}=2^3=8$. In contrast in the companion paper, [3], the variation in r_{71} with one disk is much smaller $0.82 < r_{71} < 2.13$. For the performance shown in Fig. 5 the compression ratio $r_c=10$. Higher values of r_c with $r_{71} > 2.5$ demand high-temperature operation $T_{mx} > 2200$ K, otherwise the high expansion ratio results in exhaust pressures below atmospheric.

The power [kW] graph in Fig. 5 illustrates the power and efficiency advantages of high r_{71} . However, the engine weight [kW/kg] and required space [kW/lt] increase faster than power and efficiency with increasing r_{71} , as illustrated by the engine weight [kW/kg] and required space [kW/lt] graphs of Fig. 5. High r_{71} is suitable for on-ground applications where engine volume and weight are not of concern. For airborne applications any gains in efficiency and power must be carefully balanced against the fuel required for the mission and the overall engine space, weight, and vehicle aerodynamic arrangements.

Airborne Performance

This section investigates the unique advantages of the variable compression ratio nutating engine for airborne applications (for $0 < D < 20$ km and $0 < M < 1$). In cycle optimization cases for all types of engines, including the nutating-disk engine, the most dominant parameter is maximizing $T' = T_{mx}/T_{mn}$, where T_{mn} is a function of altitude as shown in Eqs. (1)–(2). For all results

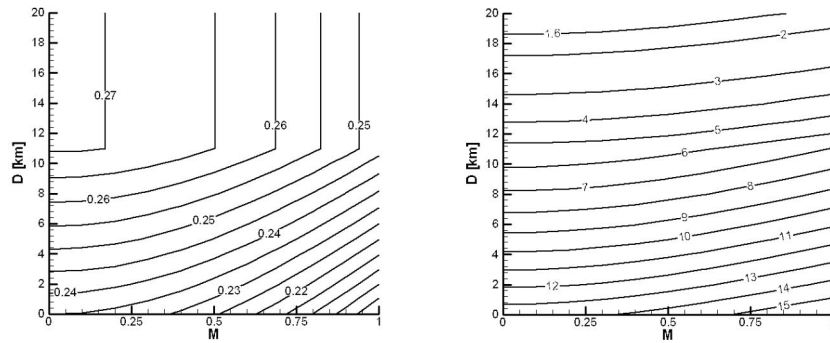


Fig. 6 Constant $r_c=15$ with $d_i=d_e=4$ in.: (left) η_{th} ; (right) power [kW]

shown in this section $T_{mx}=2000$ K. The second dominant parameter is optimization of compression ratio r_c for power, efficiency, or reasonable compromise between the two. Even though in small diesel engines the compression ratio is of the order of 15, in large diesel engines it can reach about 25. Assuming operation on jet propulsion (JP) fuels, no maximum restriction was placed on compression ratio for the results presented below (the maximum shown in about 24). The third dominant parameter is optimization of $r_{71}=(d_e/d_i)^3$, within the weight and volume restrictions implied by Fig. 5.

Figure 6 shows the performance of a two disk engine, with one disk dedicated to compression and the second to expansion, with $d_i=d_e=4$ in., operating at constant $r_c=15$. Other inputs are as follows:

$$\begin{aligned} 0 < D < 20 \text{ km} \quad 0 < M < 1 \quad f_{pl,il} &= 0.05 \quad \eta_{s,c} = 0.85 \\ \eta_{s,e} &= 0.85 \quad f_{ql,cy} = 0.200 \quad \theta_1 = 0.50 \quad \theta_2 = 0.50 \\ r_{23} &= 0.95 \quad f_{pl,23} = 0.050 \quad r_{34} = 0.95 \quad f_{pl,23} = 0.050 \end{aligned}$$

$$T = 2000 \text{ K} \quad \eta_{bu} = 0.996 \quad \eta_{me} = 0.95 \quad 12,000 \text{ rpm}$$

and leakage flows 2.5% across each seal. The sharp discontinuity in η_{th} at $D=11$ km is due to the T_{mn} effect from Eqs. (1)–(2). Maximum η_{th} and minimum power occur at low M and high D . Minimum η_{th} and maximum power occur at high M and low D . This observation already indicates the desire to further improve the cycle and match it with the power requirements of potential airframes by varying the compression ratio, as shown in Figs. 7 and 8. In Fig. 7 all inputs are the same as for Fig. 6, except r_c is varied to maximize η_{th} . In Fig. 8 all inputs are the same as for Fig. 6, except r_c is varied to maximize power. The power differences between Figs. 6, 7, and 8 are relatively small, though not negligible. The big differences occur in η_{th} , which for both Figs. 7 and 8 is significantly increased from that of Fig. 6. As expected from previous “carpet plots” of performance, maximum power occurs at lower compression ratios $13 < r_c < 7.5$ (Fig. 8 right) than maximum η_{th} , which occurs at $24 < r_c < 11$ (Fig. 7 right). How-

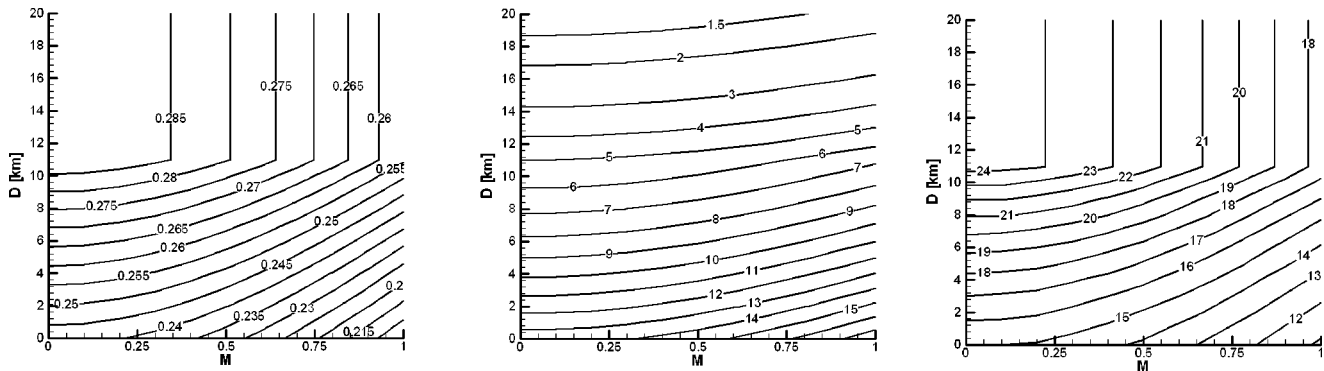


Fig. 7 Maximizing η_{th} by varying r_c with $d_i=d_e=4$ in.: (left) η_{th} ; (center) power [kW]; (right) r_c

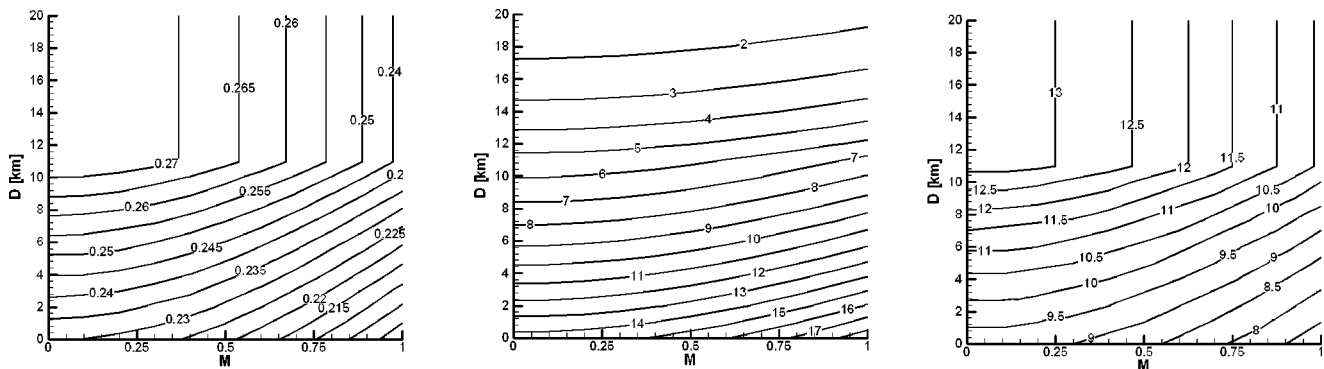


Fig. 8 Maximizing power [kW] by varying r_c with $d_i=d_e=4$ in.: (left) η_{th} ; (center) power [kW]; (right) r_c

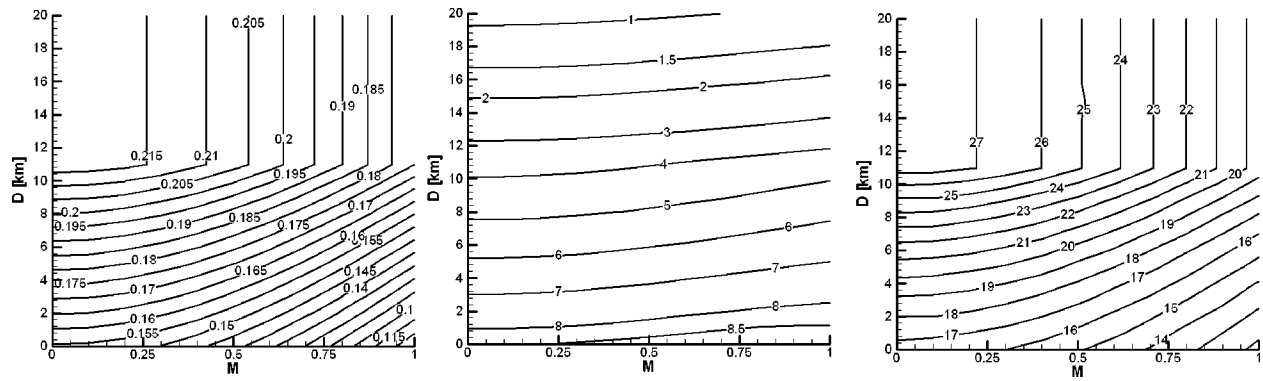


Fig. 9 Maximizing η_{th} by varying r_c with $d_i=4$ in., $d_e=3.5$ in.: (left) η_{th} ; (center) power [kW]; (right) r_c

ever, even the cases optimized for maximum power (Fig. 8 left) offer higher efficiency than those with the constant compression ratio (Fig. 6 left).

Figure 9 and 10 show the performance changes of using two disks, $d_i=4$ in. and $d_e=3.5$ in.: r_c is varied to optimize η_{th} in Fig. 9; and to optimize power in Fig. 10. The power output is significantly penalized by the reduced amount of air entering the smaller intake disk. There are no significant advantages over Figs. 6, 7, and 8 with these choices, except for perhaps very specific applications such as short-range high-altitude missions, where the reduction in engine weight with the smaller intake disk outweigh the increased fuel required for the mission.

In contrast, Figs. 11 and 12 show the significant performance improvements of using two disks, $d_i=4$ in. and $d_e=5.7$ in.: r_c is varied to increase η_{th} in Fig. 11; and power in Fig. 12. This r_{71} is the inverse of the one in Figs. 9 and 10. In both cases the power is increased significantly (almost doubled) from Figs. 6, 7, and 8,

though the general shape of the power distribution in the (M,D) flight envelope remains similar. In both of these cases the optimum r_c are much lower than in Figs. 6, 7, and 8, and indicate the likely need for a separate ignition sources even with JP fuels for most of the flight envelope. The distribution of optimum r_c curves and resulting η_{th} in the flight envelope has changed from those of Figs. 6, 7, and 8, and η_{th} has increased substantially. Clearly this approach is more desirable than either the constant r_c of Fig. 6 or the $d_i=d_e$ of Figs. 7 and 8 for most airborne applications, as the increase in engine weight is easily overcome by reductions in fuel required for most missions.

Conclusions

Multidisk configurations significantly expand the design flexibility and performance potential of the nutating-disk engine. The use of two or more disks enables the expansion ratio to be much

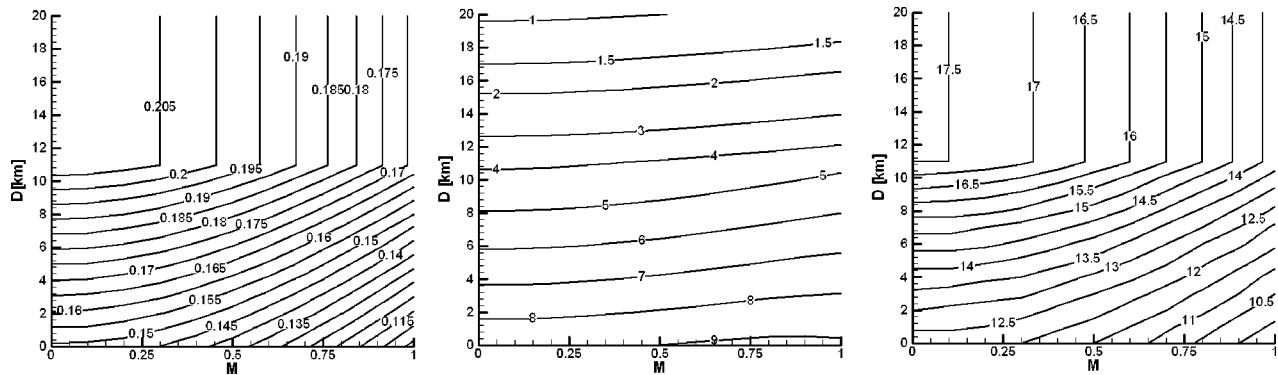


Fig. 10 Maximizing power [kW] by varying r_c with $d_i=4$ in., $d_e=3.5$ in.: (left) η_{th} ; (center) power [kW]; (right) r_c

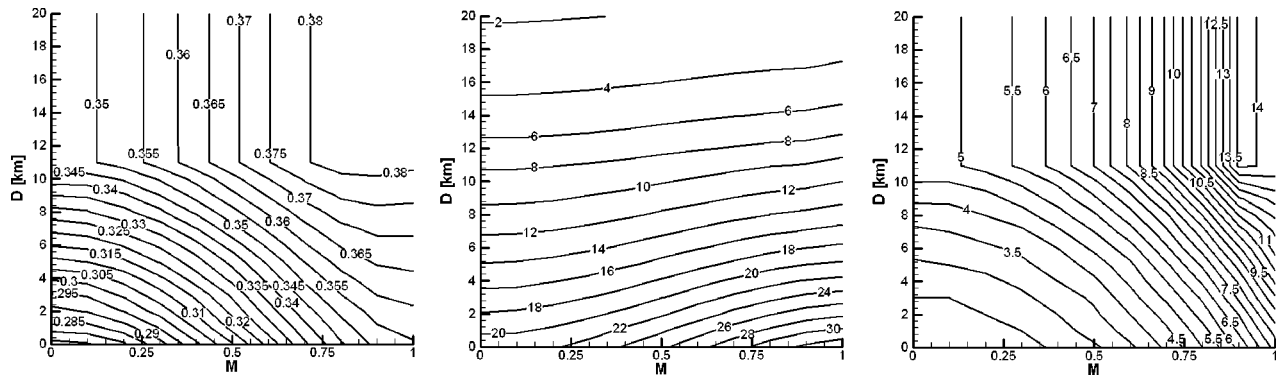


Fig. 11 Maximizing η_{th} by varying r_c with $d_i=4$ in., $d_e=5.7$ in.: (left) η_{th} ; (center) power [kW]; (right) r_c

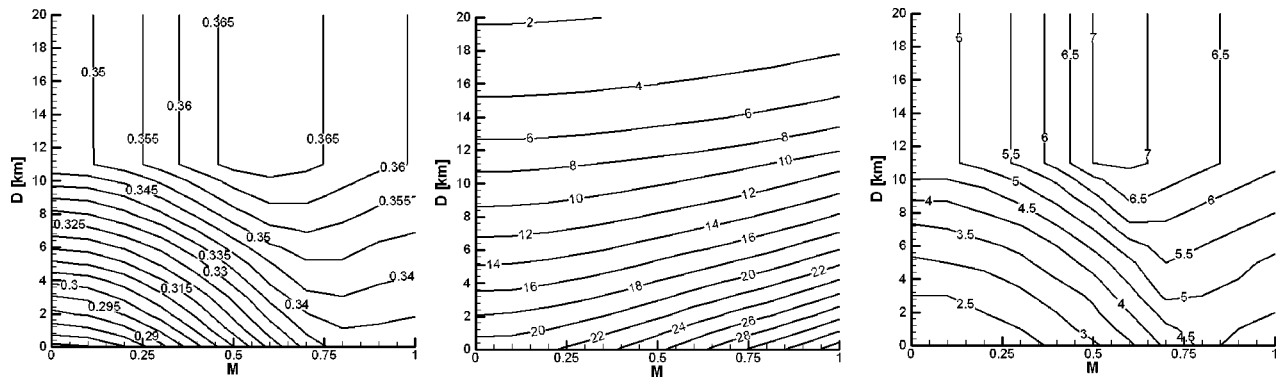


Fig. 12 Maximizing power [kW] by varying r_c with $d_i=4$ in., $d_e=5.7$ in.: (left) η_{th} ; (center) power [kW]; (right) r_c

larger than the compression ratio, offering significant power and efficiency advantages, both on ground and at various flight envelopes. These advantages must be carefully balanced with increases in weight and engine volume. The external combustion and positive-displacement compression of the engine permit a variable-compression ratio cycle. Overall optimization of engine performance would require selection of maximum temperature, then selection of compression ratio, then selection of expansion to compression ratio. However, all three variables can be controlled while the engine is operating. The maximum temperature is controlled by fuel input, and the compression ratio is controlled by the accumulator pressure valve, effectively controlling both compression ratio and expansion to compression volume ratio. The unique efficiency and power advantages of the engine are apparent for on ground, or for small-vehicle airborne applications. Overall these variations of the nutating-disk engine are a significant competitor to both gas turbines and piston engines in the 2–500 kW power range.

Nomenclature

d = disk outside diameter
 D = altitude
 f = heat loss or pressure loss factor
 (F/A) = fuel-to-air mass ratio
 k = volume sweeps (power strokes) per shaft rotation ($k=2$ for volume V_{sw} in the nutating engine)
 L_{hv} = lower heating value of the fuel
 $m, \dot{m}$ = mass, massflow rate
 N_s = revolutions per second
 p = pressure
 r = compression (volume) ratio, $r_{ij} \equiv V_i/V_j$
 T, T' = temperature, temperature ratio

V = volume
 $W, \dot{W}$ = work, power
 δr_c = r_c increment (Fig. 8)
 η = efficiency
 θ = combustion-model factor
 ρ = density

Subscripts

01 to 07 = thermodynamic cycle points (Fig. 6)
 a = air
 bu = burner
 c, e = compression, expansion
 f = fuel
 il = engine inlet
 me = mechanical
 mn, mx = minimum, maximum
 pl, ql = pressure loss, heat loss
 s = isentropic
 sw = swept
 th = thermal
 v = volumetric

References

- [1] Meyer, L., 1993, "Nutating Internal Combustion Engine," U.S. Patent No. 5,521,594.
- [2] Korakianitis, T., Meyer, L., Boruta, M., and McCormick, H. E., 2004, "Introduction and Performance Prediction of a Nutating-Disk Engine," ASME J. Eng. Gas Turbines Power, **126**, pp. 294–299.
- [3] Korakianitis, T., Meyer, L., Boruta, M., and McCormick, H. E., 2001 "One-Disk Nutating-Engine Performance for Unmanned Aerial Vehicles," ASME J. Eng. Gas Turbines Power, **126**, pp. 475–481.
- [4] Shepherd, D. G., 1972, *Aerospace Propulsion*, Elsevier, New York.
- [5] Korakianitis, T., and Wilson, D. G., 1994, "Models for Predicting the Performance of Brayton-Cycle Engines," ASME J. Eng. Gas Turbines Power, **116**, pp. 381–388.

Fabio Bozza
Maria Cristina Cameretti
Raffaele Tuccillo
e-mail: tuccillo@unina.it

Dipartimento di Ingegneria Meccanica per
l'Energetica (D.I.M.E.),
Universita di Napoli "Federico II",
Via Claudio 21,
80125 Napoli, Italy

The Employment of Hydrogenerated Fuels From Natural Gas Reforming: Gas Turbine and Combustion Analysis

An integrated method for power plant analysis, including rotating component matching and CFD simulation of the combustion process, is applied to the study of gas turbines supplied with hydrogenated fuels originating from the natural gas reforming. The method proposed by the authors allows estimation of the power plant performance and emission in the gas turbine operating range. A comparison is then carried out between the plant behavior with conventional fuelling and with decarbonised fuel supply. Attention is also paid to the study of the combustion regimes with either natural gas or fuels with increasing hydrogen contents, in order to achieve a realistic insight of both the temperature distributions and the growth of nitric oxides throughout the combustion chamber. [DOI: 10.1115/1.1691440]

Introduction

Following their recent experiences in the field of low CO₂ emission power plant analysis, [1], the authors deal, in this paper, with gas turbine-based conversion systems with devices to obtain decarbonized fuels with high hydrogen contents from natural gas reforming.

As it is well established by current technical literature, [2–6], the preliminary fuel treatment upstream of the main combustion chamber can lead to different compositions of the decarbonized fuel, depending upon the natural gas to air ratio in the partial-oxidation system and on the amount of steam supplied to the shift reactor. The resulting gaseous mixture can be therefore characterized either by a poor calorific value, in the case of a partial-oxidation oriented system, or by enhanced properties when the effect of the steam reforming is prevailing. In the first case, the hydrogen volume contents do not exceed a 50% level, while in the second one higher hydrogen contents (up to 95–100%) are expected.

The effect of low calorific value fuels obtained from the partial oxidation of natural gas has been examined by the authors in their previous paper, [1], with a fully comprehensive analysis of a medium range, aero-derivative gas turbine equipped with devices for CO₂ emission reduction. The methods included the prediction of the off-design conditions induced by the changes in mass flow rates through the rotating components and by the different combustion characteristics. The latter were studied by means of a CFD tool, i.e., the KIVA3V code, [7,8], which was expressly adapted to the case of steady reacting flows with different features of the fuel supplied.

The purpose of the proposed paper is to present the recent improvements in the authors' methodology for an integrated study of the gas turbine power plant together with its main fluid-dynamic components:

- The main improvement in terms of thermodynamic analysis (which also includes the component matching) consists of the possibility of examining the plant behavior under varied reactant

characteristics in the systems for natural gas partial oxidation and reforming. In other words, the calculations in the previous paper, [1], were simply referring to a given mass flow rate and composition of the decarbonized fuel after the natural gas treatment, while the novel options allow variation of the typical reacting ratios and temperature levels that affect the chemical processes throughout the systems for fuel reforming. The purpose is a correct estimate of the gas flows and compositions through the several rotating and stationary devices when pursuing the trend to a full conversion of natural gas to hydrogen, with the simultaneous CO₂ elimination and absorption. This results in a realistic performance prediction of the power plant with an appropriate evaluation of the energy requirements for natural gas reforming and CO₂ sequestration.

- The combustion process is more properly analyzed for different fuel compositions (starting from the base natural gas and proceeding with decarbonized fuels with increasing hydrogen contents). This involves an accurate evaluation of the reaction mechanisms, typically based on the competition between the turbulent-mixing rates and those related to the combustion kinetics, [9–11], and includes the thermal NO prediction, [12–15], together with the estimate of the carbon monoxides which may result from some carbon residuals after the fuel treatment. The purpose of this step is to examine the adaptability of existing gas turbine combustors to the case of gaseous highly hydrogenated fuels. Really, no particular problem was found in the previous paper when examining a low calorific value decarbonized fuel, but the effect of the increased flame temperature in hydrogen combustion must be more carefully analyzed because of its trend to raise the nitric oxide formation and to require more efficient cooling systems for both the burner and the first turbine blade rows, [16,17].

Basing on the above items, the results in this paper refer to a medium-range industrial gas turbines of the last generation and allow discussion of the effects of the natural gas transformation into a highly hydrogenated fuel in terms of base performance parameters, operating range and detailed energy analysis of the single plant components. The combustor CFD simulation leads to a detailed insight both of the main overall characteristics of the combustion process (i.e., combustion efficiency, development of polluting species, etc.) and of temperature and chemical species distribution throughout the chamber.

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30414. Manuscript received by IGTI, Dec. 2001, final revision, Mar. 2002. Associate Editor: E. Benvenuti.

Table 1 Base features of the PGT25 gas turbine

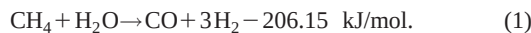
PGT25	
Mech. arrangement	twin-shaft
Pressure ratio	18.0
Turbine inlet temp.	1523 K
Rated mech. output	23,200 kW
Compressor	1 axial flow, 16-stage compressor
Turbines	2 axial flow, 2-stage turbines

Methods for Gas Turbine Plant Analysis

The plant layout innovations are considered for a gas turbine of the PGT25 type (Table 1), manufactured by G.E.–Nuovo Pignone, Italy. To refer to a typical industrial gas turbine, instead of a high-output, heavy-duty one, may be of interest for examining the actual possibilities of a wide diffusion of systems for CO₂ emission reduction. In addition, an appropriate evaluation is performed of the energy requirements for the fossil fuel treatment, and therefore of the energy and exergy fluxes which remain available for an integrated employment of the gas turbine (i.e., for cogeneration or combined cycle applications).

Figure 1 reports the general scheme of the twin-spool gas turbine, whose features are recalled in Table 1, for base rating, natural gas operation.

Figure 2 shows a scheme of the gas turbine plant, integrated with devices for natural gas reforming. The first step of the fuel treatment process is performed in the reformer supplied with a reacting steam-methane mixture. Products of the steam reforming process are basically carbon monoxide and hydrogen, as stated by the endothermic reaction:



Carbon monoxide is then converted to CO₂ in two subsequent reactors which produce the classical gas-steam shift exothermic reaction:



Overall methane conversion efficiency is promoted by high temperature as well as by high steam/methane ratio. Results in the technical literature, [3–6] show that an almost complete methane conversion requires a mean temperature level inside the reformer of approximately 700°C and a steam/methane ratio of about 3. The required steam flow can be produced from the heat released by the first shift reactor and by the reformed fuel cooling. Additional steam, if required, is produced by a conventional heat recovery system. The final composition of the reformed fuel is obtained after both water removal in a condenser, and CO₂ separation in a chemical absorber. Pure CO₂ is then compressed and liquefied for long-term storage; the required mechanical work is however not included in overall plant efficiency. The heat needed for the high-temperature level in the steam reformer can be only partially recovered from the gas turbine exhaust gases. An additional heat is usually provided by a supplementary burner, fed

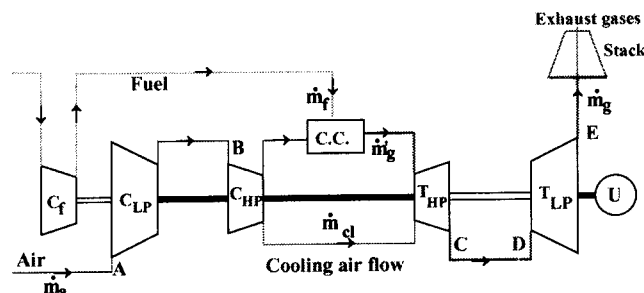


Fig. 1 Base layout of the twin-spool gas turbine

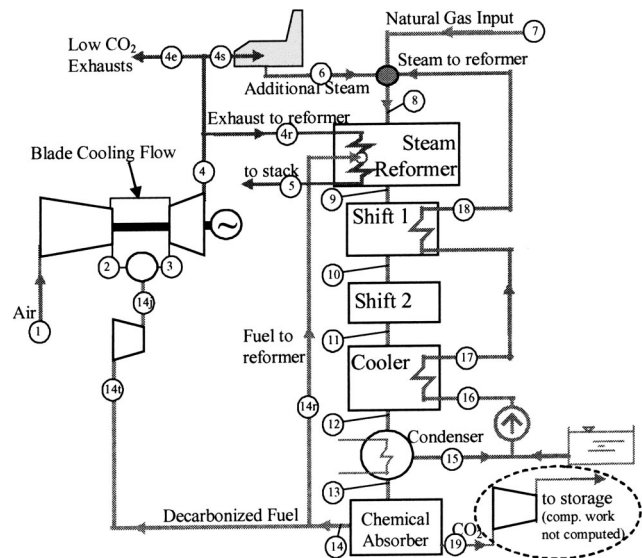


Fig. 2 Schematics of gas turbine integrated with devices for natural gas reforming

with a fraction of the reformed fuel. The remaining fraction is finally compressed and injected inside the gas turbine combustor.

The authors' experience, [1,18–21], allows the plant and component analysis to be carried out through a number of steps of increasing deepening, namely:

Thermal Cycle Analysis. The study of the thermodynamic processes that take place through the several plant components in accordance with the layouts in Figs. 1 and 2 is conducted by an accurate model based on:

- evaluation of the thermochemical equilibrium of the species involved in the several processes, like combustion, mixing between different fluids, cooled expansion, etc. Both composition and thermodynamic properties are calculated by accessing the JANAF tables, [22];
- cooling air flow estimate for the turbine stationary and rotating elements, in accordance with given limits of the blade metal temperature and under the film-cooling assumption. The model for cooled expansion and coolant mixing with the main expanding gas stream, [23], is applied to each stationary or rotating blade row. The cooling air rate, which is bled from either the compressor discharge (for the first turbine stage) or intermediate compressor stages, is determined at the base rating conditions to match a blade temperature of 1150 K.

This leads to a reliable prediction of gas turbine performance, the component characteristics (i.e., mass flow rates, pressure ratios, polytropic efficiencies, etc.) being estimated by the component matching analysis.

Thermo-Kinetic NO Model. The thermal nitric oxides production in the combustion chamber is estimated. The method is based on the hypothesis of partial chemical equilibrium, that is the main reacting species contents are calculated at equilibrium conditions while those involved in the NO formation mechanism result from the solution of the well known system of chemical kinetics equations, [12,18,20,21].

The kinetic equations are solved in the residence time domain by updating both local temperatures and species concentrations through chemical equilibrium calculations. In this way, the method, [18], is sensitive to a number of key parameters, like:

- the overall residence time of the reactants and products through the combustion chamber;

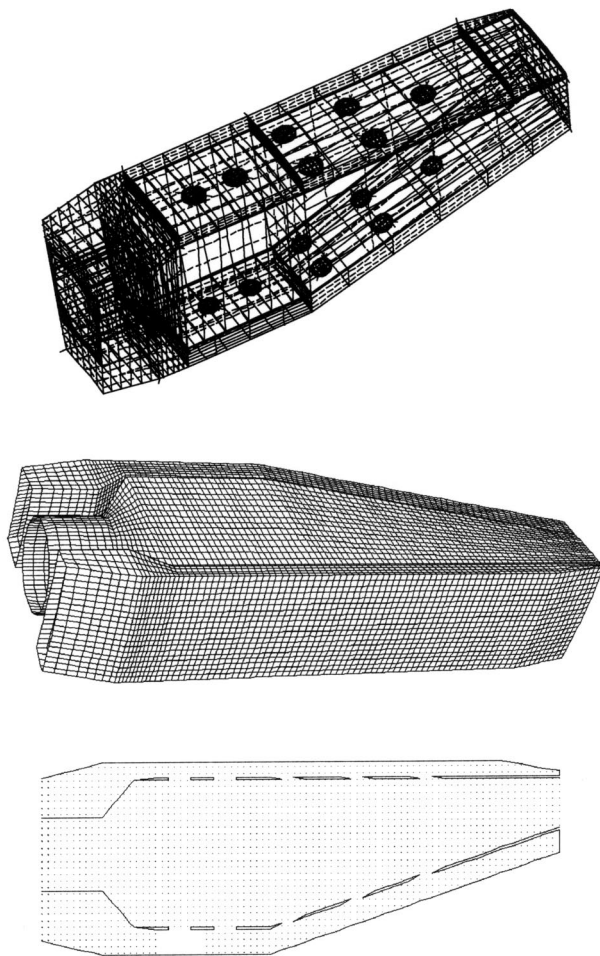


Fig. 3 Computational mesh and schematic view of the three-dimensional sector of the PGT25 combustor

- the time and mass flow rate sharing in the combustor regions (i.e., primary, secondary, and dilution zones);
- the steam concentration in the oxidant or the amount of water or steam directly injected into the primary zone.

The comparison with experiments has demonstrated that the model leads to an effective prediction of actual emission data, provided that a good calibration of the above parameters is performed, [21]. To this purpose a CFD based analysis, as the one discussed in the following, may represent a reliable identification tool.

Component Matching Analysis. This approach is of particular interest when dealing with off design conditions induced by either external load variations or changes in the plant configuration, like those under examination, so allowing a more realistic guess of the actual benefits which result from any solution proposed.

In addition, this step provides more accurate inputs to the detailed fluid dynamic analysis of a single component, like that described at the following step.

A typical matching procedure among the rotating and stationary components allows detection of the degrees-of-freedom of the system and a sequential estimation of the gas generator (GG) operating domain and of the LP turbine running lines, in the case of the PGT25 twin-shaft arrangement, [1,20]. The thermal cycle parameters directly result, therefore, from the component matching conditions.

It is worth noting to state that the off-design computation proceeds with a direct access to the compressor map (Fig. 1), while both HP and LP turbine calculations are carried out through a mean-line analysis. Despite the coarse accuracy of this method, compared with more sophisticated CFD based methods, the main characteristics of the operating gases are properly taken into account. The turbine performance estimation includes therefore the variations in gas properties which affect the expansion pattern, the Reynolds number, and the cascade and stage losses. Regarding to the blade cooling process, the same cooling air to total air ratio is assumed as at the base rating, and the blade temperature of each blade row is calculated.

CFD Analysis of the Combustion Chamber. The simulation of the reacting flows employs the widely known KIVA3V code, [7,8], which has been expressly adapted by the authors to this purpose, [1,20]. A full combustor simulation is developed, including the flow through the liner, for a better evaluation of both the air flow rate splitting between the several combustion regions and the flow conditions through secondary and dilution holes, [24]. This approach overcomes, therefore, the limits of the thermokinetic model, since residence times and local reactants temperature and composition result directly from calculation. In addition, synthetic data may be derived, to be transferred to simplified models for the emission prediction.

The chamber being of the annular type, a 12 deg three-dimensional sector was considered, due to the periodical arrangement of both 30 inlet air swirlers and 30 fuel injectors.

The intake flow surface is assumed to be of the continuous inflow type so requiring the assignment of the air entry velocity. The outlet section is a continuous outflow boundary and the related static pressure is assigned.

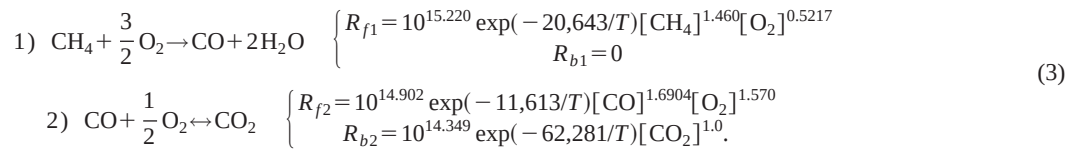
The computational mesh was entirely redesigned, compared with that in previous applications, [1,20], and it includes four secondary and ten dilution ports. Schematic views of the combustor and linear arrangement and of the port location are given in Fig. 3, together with some details of the 60,000 vertex grid. This geometry effectively approaches the actual shape of similar combustors, [25].

Both boundary and initial conditions are assigned in accordance with the gas turbine simulation results. In particular:

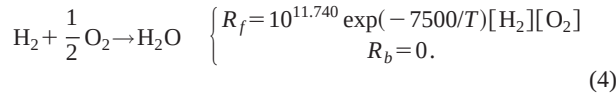
- the total pressure and temperature for combustor intake flows result from the component matching analysis;
- static pressure at combustor exit is determined by evaluating the outlet velocity level and a total pressure loss (3% of the total inlet pressure) through the combustion chamber;
- the inlet axial velocity is calculated to match the actual amount of air flow rate entering the combustor (i.e., the total air mass flow rate diminished by the cooling flows), which is automatically split into the primary air and the secondary and dilution air flows through the liner;
- the existing swirlers are replaced by a swirled intake air flow into the primary zone, of the constant flow angle type (a 60 deg value was assumed);
- regarding the fuel-injection system, a five-hole nozzle was hypothesized (with a peripheral arrangement around the nozzle center) and a full cone shape with a 30 deg angle was assumed for each fuel jet. The nozzle flow areas are scaled to ensure the same jet velocity for both natural and decarbonized gas at base rating conditions.

The code performs the solution of the unsteady Navier-Stokes system of equations with an arbitrary Lagrangian-Eulerian method, [7]. Viscous flow effects and the fuel-air mixing and turbulent chemistry rates as well, are approached with a classical two-equation (κ - ϵ) model.

Regarding to the chemical reaction rates, for the natural gas combustion, a global two-step methane oxidation mechanism was introduced, [10,11]:



Regarding to the hydrogen oxidation in the decarbonized fuel, a single step reaction was assumed, of the type



In terms of kinetically controlled mechanism, the generic kinetic reaction 'r' involving NSP species proceeds at a rate given by

$$\omega_{k,r} = R_{fr} - R_{br} = k_{fr} \prod_{m=1}^{\text{NSP}} (C_m)^{a_{mr}} - k_{br} \prod_{m=1}^{\text{NSP}} (C_m)^{b_{mr}} \quad (5)$$

where $k = C_e T^x e^{-(E_a/T)}$, E_a being the activation energy, and a_{mr} , b_{mr} are integral stoichiometric coefficients. The forward and backward reaction rates, R_f and R_b , are those in Eqs. (3) and (4).

The turbulent mixing-controlled rate, $\omega_{t,r}$, is obtained by the Magnussen and Hjertager model, [11], namely

$$\omega_{t,r} = C^* \varepsilon / \kappa^* \min(x_f, x_{ox}) \quad (6)$$

where x_f and x_{ox} are the mass fractions of fuel and oxygen, κ and ε the turbulent kinetic energy and its dissipation rate, C is the proportional factor in the mixing-controlled turbulent combustion varying with both flow conditions and fuel composition.

The actual reaction rate is then calculated as

$$\omega_r = \min(\omega_{k,r}, \omega_{t,r}). \quad (7)$$

The set of chemical kinetics equation is completed by those of the extended Zel'dovich mechanism, [12] for the prediction of thermal nitric monoxide formation. The equations are simultaneously solved in each computational cell by an algorithm that determines the formation rate of the species involved in the process, [1,18].

Computational Results

This computational section is addressed to the verification of the actual possibilities of an efficient employment of the systems for natural gas reforming, as above described, within a gas turbine based industrial power plant. Besides the base gas turbine performance, it is of interest to evaluate which margins are left for an efficient energy recovery process after the additional energy requirements for the fuel treatment and the CO_2 absorption. A special issue is to evaluate whether or not the energy conversion system, including the devices for natural gas reforming, is self-sustainable in each operating condition.

To this purpose the plant layout in Fig. 2 includes several options for both steam and heat supply to the reforming process, namely:

- a steam fraction may originate from the separation of condensed water from the gas stream in station (12). Additional steam requirements may be fulfilled by a heat recovery generator supplied with the exhaust gas fraction (4a).
- similarly, the additional heat which is needed to comply with the temperature level established for the reformer can result from the exhaust fraction (4r) and by decarbonised fuel fraction (14r). The latter amount is, of course, dependent on the exhaust and reformer temperatures.

It is self-evident that the steam and heat sharing between the above different sources will vary with the actual gas turbine load conditions and this implies a correct calculation of the component matching all over the operating range.

The computational network does not include either a bottoming steam power plant or a heat recovery device, and therefore a good criterion of the overall plant performance may be the plant efficiency and a second law efficiency, defined as follows:

$$\eta = \frac{P}{\dot{m}_{\text{NG}} \text{LHV}_{\text{NG}}}; \quad \eta_{\text{II}} = \frac{P + \dot{m}_{g4e} \xi_{g4e}}{\dot{m}_{\text{NG}} \xi_{\text{NG}}}. \quad (8)$$

The above definitions allow estimation of the total mechanical output which is actually and potentially obtainable from the whole process and automatically accounts for all exergy destructions which take place through the single units. In addition, an estimate of the exergy flows which are left available by the energy conversion system represents a reliable estimate of the additional resources for complying with the mechanical energy needed for CO_2 storage.

Component Matching and Cycle Analysis. This analysis aims to compare the gas turbine plant behavior with either a direct natural gas fueling or the employment of the decarbonised fuel obtained by the reforming process with a steam/methane ratio of 3.1. Two different temperature levels in the reformer were considered of 500°C and 700°C, respectively. The related results are referred to as DF500C and DF700C. Although the latter temperature level ensures a more efficient fuel decarbonization, the DF500C process was analyzed in order to consider a compromise between the trading off needs of CO_2 and NO_x reduction. Table 2 reports the fuel compositions after natural gas reforming at different temperature levels and evidences the CH_4 and H_2 contents.

Besides the pollutant species estimation, the matching analysis aims to check the adaptability of the components to different conditions induced by the new fuel supply. The DF 500C operation could represent an acceptable compromise also by this point of view, since less significant variations, with respect to the DF700C case, can be expected for the individual components and for the whole energy conversion plant.

The PGT25 being of the aero-derivative type, a better fitting to new arrangements and fuelling, respect to single-spool gas turbines, can be foreseen because of the variable-speed gas generator.

Figures 4 and 5 display the final results of the whole matching procedure which is briefly recalled hereafter, further details being given in [1,19,20]. The steady operating map of the axial flow compressor (Fig. 4) is employed to derive the gas generator (GG) operating domain in a (natural gas flow rate—GG speed) space. Two physical parameters can in fact represent the degrees-of-freedom of the GG group, once the mechanical equilibrium condition between the compressor and the HP turbine has been imposed.

The GG and LP turbine matching is then determined by the additional closure condition on the discharge pressure that must match the ambient pressure increased by the pressure losses through the exhaust gas path. The 2% value, assumed at base rating, was scaled in accordance with the actual gas mass flow rate at off-design operation. Additional losses were considered in the case of the heat recovery process activation.

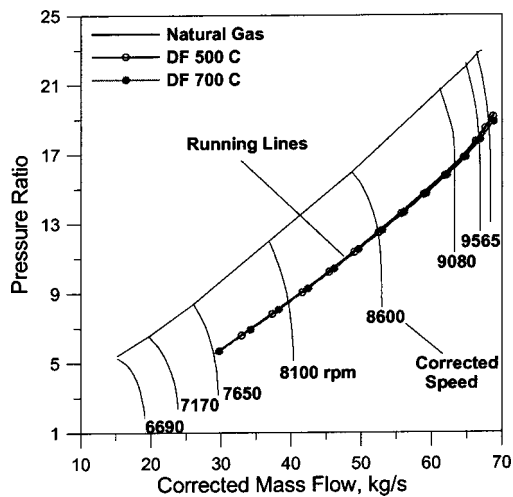
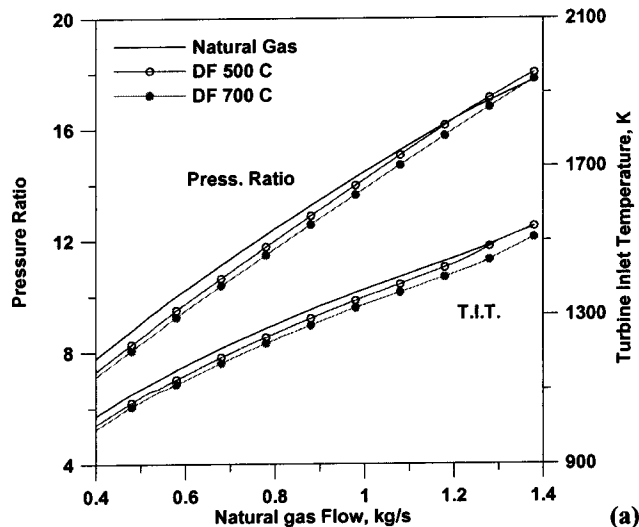


Fig. 4 Axial compressor map for the PGT 25 GT type (mass flow and speed correct by standard ISO conditions)

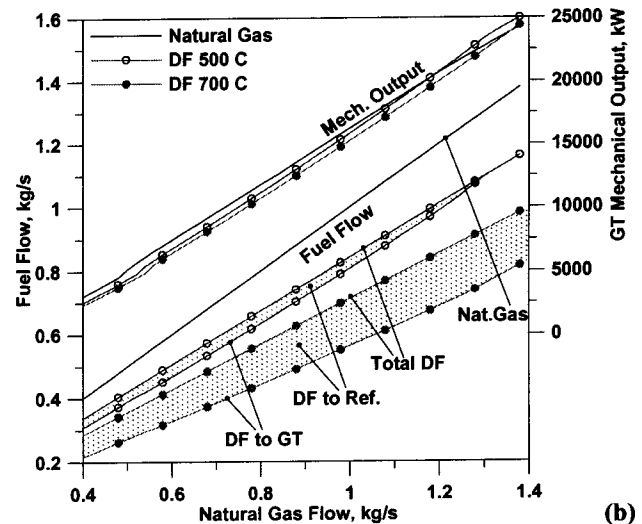
The latter step leads to draw the gas turbine *running lines* which are superimposed to the compressor curves in Fig. 4 and are plotted versus the natural gas primary input in the following Fig. 5. The related diagrams summarize therefore the gas turbine power plant behavior and performance for an operating speed of the LP turbine of 6500 rpm.

As a first consideration, in all cases examined, the turbine is able to operate in a rather wide range and the cycle main parameters, i.e., the pressure ratio and the firing temperature (Fig. 5(a)) follow similar trends both in natural gas untreated case and after the reforming process. The fairly higher pressure ratios for both DF 500C and DF 700C cases are consistent with some increase in the GG rotational speed, which probably results from the variations induced in the gas properties through the HP turbine when burning the decarbonized fuels. The decarbonized fuel compositions remain the same as in Table 2 in the whole operating field. This substantially means that the appropriate temperature levels and mass and energy supplies can be expected for the reformer and shift reactors also under off-design conditions.

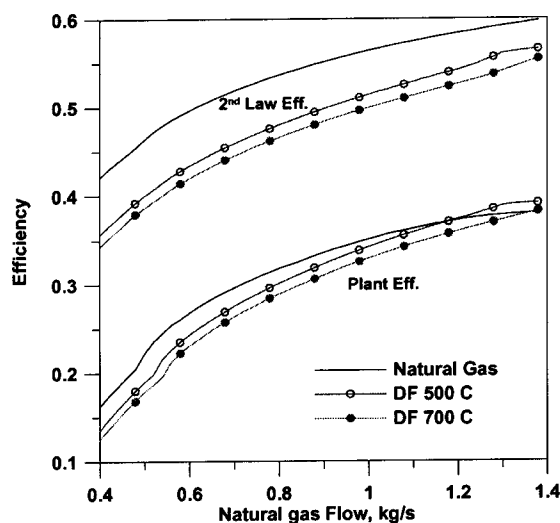
The matching analysis of the gas turbine components and of the several devices for fuel treatment provides both the actual amounts of decarbonized fuel which are supplied to the main combustor at each load level and the resulting mechanical output (Fig. 5(b)). The latter preserves similar patterns in all cases but the



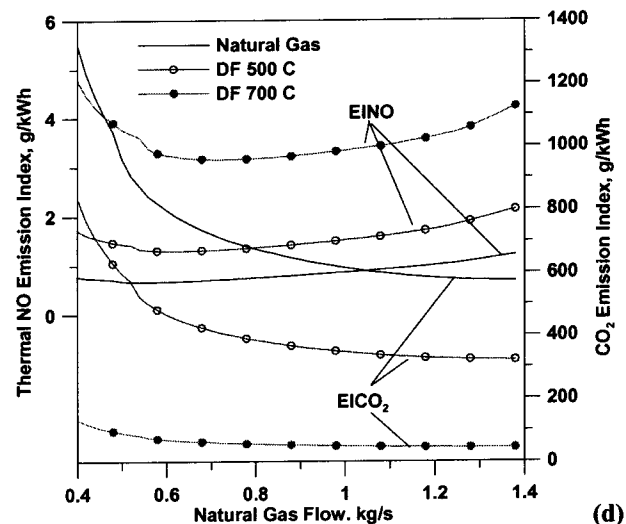
(a)



(b)



(c)



(d)

Fig. 5 Cycle parameters and performance and emission data along the gas turbine operating lines

Table 2 Natural gas and decarbonized fuel composition and properties (steam/methane ratio=3.1, 1.15 bar)

Fuel Composition (%, molar)	Natural Gas	DF 500°C	DF 600°C	DF 700°C
CH ₄	91.18	24.98	7.82	0.96
C ₂ H ₆	4.41	—	—	—
C ₃ H ₈	0.10	—	—	—
N ₂	4.31	1.84	1.29	1.08
H ₂	—	69.97	87.96	94.98
CO	—	0.087	0.275	0.539
CO ₂	—	0.386	0.443	0.473
H ₂ O	—	2.75	2.19	1.97
Mol. Mass, g/mol	17.21	6.62	4.06	3.08
LHV, kJ/kg	46,293	55,877	68,021	77,460
h _{0f} , kJ/kg	−4196.0	−3929.2	−3016.3	−1305.0
f _{st}	0.0631	0.0567	0.0495	0.0452
T _{0f} , K	2218	2271	2318	2350

DF operations present a steeper reduction at low fuel rates. Indeed, the curves of second-law and plant efficiency in Fig. 5(c), clearly indicate that a strong decrease in the effectiveness of the whole process, including the natural gas reforming, takes place at low-load conditions. This may be explained by the reduced exhaust temperature and, consequently, by a larger amount of DF which must supply the reformer (shaded regions in Fig. 5(b)), instead of the gas turbine combustor, for ensuring the prescribed temperature level. The CO₂ emission indices (Fig. 5(d)) are affected by the efficiency trends but appear to be more insensitive in the DF cases, as expected. The NO index is also dependent on the combustion temperature levels and the combined dependence on plant efficiency justifies the local minima within the gas turbine operating range.

Detailed results in Table 3 better highlight the differences be-

tween the natural gas and DF operations and those related to part-load conditions. As a general comment, it can be observed that the results at base rating confirm those in the recent literature, [3,5]. In particular, a slight increase in gas turbine output and efficiency is induced by the reforming process, due to the enhanced quality of both the fuel and the expanding gases. The DF500 C case seems to represent an effective compromise between the conflicting needs of performance preservation and CO₂ abatement, while the thermal NO raise itself is less dramatic than in the DF700 C case. Looking at the second-law efficiencies, these progressively decrease when a more refined decarbonization is carried out. The higher temperature required for the reforming process involves larger amounts of fuel to be supplied to the reformer, and an additional exergy destruction occurs. This situation is emphasized at low-load operation because of the decreased exhaust temperature, as above stated.

Table 4 better give details of the mass and energy rates through the several devices involved in the fuel treatment process, so confirming the above statement. In particular, there is evidence of the main differences among the DF500 C and the DF700 C solutions: The first one does not require, at base rating, additional fuel to the reformer while the second one is able to produce the whole steam amount required with the heat released by the fuel treatment process.

Finally, all examined DF cases in their whole operating range present the important feature of self-sustainability by both thermal and mechanical energy point of views, say, the reforming process is entirely carried out by internal heat recovery procedures. The CO₂ segregation itself appears to be viable by exploiting the mechanical energy contents in the exhaust.

CFD Analysis of the PGT25 Combustor. The results of the cycle analysis indicate encouraging possibilities of adapting the

Table 3 Results of the matching analysis of the PGT25 gas turbine

GAS TURBINE OPERATION	$\dot{m}_g$, kg/s	$\dot{m}_{NG}$, kg/s	$\dot{m}_{DF}$ to: GT/reform. kg/s	Comb. Inlet T, K	HP Turbine inlet $\dot{m}_g$, T, p kg/s, K, bar	GG Speed, r.p.m.	LP Turbine outlet T, K	Power P, kW	Efficiency η_{NG} (η_{II})	EICO ₂ , g/kWh	EINO, g/kWh	[NO] p.p.m. dry
Base NG	66.2	1.38	—/—	703.6	60.8,1537,17.1	9381	807	24,274	0.380 (0.598)	573	1.217	124
DF 500 C	66.8	1.38	1.162/—	709.0	61.1,1537,17.3	9469	803	24,987	0.391 (0.565)	320	2.129	222
DF 700 C	66.9	1.38	0.816/0.166	705.7	60.8,1508,17.2	9469	786	24,396	0.382 (0.554)	43.8	4.235	434
DF 700 C (Part-load)	46.9	0.70	0.384/0.114	598.3	42.4,1182,10.4	8430	695	8536	0.263 (0.445)	60.4	3.152	158

Table 4 Thermodynamic and flow conditions at reference stations in Fig. 2 (base rating)

pt	Description	Pressure, bar			Temperature, K			Enthalpy, kJ/kg			Mass flow, kg/s		
		NG	DF500	DF700	NG	DF500	DF700	NG	DF500	DF700	NG	DF500	DF700
1	Air	1.01	1.01	1.01	288	288	288	−64.1	−64.1	−64.1	66.2	66.8	66.9
2	Air	17.8	18.1	17.9	704	709	706	365.4	371.2	367.7	59.4	59.9	60.4
3	Products	17.1	17.3	17.1	1537	1537	1508	263.1	290.5	334.1	60.8	61.1	60.8
4	Products	1.03	1.03	1.03	807.4	803.0	786.6	−528.4	−520.9	−474.8	67.6	68.0	67.7
4r	Exh. to reform.	—	1.03	1.03	—	803.0	786.6	—	−520.9	−474.8	—	7.60	6.06
4s	Exh. to HRSG	—	1.03	1.03	—	803.0	786.6	—	−520.9	−474.8	—	2.61	0.00
4e	Exhaust gas	1.03	1.03	1.03	807.4	803.0	786.6	−528.4	−520.9	−474.8	67.6	57.8	61.6
6	HRSG steam	—	1.15	1.15	—	429	429	—	−13,176	−13,176	—	1.87	0.00
7	NG input	15	15	15	315	315	315	−4154.0	−4154.0	−4154.0	1.38	1.38	1.38
8	NG+steam	—	1.15	1.15	—	402	402	—	−10,972	−10,972	—	5.65	5.65
9	Reformed gas	—	1.14	1.14	—	773	973	—	−9040.1	−6774.4	—	5.65	5.65
10	Reformed gas	—	1.14	1.14	—	460	460	—	−9875.1	−8285.6	—	5.65	5.65
11	Reformed gas	—	1.13	1.13	—	495	495	—	−9875.1	−8285.6	—	5.65	5.65
12	Reformed gas	—	1.12	1.12	—	308	308	—	−10,374	−8836.4	—	5.65	5.65
14	Reformed gas	—	1.12	1.12	—	308	308	—	−10,374	−8836.4	—	1.162	0.982
14r	Fuel to reformer	—	1.12	1.12	—	308	308	—	−10,374	−8836.4	—	0.00	0.165
14j	Fuel to GT	21.5	21.5	21.5	347.5	354.6	357.6	−4082.6	−3793.9	−2009.5	1.38	1.162	0.817
18	Steam to ref.	—	1.15	1.15	—	429	429	—	−13,176	−13,176	—	2.40	4.27

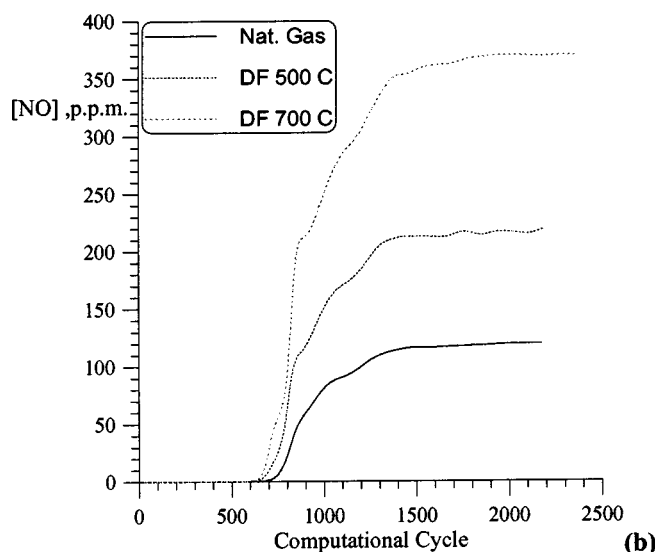
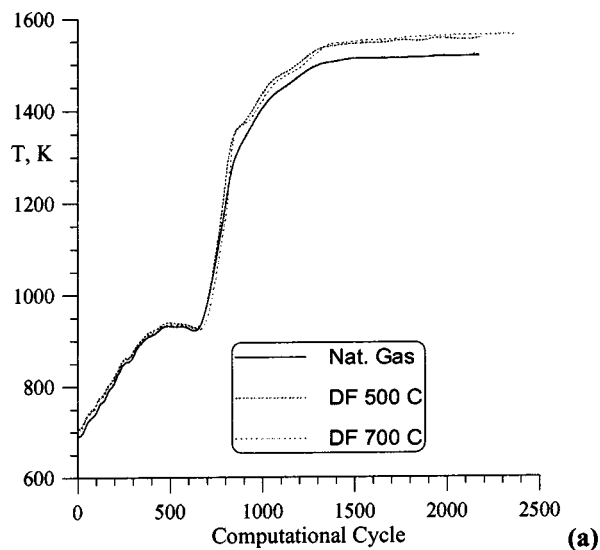


Fig. 6 Convergence histories of the CFD combustion analyses

gas turbine operation to the new conditions introduced by the natural gas reforming. An important issue concerns the verification of the combustion process with a more detailed tool, like the one based on a CFD analysis. The results in this section must be therefore intended as an attempt of the authors to get a substantial confirmation of the results based on a simply thermodynamic approach, by analysing the flow and properties distribution analysis throughout the combustion chamber.

Several limits of this analysis must however be pointed out, in order to outline the future developments of the investigations in this field:

- The combustion model is based on a mixed kinetic-turbulent mechanism but it probably needs more accurate information about the autoignition and flammability thresholds related to the different fuels adopted, [27].
- The foregoing results refer to a non-updated combustion chamber, with a primary fuel-air equivalence ratio close to unity and with a considerable supply of secondary and dilution air rates. A more efficient emission control is nowadays attained by lean-premixed combustors, [14,15], and therefore the analysis should be extended to further chamber configurations.

Despite of the above limits, this section can provide useful

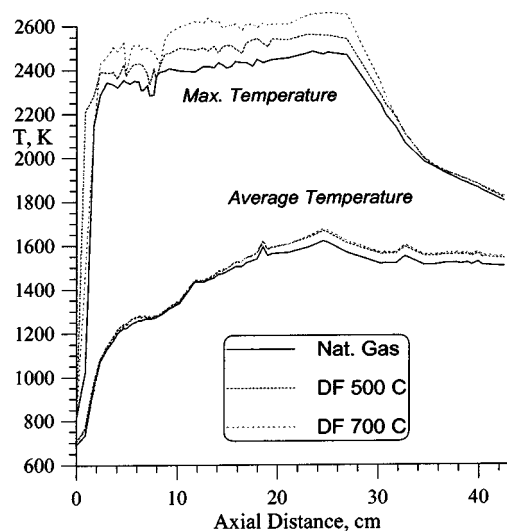


Fig. 7 Streamwise distribution of average and peak temperature in the combustor

indications about the changes in temperature levels and thermal NO production related to the hydrogenated fuel introduction. The examples refer both to the natural gas and to decarbonized fuel cases, and the related inlet and boundary conditions are those derived from the results in Table 3.

The spatial distribution of thermochemical and flow parameters refers to the last computational cycle in Fig. 6. Figure 7 reports the average temperature distributions along the combustor axial path. The trends are nearly similar, except for a fairly faster increase in the DF cases, which can be due to the increased air inlet temperatures, rather than to different combustion kinetics. The peak temperature pattern appears to correctly follow the primary rise, the combustion completion sustained by the secondary air supply and the flame extinguishments due to the dilution air entries. It is also of interest that the maximum levels agree with the adiabatic flame temperatures in Table 2, modified by the effect of both the actual combustor entry temperatures and the real combustion process. This figure gives reason for the increase in thermal NO production which occurs in the DF cases, because the related formation mechanisms is governed by an exponential dependence on the local temperature peaks, [12].

More impressive images are those presenting the multidimensional distributions in Figs. 8 to 12. Figure 8 offers a comprehensive sketch of the flow velocity field in the primary and secondary regions. The secondary and dilution air entries can be detected together with the primary vortex and those induced in the wake region.

An effective comparison is allowed by the temperature distributions in the mid-plane section (Fig. 9) which clearly show the increasing extent of the peak temperature regions when proceeding from the natural gas to the fully decarbonized fuel.

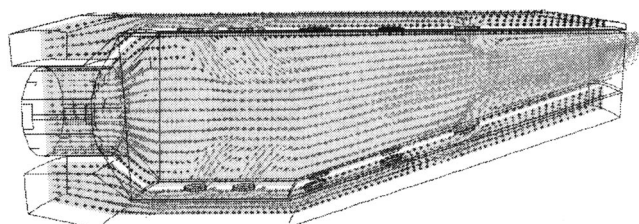


Fig. 8 Flow field in the combustor midplane

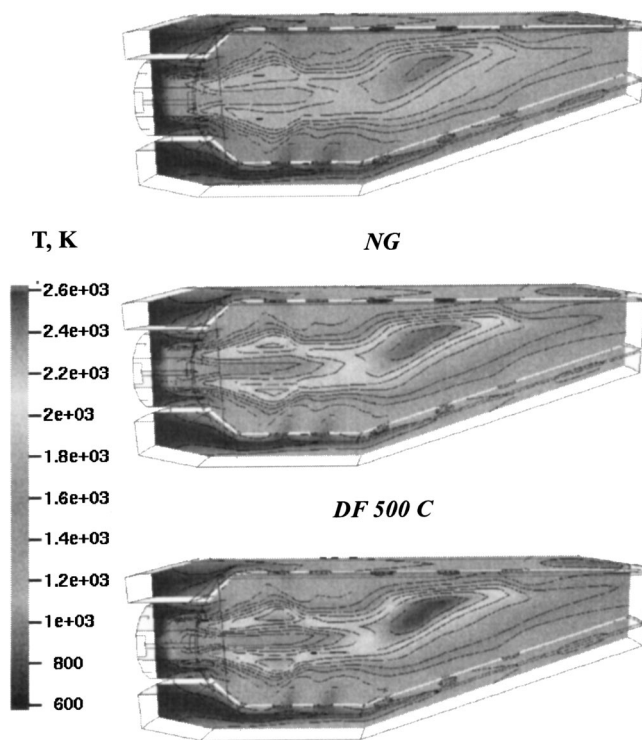


Fig. 9 Temperature distribution in the combustor midplane

The NO distributions in the same section (Fig. 10) are consistent with the temperature behavior. It can be noted that in the DF700C case the activation of the thermal NO mechanism is clearly anticipated respect to the corresponding one in the natural gas case, and this leads to a remarkably extended region with high NO contents. It should be reminded again that the average NO concentration in Fig. 6(b) and temperature development on Fig. 7 summarize the spatial distributions in Figs. 9 and 10, which better display the actual extent of regions with temperatures close to the peak level.

Finally, distributions in the combustor exit plane give a detailed insight of the local conditions at the turbine nozzle inlet. A re-

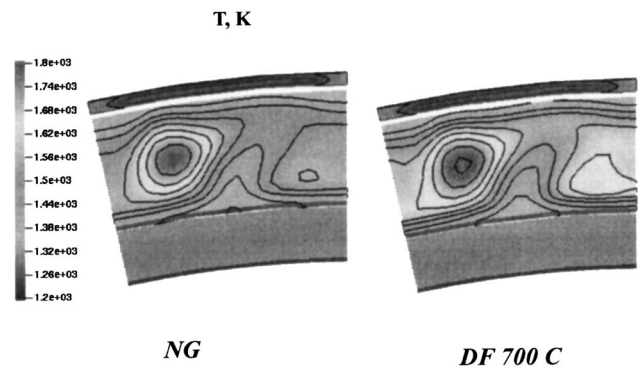


Fig. 11 Temperature distribution in the combustor exit plane

sidual high temperature core is detected in Fig. 11, which could be eliminated by a more efficient combustion development produced, as said by a lean-premixed process. The latter would also produce a NO smoother distribution, respect to the ones in Fig. 12, which in the DF700 case presents unacceptable peaks and average levels.

Conclusion

The results in this paper suggest the possibility of extending to medium-range industrial gas turbines the widely accepted concepts for the reduction of CO₂ emission from energy conversion power plants.

The authors' methodology for the analysis of the plant including the systems for natural gas reforming has allowed a comprehensive estimation of the gas turbine behavior all over its operating range when supplied with fuels with increasing hydrogen contents. The energy and exergy based performance evaluation demonstrates that attractive perspectives exist for an efficient operation of the power plant with a strongly reduced CO₂ production.

The residual issue to be dealt with concern the increased emission of nitric oxides related to the modified combustion regime. This problem has been clearly evidenced in this paper by both the thermodynamic analysis and the CFD approach. The latter needs future enhancements to comply with the requirements of more predictive results in terms of autoignition conditions and actual

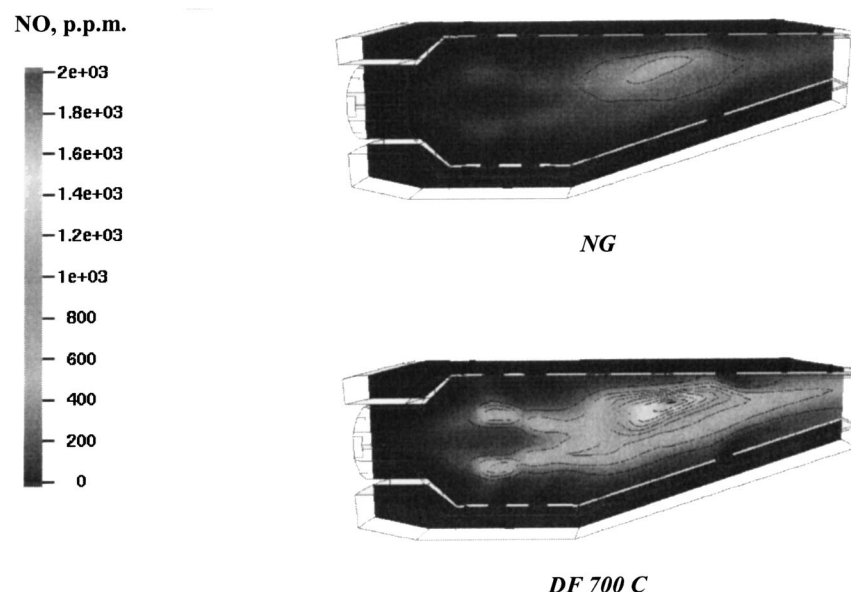


Fig. 10 Thermal NO distribution in the combustor midplane

NO, p.p.m.

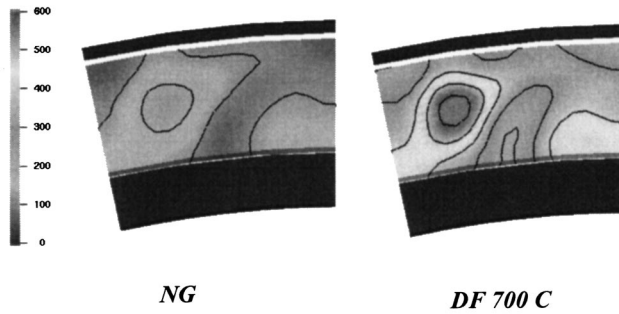


Fig. 12 Thermal NO distribution in the combustor exit plane

combustion mechanisms. The improvement of the related sub-models will provide an effective tool for analysing the most efficient systems for a definite emission abatement.

Acknowledgments

The paper refers to an Italian National Research Project (PRIN 1999) on the reduction of CO₂ emission reduction from electrical generation power plants.

To Dr. Riccardo Stolica, for contributing to computational mesh generation and CFD data post-processing.

Nomenclature

DF	=	decarbonized fuel
EICO ₂ , EINO	=	CO ₂ , NO emission indices
f_{st}	=	fuel/air stoichiometric ratio
h_{of}	=	enthalpy of formation
k_f, k_b	=	reaction rate constants
LHV	=	lower heating value
$\dot{m}$	=	mass flow rate
P	=	mechanical power
p	=	pressure
p.p.m.	=	concentration in parts per million
R_f, R_b	=	reaction rates
t	=	time
t_{res}	=	residence time
$T, T.I.T.$	=	temperature, turbine inlet temperature
x	=	mass fraction

Subscripts

a	=	air
f	=	fuel
k	=	kinetic
g	=	exhaust gas
ox	=	oxidant
DF	=	decarbonized fuel
NG	=	natural gas
t	=	turbulent

Greek

ε	=	turbulent dissipation rate
η	=	plant thermal efficiency
η_{II}	=	second-law efficiency
κ	=	turbulent kinetic energy
ξ	=	exergy
φ	=	equivalence ratio
ω	=	net reaction rate

References

- [1] Bozza, F., Cameretti, M. C., and Tuccillo, R., 2001, "Performance Prediction

- and Combustion Modelling of Low-CO₂ Emission Gas Turbines," ASME Paper No. 2001-GT-0066.
- [2] Lozza, G., and Chiesa, P., 2000, "Natural Gas Decarbonization to Reduce CO₂ Emission From Combined Cycles: Part A—Partial Oxidation," ASME Paper No. 2000-GT-0163.
- [3] Lozza, G., and Chiesa, P., 2000, "Natural Gas Decarbonization to Reduce CO₂ Emission From Combined Cycles: Part B—Steam-Methane Reforming," ASME Paper No. 2000-GT-0164.
- [4] Lozza, G., and Chiesa, P., 2001, "Low CO₂ Emission Combined Cycles With Natural Gas Reforming, Including NO_x Suppression," ASME Paper No. 2001-GT-00561.
- [5] Moritsuka, H., 2001, "CO₂ Capture Using a Hydrogen Decomposed From Natural Gas Turbine," ASME Paper No. 2001-GT-0093.
- [6] Andersen T., Kvamsdal, H. M., and Bolland, O., 2000, "Gas Turbine Combined Cycle With CO₂ Capture Using Auto-Thermal Reforming of Natural Gas," ASME Paper No. 2000-GT-0162.
- [7] Hirt, C. W., Amsden, A. A., and Cook, J. L., 1974, "An Arbitrary Lagrangian-Eulerian Computing Method for All Flow Speed," J. Comput. Phys., **14**, pp. 227–253.
- [8] Amsden, A. A., 1997, "KIVA-III v: Block Structure KIVA Program Engine With Vertical or Canted Valves," LA–Los Angeles 13313-MS, Los Alamos.
- [9] Nicol, G. D., Malte, P. C., Hamer, A. J., Roby, R. J., and Steele, R. C., 1998, "A Five-Step Global Methane Oxidation—NO Formation Mechanism for Lean Premixed Gas Turbine Combustion," ASME Paper No. 98-GT-185.
- [10] Miller, J. A., and Bowman, C. T., 1989, Prog. Energy Combust. Sci., **15**, p. 287.
- [11] Magnussen, B. F., and Hjertager, B. H., 1977, "On Mathematical Modeling of Turbulent Combustion With Special Emphasis on Soot Formation," 16th Symposium on Combustion, The Combustion Institute, Pittsburgh, PA.
- [12] Zel'dovich, Y. B., Sadochnikov, P. Y., and Frank-Kamenetskii, D. A., 1947, *Oxidation of Nitrogen in Combustion*, Academy of Science of SR, Institute of Chemical Physics, Moscow, Russia.
- [13] Talpallikar, M. V., Smith, C. E., Lai, M. C., and Holdeman, J. D., 1992, "CFD Analysis of Jet Mixing in Low NO_x Flametube Combustors," ASME J. Eng. Gas Turbines Power, **114**, pp. 416–424.
- [14] Eggels, R. L. G. M., 2001, "Modelling of NO_x Formation of a Premixed DLE Gas Turbine Combustor," ASME Paper No. 2001-GT-0069.
- [15] Eggels, R. L. G. M., and Brown, C. T., 2001, "Comparison of Numerical and Experimental Results of a Premixed DLE Gas Turbine Combustor," ASME Paper No. 2001-GT-0065.
- [16] Malecki, R. E. et al., 2001, "Application of an Advanced CFD-Based Analysis System to the PW6000 Combustor to Optimize Exit Temperature Distribution. Part A," ASME Paper No. 2001-GT-0062.
- [17] Snyder, T. S. et al., 2001, "Application of an Advanced CFD-Based Analysis System to the PW6000 Combustor to Optimize Exit Temperature Distribution. Part B," ASME Paper No. 2001-GT-0063.
- [18] Bozza, F., Fontana, G., and Tuccillo, R., 1994, "Performance and Emission Levels in Gas Turbine Power Plants," ASME J. Eng. Gas Turbines Power, **116**, pp. 53–62.
- [19] Bozza, F., and Tuccillo, R., 1996, "A Methodology for Establishing Optimal Part-Load Operation of Industrial Gas Turbines," ASME Paper No. 96-GT-333.
- [20] Bozza, F., Cameretti, M. C., Marro, A., and Tuccillo, R., 2000, "Performance and Emission Analysis of a Variable Load Operated Gas Turbine," *Advanced Energy Systems*, **40**, pp. 400–415.
- [21] Bozza F., Rizzo G., and Tuccillo, R., 1998, "Identification of a Model for Monitoring Nitric Oxide Emissions in a Gas Turbine Power Plant," *Theory and Practice of Control and Systems*, World Scientific Publishing Ltd., London UK, pp. 437–442.
- [22] Mc Bride, B. J., and Gordon, S., 1994, "Computer Program for Calculation of Complex Equilibrium Composition and Applications," NASA RP 1311, Parts I and II.
- [23] El Masri, M. A., 1986, "On Thermodynamics of Gas-Turbine Cycles: Part 2—A Model for Expansion in Cooled Turbines," ASME J. Eng. Gas Turbines Power, **108**, pp. 151–159.
- [24] McGuirk, J. J., and Spencer, A., 2000, "Coupled and Uncoupled CFD Prediction of the Characteristics of Jets From Combustor Air Admission Ports," ASME Paper No. 2000-GT-0125.
- [25] Price, G. R., Botros, K. K., and Goldin, G. M., 2000, "CFD Predictions and Field Measurements From LM1600 Gas Turbine During Part Load Operation," ASME Paper No. 2000-GT-350.
- [26] Micklow, G. J., Roychoudhury, S., Nguyen, H. L., and Cline, M. C., 1993, "Emissions Reduction by Varying the Swirler Airflow Split in Advanced Gas Turbine Combustors," ASME J. Eng. Gas Turbines Power, **115**, pp. 563–569.
- [27] Roy, C. J., Moran, A. J., and Thomas, G. O., 2001, "Autoignition Characteristics of Gaseous Fuels at Representative Gas Turbine Conditions," ASME Paper No. 2001-GT-0051.
- [28] Tolpadi, Anil K., Prakash, C., Hura, H., and Mongia, H. C., 1998, "Advanced Combustion Code: Overall Description Prediction of a Jet Diffusion Flame and Combustor Flowfields," ASME Paper No. 98-GT-229.
- [29] Crocker, D. S., Nickolaus, D., and Smith, C. E., 1998, "CFD Modeling of a Gas Turbine Combustor From Compressor Exit to Turbine Inlet," ASME Paper No. 98-GT-184.
- [30] Sivaramakrishna, G. et al., 2001, "CFD Modeling of the Aero Gas Turbine Combustor," ASME Paper No. 2001-GT-0063.

Analytical Blade Row Cooling Model for Innovative Gas Turbine Cycle Evaluations Supported by Semi-Empirical Air-Cooled Blade Data

Leonardo Torbidoni

e-mail: torbidoni@unige.it

Aristide F. Massardo

e-mail: massardo@unige.it

Thermochemical Power Group,
Dipartimento di Macchine Sistemi Energetici e
Trasporti,
Università di Genova,
Via Montallegro 1,
16145 Genova, Italia

With the objective of performing reliable innovative gas turbine cycle calculations, a new procedure aimed at evaluating blade cooling performance is presented. This complete analytical (convective and film) blade cooling modeling provides the coolant mass flow and pressure loss estimation, and is a useful tool in the field of innovative gas turbine cycle analysis, mainly when alternative fluids are considered. In this case, in fact, the conventional semi-empirical data based on the use of air as traditional coolant and working media are no longer suitable. So the analytical approach represents a way of properly investigating alternative cooling methods and fluids. In the presented analysis the effects of internal blade geometry on cooling performance are summarized by the Z parameter, which also highly affects the coolant flow pressure losses. Since existing technology represents a natural starting point for the assessment of Z , the model is able to automatically estimate a proper value relying only on available semi-empirical data which were established for air-cooled gas turbine blades. When alternative fluids are considered, the same estimated value of Z is still maintained for the calculation, with the result of investigating the performance of existing blade technology for novel operational conditions. This represents an example of how the analytical approach, supported by conventional air-cooled blade semi-empirical data, appears as an innovative tool in the analysis of novel gas turbine cycles. In fact, the simulation results for the cooled blade were easily employed on the whole system level (gas turbine). [DOI: 10.1115/1.1707030]

Introduction

In a society where economic and low-polluting power production has become of vital importance, the development of cheap, new, and clean power plants is important. In such a scenario the advent of the deregulated electricity market has reinforced the quest for cheap, clean, reliable and efficient means of heat and electric power generation.

This ongoing trend, which is very likely to continue and even increase in the future, not only includes attempts to improve existing thermal cycles but also to develop new ones, most of which consider alternative fluids and working media. In the field of cycles based on the use of gas turbines, new solutions have been proposed, such as: the humid air turbine cycle, [1,2], the SOFC-GT cycle, [3,4], and the Matiant cycle, [5]. Moreover, the realization of steam cooling in combined cycles is the main apparent example of innovative design at a cycle level, [6].

Another important aspect is that the step from the theoretical analysis to the hardware demonstration of the new technologies should require significant effort in research and technological development, [7,8] and could comprise considerable risk and high cost. This in turn means that it is uncertain how many and what innovative gas turbine cycles will actually be turned into operating power plants. Hence, it is necessary to identify what cycles are technically and economically interesting at an earlier stage, [2,9].

A natural starting point for the hardware demonstration of an

innovative cycle should be existing gas turbine technology. Therefore, it is important to assess how existing gas turbines will operate, adopting alternative coolants and working media which are often of interest in innovative cycles.

The present paper deals with the problem of evaluating the influence of changing fluid composition on the analysis of the performance of innovative gas turbine-based cycles, with the purpose of increasing the reliability of the results at the gas turbine system level.

A method for estimating the performance and the required technological development only for *convectively cooled* vanes which are operating in the presence of alternative media has been previously proposed by the authors, [10]. In the present work, this method has been carefully revised and extended to include *film cooled* blades too.

The adopted *analytical* approach allows the blade cooling analysis in presence of alternative fluids to be obtained. At the same time, the assessment of a few parameters through the use of the air cooled blade semi-empirical data not only should improve reliability, but also avoid the need of a large number of input data (mainly geometrical data) necessary to run such an analytical calculation. This makes the model a worthwhile and *user-friendly* tool when blade cooling performance has to be evaluated at the gas turbine system level for the investigation of traditional and innovative cycles.

Film Cooling Model

In a previous work, [10], the authors have focused on the case of *convective* blade cooling analysis. In the present work this procedure has been carefully revised and extended to include *film*

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30519. Manuscript received by IGTI, Dec. 2001, final revision, Mar. 2002. Associate Editor: E. Benvenuti.

cooled blade rows too. A brief description restricted to the film cooling analysis is presented below. A wider explanation of the procedure is available in the cited work.

The model provides the blade cooling analysis with an analytical approach, and it can be used to evaluate, among other things: (i) the blade row coolant mass flow rate; (ii) the blade internal geometry data, necessary for the evaluation of (iii) the pressure drop of the coolant that flows through the blade; (iv) the main flow (hot gas side) Stanton number and (v) the adiabatic wall film effectiveness; (vi) the temperature of the coolant as it leaves the blade at the film feeding outlet section, and (vii) the ratio of cooling mass flow used to feed the film to the total cooling mass flow. It should be emphasized that since the model is analytical, it can take blade row geometry, cooling methods and changing alternative fluids composition influence on blade cooling performance into account in a detailed way, and it can also calculate the pressure drop of the coolant as it passes through the cooling channels inside the blade.

On the other hand analytical models generally require a large number of input data (mainly geometrical data), which are rarely available at a cycle level analysis. Concerning the internal geometry of the blade, i.e., the configuration of the cooling channels inside the blade, a number of geometrical input data could be worth summarising in the value of the cooling system technology level parameter Z , defined as

$$Z = \Psi_i \cdot \alpha_h^{0.2} \cdot n_p^{0.8} \cdot E_h \cdot (c/d)^{1.2}. \quad (1)$$

When set, this parameter permits the full determination of the influence of the internal blade geometry on blade cooling performance. In this way, the mass flow requirements and pressure losses can be carefully determined, not only for conventional fluids but also for new coolants and new media, such as steam or carbon dioxide. Increasing Z means increasing the cooling system performance, and as a consequence the costs of the blade row itself, i.e., Z is also of importance from an economic point of view.

Values of Z are usually calculated through Eq. (1), which requires knowledge of the detailed internal blade geometry. Unfortunately such information is not easily available when the investigation is carried out at a cycle level of analysis. As described below, the alternative way adopted in the model was to estimate Z by relying on knowledge based on existing technology. This represents a point of innovation introduced by the analytical model, which allows the blade cooling system technology level parameter to be automatically determined on the basis of the air cooling semi-empirical data established for air-cooled conventional gas turbines. The simultaneous use of semi-empirical data permits a proper evaluation of the influence of blade geometry, which is fitted to actual engine performance. In this way, not only is the reliability of the results expected to improve, but also the required input to run the analytical calculation is reduced.

The analytical procedure considers the blade row geometry and the influence of alternative fluids and cooling methods on the heat transfer through the cooled blade wall in detail. The pressure loss of the coolant that passes through the cooling channels of the blade is also estimated.

The thermal analysis of the film cooled blade row is almost the same as the one presented by the authors for the convective cooled blade row, [10,11] except that the hot gas temperature T_g must be

replaced by the film average temperature T_F , and the average external heat transfer coefficient h_g by the one for the film condition, when available.

This last is a very important aspect as the external convective heat transfer coefficient varies in the presence of film cooling. The disturbance caused by the hole ribs and the injected air can cause transition in a laminar boundary layer, so that it is possible for films to increase, rather than decrease, the convective heat flux to the blade wall, [12]. Since it was far easier to measure steady state temperatures produced by the film on an adiabatic wall, film cooling investigations have produced correlation of measured film performance in terms of adiabatic wall effectiveness, and only rarely the more onerous wall heat flux measurements were made. This meant that such adiabatic wall measurements produced no data on the effect of film injection on the convective heat transfer coefficients. This lack of knowledge forces the analyst to assume the same correlation as in the case of purely convective cooling to evaluate the external heat transfer coefficient in the presence of film cooling, [12]. As a result the adiabatic wall film temperature T_F was obtained from the adiabatic wall film effectiveness η_{ad} , which is defined:

$$\eta_{ad} = \frac{T_g - T_F}{T_g - T_{co}}. \quad (2)$$

The value of η_{ad} was calculated by means of the semi-empirical correlation proposed by Goldstein and Haji-Sheikh [13], which suits the injection of film through a continuous slot over flat plates:

$$\eta_{ad} = \frac{1.9 \cdot \text{Pr}_g^{2/3}}{1 + 0.329 \cdot \left(\frac{c_{p,g}}{c_{p,c}} \right) \cdot \left(\frac{\mu_g}{\mu_c} \right)^{0.2} \cdot \xi^{0.8} \cdot \beta} \quad (3)$$

where

$$\xi = \left(\frac{x}{w \cdot M} \right) \cdot \text{Re}_w^{-0.25}$$

$$\beta = 1 + 0.00015 \cdot \text{Re}_w \cdot \left(\frac{\mu_c}{\mu_g} \right) \cdot \left(\frac{W_g}{W_c} \right) \cdot \sin \alpha$$

and where

- x is the distance in downstream chordwise direction from the injection location;
- w is the injection slot width;
- $M = (\rho_c \cdot U_c) / (\rho_g \cdot U_g)$, which is the blowing rate and U_c is the coolant velocity as it is injected to feed the film;
- $\text{Re}_w = (U_c \cdot w) / \nu_c$, which is the Reynolds number referred to the injection velocity.

The β term takes into account the injection angle α different from zero for the coolant film. When α is equal to zero, β is equal to one, and the coolant is injected in the direction parallel to the blade surface. Taking the injection geometry into account it is then possible to translate the semi-empirical correlation (Eq. (3)) into the more useful form

$$\eta_{ad} = \frac{1.9 \text{Pr}_g^{2/3}}{1 + 0.329 \left(\frac{c_{p,g}}{c_{p,c}} \right) \text{Re}_g^{-0.2} \left(\frac{x}{c} \right)^{0.8} \left[\left(\frac{2a_t \sigma}{\Psi_g} \right) \left(\frac{m_g}{m_c} \right) \left(\frac{1}{r_{fc}} \right) + 0.00015 \text{Re}_g \frac{W_g}{W_c} \sin \alpha \right]} \quad (4)$$

where r_{fc} is the ratio between the coolant mass flow that feeds the film and the total coolant mass flow entering the blade row m_c (i.e., a measure of the feeding rate of the film), while the ratio x/c was fixed equal to 0.5 in the present calculation.

The analytical procedure starts producing the film cooling analysis applied at an elementary blade surface in a radial direction, and evaluates

- the thermophysical properties of the fluids which are exchanging the heat flow;
- the parameters which define the effects of the film on the boundary layer;
- the parameters which govern the convective heat flow on the gas side of the blade;
- the conductive heat flow through the blade wall;
- the convective heat flow on the coolant side of the blade;
- the heat flow rate entering the blade.

Then integrating along the blade height, the procedure can estimate among other things:

- the distribution of the metal blade temperatures on the gas and on the coolant side of the blade;
- the temperature of the coolant itself that flows through the cooling channels inside the blade;
- the maximum blade wall metal temperature achieved;
- the pressure loss of the coolant flowing through the cooling channels, basing on the knowledge of the internal blade geometry (Z parameter) and under the assumption of the incompressibility of the coolant gas.

Therefore, an iterative procedure is necessary in order to estimate the minimum coolant mass flow m_c which allows the blade row metal temperature to be maintained below its maximum permissible value T_{bmx} (i.e., a constraint restricted by material and structural requirements).

The analytical approach of the algorithm makes it a useful tool for the analysis of cooling in the presence of different cooling methods (closed-loop, open-loop, TBC, and film cooling) and different working means and coolants (e.g., dry and humid air, steam, carbon dioxide). In this case the changes in fluid properties are not only considered through a variation of the specific heat c_p value for both the hot gas and coolant (which are assumed as semi-perfect gasses), but also by varying the viscosity μ , density ρ and all the related parameters, such as the average Stanton and Prandtl numbers.

The Stanton number value for the hot gas was determined by the relationship suggested by Louis, [14]:

$$St_g = 0.5 Re_g^{-0.37} Pr_g^{-2/3} \quad (5)$$

while for the coolant, Colburn's one was adopted:

$$St_c = E_h \cdot 0.023 \cdot Re_c^{-0.2} Pr_c^{-2/3} \quad (6)$$

Instead, the Mach number of the mainstream was iteratively calculated on the basis of the knowledge of the expansion inlet section and the mainstream mass flow.

As already stated, the key parameter for the modeling is the parameter Z . This parameter should be evaluated from the knowledge of the internal blade geometry (Eq. (1)), data that is rarely available in open literature. On the contrary, Z is the free parameter used in the present calculation to obtain concordance between the output of the analytical procedure and the semi-empirical data, which were established for air-cooled gas turbines: as a result Z is evaluated in accordance with the actual engine performance. Furthermore, Z determines among other things: (i) the number of cooling channels n_{ch} inside the blade, (ii) the number of passages n_p for each channel through the blade (1, 2, or 3), and (iii) the cooling channel hydraulic diameter d , which must be superior to an assumed minimum value given as input.

The proper Z parameter setting based on air-cooled blade semi-empirical data is possible as the analytical code is supported by an

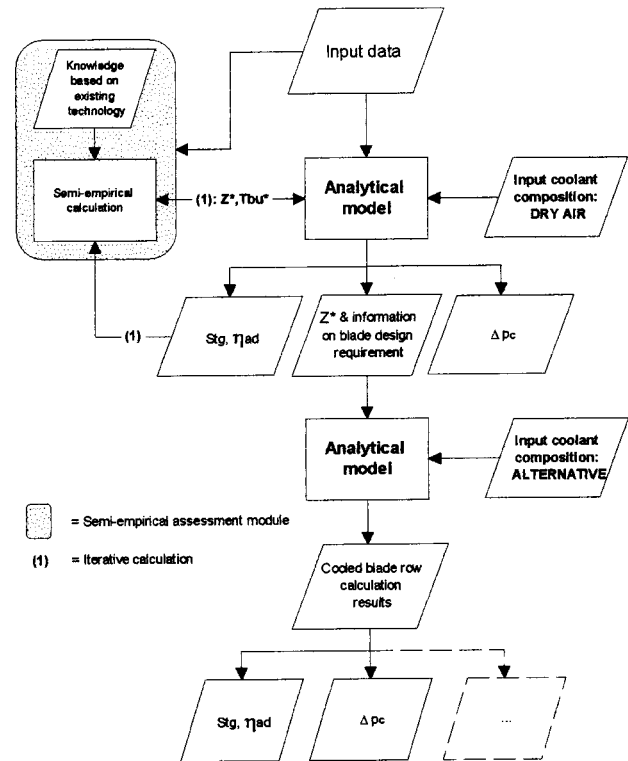


Fig. 1 Flow chart of the blade row cooling flow and pressure losses procedure (conventional and alternative fluid)

iterative procedure which considers a calculation unit called a *semi-empirical assessment module*, (Fig. 1). This module allows the estimation of (i) the air mass flow m_c required to cool the blade row and (ii) the temperature T_{co} of the coolant as it leaves the blade to feed the film layer, simply relying on information derived from available semi-empirical curves (a detailed explanation is presented in the Appendix). In Table 1, a list of input and output variables for the code is presented in a manner that distinguishes between the cited assessment module and the main analytical calculation branch: that is the reason why a few outputs are considered as input for the semi-empirical module (see also Fig. 1). When the external blade geometry parameters in Table 1 are known, together with the thermodynamic properties of the coolant and of the gas surrounding the blade, the main analytical calculation output m_c , T_{co} and the coolant flow pressure loss depend on the value assigned to the parameter Z . Also, the assessment module estimation of m_c and T_{co} depends on the free parameter T_{bu} , which is the uniform blade temperature in a *standard blade*, [10].

Since both the procedures describe the same air-cooled blade, the calculated values of the two outputs m_c and T_{co} should be equal, i.e., it is possible to write schematically:

$$m_{c,A}(Z) = m_{c,B}(T_{bu}) \quad (7)$$

$$T_{co,A}(Z) = T_{co,B}(T_{bu}) \quad (8)$$

where the subscripts A and B refer to the main analytical procedure and to the semi-empirical assessment module, respectively. If no other free parameter has been left other than the two above mentioned, it follows that just one pair of values exists Z^* and T_{bu}^* , which simultaneously verifies the two equations.

The integrated analytical procedure is illustrated in Fig. 1. It starts with the knowledge of the component level (e.g., external blade geometry) that provides, together with the assumption of *dry air* as coolant, the required input data for the model. Values of St_g and η_{ad} are provided to the semi-empirical assessment module

Table 1 Input and output for the main analytical procedure and the semi-empirical assessment module

	Main Analytical Procedure	Main Analytical and Semi-empirical Procedure	Semi-empirical Assessment Module
Input	T_{bmx}, r_{fc}, Bi_{bw} , ~10 geometry parameters	$T_g, p_g, m_g, T_{ci}, p_{ci}$, coolant and gas composition	$St_g, \eta_{ad}, A_b/A_g$, Semiemp curve (Fig. A1)
Free parameter	Z	...	T_{bu}
Output	$\Delta p_c/p_{ci}, d, St_g, \eta_{ad}$, $St_c, h_c/h_g, n_p$	$m_c, T_{co}, c_{pc}, c_{pg}$	...

by means of an iterative procedure, which allows the analytical model to be supported by the knowledge based on existing technology. In this way the unique combination of values Z^* and T_{bu}^* is found, guaranteeing the agreement with the air-cooled blade semi-empirical data (refer to Eqs. (7) and (8)). In this way the effects of the internal blade geometry on blade cooling are assessed, which provides the following advantages:

- arbitrary setting of important parameters is avoided;
- internal blade geometrical data (Z parameter) useful for cooling analysis are derived even if the detailed knowledge of the blade geometry is not available;
- the acquired knowledge of the internal blade geometry allows the calculation of the coolant flow pressure losses and gives some information on blade design requirements;
- after the Z parameter has been set, based on available semi-empirical data, which have been established for conventional air cooled blades, investigation of alternative coolants and working media can be carried out for a given internal blade geometry that should increase the reliability of the results (see Fig. 1);
- information can be obtained about the required changes in internal blade geometry, as thermodynamic properties around the studied gas turbine blade row are altered (e.g., for different values of T_g, T_{ci} , etc.).

Moreover, the presence of temperature fluctuations (i.e., hot spots) in the flow leaving the combustion chamber is taken into account by using an estimated maximum value for the gas temperature:

$$T_g^{\text{MAX}} = T_g + K_{\text{comb}} \cdot \Delta T_{\text{comb}} \quad (9)$$

where ΔT_{comb} is the temperature rise in the combustion chamber [12,15,16]. The combustion chamber pattern factor K_{comb} is an empirical constant which depends on the type of combustion chamber (aero or industrial) and the position of the blade row with respect to the combustion chamber outlet section; adopted values are: 0.1 in the first nozzle, 0.05 in the first rotor, and 0.025 in the rest of the cooled downstream.

Model Implementation

The model was originally created as a Fortran code and then loaded as a dynamic link library (dll) into the commercial software IPSEpro, [17], as described by Jordal and Torbidoni [18]. Specific heat and viscosity are calculated on the basis of data reported by Perry and Green [19], while the rest of the thermodynamic properties are calculated from the IPSEpro data base.

New Model Analysis of a Film Cooled Blade Row

The new model has been applied to evaluate the performance of an existing air film-cooled blade row, external geometry of which was known. The input data are presented in Table 2. As previously

described, for given values of T_g and T_{bmx} , the procedure produces the sole existing couple of values of Z and T_{bu} that satisfy Eqs. (7) and (8). The identified operating point ($m_c/m_g = 0.041$; $T_{co} = 983$ K), that is defined as the *reference point*, is characterized by $T_{bu}^* = 1199$ K and $Z^* = 82$.

Figure 2 shows the *reference point* in a diagram where the coolant flow rate is plotted versus the hot gas temperature. Varying T_g in the range 1400–2100 K, when the rest of the input data are considered as fixed and coincident with the *reference point*, the coolant mass flow rate is evaluated for different values of Z and the maximum permissible material temperature T_{bmx} . The points on the lines in Fig. 2 verify the conditions (i) $m_{c,A} = m_{c,B}$, and (ii) $T_{co,A} = T_{co,B}$ as described above.

The model is an efficient tool for determining not only the coolant mass flow rate, but also the appropriate value of Z for a novel blade, of the same type (film cooling) as the reference blade. The determined new value of Z gives information on the required internal blade geometry, which could be useful for initial studies on how to develop internal blade configuration.

As an example, starting from the *reference point*, if the hot gas temperature rises, the cooling system can be considered at fixed Z (fixed geometry and technology) while the material performance must be increased (iso- Z curves). On the contrary, if the material performance is fixed, the improvement of the cooling system can be obtained through a quantified variation of the Z value (iso- T_{bmx} curves).

When varying T_g , the variation of the Z parameter along the iso- T_{bmx} curves appears to have a point of maximum value. Z rises when the gas temperatures are low and vice versa. The reason for this has to be found in the shape of the adopted semi-empirical curve, that is shown in Fig. A2. It appears that with the

Table 2 Reference point input data

Hot Gas	
Gas composition	Products from dry-air combustion
T_g	1723 K
p_g	1.6 MPa
m_g	143 kg/s
Coolant	
Coolant composition	Dry air
T_{ci}	687 K
p_{ci}	1.64 MPa
Model	
Cooling method	Film cooling
External geometry	~10 input
Bi_{bw}	0.5
T_{bmx}	1273 K
r_{fc}	0.35

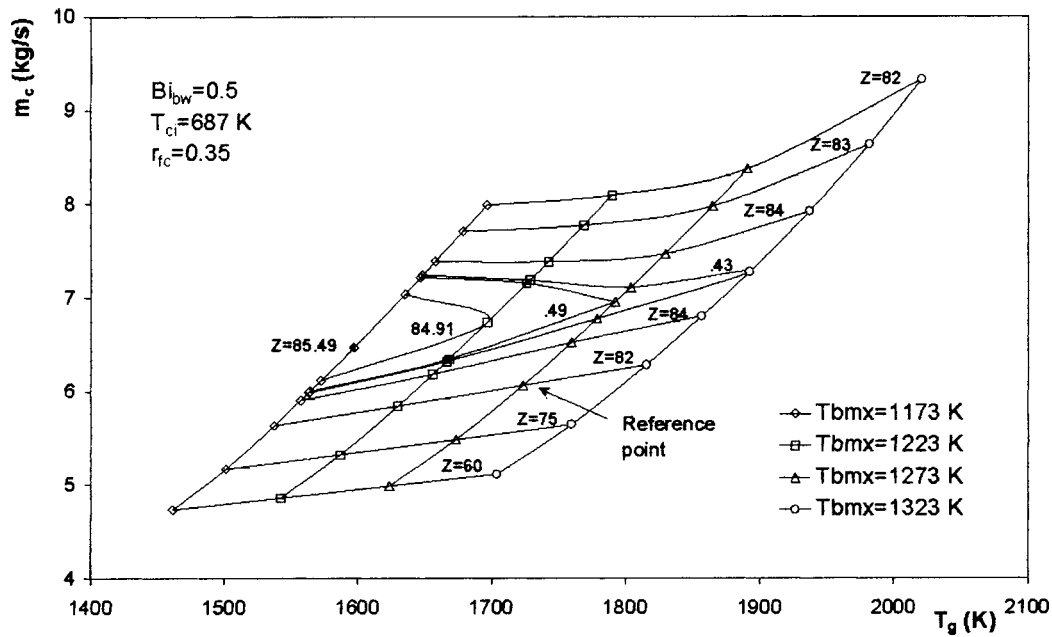


Fig. 2 Coolant mass flow, blade maximum temperature and corresponding values of Z , obtained through a variation of gas temperature T_g for the air film cooled blade row

purpose of keeping the required coolant mass flow as low as possible (and with it the gas turbine expansion losses connected with the mixing of the film layer with the main stream) Z increases with increasing T_g as long as the gas temperatures (and the heat flow entering the blade) are low. On the contrary, when the gas temperatures and the entering heat flow become too high, Z decreases with increasing T_g to provide lower convective cooling efficiency η_c (Eq. (A1)). Therefore a higher amount of coolant feeds the film (i.e., higher m_c required with respect to a constant film feeding rate r_{fc}) with lower T_{co} , this allowing higher film effectiveness η_{ad} (Eqs. (2) and (A4)) and enhances the effect of the barrier for the heat flow entering the blade.

Coolant Pressure Losses. Considering the *reference point* in Fig. 2, it is possible to evaluate the coolant flow pressure loss based on the known value of Z ($Z^* = 82$). In a wider approach it is possible to evaluate the relative pressure losses $\Delta p_c / p_{ci}$ for all the points presented in the diagram of Fig. 2. The results are shown in Fig. 3, where coolant pressure losses are plotted versus hot gas temperature T_g with Z and T_{bmx} as parameters for the air film-cooled blade row.

Even if they deal with two completely different gas turbine blade rows, it is worthwhile comparing the diagram from *film cooling* presented here with the one for *purely convective cooling*

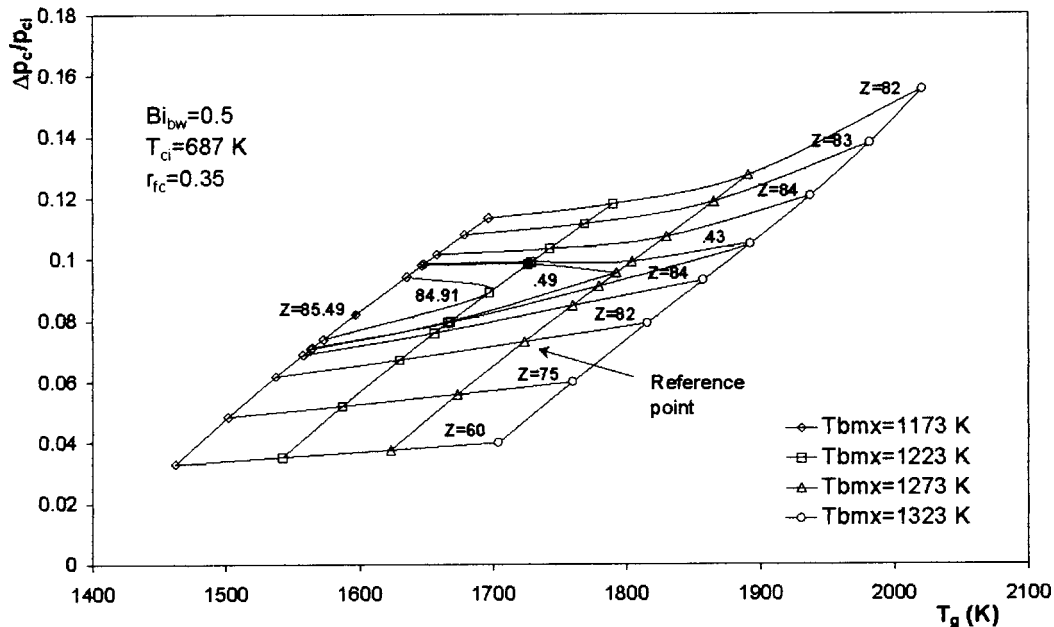


Fig. 3 Coolant pressure losses, blade maximum temperature, and corresponding values of Z , obtained through a variation of gas temperature T_g for air film cooled blade row

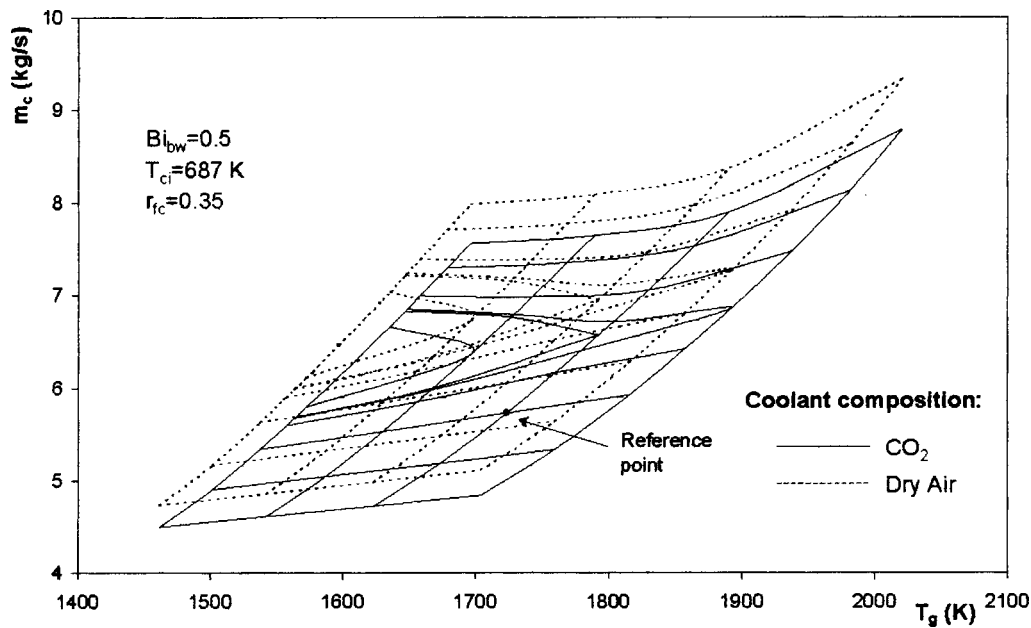


Fig. 4 Coolant mass flow required by the blade row when film cooled with air (dotted lines) or carbon dioxide (solid line), for varying gas temperature T_g , maximum blade temperature T_{bmx} and technology level parameter Z

presented by the authors in their previous work (i.e., Fig. 6 in [10]). It becomes evident that the relative pressure loss in convective cooling rises more rapidly with T_g than in film cooling. This is determined by the increasing value of Z , which is the only way to deal with the effects of the rising gas temperature in a convectively cooled blade: In fact, in this case Z can reach very high values. In accordance with Eq. (1), higher values of Z correspond to more cooling channels with lower hydraulic diameter d , which in turn means a higher pressure loss for the coolant flow. A dif-

ferent situation is presented in the diagram in Fig. 3, where the adopted film cooling method allows the Z parameter values to be kept low and so to keep pressure losses low.

Film Cooling With Alternative Coolants

A sample of the analysis for film cooling performed with alternative coolants is presented here.

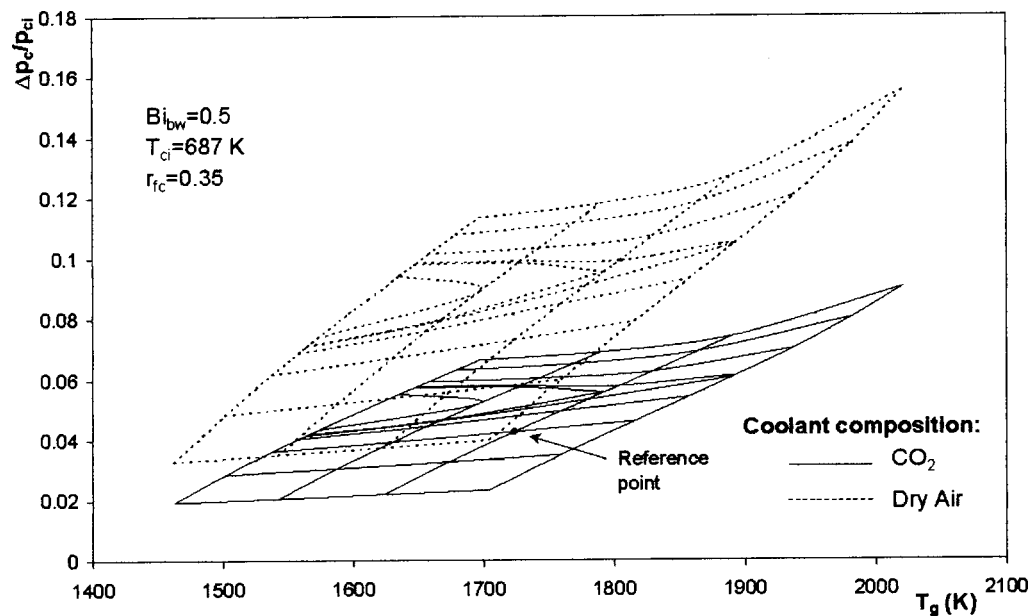


Fig. 5 Relative coolant pressure drop for the blade row when film cooled with air (dotted lines) or carbon dioxide (solid line), for varying gas temperature T_g , maximum blade temperature T_{bmx} and technology level parameter Z

The parameter Z having been determined, the internal geometry of the blade is sufficiently known to evaluate the cooling of the same blade row when alternative coolants are adopted in place of dry air (see Fig. 1). The calculations were carried out using exactly the same input data as for Figs. 2 and 3, except for the different coolant composition which was carbon dioxide. For comparison, the results for air cooling shown in Figs. 2 and 3 are reproduced in dotted lines in Figs. 4 and 5, respectively. The diagram in Fig. 4 shows the coolant mass flow required versus hot gas temperature and Fig. 5 shows coolant relative pressure drops. It can be seen that carbon dioxide cooling substantially reduces the coolant mass flow requirement with respect to that of air.

The pressure losses inside the blade are also greatly reduced. Considering that the cooling system geometry, and with it also the cooling channel diameter, are equal for the same values of Z , the reduction of the pressure loss for the carbon dioxide cooling depends on (i) the different thermophysical properties of the coolant and (ii) the different rate of the coolant mass flow required (i.e., a different coolant velocity through the constant cooling channel geometry). Since the pressure losses depend also on the square of the velocity, their reduction becomes more evident for high values of the hot gas temperature, due to the increasing coolant velocity.

It is possible to conclude that the carbon dioxide cooling performs better than that with dry air, and also that the improvement is much more evident for high values of the hot gas temperature. In that range, in fact, the alternative coolant allows for a sensitive reduction of both the coolant pressure loss and the coolant mass flow rate requirement (this last being also connected with the gas turbine expansion losses caused by the mixing of the film layer with the hot gas stream).

Conclusions

In order to perform reliable innovative gas-turbine cycle calculations, a new computational tool for the analysis of film cooled blades and vanes is proposed. It consists of an *analytical procedure* for the investigation of the film cooling starting with the knowledge of the blade row geometry required to run the simulation. The effect of the internal blade geometry (i.e., cooling channels geometry) on the cooling performance is considered through the cooling system technology level parameter Z . The procedure is able to automatically set a proper value for Z by means of a *semi-empirical assessment module*. As the assessment module relies on semi-empirical data, the Z setting is determined with respect to existing technology.

Once Z is determined, the internal blade configuration is determined, which makes it possible to calculate the pressure losses of the coolant and the behavior of a given blade configuration in an environment consisting of a new cooling medium and/or a new hot gas composition. It is also possible to study how Z should be altered for, e.g., by varying hot gas temperature, in order to avoid excessive coolant mass flow or a pressure drop. This means that valuable information on blade design requirements for alternative fluids can be obtained.

Therefore the following main conclusions may be stated:

- the method proposed here allows a very detailed analysis of blade cooling influence on innovative gas turbine cycle performance, and it is a noteworthy step in this field;
- when traditional cooling fluids are investigated, the model represents a very detailed method to evaluate not only the mass flow rate m_c , but also the coolant pressure loss;
- when alternative fluids are investigated and the semi-empirical data become not anymore available, the proposed *analytical model* overcomes the limits of the *semi-empirical approach*, thanks to the preliminary “ Z ” analysis carried out with air as coolant.

Planned future work is to extend the integration approach to include the *analytical expansion model*. This should provide a

method for the analysis of the cooled gas turbine on a cycle level with increased reliability for alternative coolants and working media.

Appendix

The Semi-Empirical Assessment Module. The *semi-empirical assessment module* relies on knowledge based on existing technology in order to allow the presented analytical code to set a proper value for the cooling system technology level parameter Z . It uses conventional air cooled blade semi-empirical curves to calculate the amount of coolant m_c required to cool one gas turbine blade row and the temperature T_{co} of the coolant as it leaves the blade and mixes with the hot gas stream. The fundamental equations for the module can be found in Holland and Thawke [20] and the procedure is based on the “*standard blade*” assumption presented by Halls [21]. A standard blade has a uniform blade temperature T_{bu} equal to the average temperature of that in the actual blade, the same geometry as the actual blade, and the cooling air is warmed up to the uniform blade temperature before leaving the blade. The cooling efficiency η_c describes the ratio between the temperature rise of the coolant flow in the actual blade and in the standard blade:

$$\eta_c = \frac{T_{co} - T_{ci}}{T_{bu} - T_{ci}} \quad (A1)$$

For film cooled blades, T_{co} is the temperature to which the coolant flow is warmed before emerging from the blade to feed the film. Instead, T_{ci} is the temperature of the coolant entering the blade row at the inlet section.

When the film cooling method is adopted, the value of the cooling efficiency is determined by the equation

$$\eta_c = \frac{\varepsilon_c - \eta_{ad}}{(B - \eta_{ad})(1 - \varepsilon_c)} \quad (A2)$$

where the cooling effectiveness ε_c is defined as

$$\varepsilon_c = \frac{T_g - T_{bu}}{T_g - T_{ci}} \quad (A3)$$

and for the adiabatic wall film effectiveness η_{ad} Eq. (2) is adopted.

The cooling parameter B , referred to as w^+ by Holland and Thawke [20], is included in the expression

$$\frac{m_c}{m_g} = B \frac{c_{pg}}{c_{pc}} \text{St}_g \frac{A_b}{A_g} \quad (A4)$$

The relationship between ε_c and B is determined through the use of semi-empirical data, a principle that was previously used by Rosén [22]. Semi-empirical diagrams of ε_c as a function of B can be found in works by Haselbacher [23] and Mukherjee [24]. The data from these two references are plotted in Fig. A1.

In the present work, the data comes from the diagram presented by Haselbacher, and the curve adopted in the calculation is the average one between the lower and upper limit curves that describe the film cooling (Fig. A2). Thermal analysis for the film cooled blade is the same as that for the convective cooled blade, [10], except that the hot gas temperature T_g must be replaced by the average film temperature T_f . Another very important aspect is how the external convective heat transfer coefficient varies in the presence of film cooling. The lack of knowledge forces the analyst to assume the same correlation as in the case of purely convective cooling to evaluate the external heat transfer coefficient also in the presence of film cooling, [12]. Thus Eq. (A2) produces the expression for the effect of film cooling, expressed in terms of adiabatic wall film effectiveness η_{ad} , on the cooling efficiency of a convective plus film cooled blade η_c . The same Eq. (A2) can be transposed in such a way as to give for a *purely convective cooled blade*:

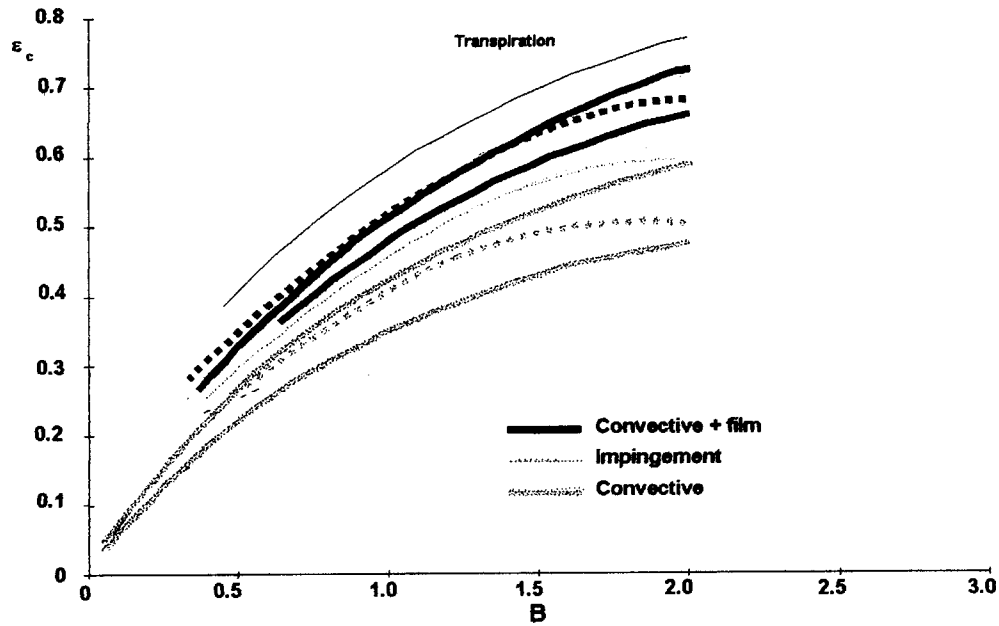


Fig. A1 ε_c as a function of B , reproduced with data from Haselbacher (dotted lines), and Mukherjee (solid lines)

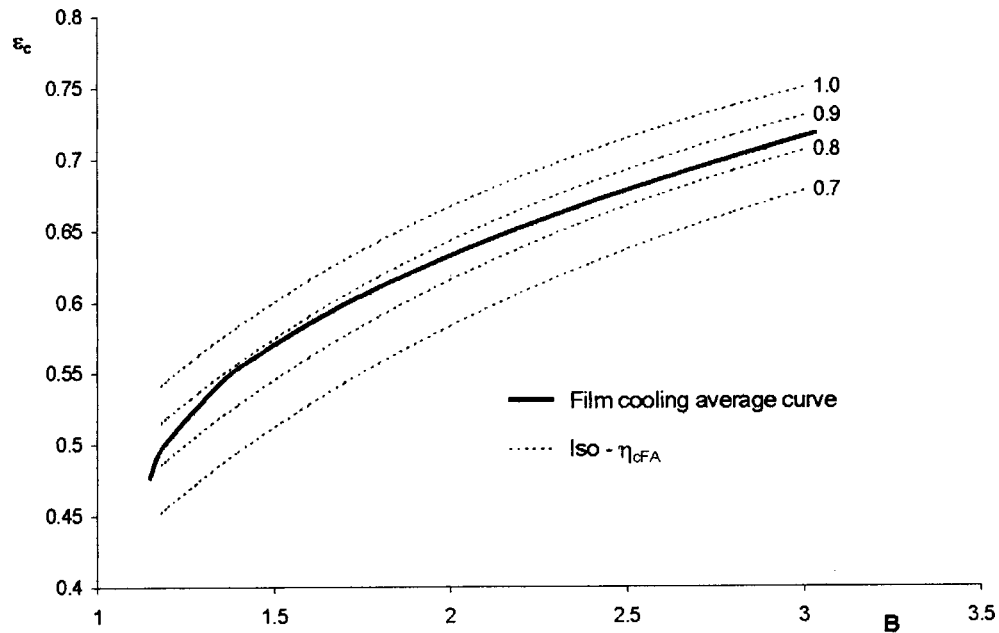


Fig. A2 ε_c as a function of B , reproduced with data from Haselbacher for film cooling (i.e., average curve)

$$\eta_{cFA} = \frac{\varepsilon_c}{B(1 - \varepsilon_c)} \quad (A5)$$

By substituting the appropriate values for the *film cooled* blades on the right-hand side of Eq. (A5), the apparent convective cooling efficiency of a film cooled blade η_{cFA} is obtained. The efficiency value deriving from Eq. (A5) is the value to be read when the film cooled blade is plotted on the conventional correlation graph such as in Figs. A1 and A2, [12].

Apart from selecting the appropriate curve from Fig. A1, the parameters to be set in order to run a solution are the geometry parameter A_b/A_g (i.e., wetted blade and adjacent surfaces to be cooled over average gas cross-sectional area), the hot gas side

Stanton number St_g , the adiabatic wall film effectiveness η_{ad} , and the uniform blade temperature T_{bu} . The value of A_b/A_g was set equal to five in the work presented by Jordal [25], a value that was previously proposed by Bolland and Stadaas [26]. The same value has been adopted based on the external geometry of the blade row investigated here. The value of St_g was provided by correlation with the main analytical code and so passed to the assessment module as shown in Fig. 1. The adiabatic wall film effectiveness η_{ad} is also adopted as it is evaluated by the code. The input and output for the semi-empirical module are listed in Table 1. The main parameter that remains to be set in order to complete the input for one blade row is the uniform blade temperature T_{bu} .

Nomenclature

A_b	= wetted surface of the blade row (blades+shrouds)
A_g	= actual gas cross-sectional area
a_t	= ratio between blade+shroud surface and blade surface
B	= dimensionless coolant parameter
Bi	= Biot number
c	= chord
c_p	= specific heat at constant pressure
d	= cooling channel hydraulic diameter
dll	= dynamic link library
E_h	= parameter that considers increased surface of cooling channels due to turbulators
h	= convective heat transfer coefficient
K_{comb}	= combustion pattern factor
M	= film blowing rate
m	= mass flow rate
n_{ch}	= number of the cooling channels inside the blade
n_p	= number of cooling channel passages
p	= pressure
Pr	= Prandtl number
Re	= Reynolds number
r_{fc}	= fraction of the coolant flow used for film cooling
St	= Stanton number
T	= temperature
t_b	= average blade thickness, (blade cross-sectional area)/ c
TBC	= thermal barrier coating
U	= velocity
W	= molar weight
w	= injection slot width
x	= distance in downstream chordwise direction from the film injection location
Z	= blade cooling system technology level parameter
α	= film injection angle
α_h	= ratio, (empty blade cross-sectional area)/ c^2
Δ	= arithmetic difference
ε_c	= cooling effectiveness
η_c	= cooling efficiency
η_{ad}	= adiabatic wall film cooling effectiveness
η_{cFA}	= apparent convective cooling efficiency of the film cooled blade
μ	= dynamic viscosity
ν	= kinematics viscosity
ρ	= density
σ	= solidity
Ψ_g	= parameter that accounts for the reduction of the gas cross-sectional area due to the blade thickness, $1 - \sigma^* t_b / c$
Ψ_i	= parameter that considers influence of heat transfer between adjacent cooling channels

Subscripts

b	= blade
bw	= blade wall
c	= coolant, cooling, coolant side
comb	= combustion chamber
F	= film layer
g	= gas, gas side
i	= inlet

mx	= maximum
o	= outlet
u	= uniform
w	= related to the film injection slot

Superscripts

*	= reference point
MAX	= maximum

References

- [1] Rao, A. D., 1989, "Process for Producing Power," U.S. Patent No. 4,829,763.
- [2] Traverso, A., and Massardo, A. F., 2002, "Thermoeconomic Analysis of Mixed Gas-Steam Cycles," *Appl. Therm. Eng.*, **22**, pp. 1–21, Elsevier, New York.
- [3] Harvey, S. P., and Richter, H. J., 1994, "Gas Turbine Cycles With Solid Oxide Fuel Cells," *ASME J. Energy Resour. Technol.*, **116**.
- [4] Massardo, A. F., and Lubelli, F., 2000, "Internal Reforming Solid Oxide Fuel Cell—Gas Turbine Combined Cycles—Part I: Cell Model and Cycle Thermodynamic Analysis," *ASME J. Eng. Gas Turbines Power*, **122**, pp. 27–35.
- [5] Mathieu, P., and Nihart, R., 1998, "Zero Emission Matiant Cycle," *ASME*, Paper No. 98-GT-383.
- [6] Stambler, I., 2000, "Steam-Cooled H-Technology Machines Set for Commercial Operation in 2002," *Gas Turb. World*, Jan–Feb.
- [7] Layne, A. W., and Hoffman, P. A., 1999, "Gas Turbine Systems for the 21st Century," *ASME*, Paper No. 99-GT-367.
- [8] Layne, A. W., 2000, "Developing the Next Generation Gas Turbine Systems—A National Partnership," *ASME* Paper No. 2000-GT-176.
- [9] Massardo, A. F., and Scialò, M., 2000, "Thermoeconomic Analysis of Gas Turbine Based Cycles," *ASME J. Eng. Gas Turbines Power*, **122**.
- [10] Jordal, K., Torbidoni, L., and Massardo, A. F., 2001, "Convective Blade Cooling Modeling for the Analysis of Innovative Gas Turbine Cycles," *ASME* Paper No. 2001-GT-0390.
- [11] Torbidoni, L., 2000, "Espansione Refrigerata nel Turbogas al Variare delle Tecniche e dei Fluidi di Raffreddamento," *Tesi di Laurea*, DIMSET, Italy.
- [12] Holland, M. J., 1992, "Cranfield Lecture Notes—Rotor Blade Cooling in HP Turbines," Technical Report, Department of Turbine Technology, Ref. GN30884, Rolls Royce plc.
- [13] Goldstein, R. J., and Haji-Sheikh, A., 1967, "Prediction of Film Cooling Effectiveness," *Proc. 1967, Semi-International Symposium (Tokyo)*, Japan Society of Mechanical Engineers, Tokyo, pp. 213–218.
- [14] Louis, J. F., 1977, "Systematic Studies of Heat Transfer and Film Cooling Effectiveness," *AGARD CP 229*.
- [15] Young, J. B., and Wilcock, R. C., 2001, "Modeling the Air-Cooled Gas Turbine—Part II: Coolant Flows and Losses," *ASME* Paper No. 2001-GT-0392.
- [16] Consonni, S., 1990, "Performance Prediction of Gas/Steam Cycles for Power Production," MAE Dept., Ph.D. thesis No. 1893-T, Princeton University, Princeton, NJ.
- [17] Anon, 1998, "IPSEpro User Documentation," Simtech Simulation Technology, Graz, Austria.
- [18] Jordal, K., and Torbidoni, L., 2000, "Linking of a Fortran Code to IPSEpro," DIMSET, Internal report TN 4-2000, Italy.
- [19] Perry, R. H., and Green, D., 1984, *Perry's Chemical Engineers' Handbook*, 6th Ed., McGraw-Hill, New York.
- [20] Holland, M. J., and Thawke, T. F., 1980, "Rotor Blade Cooling in High Pressure Turbines," *Int. J. of Aircraft Eng.*, **17**(6).
- [21] Halls, G. A., 1969, "Air Cooling of Turbine Blades and Vanes," *AGARDograph* No. 120.
- [22] Rosen, P. M., 1993, "Evaporative Gas Turbines—A Thermodynamic Evaluation of Their Potential," Report No. ISRN/LUTMDN/TMVK-7010-SE, Lund University.
- [23] Haselbacher, H., 1989, "Gas Turbines," *Standard Handbook of Power Plant Engineering*, Thomas C. Elliot, ed., McGraw-Hill, New York.
- [24] Mukherjee, D. K., 1977, *Zür Schaufelkühlung der Gasturbine*, Brown Boweri Mitteilungen.
- [25] Jordal, K., 1999, "Gas Turbine Cooling Modeling—Thermodynamic Analysis and Cycle Simulations," Report No. ISRN LUTMDN/TMVK-7034-SE, Lund University.
- [26] Bolland, O., and Stadaas, J. F., 1995, "Comparative Evaluation of Combined Cycles and Gas Turbine Systems With Water Injection, Steam Injection and Recuperation," *ASME J. Eng. Gas Turbines Power*, **117**.

Aspects of Cooled Gas Turbine Modeling for the Semi-Closed O_2/CO_2 Cycle With CO_2 Capture

Kristin Jordal¹

Olav Bolland

Department of Energy and Process Engineering,
Norwegian University of Science and
Technology,
N-7491 Trondheim, Norway

Ake Klang

Demag Delaval Industrial Turbines,
S-612 83 Finspong, Sweden
e-mail:
ake.klang@industrial-turbines.siemens.com

In order to capture the behavior of the oxyfuel cycle operating with high combustor-outlet temperature, the impact of blade and vane cooling on cycle performance must be included in the thermodynamic model. As a basis for a future transient model, three thermodynamic models for the cooled gas turbine are described and compared. The first model, known previously from the literature, models expansion as a continuous process with simultaneous heat and work extraction. The second model is a simple stage-by-stage model and the third is a more detailed stage-by-stage model that includes velocity triangles and enables the use of advanced loss correlations. An airbreathing aeroderivative gas turbine is modeled, and the same gas turbine operating in an oxyfuel cycle is studied. The two simple models show very similar performance trends in terms of variation of pressure ratio and turbine inlet temperature in both cases. With the more detailed model, it was found that, without any change of geometry, the turbine rotational speed increases significantly and performance drops for the maintained geometry and pressure ratio. A tentative increase of blade angles or compressor pressure ratio is found to increase turbine performance and decrease rotational speed. This indicates that a turbine will require redesign for operation in the oxyfuel cycle. [DOI: 10.1115/1.1762908]

Introduction

The oxyfuel cycle, i.e., the semiclosed gas turbine cycle burning a fossil fuel in oxygen (97%+ purity) with CO_2 as a diluting gas, has emerged as one of the possible gas-turbine-based power plant concepts that enable CO_2 capture. The behavior and performance of this cycle at full-load steady-state operation with natural gas as fuel has been well investigated by, e.g., Bolland and Mathieu [1], and Bolland and Saether [2]. Ulizar and Pilidis [3] investigated a semiclosed cycle with combustion of gasified coal where argon (Ar) is assumed not to be separated from the oxygen before injection in the combustion chamber. In [3], steady-state, off-design performance was investigated, and it was also mentioned that much work is required in order to understand the fundamental principles of a gas turbine operating in an oxyfuel cycle, since current knowledge relies on air as a working fluid. The compressor and combustor [4], and also the cooled turbine require particular attention in the oxyfuel cycle. An example of a study related to the cooled turbine is the blade cooling study by Carcasci et al. [5] where it was concluded that CO_2 is very well suited for film cooling, less suited for convective cooling and that a redesign of the cooling system might be necessary.

Due to the recirculation of a major part of the exhaust gases into the compressor, the dynamic behavior of the oxyfuel cycle can be complicated. Ulfsnes et al. [6] present initial results of the oxyfuel cycle's dynamic behavior, but with an uncooled turbine model. Turbine cooling is, in itself, probably not the most crucial aspect in the dynamics of the oxyfuel cycle, but it should be included with a reliable, but preferably simple model, in order to make the calculated results more accurate. As for design studies and the evaluation of differences between the oxyfuel turbine and the turbine in ordinary gas turbines, cooling must definitely be

taken into account, since a full-scale base-load design of the oxyfuel cycle is likely to have a combustor outlet temperature elevated enough to require the cooling of turbine blades and vanes.

The objective of the present paper is to compare three conceptually different thermodynamic models for the cooled gas turbine. The three models will be referred to as M1, which is the model presented in Bolland and Stadaas [7], M2, which was described and employed in Jordal [8] and M3, which is a new mean-line stage-by-stage model. The long-term goal of the present work is to establish two models for use in thermodynamic cycle calculations; (A) one simple, cooled turbine model that is suitable for dynamic simulations, and (B) one detailed, cooled turbine model, for steady-state investigations of how flow conditions and turbine material temperatures might vary with varying gas compositions. Model M3 is close to goal B, but, as discussed towards the end of the paper, some possible modifications have been identified during the progress of the present work, in particular for the loss modeling.

Comments are made with respect to model implementation, and computational results are given for simple and oxyfuel cycle performance, including changes in blade and vane average material temperatures. Requirements regarding compressor performance and design will not be treated here. Throughout the present paper, it will be assumed, in all calculations, that the compressor is operating at its design point with a polytropic efficiency of 0.92 and coolant extraction points at appropriate pressures. Jackson et al. [4], investigated the synchronous operation of a gas turbine in the oxyfuel cycle; they concluded that a significant redesign of the combustor and combustor systems would be necessary, when compared to a gas turbine designed to use air as its main working medium. Further, they mention that the compressor probably also needs redesign in multishaft configurations.

Since combustion in the oxyfuel cycle must generally take place with excess oxygen in order to ensure complete combustion, the working medium in the turbine will be referred to as a $CO_2/H_2O/O_2$ mixture.

¹To whom correspondence should be addressed. Current address: Vattenfall Utveckling, 162 87 Stockholm, Sweden. E-mail: Kristin.Jordal@vattenfall.com.

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Atlanta, GA, June 16–19, 2003, Paper No. 2003-GT-38067. Manuscript received by IGTI, October 2002; final revision, March 2003. Associate Editor: H. R. Simmons.

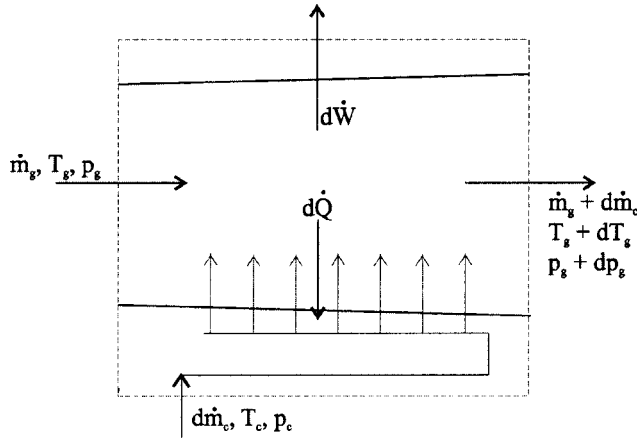


Fig. 1 Element of cooled expansion path for model M1

M1—Continuous Heat and Work Extraction

A description of the first model (in this paper, denoted M1), its background and references are given by Bolland and Stadaas [7]. In this model, expansion through the turbine is divided into a large number of steps, where the sequence

1. adiabatic expansion,
2. mixing of coolant and hot gas (total temperature loss), and
3. total pressure loss due to coolant mixing

is repeated for each step in the cooled part of the expansion path. An element of the cooled expansion path for M1 is shown in Fig. 1.

The work extraction through the walls is

$$d\dot{W} = -\dot{m}_g c_{pg} dT_g^* \quad (1)$$

On a differential form, the relationship between temperature and pressure drop in the turbine is expressed as

$$\frac{dT_g^*}{T_g^*} = \left(\frac{p + dp}{p} \right)^{R \eta_{pt} / c_{pg}} - 1 \quad (2)$$

where η_{pt} is uncooled, polytropic efficiency of the turbine. One of the features of M1 is the close interconnection between cooling and heat and work extraction through the flowpath. The required coolant mass flow for an increment of the flowpath is found from

$$\frac{d(\dot{m}_g c_{pg})}{\dot{m}_g c_{pg}} = \sigma \frac{dT_g^*}{T_{amb}^*} \frac{(T_g^* - T_m)}{(T_m - T_c^*)} \quad (3)$$

The parameter σ characterizes the relative heat-to-work loadings on the flowpath surfaces and can be regarded as a descriptor of the level of technology of the turbine, i.e., the lower the value of σ , the higher the level of technology.

$$\sigma = \frac{St(A_{b,stage}/A_g)}{C_{M1}(\kappa - 1)Ma_{amb}^2 \eta_c} \quad (4)$$

In (4), C is a constant that depends on stage geometry and velocity triangles. The ratio $A_{b,stage}/A_g$ is the stage wetted surface over gas path cross-sectional area, and the cooling efficiency η_c is

$$\eta_c = \frac{T_{co}^* - T_{ci}^*}{T_m - T_{ci}^*} \quad (5)$$

The losses due to coolant injection in the hot gas flow are considered through a total pressure loss and are taken into account by means of the following equation:

$$\frac{dp}{p} = -\frac{d\dot{m}_c}{\dot{m}_g} \kappa Ma^2 Y. \quad (6)$$

The fact that M1 does not include physical turbine stages makes it suitable for general concept studies and variations of cycle pressure ratio over wide ranges. Since the metal temperature is always an input, however, M1 is less useful for studies of how the temperature of materials might vary with different working media. The cooling efficiency η_c is, in M1, related to the flowpath surface temperature.

M2—Simple Stage-by-Stage Model

Model M2 was developed for the study of coolant-type variation in a turbine with a defined number of stages [8], meaning that focus was placed on getting the heat and mass balances correct for a specific case, rather than for concept studies covering a wide range of pressures and temperatures. Like M1, only total conditions are considered for the working medium and coolant. In M2, cooling and expansion are treated separately, where the stator cooling and mixing of hot gas and coolant are considered before the work extraction. The rotor total enthalpy drop is considered to be equal with the value of Δh before and after rotor cooling is calculated. This means that for each stage, the calculation of produced work is divided into two parts where each part of the work is determined from

$$\dot{W} = \dot{m}_g \Delta h = \dot{m}_g (h_x - h_y). \quad (7)$$

The corresponding relation between total pressure and temperature drop over this half-stage is

$$\frac{T_x^*}{T_y^*} = \left(\frac{p_x^*}{p_y^*} \right)^{R \eta_{ptr} / c_{pg}} \quad (8)$$

where η_{ptr} is the turbine-reduced polytropic efficiency, the reduction taking place due to the losses that occur due to the mixing of coolant with the hot gases. The value of η_{ptr} is calculated with an equation derived from Traupel [9]:

$$\eta_{ptr} = \eta_{pt} - \ln \left(\frac{p_{it}^*}{p_{ot}^*} \right) \frac{p_x^*}{p_{it}^* - p_{ot}^*} \frac{\dot{m}_c}{\dot{m}_{git}} S. \quad (9)$$

It should be noted that Eq. (9) makes use of the total pressure at the inlet and outlet of the entire turbine, and the ratio between total coolant mass flow and gas mass flow into the first turbine stage. Since p_x will decrease throughout the expansion, the reduction of the polytropic efficiency will also decrease, which is also generally the case in real engines. The model parameter S is a characterization of the technology level of the engine and has a lower level for large and/or modern engines than for old and/or small engines. A typical value of S is 0.1.

Like M1, M2 employs the cooling efficiency in Eq. (5), combined with the cooling effectiveness ε_c

$$\varepsilon_c = \frac{T_g^* - T_m}{T_g^* - T_{ci}^*} \quad (10)$$

which can also be expressed as

$$\varepsilon_c = \frac{B \eta_c}{1 + B \eta_c} \quad (11)$$

The parameter B (denoted as w_c^+ by Holland and Thake [10]) is defined as

$$B = \frac{c_{pg} \dot{m}_c}{\alpha_g A_g} \quad (12)$$

The required coolant mass flow is calculated from

$$\frac{\dot{m}_c}{\dot{m}_g} = B \frac{c_{pg}}{c_{pc}} St_g \frac{A_b}{A_g}. \quad (13)$$

The Eqs. (5) and (10)–(13) can be found in Holland and Thake [10] and they were employed in Jordal [8]. It should be noted here that there are similarities between M1 and M2, such as the use of the group $St_g A_b/A_g$. A default value for St_g of 0.005 can be found in El-Masri [11]; this has been employed in both M1 and M2. A typical value used previously for A_b/A_g for one blade row in M2 has been 4 or 5, whereas in M1, a value for an entire stage has typically been 8. A typical value for the cooling efficiency of 0.3, in the case where coolant is extracted continuously from the compressor, is employed in [7] with reference to Louis [12] and will also be employed in the present paper to determine the connection between coolant inlet temperature, outlet temperature and the materials surface temperature on the hot gas side.

In M1, the materials surface temperature is given as an input (Typically 1123 K) that is kept constant even when the gas temperature varies. In reality though, it has historically been the trend that a more advanced gas turbine with a higher combustor outlet temperature most likely will employ more advanced materials that can operate at higher temperatures. This trend is represented, e.g., in a diagram by Geipel et al. [13], and from this diagram it is possible to deduce the following linear correlation between gas temperature and materials temperature (in K):

$$T_g = 3.77T_b - 2.19 \cdot 10^3. \quad (14)$$

Equation (14) was implemented as an option in M2.

In M2, the heat extraction from the hot gas flow due to cooling takes place at the point where cooling is calculated. With help from the calculated value of T_{ce} in Eq. (5), a heat balance is simply set up in that point, to determine the new enthalpy and temperature of the hot gas flow. Note that this heat extraction takes place once for the vane and once for the blade in each stage, whereas in M1, it occurs in close connection to the work extraction throughout the expansion. It is only the cooled part that is calculated with a stage-by-stage approach in M2, the uncooled part of the turbine is calculated in one step with a constant polytropic efficiency.

M3—Mean Line Stage-by-Stage Model

Like M2, model M3 is also a model where the number of stages has to be defined, and where cooling and expansion are treated separately. The fact that M3 contains more geometry parameters than M1 and M2 makes it suitable for investigating, e.g., changes in flow and performance due to changing gas composition, and, with loss correlations that take nonideal inlet angles into account, it should also be possible to obtain good part-load performance results. With M3, it is also possible to study changes in average vane or blade temperature due to changing coolant and/or hot gas composition. At the same time, the higher level of complexity of M3 makes it less general.

M3 Cooling Model. The cooling calculations in M3 are based on Eqs. (5) and (10)–(12). To keep the Stanton number at a constant value of 0.005 is not quite satisfactory when considering a working medium with a completely different composition than ordinary combustion gases, such as a $CO_2/H_2O/O_2$ mixture. Instead, the heat transfer coefficient α_g in Eq. (12) must be calculated. A value of the gas-side heat-transfer coefficient α_g can be estimated with help from the Nusselt number:

$$Nu_g = \frac{\alpha_g C}{\lambda_g} \quad (15)$$

where C is the blade chord. In the present paper, the Nusselt number for the stator vanes is calculated from the equation for heat transfer over a flat plate, as previously employed by, e.g., El-Masri and Pourkey [14]:

$$Nu = St Re Pr = 0.037 Re^{0.8} Pr^{1/3} \quad (16)$$

where the Reynolds number is based on the chord and the average velocity over the vane. For the rotating blades, an expression given by Halls [15] has been employed:

$$Nu = 0.235 Re^{0.64}. \quad (17)$$

The expression is clearly related to Eq. (16), but the value of the Pr number has been assumed to be constant and included in the parameter 0.235. Since variation in Pr number is very small between ordinary gas turbine combustion gases and the $CO_2/H_2O/O_2$ mixture found in an oxyfuel cycle, Eq. (17) was employed without any modification. Also, in this case, the Re number is based on the average velocity and the chord length.

Equation (13) is useful for conceptual studies where many details of the geometry are unknown. In a gas turbine with a known geometry, however, the flow path of the coolant from the compressor extraction until it is injected into the hot gas flow can be modeled with a series of throttlings and pipe flow units, where the final part of the flow path is the throttling that the coolant experiences as it leaves the blade or vane. This throttling can be described with the following equation:

$$\dot{m}_c = A_{co} p_{co}^* \sqrt{\frac{2\kappa}{\kappa-1} \frac{1}{RT_{co}^*} \left[\left(\frac{p_{co}}{p_{co}^*} \right)^{2/\kappa} - \left(\frac{p_{co}}{p_{co}^*} \right)^{\kappa+1/\kappa} \right]}. \quad (18)$$

This means that the coolant is considered to pass through the area A_c as it leaves the blade or vane. The total pressure p_c^* is the pressure directly upstream of A_c . The velocity v of the coolant as it leaves the blade or vane is calculated from

$$\dot{m}_c = \rho_c A_{co} v \quad (19)$$

where the density is the static density at the moment the coolant leaves the blade through the total effective coolant hole area. In the case of a film-cooled blade in M3, coolant is assumed to be ejected at the leading edge and at the blade trailing edge, but nowhere in between, although in reality the opposite might very well be the case. Like M2, uniform conditions are assumed for the blade or vane, meaning that T_c^* and p_c^* are equal at leading and trailing edges, whereas p_c equals the surrounding static pressure, which differs between leading and trailing edges, and that two separate values for A_c , one at the leading edge and one at the trailing edge, must be determined. In the case of a convectively cooled blade with all of the coolant ejected through the trailing edge, p_c equals the trailing-edge static pressure.

T_g in Eqs. (5) and (10) is for the vane assumed to be the total temperature at the vane inlet. For the rotating blades, it is more suitable to set T_g to equal the recovery temperature T_{rec} :

$$T_{rec} = T_2 + \frac{Pr^{1/3}}{2c_p} w_2^2. \quad (20)$$

M3 Expansion Model. An enthalpy-entropy diagram for the first stage with film-cooled blade and vane row is shown in Fig. 2. Starting from this model, simplifications and modifications can be made to describe a convectively cooled stage or an uncooled stage.

In thermodynamic models for cooled expansion, including M1 and M2, all of the flow that is mixed into the main flowpath is assumed to come from the cooling of blades and vanes, although, in real gas turbines a non-negligible part of the flows enter the flowpath between blade and vane rows after, e.g., cooling the disks. These flows are referred to as disc-cooling flows with index “ dc ” in the description below, but are not shown in Fig. 2.

Arbitrary Film-Cooled Stage. The notation for the film-cooled stage can be seen in the h - s -diagram in Fig. 2, and the velocity triangles in Fig. 3. Momentum losses and reductions in main flow temperature due to mixing and cooling are all expressed explicitly in the model, and other losses are considered

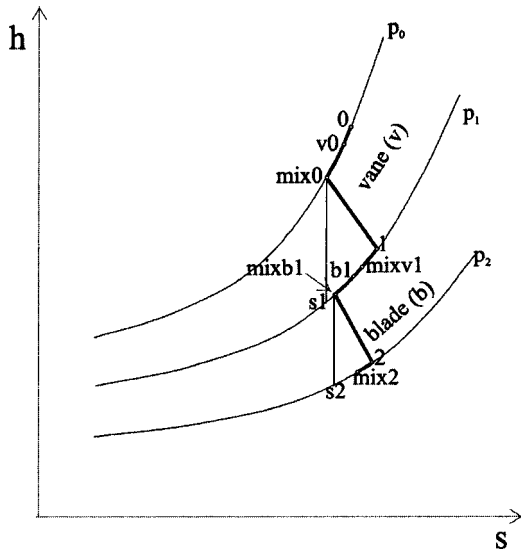


Fig. 2 Enthalpy-entropy diagram for the cooled stage model

through a blade row isentropic efficiency. Furthermore, as shown in Fig. 4, to determine the momentum losses that occur due to mixing of hot gas and coolant, it is assumed that the leading edge coolant, on average, has a direction that is perpendicular to the main flow, and that the flows are equally distributed in $+90^\circ$ and -90° (i.e., the coolant velocity in the flow direction is zero and it does not contribute to a change in flow direction) and that the

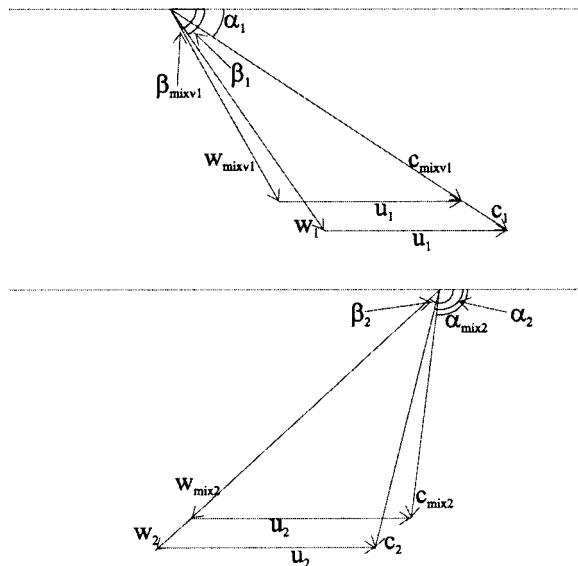


Fig. 3 Velocity triangles for vane (above) and blade (below) outlet flows before and after mixing with coolant

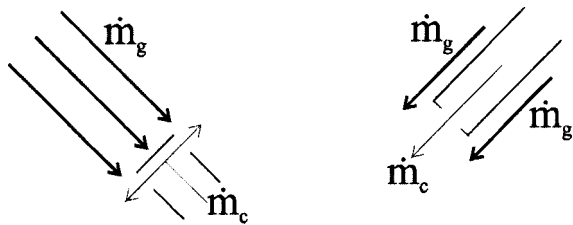


Fig. 4 Models for directions of coolant and hot gas at leading edge (left) and trailing edge (right)

trailing edge coolant-flow direction equals that of the main flow, i.e., it reduces the main flow velocity, but it does not change the flow direction.

Disk cooling flows are assumed to be mixed with the main flow at the points mix0, mixv1 and mix2, and are assumed to have negligible velocities. Heat extraction from the hot gas is assumed to occur before the mixing of leading edge coolant and main gas flow, from point 0 to point v0, although it is actually a continuous process throughout the cooled expansion path, a phenomenon that is captured in the interconnection between heat and work extraction in model M1.

The tangential speeds u_1 and u_2 are calculated at the mean radii R_{mv} and R_{mb} through

$$u_1 = \frac{n\pi}{30} R_{mv} \quad \text{and} \quad u_2 = \frac{n\pi}{30} R_{mb} \quad (21)$$

A mass balance upstream of the vane returns

$$\dot{m}_1 = \dot{m}_{mix0} = \dot{m}_{v0} + \dot{m}_{cv0} + \dot{m}_{dc0} \quad (22)$$

The corresponding energy balance is

$$\dot{m}_1 \left(h_{mix0} + \frac{c_{mix0}^2}{2} \right) = \dot{m}_0 \left(h_0 + \frac{c_0^2}{2} \right) + \dot{m}_{cv0} \left(h_{cv0} + \frac{v_{cv0}^2}{2} \right) + \dot{m}_{dc0} h_{dc0} \quad (23)$$

and, the momentum balance becomes

$$\dot{m}_0 c_0 + \dot{m}_{cv} \times 0 + \dot{m}_d \times 0 = \dot{m}_{mix0} c_{mix0} \quad (24)$$

The preservation of mass gives the absolute velocity c_1 out of the vane:

$$c_{a1} = c_1 \sin \alpha_1 = \frac{\dot{m}_1}{\rho_1 2 \pi R_{mv} H_v} \quad (25)$$

Referring to Fig. 2 for notations, the general expression that defines the use of the vane isentropic enthalpy efficiency is

$$\frac{c_1^2}{2} = \eta_v \left(\Delta h_{is,v} + \frac{c_{mix0}^2}{2} \right) \quad (26)$$

where it should be noted that the vane inlet velocity is included in the brackets for the isentropic blade row efficiency, as suggested by Cordes [16], and that this definition also means that Soderberg's loss model can be employed. For the first vane, c_{mix0} , in Eq. (26) is assumed to be zero, i.e., stagnation properties are assumed at points 0, v0 and mix0.

The static pressure p_1 at the vane outlet is found through

$$s_{mix0} = s_{1,is} = s(p_1, (h_{mix0} - \Delta h_{is,v})) \quad (27)$$

where, for the first stage, $h_{mix0} = h_{mix0}^*$.

The disk cooling flow between vane and blade is referred to the vane trailing edge, which will lead to slight simplifications in the case where only convective cooling is considered, since no heat, mass and momentum balances will be required in that case for the blade leading edge mixing. This means that the mass balance at the vane trailing edge will be

$$\dot{m}_1 + \dot{m}_{cv1} + \dot{m}_{dc1} = \dot{m}_{mixv1} = \dot{m}_{b1} \quad (28)$$

The energy balance at point mixv1 is

$$\begin{aligned} \dot{m}_1 \left(h_{mix0} + \frac{c_{mix0}^2}{2} \right) + \dot{m}_{cv1} \left(h_{cv1} + \frac{v_{cv1}^2}{2} \right) + \dot{m}_{dc1} h_{dc1}^* \\ = \dot{m}_{mixv1} \left(h_{mixv1} + \frac{c_{mixv1}^2}{2} \right) \end{aligned} \quad (29)$$

Provided that the losses caused by the disk cooling flow are taken into account by the loss model, the momentum balance at the vane trailing edge becomes

$$\dot{m}_1 c_1 + \dot{m}_{cv1} v_{cv1} = (\dot{m}_1 - \dot{m}_{dc1}) c_{mix1v}. \quad (30)$$

The continuity equation at the vane trailing edge returns

$$c_{mixv1} \sin \alpha_1 = \frac{\dot{m}_{mixv1} - \dot{m}_{dc1}}{\rho_{mixv1} \pi R_{mv} H_{mv}}. \quad (31)$$

Since c_1 and c_{mixv1} can be calculated through the above equations and since α_1 is assumed to remain unchanged after the mixing with the coolant flow, this means that w_1 and w_{mixv1} can now be calculated through trigonometry.

From point $mixv1$ to $b1$ (refer to Fig. 2), it is assumed that, without any change in velocities, heat is removed from the main flow due to the calculation of the cooled rotating blade, where the heat balance gives that the heat absorbed by the coolant as it passes through the blade is rejected by the main flow. Of course, this heat is added again (but at a lower temperature) to the main flow through mixing with the coolant at the blade leading and trailing edges. The heat balance for mixing at the blade leading edge becomes

$$\dot{m}_{b1} \left(h_{b1} + \frac{w_{mixv1}^2}{2} \right) + \dot{m}_{c1} \left(h_{cb1} + \frac{v_{c1}^2}{2} \right) = \dot{m}_{mixb1} \left(h_{mixb1} + \frac{w_{mixb1}^2}{2} \right) \quad (32)$$

and the momentum balance becomes

$$\dot{m}_{b1} w_{mixv1} + \dot{m}_{cb1} v_{cb1} = \dot{m}_{mixb1} w_{mixb1}. \quad (33)$$

The entropy at point $mixb1$ (after the mixing of leading edge coolant but before expansion) is determined by

$$s_{mixb1} = s(p_1, h_{mixb1}). \quad (34)$$

The blade enthalpy drop is expressed with

$$\frac{w_2^2}{2} = \eta_b \left(\Delta h_{bs} + \frac{w_{mixb1}^2}{2} \right). \quad (35)$$

The pressure p_2 after expansion over the blade is determined by

$$s_{mixb1} = s_{2s} = s(p_2, h_{mixb1} - \Delta h_{bs}). \quad (36)$$

In order to determine the energy balance at the blade trailing edge, it is necessary to explicitly calculate h_2 , which is done by

$$h_2 = h_{mixb1} - \left(\eta_b \Delta h_{bs} - \frac{w_{mixb1}^2}{2} (1 - \eta_b) \right). \quad (37)$$

The energy balance at the blade trailing edge is

$$\begin{aligned} \dot{m}_{mixb1} \left(h_2 + \frac{w_2^2}{2} \right) + \dot{m}_{cb2} \left(h_{cb2} + \frac{v_{cb2}^2}{2} \right) + \dot{m}_{dc2} h_{dc2} \\ = \dot{m}_{mix2} \left(h_{mix2} + \frac{w_{mix2}^2}{2} \right) \end{aligned} \quad (38)$$

and the momentum balance is

$$\dot{m}_{mixb1} w_2 + \dot{m}_{cb2} v_{cb2} = (\dot{m}_{mix2} - \dot{m}_{dc2}) w_{mix2}. \quad (39)$$

The velocity c_{mix2} will be set equal to the velocity c_0 into the subsequent stage and is calculated from trigonometry, with the condition that β_2 is unchanged by the mixing of blade trailing edge coolant and main flow. Also, the blade absolute velocity outlet angle α_{mix2} and the absolute velocity c_2 before trailing edge mixing are obtained through pure trigonometry. This means that the continuity equation at the blade outlet can be calculated from

$$c_{mix2} \sin \alpha_{mix2} = \frac{\dot{m}_{mix2} - \dot{m}_{dc2}}{\rho_{mix2} 2 \pi R_{mb} H_{mb}}. \quad (40)$$

Finally, the power output from the stage is determined by

$$P = \dot{m}_{mixb1} \left[\left(h_{mixb1} + \frac{c_{mixv1}^2}{2} \right) - \left(h_2 + \frac{c_2^2}{2} \right) \right]. \quad (41)$$

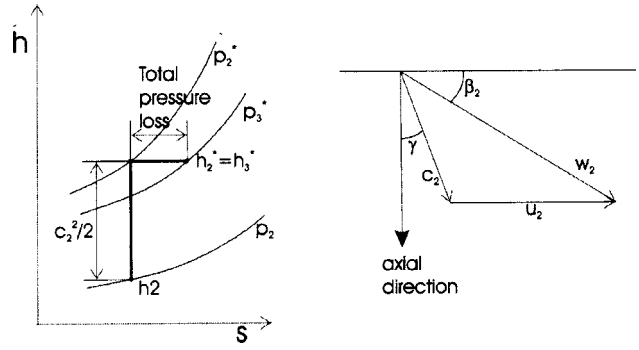


Fig. 5 Enthalpy-entropy diagram and definition of γ for the diffuser

With the expression in Eq. (41), it is assumed that the trailing edge coolant performs no work and that the impact of tip leakage on work extraction is taken into account through an inclusion of tip leakage losses in the loss model.

Adaptation of M3 to the Convectively Cooled Stage. In the case of a convectively cooled stage, it could be assumed that there is no mixing at the vane or blade leading edge, in order to simplify the model; i.e., it should be imagined that points $mix0$ and $v0$ coincide for the vane, and that points $b1$ and $mixb1$ coincide for the blade, and that the disk cooling flow is mixed at the vane or blade trailing edge. It is still assumed that heat is extracted from the main gas flow before the expansion over each blade row.

Adaptation of M3 to the Closed-Loop Cooled Stage. In the case of closed-loop cooling with no disk cooling flows, points 1 and $mixv1$ will coincide, as well as points 2 and $mixb2$, since there will be no mixing of coolant and main flow.

Adaptation of M3 to the Uncooled Stage. In the case of an uncooled stage with no disc cooling flows, points 0, $v0$ and $mix0$ will coincide, points 1, $mixv1$, $b1$, $mixb1$ will coincide and points 2 and $mixb2$ will coincide.

The Diffuser for Model M3. The diffuser unit describes how the gas turbine diffuser recovers the dynamic pressure from the final turbine stage. The diffuser cannot transform all of the dynamic pressure into static pressure, i.e., part of the total pressure is lost in this component without any work being extracted. The diffuser is characterized by the angle γ between the flow out of the final turbine stage and the axial direction, as shown in Fig. 5.

The pressure loss in the diffuser is an isenthalpic process, which means that

$$h_2 + \frac{c_2^2}{2} = h_3^*. \quad (42)$$

The isentropic condition helps to determine the total pressure p_2^* at the diffuser inlet.

$$s_2 = s_{is,3} = s(p_2, h_2) = s(p_2^*, h_3^*). \quad (43)$$

The diffuser outlet total pressure is determined by

$$p_3^* = p_2^* - (p_2^* - p_2) \cdot k_1(\gamma) \quad (44)$$

where $p_2^* - p_2$ is the dynamic pressure at the diffuser inlet, and $k_1(\gamma)$ is a tabulated function, which is specific to the diffuser studied.

Finally, based on the static conditions, a check is made of the Mach number at the diffuser inlet:

$$Ma = \frac{c_2}{\sqrt{\kappa R T_2}}. \quad (45)$$

This Mach number should not exceed 0.9 and must not exceed 1.0.

In models M1 and M2, no diffuser is included, instead, as is common practice in thermodynamic simulations, a total pressure drop (typically 2–5% or 0.02–0.05 MPa) is set at the turbine outlet.

Loss Correlations for M3. Different families of blades have different kinds of loss correlations, and it is virtually impossible to find one general set of equations that fully covers all types of blades, one example of this being the difference in tip clearance losses between shrouded and unshrouded blades. In an extensive review of loss correlations presented by Wei [17], it is noteworthy that existing correlations can be grouped in two families. Either the losses are expressed in terms of enthalpy losses (which is convenient for thermodynamic simulations) or in terms of total pressure losses (which is convenient for experimental work).

One method for describing enthalpy losses is the one generally known as Soderberg's model, which has been previously employed by, e.g., Hånde [18], who in turn refers back to Horlock [19]. It is a model that is easy to implement, and although it may be insufficient for blade design, it is supposed to give sufficiently accurate results when it comes to estimating turbine performance [20]. With Soderberg's model, the isentropic blade row efficiency employed in Eqs. (26) and (35) above is expressed as

$$\eta_s = \frac{1}{1 + \zeta} \quad (46)$$

The loss coefficient ζ is calculated from

$$\zeta = \left(\frac{\text{Re}_C}{10^5} \right)^{0.25} \left[(1 + \zeta^*) \left(a + b \frac{C}{H} \right) - 1 \right] \quad (47)$$

where $a = 0.993$ for rotors and 0.975 for stators and $b = 0.021$ for rotors and 0.075 for stators. The coefficient ζ^* expresses the loss due to blade or vane deflection $\Delta\beta$:

$$\zeta^* = d + e \left(\frac{\Delta\beta}{100} \right)^2 \quad (48)$$

Although always referred to as Soderberg's model, the coefficients d and e in Eq. (48) vary depending on the source. Wei [17], referring to Dixon [21], reports $d = 0.04$ and $e = 0.06$, whereas $d = 0.05$ and $e = 0.07$ are given in [18].

Soderberg's model covers profile and secondary flow loss. It is important, in the present context, that the model takes the effect of Reynolds number variation into account, which is of interest when the working medium changes. As stated by Kacker and Okapuu [22], tip clearance losses on unshrouded rotor blades constitute a major source of turbine inefficiency, and should therefore be included in a loss model, which is the case in the present paper. In addition to the losses described by Eqs. (47) and (48), equations for, e.g., trailing edge losses, film-cooling losses, secondary flow mixing losses and disc friction losses can be included in the loss correlation. Since the loss model reported in the present paper is incomplete, the values of the coefficients d and e from [18] will be employed in Eq. (48) to accentuate the losses that are actually implemented. It should be noted, though, that momentum loss due to coolant injection and heat extraction through the blade and vane are considered in M3 without being included in the actual loss model.

Model Implementations

M1 was implemented with an in-house Fortran code, whereas M2 and M3 were implemented in the commercial graphic programmable equation-solving software IPSEpro [23]. Each approach has its advantages—with a sequential Fortran code, the division of the expansion path into a large number of small segments is easy to implement, whereas it would require a tremendous effort to implement this in an equation solver such as IP-

Table 1 Data for reference gas turbine

	Reference	M1	M2	M3
P_{HPT} [MW]	29.0	29.1	28.8	29.2
P_{LPT} [MW]	23.2	23.7	23.2	23.7
$T_{\text{exh,LPT}}$ [K]	786.7	796.1	799.4	772.5
n_{HPT} [rpm]	9746	...	...	9055
ϕ [%]	19.0	19.9	19.0	19.0
$\eta_{\text{th,GT}}$ [%]	38.0	37.9	37.3	37.7

SEpro. On the other hand, in a real gas turbine, the pressure downstream of the diffuser is one of the conditions that determine the performance of the gas turbine, and with an equation solver, this pressure can easily be given as an input to the simulations. Furthermore, with the use of an object-oriented equation solver, there is no need to set up a solution algorithm for the entire gas turbine engine, since each object is programmed separately. Also, full flexibility is obtained concerning what variables should be input and what should be output, since this is determined at the time of each simulation and not during the programming.

All physical properties in M1, M2, and M3 were calculated with the JANAF Thermochemical Tables.

Cooled Gas Turbine Simulations

Design Point Results for M1, M2, and M3. Based on data estimated by Hånde [18] for a machine similar to an old version of the LM2500, (including estimated values of blade and vane geometry), design point simulations were made with M1, M2, and M3 as shown in Table 1.

Combustor outlet temperature is 1580 K, compressor pressure ratio is 19.1 and compressor inlet mass flow is 68.3 kg/s. It can be seen that the results obtained with the three different models vary slightly. For instance, too low exhaust temperature and too high power output are obtained with M3, due to the incomplete loss model.

Variations of PR and Temperature With M1 and M2. To compare the behavior of M1 and M2, combustor outlet temperature and compressor pressure ratio were varied, with both an air-breathing engine and an oxyfuel engine, the principle of the latter being shown in Fig. 6.

In the oxyfuel cycle, combustion of CH_4 in pure oxygen (5 molar percent excess) was simulated, after the diffuser (with maintained outlet pressure of 101.3 kPa) a heat sink to model the HRSG was applied, thereafter the $\text{CO}_2/\text{H}_2\text{O}/\text{O}_2$ mixture is cooled down to 288 K so that most of the water is condensed, part of the remaining gas mixture is extracted (for further processing and end disposal of CO_2) and the rest of the gas stream is recirculated into the compressor. Due to this simulation of the entire cycle, a realistic gas composition is ensured in the turbine.

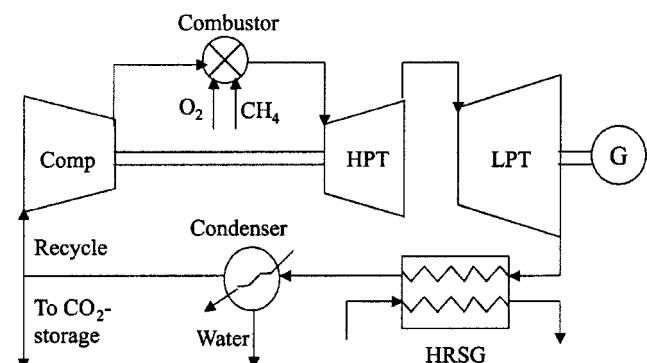


Fig. 6 Oxyfuel cycle flow sheet

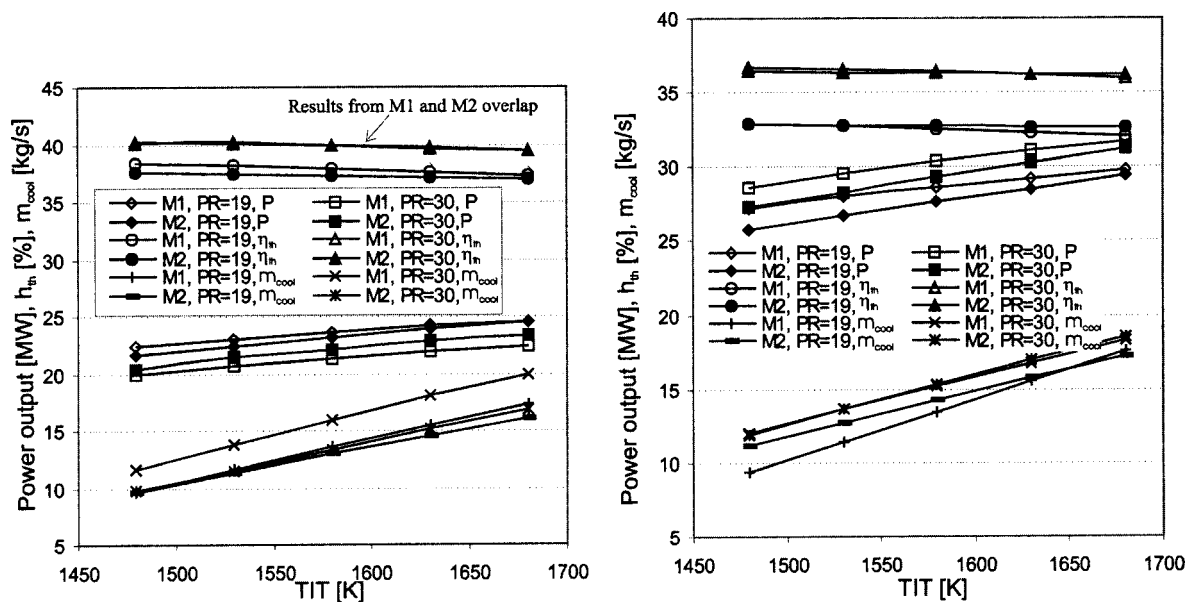


Fig. 7 Results from M1 and M2 for gas turbine operating with air (left) and $\text{CO}_2/\text{H}_2\text{O}/\text{O}_2$ mixture (right)

Values of St_g , A_b/A_g , η_c and T_b were maintained constant in all cases, as well as the compressor inlet mass flow. With help from M3, it was concluded that, for an engine with constant geometry (i.e., $A_b/A_g = \text{constant}$), St_g varies only with a couple of percentage points as the working medium changes from ordinary combustion gases to a $\text{CO}_2/\text{H}_2\text{O}/\text{O}_2$ mixture. This means that keeping St_g at 0.005, as the working medium changes in the turbine, is a much smaller approximation than the approximation that is already made when St_g is set constant independent of the temperature through the turbine.

Results with air for power output (P), thermal efficiency (η_{th}), and required coolant mass flow (m_{cool}) are shown in the diagram to the left in Fig. 7. It can be seen that, with increasing combustor outlet temperature, M1 and M2 show similar behavior, i.e., increasing power output and slightly decreasing thermal efficiency. This decrease is most likely due to the fact that the cooling is overestimated for high gas temperatures due to the constant value of the temperature of materials. The impact of varying pressure ratio differs slightly between M1 and M2, but the trend is the same, i.e., that the power output is decreased as the pressure changes from 19 to 30 bar. The same behavior was observed for M1 when simulating aeroderivatives in [7].

To simulate the oxyfuel case, no model parameters were altered from M1 or M2, only the gas composition changes. The compressor-inlet mass flow of 68.3 kg/s was maintained as a constant. The results can be seen in the diagram to the right in Fig. 7. It can be seen that, in terms of the thermal efficiency rating, the predictions are very similar for M1 and M2, whereas there is a somewhat larger discrepancy in the power output prediction. Generally, it can be observed that thermal efficiency decreases and power output increases as in the airbreathing 1 case. The reduced gas turbine thermal efficiency is due to the lower enthalpy drop in the oxyfuel case, but this also enables a steam bottoming cycle with a high live steam temperature, which, in present combined cycles, only can be achieved with supplementary firing. The higher power output with maintained mass flow is due to the lower power requirement for the compressor, since its enthalpy increase is lower for a given pressure ratio in the oxyfuel case.

As for the estimation of coolant mass flow requirements, the inherent division of M2 into a fixed number of stages, regardless of the pressure ratio, means that the surface that has to be cooled is underestimated, and thus the required coolant mass flow is underestimated, since A_b/A_g is kept constant.

Keeping the model parameters constant in these simulations includes ensuring that the turbine uncooled polytropic efficiency was maintained at 0.88. This, in turn implies an assumption that a new $\text{CO}_2/\text{O}_2/\text{H}_2\text{O}$ turbine with the same efficiency as the existing airbreathing turbine can be designed.

Investigation of Oxyfuel Operation With M3. As an example of the kind of results that it is possible to obtain with M3, simulations were made where the data employed to obtain the results in Table 1 were employed in an oxyfuel cycle. All geometry parameters of the cooled blade were maintained constant, as well as the cooling efficiency of 0.3. A sketch of the cycle is shown in Fig. 6.

Three different cases were investigated: in the first case, compressor pressure ratio and blade angles were maintained constant, in the second case the values of α_1 and β_2 were increased with 3 deg for the two stages in the power turbine, and in the third case, no geometry was altered, but the compressor pressure ratio was increased to 21. The results are shown in Table 2. The rotational speed of the low-pressure turbine is kept constant at 3600 rpm in all cases. Like the results shown in Table 1, the compressor model is, in all three cases, a simple model with no compressor map, where the polytropic efficiency is kept constant at 0.92, and where the required coolant is extracted at an appropriate pressure.

The first observation to be made from the results in Table 2 is that there is a substantial decrease in gas turbine thermal effi-

Table 2 Data for reference gas turbine in oxyfuel operation, modeled with M3

	Base Case	Angles+3 deg	pr _{cmp} =21
P_{HPT} [MW]	18.2	20.6	20.9
P_{LPT} [MW]	23.5	29.7	27.6
$T_{exh,HPT}$ [K]	1124.9	1166.8	1133.1
$T_{exh,LPT}$ [K]	934.6	943.9	921.9
n_{HP} [rpm]	12729	10575	11817
$\dot{m}_{cool}$ [kg/s]	71.8	80.3	78.5
ϕ [%]	21.9	20.4	21.67
$\eta_{th,GT}$ [%]	26.7	29.5	28.7
ΔT_{v1}	-105.4	-114.6	-107.6
ΔT_{b1}	-11.4	-12.1	-12.9
ΔT_{v2}	0.0	14.5	6.3
ΔT_{b2}	14.9	12.3	17.9

ciency when operating with a $\text{CO}_2/\text{H}_2\text{O}/\text{O}_2$. The exhaust gas temperature however is high enough to allow for advanced steam data in the bottoming cycle, which should partly compensate for this efficiency loss. It can also be seen that the power turbine output is hardly affected by the change of working medium, although the power increased with M1 and M2 when the uncooled polytropic efficiency was kept constant. This, in turn, indicates that the losses increase when the blade and vane angles are unchanged and the working medium changes. Furthermore, it can be seen in Table 2 that, with maintained compressor pressure ratio and turbine geometry, a substantial increase in gas generator rotational speed is required. Since κ is lower in this engine than in the airbreathing one, the enthalpy increase in a compression (to a given pressure) is less in the oxyfuel cycle than in an air compressor, but also the enthalpy drop of the turbine is lower. The resulting effect is that, although density is higher for the working medium in the oxyfuel cycle, it is not high enough for the turbine to be able to drive the compressor at a maintained rotational speed.

In order to increase the flow area of the turbine, the blade angles α_1 and β_2 can be increased, and as is shown for the second case in Table 2, an increase of 3 deg for the first two stages will give a certain reduction of the gas generator rotational speed. Opening up the turbine in this way means that the mass flow will increase, and hence, the power turbine output will increase, although the exhaust gas temperature also will increase. This increase in temperature level causes a higher temperature at the power turbine inlet, which might call for more expensive materials or even cooling of the first vane. In the case where the compressor pressure ratio is slightly increased (with maintained combustor relative pressure drop), the same trends as for increased blade angles can be observed, with one exception, which is that, with increased pressure ratio, the low pressure turbine outlet temperature decreases.

The deviation in calculated blade and vane average surface temperatures from the airbreathing case is also shown in Table 2. The substantial decrease in first vane temperature is mainly due to the large decrease in coolant temperature, since the temperature increase in a $\text{CO}_2/\text{H}_2\text{O}/\text{O}_2$ compressor is significantly lower than that of an air compressor with an equal compression ratio. One possible design change here would be to modify the geometry of the flowpath from compressor to turbine so that coolant consumption is reduced and the full potential of the vane material is employed. It should be noted here that the cooling efficiency in Eq. (5) was maintained constant in the calculations. The thermal conductivity of CO_2 is lower than that of air, however, and as is pointed out in [5], CO_2 does not have very good properties for convective cooling (but is well suited for film cooling), and it might be the case that the value of η_c should have been decreased for the oxyfuel cycle studies.

The low coolant temperature may cause increased materials temperature gradients, which should not be ignored. In all cases in Table 2, the blade and vane relative gas velocities are lower and trailing edge Mach numbers are fairly unchanged, compared to the airbreathing engine in Table 1.

Possible Future Model Development

Much of the published work on thermodynamic modeling of cooled gas turbines over the past two decades, including the three models M1, M2, and M3 can be traced back to Louis et al. [24], where they defined the phenomena to model in the cooled expansion as cooling (of the blade or vane, which leads to total temperature losses), expansion (with work extraction) and losses (due to cooling and aerodynamic phenomena). Losses are, in the work by Louis, defined in terms of total pressure losses, and much work on turbine losses is presented in terms of total pressure loss (generally denoted with Y), such as in [22]. It might be useful to modify the M3 loss model so that total pressure loss correlations can be employed. The relation between Y and ζ in Eq. (46) is, for compressible flow with $\text{Ma} < 1$ ([20], p. 64)

$$Y \cong \zeta \left(1 + \frac{\kappa \text{Ma}^2}{2} \right). \quad (49)$$

Here, the Mach number is based on c_1 for the stator and w_2 for the rotor. Still, even with this approach, it must be remembered, as pointed out in [3], that current knowledge for experimental and computational techniques relies on air as a working fluid, including loss models.

Noninteger number of stages [25] could be a solution for M2 or M3 for a future—more general—implementation that enables pressure ratio variations, although it was found with M2 that its behavior is quite similar to that of the continuous expansion in M1.

The introduction of dynamic behavior into the cooled gas turbine model, covering both rotor dynamics and unsteady heat transfer, could possibly be managed with the approach outlined in [18].

Conclusions

The modeling of a cooled turbine for operation in an oxyfuel cycle was described. Two models that require little geometry and only employ gas total conditions were reviewed, and a third, more detailed model based on velocity triangles was presented.

It was found that, although the two simple models are quite different in their way of dealing with expansion and losses, they show quite similar behavior with varying pressure and temperature, regardless of whether the working medium is ordinary combustion gases or a $\text{CO}_2/\text{H}_2\text{O}/\text{O}_2$ mixture. This means that a dynamic model could be based on either of the two concepts.

With the most detailed model, it was found that without any change of geometry, the turbine rotational speed increased significantly and the performance decreased if the pressure ratio is maintained. Tentative increase of blade angles or compressor pressure ratio was found to counteract this behavior, which indicates that a turbine will require design for operation in the oxyfuel cycle. For more specific conclusions concerning the design, a more detailed compressor model should be employed.

Concerning the future model for dynamic simulations, M3 is probably far too detailed, but it might be employed to verify the off-design behavior of a simple model, and it is a useful tool for studies on design requirements for new working media, provided that sufficient geometry input can be obtained. In the present paper, thanks to the use of M3, it can be verified that the value of the Stanton number need not be modified as the gas turbine working medium is shifted to a $\text{CO}_2/\text{H}_2\text{O}/\text{O}_2$ mixture.

Acknowledgments

Thanks are due to Demag Delaval Industrial Turbines for allowing Ake Klang to participate in the writing of this paper. The work was sponsored by the Norwegian Research Council, Klimatek program.

Nomenclature

A	= area (m^2)
B	= cooling parameter
C	= chord (m)
C_{M1}	= model M1 turbine parameter
G	= generator
H	= blade height (m)
HPT	= high pressure turbine
HRSG	= heat recovery steam generator
LPT	= low pressure turbine
Ma	= Mach number
Nu	= Nusselt number
P	= power (kW)
PR	= pressure ratio
Pr	= Prandtl number
Q	= heat (kW)

R = specific gas constant (kJ/kgK)
 R_m = average annulus radius (m)
 Re = Reynolds number
 St = Stanton number
 T = temperature (K)
 $\dot{W}$ = work (kW)
 Y = loss coefficient related to total pressure drop
 a, b, d, e = loss model coefficients
 c = absolute flow velocity (m/s)
 c_p = specific heat (kJ/kgK)
 h = enthalpy (kJ/kg)
 k_1 = diffuser loss function
 $\dot{m}$ = mass flow (kg/s)
 n = rotational speed (rpm)
 p = pressure (Pa)
 s = entropy (kJ/kgK)
 u = tangential velocity (m/s)
 v = coolant velocity (m/s)
 w = blade relative velocity (m/s)
 α = angle between blade absolute velocity and tangential direction
 α_g = convective heat transfer coefficient on hot gas side
 β = angle between blade relative velocity and tangential direction
 ε_c = cooling effectiveness
 γ = angle between flow into the diffuser and axial direction
 φ = total coolant to compressor inlet mass flow ratio
 η = efficiency
 κ = specific heat ratio, c_p/c_v
 λ = conductivity (W/mK)
 μ = dynamic viscosity (Pa s)
 ρ = density (m³/kg)
 s = relative heat-to-work loading in M1
 ζ = loss coefficient

Subscripts

0 = stage inlet (upstream of vane)
 1 = between vane and blade
 2 = downstream of blade
 3 = diffuser outlet
 a = axial
 amb = ambient
 b = blade
 c = cooling, coolant
 dc = disc cooling
 g = hot gas
 i = inlet
 m = metal
 mix = mixture
 o = outlet (for coolant leaving the blade/vane)
 p = polytropic
 s = isentropic
 rad = radiation
 rec = recovery
 rot = rotational
 t = turbine

v = vane
 x = turbine half-stage inlet point in M2
 y = turbine half-stage outlet point in M2

Superscript

* = gas total conditions

References

- [1] Bolland, O., and Mathieu, Ph., 1998, "Comparison of Two Removal Options in Combined Cycle Power Plants," *Energy Convers. Manage.*, **39**, pp. 1653–1663.
- [2] Bolland, O., and Saether, S., 1992, "New Concepts for Natural Gas Fired Power Plants Which Simplify the Recovery of Carbon Dioxide, in *Energy Conversion and Management*," *Energy Convers. Manage.*, **33**(5–8), pp. 467–475.
- [3] Ulizar, I., and Pilidis, P., 1997, "Design of a Semiclosed Gas Turbine With Carbon Dioxide-Argon as Working Fluid," ASME Paper No. 97-GT-125.
- [4] Jackson, A. J. B., Codeceira Neto, A., Whellens, M. W., and Audus, H., 2000, "Gas Turbine Performance Using Carbon Dioxide as Working Fluid in Closed Cycle Operation," ASME Paper No. 2000-GT-153.
- [5] Carcasci, C., Zecchi, S., and Oteri, G., 2002, "Comparison of Blade Cooling Performance Using Alternative Fluids," ASME Paper No. GT-2002-30551.
- [6] Ulfesnes, R., Jordal, K., Bolland, O., and Kvamsdal, H. M., 2003, "Modeling and Simulation of Transient Performance of the Semi-Closed O₂/CO₂ Gas Turbine Cycle for CO₂ Capture," ASME Paper No. GT-2003-38068.
- [7] Bolland, O., and Stadaas, J. F., 1995, "Comparative Evaluation of Combined Cycles and Gas Turbines With Water Injection, Steam Injection and Recuperation," ASME J. Eng. Gas Turbines Power, **117**.
- [8] Jordal, K., 2001, "Modeling and Performance of Gas Turbine Cycles With Various Means of Blade Cooling," Ph.D. thesis, Lund University, Sweden.
- [9] Traupel, W., 1988, *Termische Turbomaschinen*, 3rd Ed., Springer-Verlag, Berlin.
- [10] Holland, M. J., and Thake, T. F., 1980, "Rotor Blade Cooling in High Pressure Turbines," *J. Aircr.*, **17**, pp. 412–418.
- [11] El-Masri, M. A., 1986, "On Thermodynamics of Gas Turbine Cycles: Part 2—A Model for Expansion in Cooled Turbines," ASME J. Eng. Gas Turbines Power, **108**, pp. 151–159.
- [12] Louis, J. F., 1977, "Systematic Studies of Heat Transfer and Film Cooling Effectiveness," AGARD CP 2291.
- [13] Geipel, H., Keppel, W., and Weyer, H. B., 1998, "Verbundforschung zur Hochtemperatur-Gasturbine," VGB-Konferenz Forschung für die Kraftwerkstechnik.
- [14] El-Masri, M. A., and Pourkey, 1986, "Prediction of Cooling Flow Requirements for Advanced Utility Gas Turbines—Part 1: Analysis and Sealing of the Effectiveness Curve," ASME Paper No. 86-WA/HT-43.
- [15] Halls, 1969, "Air Cooling of Turbine Blades and Vanes," *Supersonic Turbojet Propulsion Systems and Components*, AGARDograph 120.
- [16] Cordes, G., 1963, *Strömungstechnik der Gasbeaufschlagten Axialturbine*, Springer-Verlag, Berlin.
- [17] Wei, N., 2000, "Significance of Loss Models in Aerothermodynamic Simulation for Axial Turbines," PhD thesis, Royal Institute of Technology, Sweden.
- [18] Hånde, B. M., 1992, "Simulation of Steam Injected Gas Turbines," PhD thesis, Norwegian Institute of Technology, University of Trondheim, Norway.
- [19] Horlock, J. H., 1960, "Losses and Efficiencies in Axial-Flow Turbines (Review)," *Int. Metall. Rev.*, **2**.
- [20] Horlock, J. H., 1966, *Axial Flow Turbine*, Butterworth, London, pp. 86–87.
- [21] Dixon, S. L., 1989, *Thermodynamics of Turbomachinery*, Pergamon Press, London.
- [22] Kacker, S. C., and Okapuu, U., 1981, "A Mean Line Prediction Method for Axial Flow Turbine Efficiency," ASME Paper No. 81-GT-58.
- [23] Anon., 1991–2000, *IPSEpro User Documentation*, Simtech Simulation Technology, Graz, Austria.
- [24] Louis, J. F., Hiaroka, K., and Elmasri, M. A., 1983, "A Comparative Study of the Influence of Different Means of Turbine Cooling on Gas Turbine Performance," ASME Paper No. 83-GT-180.
- [25] Coppinger, M., Cox, G., Hannis, J., and Cox, N., 2002, "Cycle Optimization Using an Advanced, Real Engine Performance Prediction Model," ASME Paper No. GT-2002-30515.

Loredana Magistri
Riccardo Bozzo
Paola Costamagna
Aristide F. Massardo

Thermochemical Power Group,
DIMSET DICHEP,
Università Degli Studi de Genova
Via Montallegro, 1,
I-16145 Genova, Italy

Simplified Versus Detailed Solid Oxide Fuel Cell Reactor Models and Influence on the Simulation of the Design Point Performance of Hybrid Systems

High-efficiency hybrid systems (HS) based on the coupling of solid oxide fuel cells (SOFCs) and gas turbines (GT) are analyzed in this paper through the use of two different approaches: simplified and detailed SOFC models. The simplified model, already presented by the authors, is very useful for HS design and off-design analysis. The detailed model, developed by the authors and verified through the use of available experimental data, allows the complete description of the SOFC reactor's internal behavior to be obtained. The detailed model can also be utilized for HS modeling. Both models are presented and discussed in this paper, and they are compared to each other. The results obtained for the stand-alone SOFC reactor, and the HS design point configuration are presented and carefully discussed, also taking into account the nonlinear SOFC response. [DOI: 10.1115/1.1719029]

Introduction

High-temperature fuel cell–gas turbine (FC/GT) hybrid systems have exceptional potential to improve both the local and global environmental impact of power generation. Their near-zero emissions and low noise will enable their use in populated areas, reducing the community's dependence on overhead electrical transmission and distribution systems and leading to improvements in the quality of life. Furthermore, their potential to generate electricity at a greater efficiency than conventional CHP plants even at relatively small scales, makes them economically attractive. The key point for the economic and technical development of FC/GT power generation plants is the possibility of predicting performances both at design and off-design conditions. Also a good knowledge of the system transients at critical operating points such as the system startup and shutdown helps to prevent breakdown or serious damage to the overall plant. As a result, simulation plays an important role in the design of these systems and, in particular, the simulation of the SOFC electrochemical reactor (which is a key component of the plant) is a critical aspect, which is under intensive study at present, [1–5]. Different types of simulation models have been proposed for solid oxide fuel cells (SOFCs), and they can be classified as (i) models based on the interpolation of experimental data of reactor behavior under different operating conditions, [6]; (ii) approaches based on balance equations in the form of macroscopic balances, i.e., finite equations. These equations simply express a balance between inlet and outlet flows of mass and energy, and allow the evaluation of the average values of the physical-chemical variables (i.e., temperatures, concentrations, etc.), and the electrochemical performance of the SOFC reactor as a function of the operating conditions, [7]; and (iii) modeling approaches based on balance equations written as local balances, [8,9] in the form of partial differential equations. These equations need numerical integration along the SOFC

geometrical co-ordinates, allowing a detailed evaluation of the distribution of the physical-chemical variables (temperature, gas compositions, electrical current density, etc.) within the SOFC.

Considering that the SOFC model is often included as a module in the simulation of a hybrid system, even if the simplified approaches are less accurate than the detailed ones, this loss of accuracy is sometimes acceptable within the framework of the overall plant evaluation. In those cases a significant reduction of computational time is achieved obtaining reasonably approximated results. On the other hand, it has to be said that simplified models do not allow a deep investigation of the status of the SOFC reactor, and there are several significant cases where critical issues, such as hot spots and/or steep temperature gradients, dangerous for the stability of the materials involved, can arise and cannot be detected through the simplified models. These issues, and in particular the limitations of the different types of simulation tool, are investigated in the present work, with particular reference to a pressurized tubular SOFC coupled to a recuperated micro gas turbine (MGT) with variable speed control. Critical aspects such as the lightoff and the extinction of the SOFC, and the sensitivity of the plant model to the accuracy of the simulation in this respect have been investigated both at the design point and at off-design conditions.

Simplified Versus Detailed SOFC Reactor Models

The cell geometry taken into consideration is tubular, with a design similar to the one proposed by Westinghouse, [10]; a simplified scheme of the cell and of some surrounding facilities, such as the pre-reformer, the pre-heater, etc., is reported in Fig. 1. We wish to point out that the dimensions of the SOFC under consideration are typical of the pre-commercialization phase currently in progress (for example, the cell length is 1.5 m). The cell is considered to be coupled to a pre-reforming unit, which feeds the SOFC with a mixture of H_2 , CH_4 , CO , CO_2 , H_2O . As a consequence, chemical reactions such as shifting and methane reforming occur in the anodic compartment of the SOFC. Two models have been developed and they are presented in the following: Since the equations of the simplified model are easily explained by considering them as a simplification of the detailed model, the latter model will be presented first.

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30653. Manuscript received by IGTI, Dec. 2001, final revision, Mar. 2002. Associate Editor: E. Benvenuti.

Detailed SOFC Model. Some models previously developed for tubular SOFCs, [8], were specific for experimental cells of small size, with a very particular type of interconnection geometry and operated within an oven. Since all these features strongly influence the structure of the simulation model and do not apply to the cells taken into consideration in the present study, a completely new simulation tool has been developed. Only the main aspects of the model are reported here, while a more complete description is given in another paper, [11], together with a discussion of the simulation results obtained. In particular, a comparison has been carried out between modeling results and literature experimental data, [12], with very satisfactory agreement.

Main hypotheses of the single cell model are:

- the cell is adiabatic;
- the cell voltage is uniform;
- all the chemical reactions within the anodic stream are at equilibrium;
- the electrochemical reaction of H_2 is taken into consideration; the electrochemical reaction of CO is neglected.

The model is one-dimensional; the cell coordinate x is the axis of the tube and its origin corresponds to the bottom of the cell. The cell model includes the following.

- Electrochemical performance: The current-voltage behavior of the stack is evaluated by subtracting from the thermodynamic potential (i.e., the voltage of the stack under conditions of thermodynamic reversibility, Table 1) the overall voltage losses (Eqs. (2)–(6)). In our model, we simulate both ohmic losses (Eqs. (2)–(4)) and activation losses, due to sluggish electrode kinetics (Eqs. (5)–(6)); concentration losses, occurring when the diffusion of the reactants through the electrodes is slower than the electrochemical reaction, have not been taken into consideration since diffusion is usually very efficient at the high SOFC operating temperatures. In this way, Eqs. (7)–(10) allow the evaluation of the electrical power supplied by the cell and of the electrochemical reaction rate r . We wish to underline that the evaluation of the electrical performance is made locally along the cell length: indeed, in order to integrate the model, the cell is divided into a number of grid points, and the evaluation of the electrical performance is repeated in each of them.

- Equilibrium of reforming and shifting chemical reactions: which is made through Eqs. (11)–(12); these equations allow the calculation of the reaction rates r of reforming and shifting.

- Mass balances of anodic and cathodic flows (Eq. 13): taking into account that molar flow rate variations are due to the reaction rates r of both chemical and electrochemical reactions.

- Energy balances of gaseous flows: including the anodic gas (Eq. (15)), the air flowing within the quartz tube (Eq. (16)) and the cathodic gas (air) flowing into the annular space between the quartz tube and the cathode (Eq. (17)). The equations include the convective exchanges due to mass transfer with the solid (only for the anodic and cathodic gas flow), and the heat exchanges between the gases and the solid surfaces (either the inner tube or the electrochemical cell itself). The evaluation of the heat exchange coefficient is made on the basis of the Nusselt number, assuming that the flow regimen is laminar.

- Energy balance of the tube (Eq. (18)) and of the solid PEN (positive(P)-electrolyte(E)-negative(N)) structure (Eq. (19)): including forced convection between the solid and gases, in-plane conduction and, in Eq. (19), the dissipative heat sources due to chemical and electrochemical reactions. The thermal balance of the solid results in a monodimensional Fourier problem with distributed heat sources.

Even if this model addresses a single cell, we can still use it to draw conclusions about the simulation of the overall stack to be included in the HS. However, the passage from the single cell to the stack size cannot simply be made by assuming that all the cells of the stack show behavior identical to the single cell operated under adiabatic conditions. Indeed, in the stack the effects of ra-

diation between contiguous cells cannot be neglected, which tend to smooth the temperature profile along each cell. As the details of the internal geometry of a tubular SOFC stack are not available for academic work, radiation effects have been considered by increasing the thermal conductivities of the SOFC materials (1000 times the real thermal conductivity $W m^{-1} K^{-1}$). However, the possibility of including a detailed simulation of radiation effects is under investigation.

The differential equations of the model have been integrated using a relaxation method for the Fourier problem and the charge balance, combined with a finite difference method to solve the other equations. Input data are geometrical characteristics, operating conditions, inlet flow conditions and gas and material properties: all the data have been assumed to be the same for both models, and are identical to the ones reported in a previous work, [7].

Simplified SOFC Model. In this case, the electrical current-voltage performance is evaluated following the same procedure described in the previous chapter, with the difference that in the present case the evaluation of the electrical performance is made only once, based on the average values of the physical-chemical variables. As far as the other equations are concerned, in the simplified model they are in the form of macroscopic balances, i.e., finite equations, in a form which is the integration of the differential equations between the inlet and outlet of the fuel cell. Indeed, these equations simply express a balance between inlet and outlet flows of mass and energy, and only allow the evaluation of the average values of the physical-chemical variables within the electrochemical reactor. Further details about the simplified model are given elsewhere, [7]; however, we wish to underline here that the detailed model gives the same results as the simplified one if the detailed one is modified in such a way that the thermal conductivity of the solid is set to infinity, as well as the heat exchange coefficient between the solid and the gases.

Like the detailed model, the simplified model is also integrated through a relaxation method for the evaluation of the cell performance at the design point. This kind of approach is also useful for the investigation of the system transients because of its temporal step conception that helps to trace the evolution of the system until steady-state conditions are reached.

In the next paragraph the results obtained using both models are presented and discussed for a stand-alone SOFC reactor.

Results for the SOFC Reactor in Stand-Alone Configuration

The comparison between model results has been carried out under the operating conditions reported in Table 2, which are also the operating conditions of the SOFC when operated in the HS (section on the Influence on hybrid system performance evaluation, Case C). The results are reported in Figs. 2–7; Fig. 2 shows the distribution of solid temperature, evaluated using both the detailed and the simplified models. The first observation is that the solid temperature evaluated from the detailed model continuously increases along the cell length, from the cell bottom (where the reactants enter the electrochemical part of the cell at low temperature, and where the endothermic reforming reaction occurs) to the top. This continuous increase of temperature is associated with heat dissipations due to the electrochemical reaction. The overall ΔT along the cell is about 380 K, which is a considerable value, and might cause high mechanical stresses in the ceramic structure; however, the cell length is considerable as well (1.5 m), and thus the maximum thermal gradient is not extremely high, i.e., about 3 K/cm. Figure 2 also shows that the solid temperature evaluated with the simplified model is in acceptable agreement with the average temperature evaluated with the detailed model, the difference being about 48 K.

The gas temperature distribution obtained from the detailed model is reported in Fig. 3. Figure 3 shows that the air entering the inlet quartz tube has a temperature which is almost constant along its path, while the gases flowing into the anodic and ca-

Table 1 Equations of the detailed and simplified SOFC models

DETAILED AND SIMPLIFIED		
Electrochemical Reaction	$V_{oc} = \frac{-\Delta G}{2F} = \frac{-\Delta G^0}{2F} + \frac{R_g T}{2F} \ln \frac{p_{H_2} \cdot (p_{O_2})^{1/2}}{p_{H_2O}} \quad (1)$	
	$\rho_{H_2} = A_a \exp\left(-\frac{B_a}{T}\right) \quad (2)$	$\rho_a = A_c \exp\left(-\frac{B_c}{T}\right) \quad (3) \quad \rho_c = A_s \exp\left(-\frac{B_s}{T}\right) \quad (4)$
	$\frac{1}{\Omega_{pol,a}} = D_1 \cdot \frac{2F}{R_g T} \cdot \left(\frac{p_{H_2}}{p^0}\right)^{n_1} e^{-(E_a/RT)} \quad (5)$	$\frac{1}{\Omega_{pol,c}} = D_2 \cdot \frac{4F}{R_g T} \cdot \left(\frac{p_{O_2}}{p^0}\right)^{n_2} e^{-(E_c/RT)} \quad (6)$
	$\Omega_{tot} = \sum \Omega_{ohm} + \sum \Omega_{pol} + \Omega_{con} \quad (7)$	$V_p = \Omega_{tot} I \quad (8) \quad V_p = V_{oc} - V \quad (9) \quad W_{el} = VI \quad (10)$
Equilibrium Chemical Reactions	$K_{p,ref} = \frac{p_{H_2}^3 p_{CO}}{p_{CH_4} p_{H_2O}} \quad \text{reforming} \quad (11)$	$K_{p,shift} = \frac{p_{CO_2} p_{H_2}}{p_{CO} p_{H_2O}} \quad \text{shifting} \quad (12)$
DETAILED MODEL		SIMPLIFIED MODEL
Mass Balance	$\frac{dN}{dx} = S \sum_k v_{i,k} r_k \quad (13)$	$N_i^{out} = N_i^{in} + S \sum_k v_{i,k} r_k \quad (14)$
Energy Balance	$\frac{1}{S_a} \left[\sum_i N_i \cdot c_{p,i} \cdot \frac{dT_a}{dx} + \sum_i c_{p,i} \cdot \frac{dN_i}{dx} (T_a - T_i) \right] + \frac{h_a}{b_a} (T_a - T_i) = 0 \quad (15)$ $\frac{1}{S_{ar}} \sum_i N_i \cdot c_{p,i} \cdot \frac{dT_{ar}}{dx} + \frac{h_{ar-nube}}{b_{ar}} (T_{ar} - T_{nube}) = 0 \quad (16)$ $\frac{1}{S_c} \left[\sum_i N_i \cdot c_{p,i} \cdot \frac{dT_c}{dx} + \sum_i c_{p,i} \cdot \frac{dN_i}{dx} (T_c - T_i) \right] + \frac{h_c}{b_c} (T_c - T_i) + \frac{h_{c-nube}}{b_c} (T_c - T_{nube}) = 0 \quad (17)$ $\frac{1}{S_{nube}} \cdot [h_{ar-nube} (T_{nube} - T_{ar}) + h_{c-nube} (T_{nube} - T_c)] - k_{nube} \frac{dT_{nube}}{dx^2} = 0 \quad (18)$ $\frac{1}{S_s} \cdot [h_s (T_s - T_a) + h_c (T_s - T_c) + Vi + \frac{1}{\mu d} \sum_i H_i \frac{dF_i}{dx}] - k_s \frac{dT_s}{dx^2} = 0 \quad (19)$	$T^{out} = \frac{\sum_i N_i^{out} C_{p,i} T^{in} + S \sum_k r_k (-\Delta H_k) - W_d}{\sum_i N_i^{out} C_{p,i}} \quad (20)$

thodic compartments of the cell have a temperature which tends to approach the solid temperature at the cell outlet (see also Fig. 2). Indeed, in the simplified model, the gases are hypothesized to leave the cell at exactly the same temperature of the solid, while in the detailed model the heat exchange between the gases and solid is carefully evaluated through specific equations, including an accurate evaluation of the heat exchange coefficient between gases and solid, which leads to the result that the gases leave the cell (right-hand side of Fig. 3 representing the top of the tubular fuel cell) with a temperature which is about 110 K lower than the solid temperature.

The detailed simulation of the anodic gas compositions along the cell is depicted in Fig. 4, showing that methane is completely consumed in the first part of the cell (left hand side of the plot), due to the fact that at the high cell temperature the equilibrium of the reforming reaction is completely shifted towards the reaction product, leading to a complete consumption of methane and to significant hydrogen production and water consumption.

Subsequently, hydrogen and water being a reactant and a product of the electrochemical reaction respectively, this trend is reversed

and hydrogen tends to decrease and water to increase towards the top of the cell (right-hand side of Fig. 4). The reversible voltage, plotted in Fig. 5, quite closely follows the trend of the hydrogen concentration. It increases as long as hydrogen is produced by the reforming reaction, whereas it starts to decrease at a quite constant slope when the hydrogen decreases.

The electrochemical results reported in the following can easily be explained considering that the cell voltage is constant (0.61 in the case study), and that the current density is obtained through Ohm's law (Eq. (8)) at any point along the cell length. However, we must keep in mind that the cell resistance is not constant along the cell length, since it is an exponential function of temperature; the distribution of the resistances along the SOFC is shown in Fig. 6. As a result, the distribution of electrical current along the cell (Fig. 7) shows a minimum at the bottom of the cell, where the electrical resistances are very high due to the low temperature, and another minimum at the top of the cell, where the reversible Nernst potential is low due to the reactant consumption. In the middle, the current density displays a maximum at a distance of about 1/3 of the cell length from the top of the cell itself.

The average current density obtained from the simplified simulation, which is about 4790 A/m² versus 4350 A/m² obtained as the average value of the detailed cell simulation, is reported as well. The cell electrical current and temperature obtained from the simplified model are both higher than the average values obtained from the detailed simulation. This slight discrepancy between the models is due to the nonlinearity of the equations involved; in any case the error made in using the simplified model instead of the detailed one is fully acceptable in view of its high execution speed and the simplicity of setting it up compared to the latter.

Nevertheless, we must also point out that there are cases where the simplified and the detailed model give very different results. This situation can be well explained by analyzing the results shown in Fig. 8. This figure reports a parametric study of the average solid temperature obtained by varying the inlet temperature of the anodic and cathodic gases entering the cell (the same value has been assumed for both gases). While keeping the gas

Table 2 Operating conditions for both the detailed and the simplified cell models

Cell voltage		0.61 [V]
Inlet fuel flowrate and temperature		0.079 [g/s], 606 [°C]
Inlet air flowrate and temperature		0.33 [g/s], 829 [°C]
Inlet fuel composition	H ₂	0.264
	H ₂ O	0.339
	CH ₄	0.095
	CO	0.060
	CO ₂	0.242
Initial PEN temperature		1027 [°C]
Gases pressure		3.84·10 ⁵ [Pa]

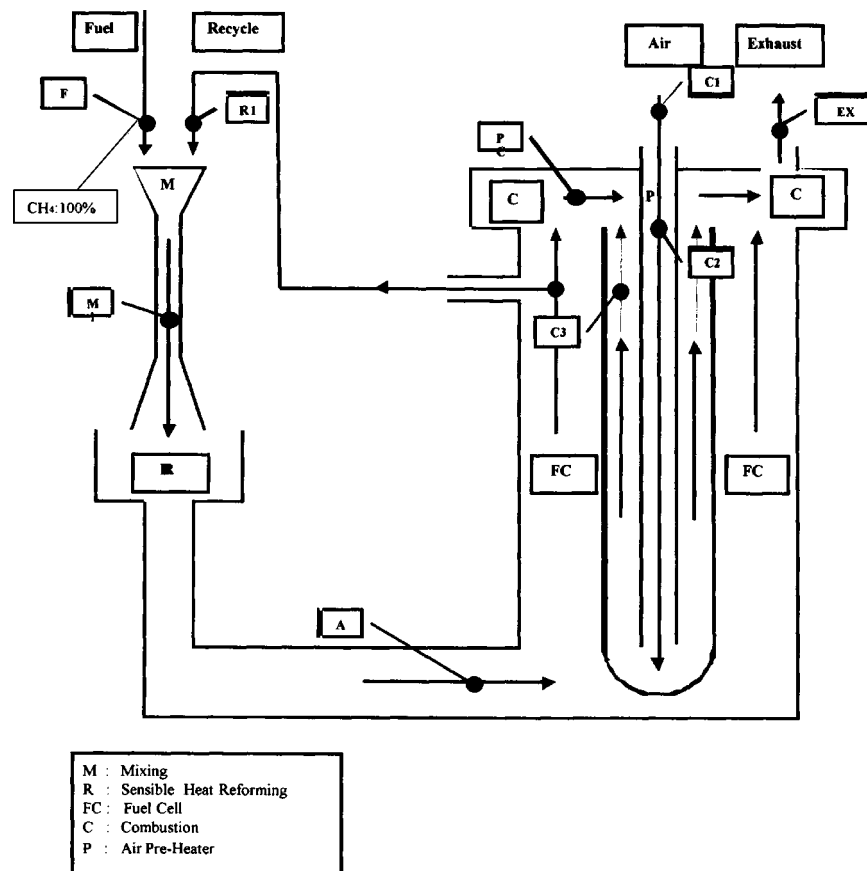


Fig. 1 Scheme of the SOFC group, including the SOFC stack, the mixer and the sensible heat reformer

flow rate and all the other operating condition as constants, the same calculations have been repeated twice starting with two different initial temperatures of the solid, namely 1500°C and 700°C. In turn, this has been repeated with both models, in order to investigate the tendency of each to reveal steady-state multiplicity. In the case where the detailed model is considered if the inlet gas temperature is lower than 770°C the reactor is stable in a “cold” situation where no current is supplied whereas if it is higher than 870°C a “hot” steady state with high conversion of the reactants

(i.e., high electrical current) is reached. Moreover the solid temperature at the startup doesn't affect the stable condition reached. Instead, the simplified model shows a multiplicity of steady states; indeed the curves in Fig. 8 show a typical hysteresis loop. If the feeding gases are at a temperature between 600°C and 860°C the reactor can work either in a cold-low current stable steady state (lower lines of the diagrams) or in a hot-high current stable steady state (upper lines of the diagrams), depending on the solid temperature at the startup.

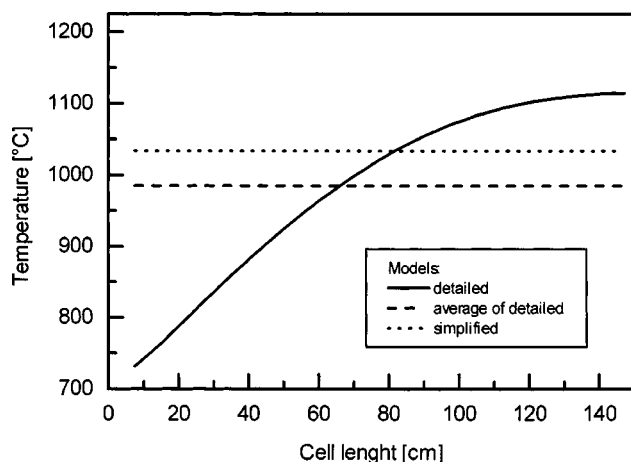


Fig. 2 Distribution of temperature in the solid structure of the SOFC cell, evaluated through both the detailed and the simplified model

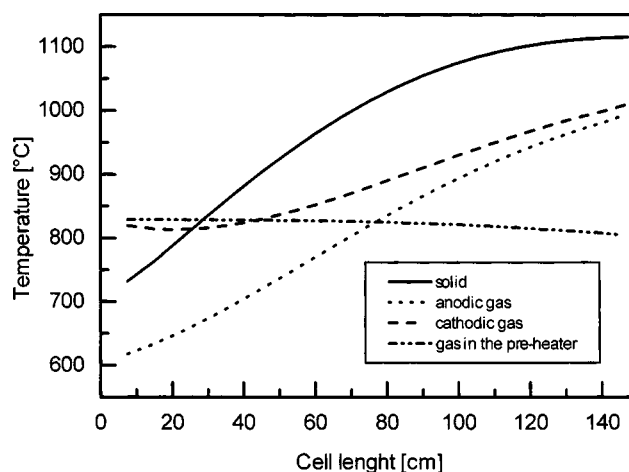


Fig. 3 Distribution of gases temperatures along the cell length

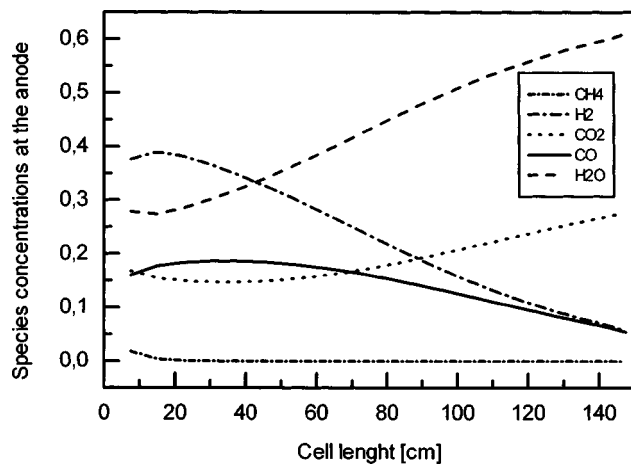


Fig. 4 Distribution of flow composition along the SOFC (results from the detailed model)

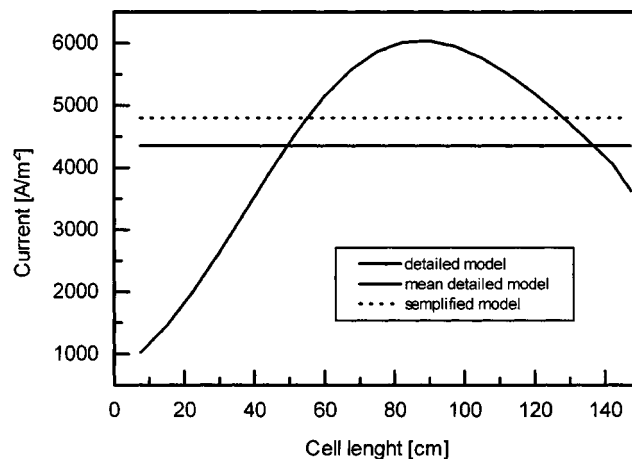


Fig. 7 Distribution of electrical current along the SOFC cell, evaluated through both the detailed and the simplified model

Influence on Hybrid System Performance Evaluation

Solid oxide fuel cells (SOFCs) integrated with gas turbines of small size (less than 1 MW) are attracting wide interest as these systems are able to simultaneously solve some key problems of

small gas turbines (low efficiency and NO_x emissions due to the combustor) and SOFCs (high cost, predicted to be around 1000–1500 \$/kW). In the HS configuration the pressurized SOFC module substitutes the combustor of the regenerated MGT, and the clean effluent at a temperature of about 900°C, which matches well the requirements on the inlet temperature of the microturbines, enters the expander to generate both electrical and mechanical power for the compressor.

Plants employing pressurized SOFC modules are considered the most interesting option, since this configuration allows very high efficiency to be achieved, even for systems of small size.

Figure 9 shows a simplified layout of the HS. The plant design and the size of all the components have been chosen on the basis of indications from the permanent literature on combined SOFC/MGT plants; the MGT has a size of about 45 kW_e, and the SOFC gives a power of about 220 kW at its design point. The hybrid plant has been simulated using a combination of the MGT model, developed by the authors and described in a previous paper, [13], with the SOFC models, both simplified and detailed, previously mentioned. The following design point calculations show how discriminating the accuracy of the simulation is in some cases.

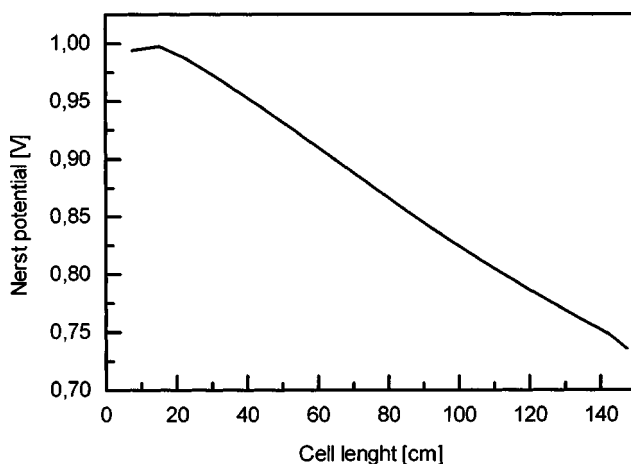


Fig. 5 Distribution of thermodynamic voltage along the SOFC (detailed simulation)

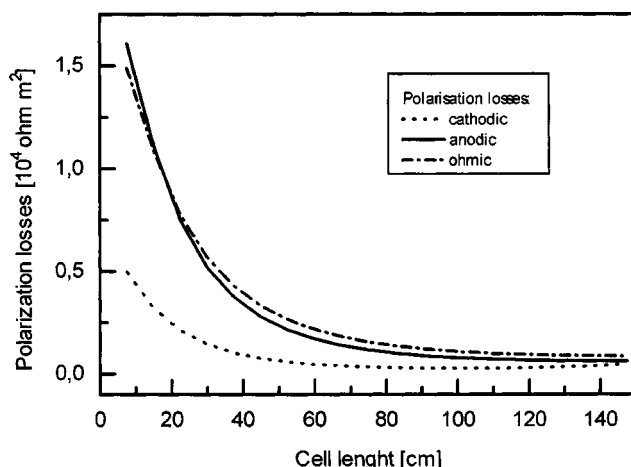


Fig. 6 Distribution of electrical resistances along the SOFC (detailed simulation)

Case Where Different Plant Results are Obtained With the Simplified and the Detailed SOFC Simulation

(1) *Case A (Table 3).* In this case, the average simulation results obtained for the fuel cell and for the whole plant do not appear very different if obtained with the detailed or the simplified SOFC model; at a first glance, the results look reasonable and compatible with the technological limits of the components of the hybrid system. But a careful analysis of the results obtained from the SOFC detailed model reveals that, as shown in Fig. 10, the temperature inside the stack is not uniform and, although the average value is acceptable, the maximum values are too high. These temperature peaks could lead to deterioration of the cell components (especially the anode, where Ni is present) or even breakages within the stack. With the simplified model, that estimates only the average value of the stack temperature, it is not possible to have information about these temperature peaks, and thus the plant design might lead to the choice of design point conditions dangerous for the SOFC group operation.

(2) *Case B (Table 4).* The table shows another case where the accuracy of the simulation is very important. The results have been obtained with the simplified SOFC model and show an operating point for the plant. The fuel flow rate and the SOFC voltage are different from the values reported in Case A, and consequently the operating temperatures are different as well. However,

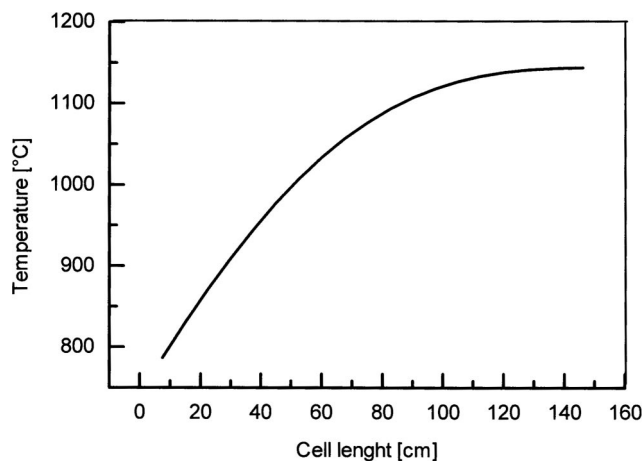


Fig. 10 Mean solid temperature distribution along the cell length

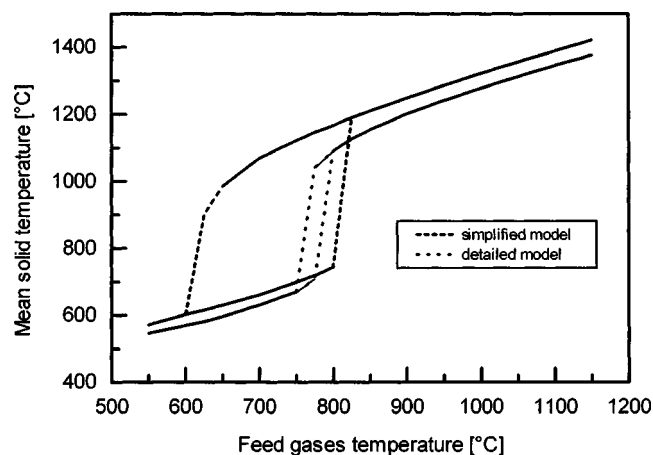


Fig. 8 Mean solid temperature at steady-state conditions for different inlet gas temperatures and initial PEN temperature

this design point also doesn't represent the real operating conditions of the plant: indeed, if the hybrid system analysis is carried out through the detailed model, it is found that in these conditions the fuel cell operates at very low temperature and no power is produced. This situation is easily explained considering Fig. 8: in this case, the cathode inlet temperature is in the operating range

where the cell simulated through the detailed model shows a very low temperature, while the cell simulated through the simplified model can display either low or high temperature operation, depending on the starting temperature used for the numerical integration procedure.

Thus, also in this case the use of the simplified model can lead

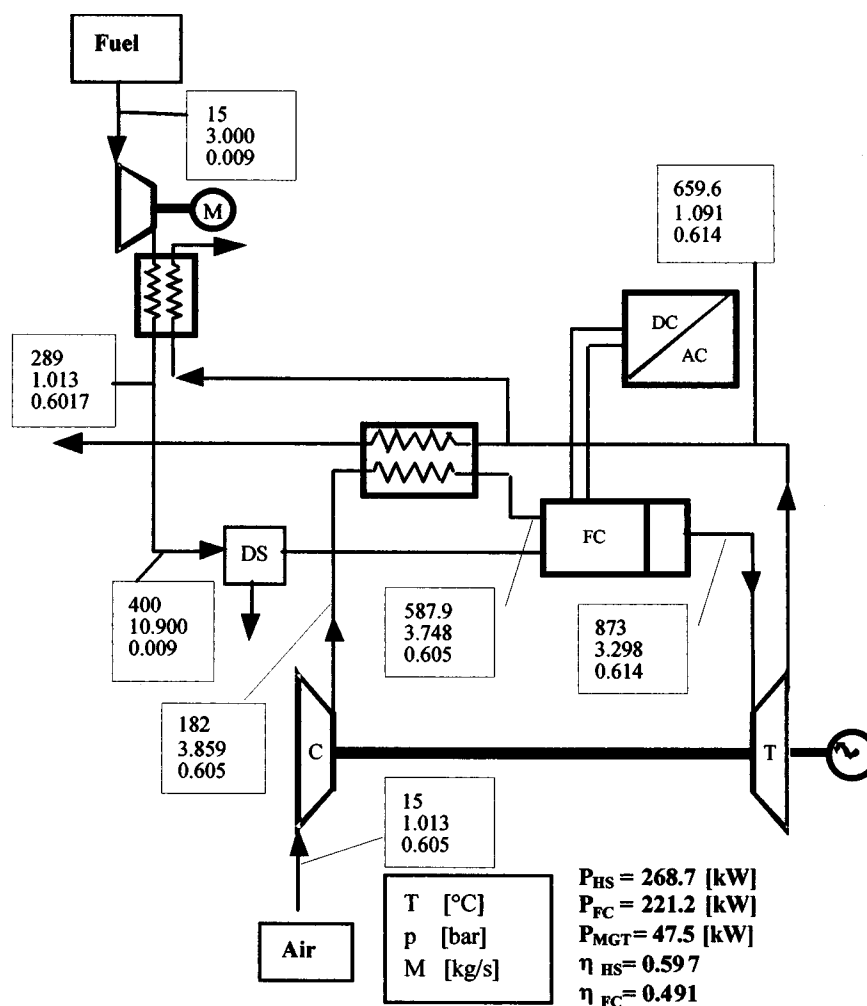


Fig. 9 Simplified layout of the hybrid plant and design point data

Table 3 Case A

	Simplified SOFC Model	Detailed SOFC Model
Air Flow Rate	0,605 [kg/s]	0,605 [kg/s]
Fuel Flow Rate	0,0098 [kg/s]	0,0098 [kg/s]
Compressor Pressure Ratio	3,81	3,81
Recuperator		
Effectiveness	0,85	0,85
Air Inlet Temperature	182 [°C]	182 [°C]
Air Outlet Temperature	589,2 [°C]	612,2 [°C]
Exhaust Inlet Temperature	661 [°C]	688,1 [°C]
Exhaust Outlet Temperature	290,6 [°C]	298 [°C]
Fuel Cell		
Cell Voltage	0,61 [V]	0,61 [V]
Average Electr. Current Density	476,18 [mA/cm ²]	464,87 [mA/cm ²]
An. Inlet Comp: H ₂ - H ₂ O - CH ₄	0,2437 0,3579 0,09762	0,2563 0,3461 0,09641
CO - CO ₂ (Molar)	0,05207 0,2487	0,05768 0,2435
Cathode Gas Inlet Temperature	804 [°C]	836 [°C]
Cathode Gas Outlet Temperature	1028,6 [°C]	1054 [°C]
Anode Gas Inlet Temperature	597,8 [°C]	606,7 [°C]
Anode Gas Outlet Temperature	1028,6 [°C]	1027,8 [°C]
Maximum Stack Temperature	1028,6 [°C]	1144 [°C]
Average Stack Temperature	1028,6 [°C]	1027 [°C]
Fuel Cell Efficiency	0,5068	0,4948
Fuel Cell Power	248,4 [kW]	242,5 [kW]
System		
Turbine Inlet Temperature	874 [°C]	908 [°C]
HS Efficiency	0,604	0,601
HS Power	296,15 [kW]	294,87 [kW]

to the determination of a plant operating point that is completely unrealistic.

Another important plant consideration that can be derived from the analysis of Fig. 8 is that the simulation results of the detailed SOFC model show a sharp transition between lightoff and extinction of the SOFC. This might have serious consequences in terms of plant operation since a change in the operating conditions of the HS could easily lead to a shutdown of the SOFC that might be very dangerous for the components downstream from the SOFC group. For example, if the main part feeding methane is not con-

Table 4 Case B

	Simplified SOFC Model
Air Flow Rate	0,605 [kg/s]
Fuel Flow Rate	0,009 [kg/s]
Compressor Pressure Ratio	3,81
Recuperator	
Effectiveness	0,85
Air Inlet Temperature	182 [°C]
Air Outlet Temperature	572,5 [°C]
Exhaust Inlet Temperature	641 [°C]
Exhaust Outlet Temp.	284 [°C]
Fuel Cell	
Cell Voltage	0,62 [V]
Av. Electr. Current Density	423,1 [mA/cm ²]
Cathode Gas Inlet Temp.	782 [°C]
Cathode Gas Outlet Temp.	979 [°C]
Anode Gas Inlet Temp.	605 [°C]
Anode Gas Outlet Temp.	979 [°C]
Average Stack Temp.	979 [°C]
Fuel Cell Efficiency	0,4984
Fuel Cell Power	224,3 [kW]
System	
Turbine Inlet Temperature	851 [°C]
HS Efficiency	0,5972
HS Power	224,3 [kW]

sumed in the SOFC, it burns in the combustion chamber causing a very high turbine inlet temperature, with serious damage for this component and consequently for the regenerator. Thus, the plant control system must be carefully designed in order to avoid mal-functionings of the systems.

Case Where Both the Simplified and the Detailed SOFC Simulation Lead to Similar Plant Performance. (Case C, Table 6). Apart from the problems illustrated in the previous sections, there is a wide number of cases where both the simplified and the detailed SOFC models lead to similar results. In particular, the case reported in the table above, the operating conditions (i.e., air and fuel flow rates) are equal to those reported in Case B, but the

Table 5 Case C

	Simplified SOFC Model	Detailed SOFC Model
Air Flow Rate	0,605 [kg/s]	0,605 [kg/s]
Fuel Flow Rate	0,009 [kg/s]	0,009 [kg/s]
Compressor Pressure Ratio	3,81	3,81
Recuperator		
Effectiveness	0,85	0,85
Air Inlet Temperature	182 [°C]	182 [°C]
Air Outlet Temperature	568,2 [°C]	587,9 [°C]
Exhaust Inlet Temperature	636,3 [°C]	659,6 [°C]
Exhaust Outlet Temperature	282 [°C]	289 [°C]
Fuel Cell		
Cell Voltage	0,61 [V]	0,61 [V]
Average Electr. Current Density	437,05 [mA/cm ²]	424,1 [mA/cm ²]
An. Inlet Comp: H ₂ - H ₂ O - CH ₄	0,2462 0,3559 0,09683	0,2646 0,3386 0,09524
CO - CO ₂ (Molar)	0,05219 0,2489	0,06036 0,2412
Cathode Gas Inlet Temp.	776 [°C]	803 [°C]
Cathode Gas Outlet Temp.	995 [°C]	1007,19 [°C]
Anode Gas Inlet Temp.	595 [°C]	607 [°C]
Anode Gas Outlet Temp.	995 [°C]	999,01 [°C]
Maximum Stack Temp.	995 [°C]	1107 [°C]
Average Stack Temp.	995 [°C]	971,85 [°C]
Fuel Cell Efficiency	0,506	0,4915
Fuel Cell Power	227,9 [kW]	221,2 [kW]
System		
Turbine Inlet Temperature	844 [°C]	873 [°C]
HS Efficiency	0,603	0,597
HS Power	271,5 [kW]	268,7 [kW]

SOFC voltage is lower. In this conditions the operating temperatures calculated through both the detailed and the simplified models are quite high and very similar to each other; all the other average results (power, etc.) obtained through the two models are also very similar to each other. Furthermore, the difference between the calculations with the two models are slight also in terms of HS efficiency and total power produced. (See Tables 3–5.)

Conclusions

The hybrid plant has been simulated using a combination of the MGT model, developed by the Authors and described in a previous paper, [7], with the SOFC models, both simplified and detailed, previously mentioned. The use of two different SOFC models is very important in order to investigate how the study of the HS is affected by the approximations of the SOFC models. Indeed, detailed fuel cell models require a long computational time and knowledge about geometrical data, materials and lay out of the fuel cell, that are rarely available. For these reasons, simplified fuel cell models are generally used in HS simulations, but in this way critical aspect such as the lightoff and the extinction of the SOFC cannot be investigated, and the calculation could give results very different from the real performance of the fuel cell and whole system. On the other hand, there are several cases where the results of the two types of simulation coincide, and it is difficult to have information a priori on the range of operating conditions where this occurs. Another interesting aspect is that cells with different geometries, [14], might show different behavior in this respect, and this will be the subject of further studies.

Nomenclature

A, B	= coefficients in Eqs. (2)–(4) (ohm m, K)
B	= hydraulic diameter (m)
C_p	= molar specific heat ($\text{J mol}^{-1} \text{K}^{-1}$)
D_1, D_2	= coefficients (A)
d	= cell diameter (m)
E	= activation energy (J mol^{-1})
F	= Faraday constant (C mol^{-1})
ΔG	= Gibbs free energy change (J mol^{-1})
ΔG°	= standard Gibbs free energy change (J mol^{-1})
H	= enthalpy (J mol^{-1})
h	= heat transfer coefficient ($\text{W m}^{-2} \text{K}^{-1}$)
I	= electrical current (A)
l	= cell length (m)
N	= molar flow rate (mol s^{-1})
$K_{p,\text{ref}}, K_{p,\text{shift}}$	= equilibrium constants ($\text{Pa}^2, -$)
k	= thermal conductivity of the solid ($\text{W m}^{-1} \text{K}^{-1}$)
p	= partial pressure (Pa)
p°	= reference pressure (Pa)
P	= overall pressure (Pa)
r	= reaction rate ($\text{mol m}^{-3} \text{s}^{-1}$)
R_g	= gas constant ($\text{J mol}^{-1} \text{K}^{-1}$)
S	= cross section (m^2)
s	= thickness (m)

T	= temperature (K)
V_{oc}	= thermodynamic electrical potential (V)
V_p	= loss of electrical potential (V)
V	= electrical potential of stack (V)
W_{el}	= electrical power (W)
x	= cell longitudinal coordinate

Greek

ν	= stoichiometric coefficient
Ω	= electrical resistance (ohm)
ρ	= electrical resistivity (ohm^{-1})

Subscripts

a, c, el	= anode, cathode, electrolyte
air	= air in the quartz tube
i	= chemical component
k	= chemical reaction
pol	= polarization
s	= solid structure
tube	= quartz tube
ohm	= ohmic

References

- [1] Iwata, M., Hikosaka, T., Morita, M., Iwanari, T., Ito, K., Onda, K., Esaki, Y., Sakaki, Y., and Nagata, S., 2000, "Performance Analysis of Planar-Type Unit SOFC Considering Current and Temperature Distributions," *Solid State Ionics*, **132**, p. 297.
- [2] Hall, D. J., and Colclaser, R. G., 1999, "Transient Modeling and Simulation of a Tubular Solid Oxide Fuel Cell," *IEEE T. Energy Convers.*, **14**, p. 749.
- [3] Bessette, N. F., and Wepfer, W. J., 1996, "Electrochemical and Thermal Simulation of a Solid Oxide Fuel Cell," *Chem. Eng. Commun.*, **147**, p. 1.
- [4] Ferguson, J. R., Fiard, J. M., and Herbin, R., 1996, "Three-Dimensional Numerical Simulation for Various Geometries of Solid Oxide Fuel Cells," *J. Power Sources*, **58**, p. 109.
- [5] Achenbach, E., 1994, "3-Dimensional and Time-Dependent Simulation of a Planar Solid Oxide Fuel-Cell Stack," *J. Power Sources*, **49**, p. 333.
- [6] Campanari, S., 1998, "Power Plants Based on Solid Oxide Fuel Cells Combined With Gas Turbine Cycles," Ph.D. thesis (in Italian), Politecnico di Milano.
- [7] Costamagna, P., Magistri, L., and Massardo, A. F., 2001, "Design and Part-Load Performance of a Hybrid System Based on a Solid Oxide Fuel Cell Reactor and a Micro Gas Turbine," *J. Power Sources*, **96**, p. 352.
- [8] Costamagna, P., Arato, E., Antonucci, P. L., and Antonucci, V., 1996, "Partial Oxidation of CH₄ in Solid Oxide Fuel Cells: Simulation Model of the Electrochemical Reactor and Experimental Validation," *Chem. Eng. Sci.*, **51**, p. 3013.
- [9] Costamagna, P., and Honegger, K., 1998, "Modeling of Solid Oxide Heat Exchanger Integrated Stacks and Simulation at High Fuel Utilization," *J. Electrochem. Soc.*, **145**, p. 3995.
- [10] Singhal, S. C., 2000, "Advances in Solid Oxide Fuel Cell Technology," *Solid State Ionics*, **135**, p. 305.
- [11] Costamagna, P., Bozzo, R., Magistri, L., and Massardo, A. F., 2004, manuscript in preparation.
- [12] Singhal, S. C., 1997, Proc. 5th Int. Symposium on SOFC, "Recent Progress in Tubular Solid Oxide Fuel Cell Technology," *Electrochemical Proceedings*, Volume 97–18, Aachen, June, p. 37.
- [13] Massardo, A. F., and Lubelli, F., 2000, "Internal Reforming Solid Oxide Fuel Cell-Gas Turbine Combined Cycles, Part I," *ASME J. Eng. Gas Turbines Power*, **122**, pp. 1–27.
- [14] Gardner, F. J., Day, M. J., Brandon, N. P., Pashley, M. N., and Cassidy, M., 2000, "SOFC Technology Development at Rolls-Royce," *J. Power Sources*, **86**, pp. 122–129.

Siemens Westinghouse Advanced Turbine Systems Program Final Summary

Ihor S. Diakunchak

Greg R. Gaul

Gerry McQuiggan

Leslie R. Southall

Siemens Westinghouse Power Corporation,
4400 Alafaya Trail,
Orlando, FL 32826-2399

This paper summarizes achievements in the Siemens Westinghouse Advanced Turbine Systems (ATS) Program. The ATS Program, co-funded by the U.S. Department of Energy, Office of Fossil Energy, was a very successful multiyear (from 1992 to 2001) collaborative effort between government, industry, and participating universities. The program goals were to develop technologies necessary for achieving significant gains in natural gas-fired power generation plant efficiency, a reduction in emissions, and a decrease in cost of electricity, while maintaining current state-of-the-art electricity generation systems' reliability, availability, and maintainability levels. Siemens Westinghouse technology development concentrated on the following areas: aerodynamic design, combustion, heat transfer/cooling design, engine mechanical design, advanced alloys, advanced coating systems, and single crystal (SC) alloy casting development. Success was achieved in designing and full scale verification testing of a high-pressure high-efficiency compressor, airfoil clocking concept verification on a two-stage turbine rig test, high-temperature bond coat/TBC system development, and demonstrating feasibility of large SC turbine airfoil castings. The ATS program included successful completion of W501G engine development testing. This engine is the first step in the W501ATS engine introduction and incorporates many ATS technologies, such as closed-loop steam cooling, advanced compressor design, advanced sealing, and high-temperature materials and coatings.

[DOI: 10.1115/1.1719030]

Introduction

The Advanced Turbine Systems Program co-funded by the U.S. Department of Energy, Office of Fossil Energy, was an ambitious ten-year effort to develop the necessary technologies which will result in a significant increase in natural gas-fired power generation plant efficiencies, decrease in cost of electricity, and a reduction in emissions, while maintaining the current state-of-art reliability, availability, and maintainability (RAM) levels. This three-phase technology development and demonstration program was started in 1992 and was completed in 2001.

The ATS Program objective was to develop ultra-high efficiency, environmentally superior, and cost competitive systems for base load application in utility, independent power producer, and industrial markets. The following specific performance targets were set using natural gas as the primary fuel:

1. system efficiency exceeding 60% (net, lower heating value basis) on natural gas for large scale utility turbine systems; for industrial applications, systems that will result in a 15% improvement in heat rate compared to currently available gas turbine systems.
2. an environmentally superior system that will not require the use of post combustion emissions controls under full load operating conditions.
3. busbar energy costs that are 10% lower than current state-of-the-art turbine systems, while meeting the same environmental requirements.
4. fuel-flexible designs that will operate on natural gas but are capable of being adapted to operate on coal-derived or biomass fuels.
5. RAM that is equivalent to the current turbine systems.

6. water consumption minimized to levels consistent with cost and efficiency goals.
7. commercialization in the year 2000.

In Phase 1 of the ATS Program, preliminary investigations on different gas turbine cycles demonstrated that net plant LHV based efficiency greater than 60% was achievable, [1]. In Phase 2 the more promising cycles were evaluated in greater detail and the closed-loop steam-cooled combined cycle was selected for development because it offered the best solution with least risk for achieving the ATS Program goals of plant efficiency, emissions, cost of electricity and RAM, [2]. Phase 2 also involved conceptual ATS engine and plant design and technology developments in aerodynamics, sealing, combustion, cooling, materials, coatings, and casting development, [3,4].

Phase 3 and Phase 3 Extension involved further technology development, component testing and W501ATS engine design. The technology development efforts consisted of ultra low NO_x combustion, catalytic combustion, heat transfer, advanced coating systems, advanced alloys, single crystal casting development, and determining the effect of steam on turbine alloys. Included in this phase was full-load testing of the W501G engine at the McIntosh No. 5 site in Lakeland, FL, [5].

The W501ATS engine incorporates new technologies, as well as design features developed over the last 50 years and employed successfully in the W501 series of heavy-duty industrial and utility engines, [6]. These design features include single-shaft two-bearing rotor, cold-end generator drive, compressor blade rings, individual combustor baskets, low alloy steel rotor discs, curvic clutched turbine rotor, four-stage turbine, cooled and filtered rotor cooling air, single first-stage turbine vane segments, and tangential exhaust struts. The evolution of large gas turbines started at Westinghouse with the introduction of the 45 MW W501A engine in 1968 (see Fig. 1). Continuous enhancements in performance were made up to the 100 MW W501D5, introduced in 1981. The

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30654. Manuscript received by IGTI, Dec. 2001, final revision, Mar. 2002. Associate Editor: E. Benvenuti.

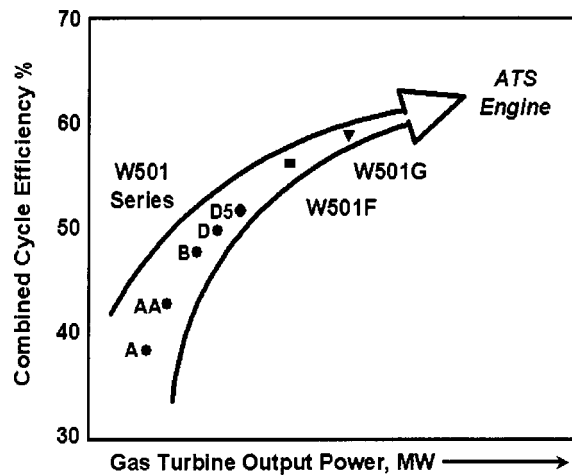


Fig. 1 Evolution of large gas turbines

W501ATS engine is the latest evolutionary design that builds on the success of its predecessors, such as the 186 MW W501F, introduced in 1991, and 250 MW W501G, [7,8].

Siemens Westinghouse's strategy to achieve, and exceed ATS Program goals is to build on the proven technologies used in the successfully operating fleet of its utility gas turbines, such as the W501F, and to extend the technologies developed for the W501G. To help in defining the W501ATS engine and plant, Siemens Westinghouse constituted a 12-member ATS Advisory Board composed of major utilities and independent power producers from the U.S. and U.K. The objective was to ensure that the W501ATS engine will address the customers' needs and find early acceptance in the market place.

Cycle Selection

An extensive study was carried out to select the optimum cost-effective cycle that will achieve the ATS Program goals. The following major cycles were analyzed in considerable detail: increased firing temperature, intercooling, recuperation, intercooling with recuperation, reheat, chemical recuperation, steam injection, and closed-loop steam cooling (CLSC). Based on performance, emissions, cost of electricity and RAM considerations, the cycle selected for the ATS plant consisted of an advanced, high firing temperature, closed-looped steam cooled gas turbine, with a high efficiency bottoming cycle and high efficiency generator.

Technology Development

The main objective of the technology development effort was to develop and verify technologies considered necessary to ATS Program success. To achieve this objective, research was carried out in the following areas: aerodynamics, sealing, combustion, cooling/heat transfer, materials, coatings, and single crystal casting development.

Aerodynamics. High-pressure ratio, high efficiency compressor was designed using the latest three-dimensional viscous code and controlled diffusion airfoils. Variable stators were incorporated into the first two stages to improve starting capability and part-load performance. The mechanical integrity of each stationary and rotating airfoil was verified by finite element analyses to satisfy steady stress and endurance strength criteria. Each airfoil was tuned to avoid potentially harmful resonant frequencies. A plastic model based on the W501F combustion cylinder was constructed to investigate in detail the flow conditions between the compressor exit and the turbine inlet. One of the sixteen combustors was the exact scaled replica of the W501F design while the other combustors simulated the correct combustor pressure loss with the aid of orifices. The model testing and extensive compu-

tational analysis addressed the detail flow distributions around the combustors with and without air extraction for applications in an integrated gasification combined cycle, [9]. Even air flow distributions around the combustor baskets, and hence uniform fuel/air ratios, are very important in achieving ultra low NO_x required in the ATS combustion system.

Sealing. Gas turbine performance is adversely affected by internal leakages. One percent air leakage results in about 1.5% decrease in combined cycle output power and about 0.5% decrease in thermal efficiency. In order to minimize leakages and hence optimize performance, an extensive sealing development program was carried out in the following areas: brush seals, face seals, rope seals, and abradable coatings applied to outer air seal surfaces to allow reduced compressor and turbine blade tip clearances. To reduce air leakage, as well as hot gas ingestion into turbine disk cavities, brush seals will be incorporated under the compressor diaphragms, turbine disk front, turbine rims, and turbine interstage locations. Tests were carried on test rigs for the different brush seal locations to develop effective, rugged, reliable, and long service life brush seal systems. These tests verified the brush seal low leakage and wear characteristics.

A face seal was designed and developed for the rotor rear location in closed-loop rotor cooling applications to prevent rotor cooling air leakage. The noncontacting, dry running face seal has been used in aircraft gas turbines and other turbomachinery applications. Development was necessary to demonstrate that such a seal could meet the requirements of life, durability, large axial movement and turning gear operation in the W501ATS applications. Two full-size prototype seals were designed and tested under simulated engine conditions. Test results showed that the leakage was a fraction of the target value, there was no seal wear and large axial movement could be accommodated.

Considerable performance benefits result from reduced compressor and turbine blade tip clearances. Abradable coatings are being used to reduce tip clearances by allowing minimum build clearances without fear of damaging hardware and by providing more circumferentially uniform tip clearances. Abradable coatings, identified for compressor and turbine applications, were tested to determine abrasability, tip-to-seal wear rate, and erosion characteristics.

Several active turbine blade tip clearance control schemes were investigated. One of them, based on the closed-loop steam cooling concept, showed considerable promise in minimizing the steady-state blade tip clearances, while allowing large clearances on start-up so as to avoid blade tip rubs.

Combustion. To achieve single-digit NO_x emissions at the ATS firing temperature required a considerable development effort, balancing the design for efficiency, emissions, mechanical integrity, and cost. Three different combustor concepts were included in this development. The most successful candidate is the dry low NO_x (DLN) combustor, which consists of eight premixed swirler assemblies arranged around a diffusion/pilot nozzle/swirler assembly. The premixed swirlers are designed to enhance mixing without recirculation, whereas the pilot swirler provides strong swirl to produce a recirculation zone and, hence, good flame stability. The DLN combustor development is concentrated on air management and fuel/air mixing optimization. Although the combustor design allows the pilot to run at very low fuel flows, the pilot is primarily responsible for the NO_x production. To achieve low NO_x levels, a catalytic pilot is being developed to allow operation below the lean extinction limit.

To aid in combustion system development flow visualization tests were carried out on a dry low NO_x combustor using both a single combustor rig and a full-scale sector rig, in order to verify qualitatively CFD predictions. The agreement between the experimental and CFD results was excellent.

Optical diagnostics allow measurement of pertinent parameters, such as the composition and concentration of combustion products

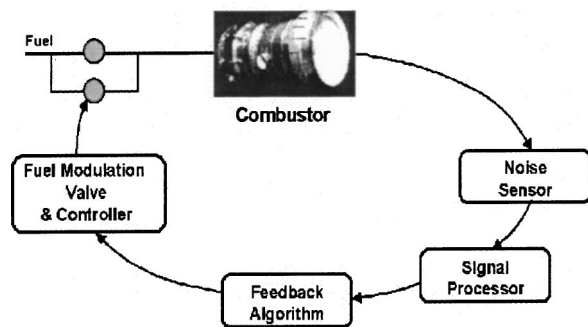


Fig. 2 Active combustion noise control loop

in addition to velocities and flow angles, without disturbing the main flow. A laser-induced fluorescence probe was developed. The probe, which will be used in both cold flow and fired tests, will be a very useful tool in enhancing combustion development productivity.

Lean-premix combustion system will be employed to achieve the NO_x emissions goals. The lean combustion with its inherent flame instability results in more combustion generated noise, and hence, in vibration problems in the combustion system as well as in the downstream components. A program to develop the theoretical background for combustion instabilities, to carry out experiments to aid in the understanding of the problem, to develop a generalized analysis procedure, and to develop stability criteria is under way. An active noise control system is being developed to eliminate combustion instabilities. It consists of a sensor to detect the combustion instabilities, signal processor, feedback algorithm generator, and a fuel modulation valve and controller (see Fig. 2).

To ensure that single digit NO_x emissions are achieved a catalytically enhanced combustor development program is being pursued in parallel with other combustion development efforts. Catalytic combustion is expected to play an important role in achieving ultra low NO_x emissions at the ATS engine firing temperature. The catalyst allows ultra lean-premix combustion without flame instability and flame outs. Therefore, NO_x production is restricted to low single digits at ATS firing temperature with stable operation. Development is in progress to gain theoretical understanding of catalytic combustion, to design a catalytic combustion system and to develop a practical catalytic combustor. The catalytic combustor development effort involves lab testing and field testing prior to the components being installed for commercial operation.

Cooling. Closed-loop cooling is the largest contributor to the ATS plant performance. Eliminating cooling air ejection into the turbine gas path raises the gas temperature downstream of the first stage vane, hence increasing the gas energy level during the expansion process and eliminating cooling air mixing losses. Closed-loop cooling provides an additional benefit in NO_x reduction, by making more air available at the combustor inlet for the lean premix combustion, while maintaining the same burner outlet temperature. Without a cooling air film to shield the turbine components from the hot gases and trailing edge cooling air ejection to enhance cooling in the critical trailing edge region, it is quite a challenge to develop a successful closed-loop cooling design. This challenge was overcome in the W501ATS turbine airfoil cooling design by using thin outer wall design concept. In order to verify the critical closed-loop cooled designs, the following development programs were undertaken: outside heat transfer coefficient measurement, internal heat transfer coefficient and pressure loss measurement, and first-stage vane hot cascade test.

The outside heat transfer coefficients on airfoil and endwall surfaces were measured on the model turbine tests performed at Ohio State University. Internal heat transfer coefficients and flow

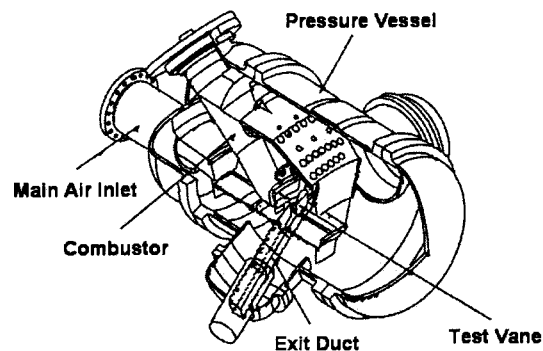


Fig. 3 Cutaway view of vane cascade test rig

characterization tests were carried out at Carnegie-Mellon University on the first-stage vane and blade-cooling designs. These tests involved ten plastic models representing six cooling techniques utilized in the various sections of the first stage turbine vane and blade. The transient liquid crystal technique was the primary measurement method. Test results confirmed analytical predictions. The first-stage vane cooling design will be verified at ATS operating conditions in an integrated combustion and hot cascade test rig, located at the Arnold Engineering Development Center, Arnold AFB, TN (see Fig. 3).

Heat transfer development included plastic model tests to measure internal heat transfer coefficients and pressure drops in multipass blade cooling channels and stator airfoil and endwall film cooling. Third-stage turbine blade shroud cooling design was carried out to ensure adequate cooling for the highly stressed portion of the blade.

Materials. ATS operating conditions extend the technology envelope of current materials, hence materials development work is an important element in the evolution and success of the W501ATS engine. To ensure this success, development programs were carried out on the effect of steam cooling on materials, blade life prediction, advanced vane alloy, nickel-based superalloy for rotors, directionally solidified blade alloy properties, and single-crystal material data.

Coatings. Thermal barrier coatings are critical to the mechanical integrity of the W501ATS engine design. An advanced bond coat/TBC system was developed for more than 24,000 hours service life. Different bond coats and ceramic materials were evaluated under accelerated oxidation test conditions and down selected. The advanced bond coat/TBC system mechanical integrity and durability were demonstrated in more than 24,000 hours of cyclic testing at 1010°C (1850°F).

Novel ceramic chemistries were investigated in an effort to improve upon the phase stability and sintering resistance of 8% yttria-stabilized-zirconia TBC. Under a related program, DOE-Oak Ridge National Laboratory Thermal Barrier Coatings Program, new ceramic TBC's have been identified with superior performance. With the selection of a new bond coat, identified earlier in the program, and a new TBC composition, a TBC system is available for application at the estimated ATS airfoil surface temperatures.

An advanced abrasible TBC system was developed under the ATS program. The system was designed for the ATS first-stage turbine ring segment, which forms the outer flow path surface around the first stage turbine blade. In this application, the system requires state-of-the-art thermal barrier properties and sufficient abrasibility to prevent excessive blade tip wear. Laboratory component testing verified its ability to withstand high surface temperatures, large thermal gradients, and engine-typical blade incursion. To verify performance in an operating engine, seven first-stage turbine ring segments were installed in an engine. These

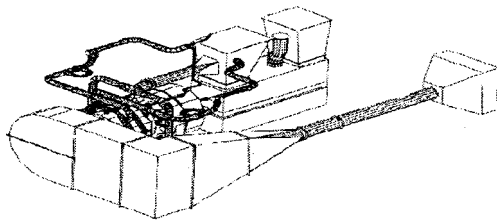


Fig. 4 Compressor test facility

segments were inspected with excellent results after 6000 hours of operation. No detectable erosion or blade tip wear was observed.

Single Crystal Casting Development. To achieve ATS Program performance and mechanical integrity goals, SC vanes and blades are used in the W501ATS engine. Casting development programs were carried out to demonstrate castability of large industrial turbine airfoils in CMSX-4 alloy. Casting trials on first-stage turbine vanes and blades incorporating thin wall cooling design features demonstrated the viability of this concept. Casting trials carried out on the thick-walled second-stage blade and shrouded third-stage blade were also successful. This development effort will be continued to further improve the casting process, and hence improve the yield and make large SC turbine airfoil castings cost effective.

Component Testing

Compressor. To verify the aerodynamic performance and mechanical integrity of the new high-pressure ratio design, the full-scale W501ATS compressor was manufactured and tested in a specially designed facility constructed at the U.S. Navy Base in Philadelphia (see Fig. 4). The compressor test was carried out at subatmospheric inlet conditions to reduce the power required to drive the test compressor to that available at the test facility.

The compressor was instrumented with static pressure taps, fixed temperature and pressure rakes, thermocouples, tip clearance probes, blade vibration monitoring probes, rotor vibration probes, acoustic probes, and strain gages installed on several stages of stators. Provisions were made for radial traverses in eight axial locations in the compressor and four radial locations in the inlet duct. More than 500 individual measurements were recorded. A dedicated data acquisition system was used to collect and reduce the test data. Important performance and health monitoring parameters were displayed on computer screens in real time. After the compressor test facility was commissioned, an extensive test program was performed. The test program included design point performance verification, blade vibration, and diaphragm strain gage measurements, inlet guide vane and variable stator optimization, compressor map definition and starting characteristics optimization. The compressor testing, which was successfully completed ahead of schedule, confirmed all mechanical and aerodynamic performance predictions.

Turbine Testing. The objectives of this development program were the experimental verification of turbine performance with airfoils designed for reduced solidity (and hence reduced cooling requirement) and optimized for closed-loop cooling, performance benefits due to optimum circumferential alignment of turbine airfoils, and airfoil surface heat transfer coefficients. A 1/3 scale model of the first two turbine stages was designed, manufactured, and tested in the shock tube test facility located at Ohio State University (see Fig. 5). Approximately 400 individual sensors were installed on the model turbine, including miniaturized pressure transducers, thermocouples, and thin-film heat flux gages on both stationary and rotating airfoils. Provision was made for exit

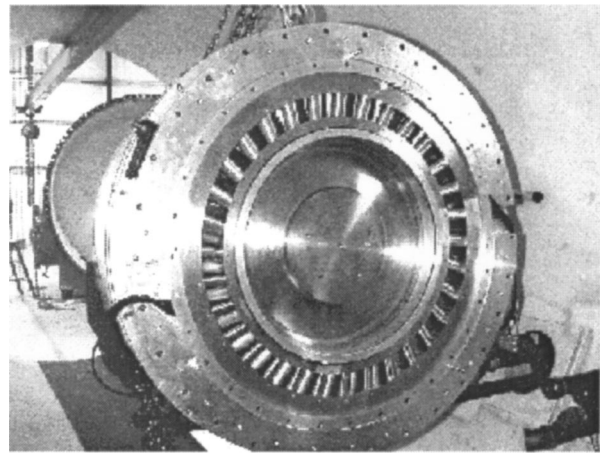


Fig. 5 Model turbine test rig

traversing. Test results confirmed efficiency and heat transfer coefficient predictions and indicated that clocking benefits were greater than expected.

ATS Plant

The ATS plant utilizes a single shaft design concept which incorporates a gas turbine on one end of the generator and a steam turbine on the other (see Fig. 6). The gas turbine is coupled to the generator in the typical manner. However, the steam turbine is coupled to the generator through a self-shifting, and self-synchronizing clutch which is connected to the generator's collector shaft. The gas turbine exhaust passes through the three-pressure level heat recovery steam generator (HRSG) before being exhausted through the stack. The high-pressure steam turbine exhaust steam is utilized to cool the transitions and the first two stages of turbine vanes. The reheated steam is returned to the steam cycle for reheat and induction into the intermediate pressure steam turbine (see Fig. 7).

The two-case multistage single-flow reheat, axial exhaust, condensing steam turbine employs advanced aerodynamic design methods. High-performance bowed impulse and reaction blades are used on the high pressure and intermediate pressure turbines, respectively. Optimized reaction blading is used on the low pressure turbine which includes 1.07 meter (42 in.) long last-stage rotating blades.

The two-pole, 60 Hz, hydrogen inner-cooled generator design absorbs the combined gas turbine and steam turbine power output. The generator operates with static excitation and supports static starting of the gas turbine. To achieve high efficiency, several design enhancements, such as reduced windage and core losses and improved insulation, were incorporated.

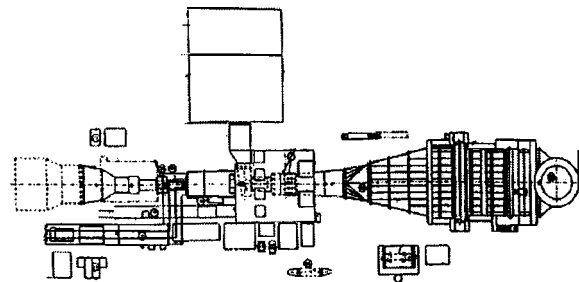


Fig. 6 ATS plant layout

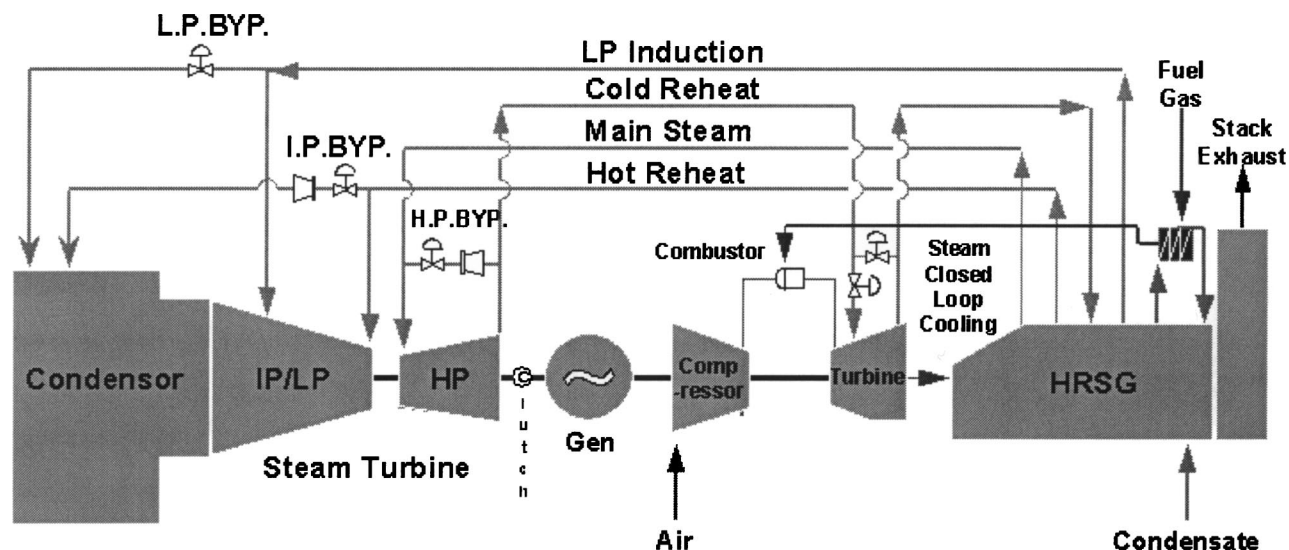


Fig. 7 ATS plant steam system

W501G Engine Development

Siemens Westinghouse solicited input from an industry advisory panel comprised of members from major U.S. and international utilities and independent power producers. Based on the input from this panel and market analyses, Siemens Westinghouse is pursuing an evolutionary introduction of the W501ATS, which incorporates ATS technology in stages culminating in an engine that meets or exceeds all of the program objectives. This approach has two main advantages. First, the evolutionary approach mitigates the risk associated with introducing multiple, advanced technologies simultaneously. Second, the early introduction of ATS technology expands the net benefit of the program, as compared with limiting the technology incorporation to only the W501ATS engine.

The evolutionary approach is shown schematically in Fig. 8. First, the introduction of the ATS frame begins with the 250 MW W501G. This engine is the latest in the series of heavy-duty utility gas turbines and incorporates many ATS technologies. This engine utilizes advanced materials, coatings, cooling technology and advanced aerodynamic design. The 19.5:1 pressure ratio 16-stage compressor, which is derived from the ATS compressor, uses advanced profile high-efficiency airfoils. The combustor design is similar to W501F and hence has the same low NO_x emissions. The 4-stage turbine uses full three-dimensional design airfoils. Brush seals are used in the turbine interstage locations and abradable coatings are used in both the compressor and the turbine to reduce blade tip clearances. Second, from the initial W501G fu-

ture enhancements include steam-cooled turbine vanes, leakage improvements, and increased combustor outlet temperature. Third, the W501ATS engine evolves from the W501G which reduces development risks through early demonstration of many critical technologies.

W501G Test Results. The first W501G was ignited in April 1999, at the City of Lakeland, MacIntosh No. 5 site. The unit has undergone extensive testing and verification. Since March 2000, the customer has dispatched the unit based on power demands. Currently, the unit operates in a simple cycle mode with a once-through-steam-generator for cooling steam production. Construction of the combined cycle plant is underway and will be completed in 2002.

The W501G test program included starting optimization, performance and emissions verification, hot parts metal temperature measurement, including telemetry and thermal paint testing, vibration measurement, tip clearance measurement, etc. The engine instrumentation included over 3000 sensors and measured parameters. An engine schematic showing the various sensors is shown in Fig. 9. The test program consisted of two distinct phases: emissions/performance mapping phase and thermal paint testing. In the emissions/performance mapping phase, testing targeted combustion system variables and provided engine performance mapping from different operating conditions, such as IGV position and exhaust temperature.

Following the initial testing, turbine flowpath and combustion components were painted with thermal paints and installed into the engine. The thermal paint changes colors based on exposed temperature. This method is used extensively in aeroengine validation since it provides a complete and accurate temperature map of the components at operating conditions. To react the thermal paint, the engine was ramped up to full load, run for approximately five minutes at full load and then shut down. The thermal paint test was conducted in two phases. In July 2000, the transitions and first stage turbine vanes were painted and tested. These components are removable without a major cover lift. In October 2000, a full paint test was carried out. This test included all turbine blades and vanes and areas of the rotor subjected to high temperatures. Figure 10 shows the scope of the painted components. Both tests were conducted successfully and results were evaluated in great detail to verify the hot parts' cooling design and to validate computer codes used in W501G engine design. The testing validated the transition duct closed-loop steam cooling in a commercial application.

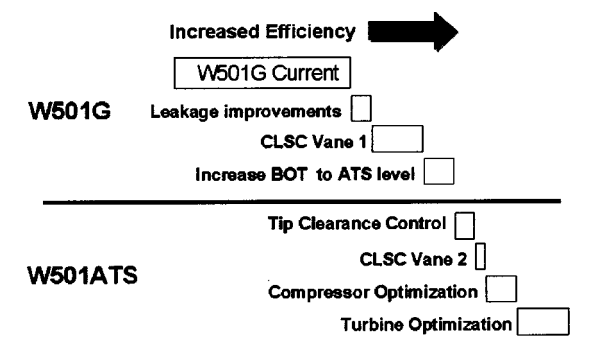


Fig. 8 W502ATS engine evolution

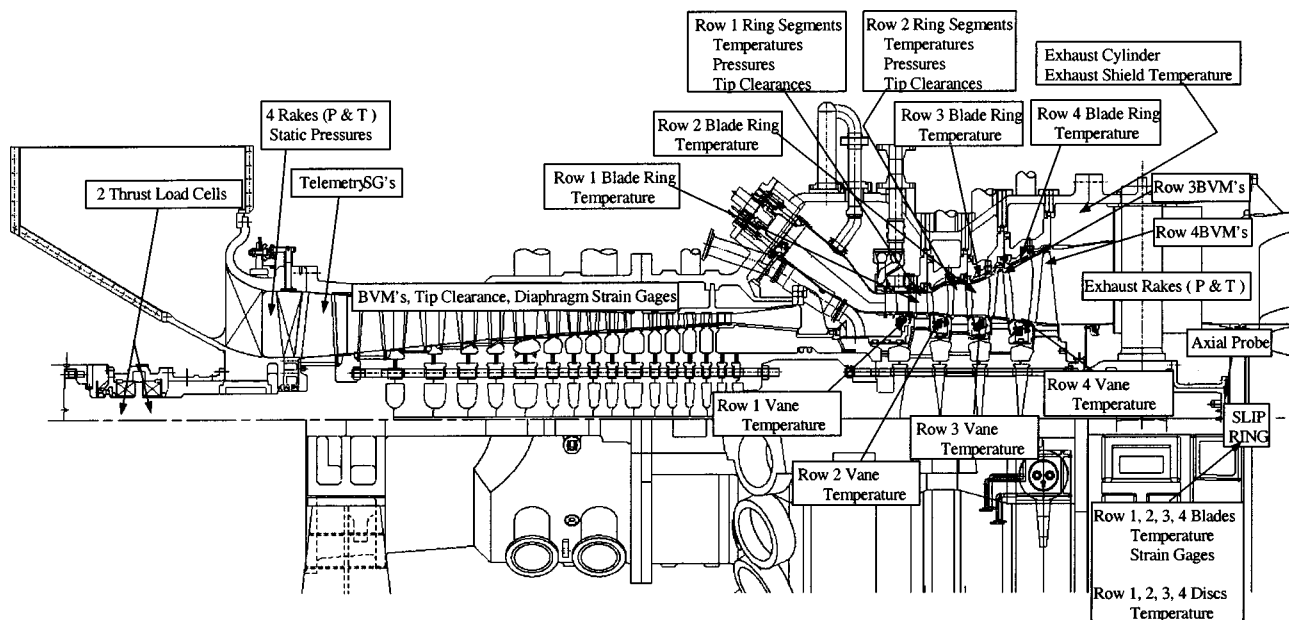


Fig. 9 W501G field test instrumentation

Technology Infusion Into Mature Frames

New technologies developed in the ATS Program were retrofitted into the whole Siemens Westinghouse gas turbine product line to improve efficiency, reduce emissions and improve mechanical integrity.

The ATS-developed compressor technology has been retrofitted into the W501F product line. Using the analytical techniques developed and proven in the ATS program, the W501F compressor was upgraded in the latest improvement to this successful frame. This advanced compressor is used on all new 501F engines. In addition, the redesigned compressor can be retrofitted to any of the 42 W501F engines that were built with the original W501F compressor. Applying this ATS technology to the W501F engine expands the benefit of the ATS program, since the W501F comprises more than 70% of future units that are sold or on order at Siemens Westinghouse.

To date, ATS-developed brush seals have been successfully incorporated and operated in W501G and W501F product lines. Pre- and post-upgrade tests have demonstrated performance improvement in retrofit applications. The ATS-developed abrasion coatings have been incorporated into W501F and W501G compressors and front turbine stages (first and second stages). The later turbine

stages (third and fourth stages) employ shrouded blades with stationary honeycomb seals, and do not require abrasion coatings.

Summary

Technology development efforts have demonstrated that ATS Program goals are achievable. Full-scale compressor tests verified the ATS compressor aerodynamic performance and mechanical integrity. Model turbine tests and internal heat transfer tests on first stage turbine vane and blade cooling designs were completed. Development of brush seals, rotor rear face seal, and abrasion coatings were completed. Development of the ultra low NO_x combustor and catalytically enhanced combustor is progressing. Several materials development programs were successfully completed. Preproduction casting development will continue on the single crystal thin wall vanes and blades. Long-term verification tests are continuing on the advanced bond coat/TBC system. W501G field verification testing was completed. New technologies developed in the ATS Program were retrofitted into the whole Siemens Westinghouse gas turbine product line. Future plans include the W501G engine evolution into the W501ATS engine by incorporating first and second-stage turbine vane closed-loop steam cooling, optimized compressor and turbine aerodynamic designs, active tip clearance control, advanced sealing, and catalytic combustion.

Acknowledgments

This research was prepared with the support of the U.S. Department of Energy, under Cooperative Agreement No. DE-FC21-95MC32267. However, any opinions, findings, conclusions, or recommendations expressed herein are those of the authors and do not necessarily reflect the views of the DOE.

References

- [1] Little, D. A., Bannister, R. L., and Wiant, B. C., 1993, "Development of Advanced Turbine Systems," *ASME Cogen Turbo Power '93 Proceedings*, IGTI, ASME, New York, 8, pp. 271–280.
- [2] Briesch, M. S., Bannister, R. L., Diakunchak, I. S., and Huber, D. J., 1994, "A Combined Cycle Designed to Achieve Greater Than 60 Percent Efficiency," *ASME J. Eng. Gas Turbines Power*, 117, pp. 734–741.
- [3] Bannister, R. L., Cheruvu, N. S., Little, D. A., and McQuiggan, G., 1995, "Development Requirements for an Advanced Turbine System," *ASME J. Eng. Gas Turbines Power*, 117, pp. 723–732.

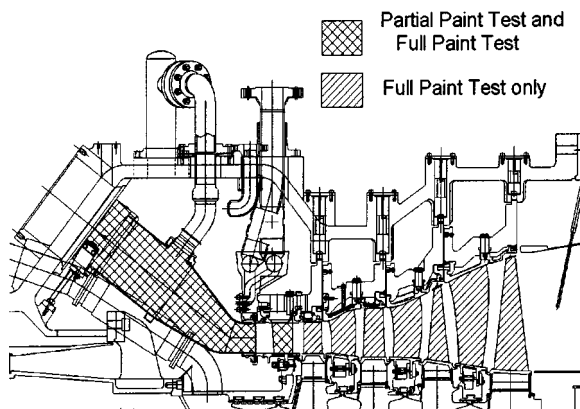


Fig. 10 W501G thermal paint test

- [4] Diakunchak, I. S., Bannister, R. L., Huber, D. J., and Roan, D. F., 1996, "Technology Development Programs for the Advanced Turbine Systems Engine," ASME Paper No. 96-GT-5.
- [5] Gaul, G., Diakunchak, I. S., and Dodd, A. M., 2001, "The W501G Testing and Validation in the Siemens Westinghouse Advanced Turbine Systems Program," ASME Paper No. 02-GT-399.
- [6] Scalzo, A. J., Bannister, R. L., DeCorso, M., and Howard, G. S., 1994, "Evolution of Westinghouse Heavy-Duty Power Generation and Industrial Combustion Turbines," ASME J. Eng. Gas Turbines Power, **118**, pp. 316–330.
- [7] Scalzo, A. J., McLaurin, L. D., Howard, G. S., Mori, Y., Hiura, H., and Sato, T., 1988, "A New 150 MW High Efficiency Heavy-Duty Combustion Turbine," ASME Paper No. 88-GT-162.
- [8] Southall, L., and McQuiggan, G., 1995, "New 200 MW Class 501G Combustion Turbine," ASME Paper No. 95-GT-215.
- [9] Wang, T., Kapat, J. S., Ryan, W. R., Diakunchak, I. S., and Bannister, R. L., 1999, "Effect of Air Extraction for Cooling and/or Gasification on Combustor Flow Uniformity," ASME J. Eng. Gas Turbines Power, **121**, pp. 46–55.

R. Bhargava
Universal Ensco, Inc.,
1811 Bering Drive,
Houston, TX 77057

M. Bianchi

A. Peretto

P. R. Spina

DIEM—University of Bologna,
Viale Risorgimento 2,
40136 Bologna, Italy

A Feasibility Study of Existing Gas Turbines for Recuperated, Intercooled, and Reheat Cycle

In the present paper, a comprehensive and simple in application design methodology to obtain a gas turbine working on recuperated, intercooled, and reheat cycle utilizing existing gas turbines is presented. Applications of the proposed design steps have been implemented on the three existing gas turbines with wide ranging design complexities. The results of evaluated aerothermodynamic performance for these existing gas turbines with the proposed modifications are presented and compared in this paper. Sample calculations of the analysis procedures discussed, including stage-by-stage analysis of the compressor and turbine sections of the modified gas turbines, have been also included. All the three modified gas turbines were found to have higher performance, with cycle efficiency increase of 9% to 26%, in comparison to their original values. [DOI: 10.1115/1.1707033]

Introduction

In recent years, with deregulation and privatization in the power industry becoming common than exception and highly fluctuating fuel prices, the gas turbine market has become extremely competitive. This market dynamics is requiring gas turbine manufacturers to be more aggressive in improving performance of their machines to the maximum achievable within the current technological limits.

Typically, design and development of a new gas turbine can take ten years or more from its conceptual phase. To reduce design and development cost, new approaches have been used by manufacturers. One of the approaches to develop high performance gas turbines for industrial applications has been to modify the engines developed for aircraft applications.

A good example of this approach is the development of a GE LM-6000 gas turbine in early 1990s. In a two-spool LM-6000 gas turbine, low pressure compressor (LPC) was derived from CF6-50 aircraft engine, whereas, high pressure compressor (HPC) and low and high pressure turbines were obtained from CF6-80C2 high bypass ratio turbofan aircraft engine that went into service in 1985, [1]. This approach resulted in the first industrial gas turbine with ISO rated simple cycle thermal efficiency in excess of 40%. The new industrial gas turbine was brought in the market in less than five years from its design initiation.

Another approach, utilized to obtain higher performance gas turbine, was demonstrated by ABB (currently, Alstom Power) in the development of an advanced cycle gas turbine GT-24. The subsonic compressor design has been adapted from GT13E2 gas turbine with optimized efficiency in the first 15 stages and reduced sensitivity to Mach numbers and incidence variations including increased surge margin, [2]. The compression ratio of the first 15 stages is 15 and in the next 7 stages (from 16 to 22), air is compressed from 15 bar to 30 bar giving a pressure ratio of 2. Another major modification to the existing gas turbine was the implementation of sequential combustion which was optimized for high-efficiency high-power density and similar temperatures before the first and second turbine stage. This gas turbine, similar to LM-6000, was brought in the market in a period of four years.

The development of WR-21 gas turbine, jointly developed by Westinghouse and Rolls-Royce, is yet another example of designing a high performance advanced cycle machine by combining aircraft technology and Brayton cycle modification. The gas generator section of WR-21 has used components from Rolls-Royce RB-211 series aircraft engines with some modifications. For example, high pressure turbine (HPT), low pressure turbine (LPT), and power turbine (PT) were designed from RB211-524 HP turbine, RB211-535 IP turbine, and RB211-Trent 700/800 aircraft engines, respectively, [3]. The fact is that 78% of the WR-21 components were derived from RB211 series aircraft engines. The prototype of WR-21 gas turbine became available for testing in approximately 3–4 years.

The examples given above are just few cases among many where existing aircraft engine or industrial gas turbine technologies in combination with modifications to the Brayton cycle have been used to achieve higher performance.

The present paper discusses another approach in which existing gas turbines either in simple cycle or with advanced cycles have been examined to improve their performance by converting them to a recuperated, intercooled and reheat cycle (ICRH) system.

This cycle, recuperated ICRH, has been shown by many studies to achieve very high thermodynamic performance, [4–8].

The main objective, as well as advantage, of the proposed approach is to present a “soft and economical” design methodology that relies on the limited design data to convert an existing machine into a complex cycle high efficiency machine. It must be noted that in the present paper emphasis is given to aerothermodynamic design analysis only. The mechanical design changes due to the proposed modifications to the existing gas turbines have not been addressed in this paper. It is also important to note that the discussed methodology, even though applied for 60 Hz gas turbines in the present study, is equally applicable for 50 Hz gas turbines.

The Existing Machines Considered

In order to identify existing gas turbines that may be suitable candidates for their conversion into a recuperated, intercooled, and reheat (ICRH) gas turbine cycle, a detailed investigation was performed using all the possible resources (internet, technical papers and reports, journal publications, manufacturer’s publications, etc.). The criteria for selection of the existing gas turbines

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30558. Manuscript received by IGTI, Dec. 2001, final revision, Mar. 2002. Associate Editor: E. Benvenuti.

included number of factors, main among them included the following: one or more of the required design features already existing, easy analysis procedures for modified gas turbines, advanced technology features such as high firing temperature or newly developed blade aerodynamic design features and wide range of parameters such as overall pressure ratio, turbine inlet temperature and power output.

A careful review process resulted in the selection of three gas turbines whose schematic layouts and main design features are reported in Figs. 1–3 and Table 1, respectively. The selected gas turbines include Alstom GT-24, GE LM-6000 Dry (that is without fogging injection) and Mitsubishi-Rolls-Royce WR-21. A brief developmental history of these selected gas turbines has already been given in the Introduction. Each of the selected gas turbine has some interesting design feature(s) that makes it particularly suitable to be converted into the desired non-recuperated and recuperated ICRH gas turbine system.

The data in Table 1 clearly show the wider selection covered by the chosen gas turbines: power from 25 MW to 179 MW; overall pressure ratio from 16 to 30; turbine inlet temperature from 980°C (1800°F) to 1260°C (2300°F); and single and multispool designs.

The first selected gas turbine (Alstom GT-24), for example, is already equipped with the reheat provision, so that only the compressor section must be modified to incorporate an intercooler (IC) and convert the existing machine into an ICRH cycle system. Subsequently, addition of a recuperator (REC) will allow the existing machine to be converted into the required recuperated ICRH gas turbine system.

The second selected gas turbine (GE LM-6000) is a simple Brayton cycle machine. However, its two spools design feature seems to be particularly advantageous to incorporate both the intercooler between the two compressors (LPC and HPC, in Fig. 2), and a second combustor between the two turbines (HPT and LPT,

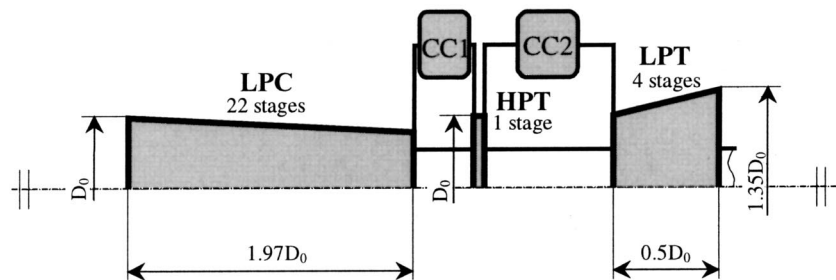


Fig. 1 Schematic layout of the Alstom GT-24 gas turbine

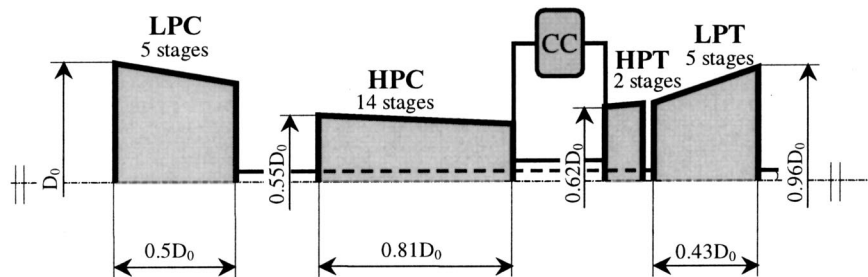


Fig. 2 Schematic layout of the GE LM-6000 gas turbine

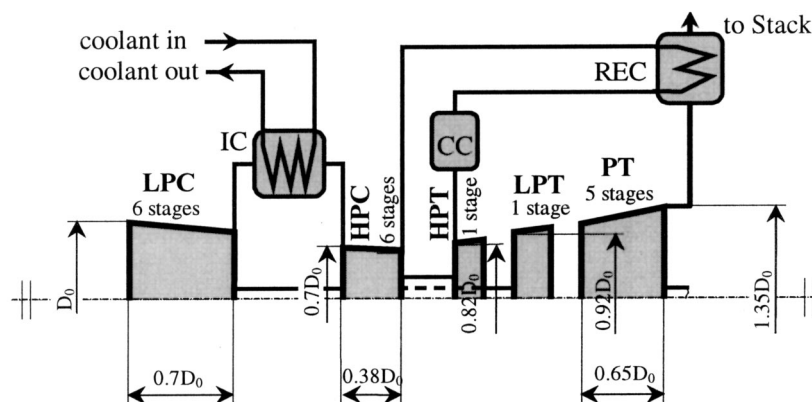


Fig. 3 Schematic layout of the Mitsubishi-Rolls-Royce WR-21 gas turbine

Table 1 Key design and performance data for the selected gas turbines

Parameter	GT-24	LM-6000 (DRY)	WR-21
Power output (ISO) (kW _{et})	179000	43400	25252
Heat rate LHV (kJ/kWh)	9600	8613	8572
Overall efficiency (%)	37.5	41.8	41.2
Overall pressure ratio	30.0	29.1	16.2
Exhaust gas flow (kg/s)	391	127	73
Output shaft speed (rpm)	3600	3600	3600
Turbine inlet temp. (°C)	1260	1260	982*
Exhaust temp. (°C)	630	438	356
Combustor type	Annular with sequential combustion	Annular	Can annular
Estimated characteristic dimension D_0 [mm]	2546	2298	1350
No. of shafts	1	2	3

*estimated turbine inlet temperature to meet manufacturer data

in Fig. 2). Additionally, a suitable recuperator needs to be incorporated for the existing LM-6000 to be converted into the required recuperated ICRH gas turbine system.

The third gas turbine (Mitsubishi-Rolls-Royce WR-21) is already equipped with an intercooler and a recuperator, so that only the second combustor must be incorporated either between LPT and PT or between HPT and LPT to convert the existing WR-21 into the required recuperated ICRH gas turbine system.

In Figs. 1–3, the values of geometrical data at certain sections of each gas turbine, expressed as a fraction of a characteristic dimension D_0 (selected as blade tip diameter of the first stage compressor), are also shown and have been used in the design modification process of the existing machines. Furthermore, data on number of stages of compressor and turbine sections for the selected gas turbines are included in Figs. 1–3.

The next step, after selection of the candidate gas turbines and identification of important changes required to obtain the desired gas turbine system, is to develop a systematic design methodology and performance evaluation procedures.

It must be noted that for the selected machines, except for the WR-21 gas turbine, the desired modifications have been achieved in two steps: (1) conversion of existing gas turbine to the ICRH system and (2) conversion to the recuperated ICRH system. For the WR-21 gas turbine, the existing machine is instead directly converted into the recuperated ICRH cycle, mainly due to the fact that the existing gas turbine is already equipped with intercooling and recuperation.

Design Methodology and Performance Evaluation Procedures for a Recuperated ICRH Cycle Using an Existing Gas Turbine

To obtain the required nonrecuperated and recuperated ICRH gas turbine cycle system from an existing gas turbine, the following design and performance evaluation phases should be carried out:

- Analysis of the identified design modifications on the existing gas turbines to obtain an ICRH cycle, such as, splitting of compressor and/or turbine in two parts to add an intercooler and/or a reheat combustor, redesigning of some compressor and turbine stages, etc. This phase may also require a preliminary design of the existing compressor and turbine sections, in order to assess the stage at which the compressor or turbine has to be divided to incorporate the intercooler or the reheat combustor and calculate the values of design pressure, temperature, and mass flow rate at the entry of this stage.

- Designing of the new gas turbine components that are emerged from the previous analysis in phase (a). This phase can involve the design process of some compressors and turbines (such as the low pressure compressor before the intercooler and the low pressure turbine at the reheat combustor exit) and of the heat exchangers. The main steps of the design process for new and modified compressors and turbines are discussed below. The design analyses of heat exchangers (intercooler and recuperator) have been performed using the ε -NTU method.

- The performance evaluation of the obtained ICRH cycle gas turbine, considering that the new designed components work in design conditions, while the old components may work in off-design conditions, due to the possible changes in operating gas temperature, pressure, and mass flow rate at the component inlet or due to the modification made on the component itself (such as elimination of some stages of the existing compressors and turbines). The thermodynamic performance calculations were performed by using a commercial software package, [9]. In particular, off-design performance of modified and existing compressors and turbines were evaluated by using nondimensional performance maps, predicted by means of scaling techniques from performance maps of similar machines, [10,11].

An example of the step-by-step design and analysis methodology discussed above and applied to LM-6000 with results obtained is presented in Appendix A.

New Compressor Design. The design process of the new compressor stages permits to estimate, in particular, compressor efficiency, that is used for the performance evaluation of the ICRH cycle. However, the complete design of the new compressor is required to evaluate the feasibility of the new machine. Moreover, a preliminary design of the existing compressor may be required in phase (a) of the design and analysis methodology, in order to assess the stage at which the existing compressor has to be divided to insert the intercooler, and the design pressure, temperature, and mass flow in this compressor section. These thermodynamic data will be used in both phase (b) and (c) of the design and analysis methodology. Appendix C provides the nomenclature used in aerodynamic analysis of a compressor stage.

The new compressor design process involves the following steps, [12,13]:

- Evaluation of the annulus dimensions.* Based on the previous experience and the available literature, this step requires choice of the following parameters: (i) first rotor blade tip speed, (ii) compressor inlet axial velocity, (iii) first rotor hub/tip ratio. Obviously, if the rotational speed is fixed, one of the three parameters is

determined when the other two are chosen. Moreover, it is necessary to assume the variation of the mean diameter from compressor inlet to outlet (the most widespread solutions are: constant mean diameter, adopted here, constant outer diameter or constant inner diameter).

Once the annulus dimensions at the compressor inlet and outlet are calculated, an important parameter to be checked is the blade height/diameter ratio (h/D), which must be lower bounded to limit blade tip leakage and secondary losses. Usually, in common design practice, this lower bound is about 0.04. In this paper, however, values of (h/D) of about 0.02 were considered acceptable for the analyses performed, since they can be corrected further by means of a more in-depth compressor aerodynamic design.

2 Determination of number of stages. In this step, it is necessary to assume the compressor stage efficiency and a stage enthalpy rise. The values of these parameters are evaluated based on the available statistical information on axial flow compressors, for example, that presented by Freeman and Dawson [14] and Wisler [15].

3 Calculation of the absolute and relative flow angles for each stage at mean radius. These parameters are evaluated starting from the assumed enthalpy rise and accounting for work reduction due to nonuniform distribution of axial flow velocity along radial direction (caused by rapid thickening of the annulus wall boundary layers through the first few compressor stages) by using the parameter called the "work-done-factor," [16].

The maximum allowable diffusion in each stage cascade is assessed using the "de Haller" criteria, which was considered suitable for the preliminary design calculations. This criteria imposes that, for each cascade, the ratio between outlet and inlet flow velocity (relative and absolute for rotor and stator, respectively) be higher than 0.72. Note, a number of iterations of Steps 1 to 3 are often required by changing the choice of some parameters (such as first rotor blade tip speed, compressor inlet axial velocity, first rotor hub/tip ratio, number of stages, etc.), until the suitable stage-by-stage design is completed.

4 Determination of flow angle variations from blade's root to tip. Usually, free-vortex bladings are used, [13], (also, adopted here). However, this type of blade design can determine critical conditions at the blade root section, such that it is not possible to meet the "de Haller" criteria and/or the stage degree of reaction becomes zero or negative. In these cases, it is possible to adopt different types of blade design, for example, using exponential bladings or bladings with constant degree of reaction from the root to tip.

New Turbine Design. The design analysis of the new turbine stages provides the turbine efficiency, the number of stages, the stage loading factor and the degree of reaction of each stage that are all used for the performance evaluation of the ICRH cycle. In particular, values of the number of stages, the stage loading factors and the degree of reactions are required to model the cooled expansion through the turbine. Appendix C provides the nomenclature used in aerodynamic analysis of a turbine stage.

The main steps of turbine section design process are:

1. **Choice of number of stages, stage loading factors and flow factors.** From the turbine overall enthalpy drop, it is possible to fix the number of stages, the stage loading factors and the flow factors of each stage. These choices are done with the purpose of obtaining the best compromise between high overall efficiency of the turbine and low number of stages (implying low component cost), with the constraint that the value of the blade tip speed is limited to a maximum acceptable value of about 450 m/s. As a result, the maximum acceptable value of the blade speed (U) at the mean radius, by assuming a constant mean radius for all stages, is linked to the turbine overall enthalpy drop, number of stages, and stage loading factors by means of the equation:

$$U = \sqrt{\frac{\sum_i \Delta h_i}{\sum_i \Psi_i}} = \sqrt{\frac{\Delta h_{TOT}}{\sum_i \Psi_i}} \quad (i = i_{th} \text{ turbine stage}).$$

In the choice of number of stages, stage loading factor and flow factor of each stage it should be considered that, by increasing the number of turbine stages, the stage loading factor of each stage decreases and the stage efficiency increases. In this phase, it is important to consider also the effects of turbine blade cooling on ICRH thermodynamic cycle, since cooling mass flow rates depend on how the overall turbine enthalpy drop is split among stages, and thus on the choice of stage loading factors. The cooling mass flow rate estimation and the cooled expansion calculations are performed using the models presented by Benvenuti et al. [17].

These above discussed considerations usually provide the first and the last turbine stages with high and low values of stage loading factors, respectively. For example, in Fig. 4, the values of stage loading factors for some of the GE-Nuovo Pignone gas turbines, [18,19], are shown in the Smith diagram, [20].

2. **Evaluation of stage passage areas and blade height.** The blade speed (U) at the mean radius and the flow factors estimated in the previous step 1 are used for the evaluation of the axial component of flow velocity in each turbine stage, which, in turn, allows the determination of stage passage area and blade height.
3. **Choice of degree reaction.** The value of degree of reaction is usually chosen to maximize the total-to-total efficiency of the intermediate turbine stages, and the total-to-static efficiency of the stage before the reheat combustor and of the last turbine stage.
4. **Calculation of the absolute and relative flow angles for each stage at mean radius.** These are calculated as a function of stage loading factor, flow factor and degree of reaction by using the following equations:

$$\tan \beta_2 = \frac{1}{\phi} \left(\frac{\Psi}{2} - R \right)$$

$$\tan \beta_3 = \frac{1}{\phi} \left(\frac{\Psi}{2} + R \right)$$

$$\tan \alpha_2 = \tan \beta_2 + \frac{1}{\phi} = \frac{1}{\phi} \left(\frac{\Psi}{2} - R + 1 \right)$$

$$\tan \alpha_3 = \tan \beta_3 - \frac{1}{\phi} = \frac{1}{\phi} \left(\frac{\Psi}{2} + R - 1 \right).$$

5. **Determination of flow angle variations from root to tip.** For turbine stages, free vortex bladings are always used.

The efficiency of each stage is estimated using the Smith diagram [20], as shown in Fig. 4, which reports the stage loading factor versus flow factor, for different values of total-to-total stage efficiency, without the effect of blade tip clearance losses.

Therefore, the stage efficiency determined using the Smith diagram has been corrected to account for the blade tip clearance losses. The blade tip clearance losses (Y_c) are evaluated using the Ainley and Mathieson correlation, [21], given below:

$$Y_c = B \frac{\varepsilon}{h} [2(\tan \beta_2 + \tan \beta_3)]^2 \frac{\cos^2 \beta_3}{\cos^5 \beta_m}$$

where

$$B = \begin{cases} 0.50 & \text{for radial tip clearance} \\ 0.25 & \text{for shrouded blade with axial clearance} \end{cases}$$

$$\beta_m = \tan^{-1}[(\tan \beta_3 - \tan \beta_2)/2].$$

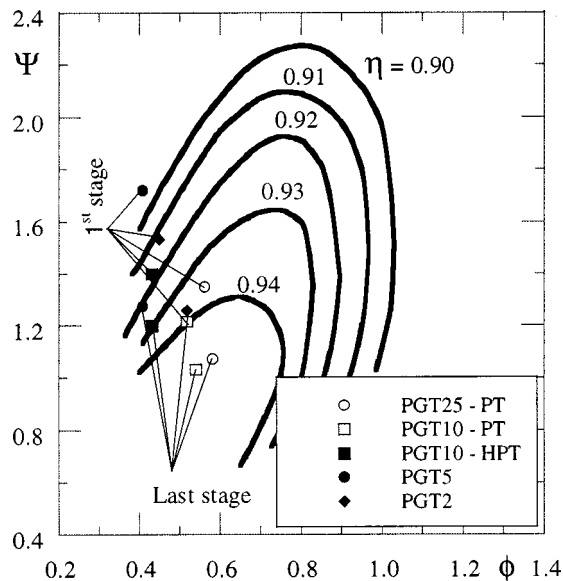


Fig. 4 Smith diagram, [20], with points representative of GE-Nuovo Pignone gas turbine stages, [18,19]

Modifications Performed on the Existing Gas Turbines and Obtained Results

On the basis of the gas turbine layouts shown above, the plausible and realistic modifications to perform on the machines to obtain an ICRH cycle have been evaluated. In all the figures (5,6,8,9) showing gas turbine layouts, gray surface means an unmodified component of the existing machine, white surface means a new designed component and dashed surface means a component modified from the existing configuration (elimination or modification of stages, etc.). All dimensions identified in these figures are referred to the blade tip diameter at the compressor inlet of the existing machine.

In the thermodynamic analysis, the calculations of the real thermodynamic transformations and the energy balance have been performed by considering the specific heats at constant pressure and volume as a function of temperature and gas composition. Pressure losses at the gas turbine inlet and outlet and through combustors and heat exchangers have also been taken into account.

The effects of nozzle and rotor blade cooling flows on gas turbine thermodynamic cycle performance have been evaluated by considering that nozzle cooling flow mixes upstream the rotor, while rotor cooling flow mixes half upstream and half downstream of the rotor itself.

The results of the aerodynamic analysis due to design changes, like compressor and turbine efficiencies, number of stages, stage loading factors, and degree of reactions of turbine stages have been assumed as input to perform the thermodynamic analyses.

Alstom GT-24 Gas Turbine—The Proposed Design Modifications. For this machine, two different approaches for the proposed design modifications have been examined:

- A. In the first approach (A), the design changes include a new LP compressor with a number of stages to be evaluated, the insertion of an intercooler and transformation of the existing compressor into the HPC by taking away a suitable number of stages from it (see Figs. 5(a), 5(b)). In order to transform the existing compressor in the HPC, it was considered that

the volumetric flow rate at the HPC inlet is equal to the design volumetric flow rate at the inlet of the corresponding stage of the existing compressor, since the compressor rotational speed has not changed. Using this approach, the new LP compressor operates under design conditions, while the HP compressor obtained from the existing compressor always works in off-design conditions with a lower efficiency than in the design case. In fact, insertion of the IC causes a reduction of the air temperature at the HPC inlet compared to the design air temperature at the inlet of the corresponding stage of the existing compressor, causing an increase in the HPC corrected speed.

- B. The second approach (B) considers transformation of the existing compressor in the LPC by taking away a suitable number of stages in its later section, the insertion of an IC and a new HP compressor with a number of stages to be evaluated (see Figs. 5(c), 5(d)). Thus, in this approach, in comparison to the approach (A), both LPC and HPC work in design conditions.

The number of stages for LP compressor in both the approaches was calculated in order to obtain a pressure at the HPC inlet that maximizes the ICRH cycle overall efficiency, [6]. It must be noted that the LPC pressure ratio, corresponding to the optimum overall cycle efficiency, is higher for the recuperated ICRH cycle compared to the non-recuperated ICRH cycle as shown by Bhargava et al. [6]. Consequently, the number of LPC stages required in the case of recuperated ICRH cycle is higher than in the nonrecuperated case (see Figs. 5(a) and 5(c) versus 5(b) and 5(d), respectively). The number of HPC stages required in both the approaches was determined in order to obtain a value of HP turbine inlet pressure close to its design value (of the existing machine). In this manner, the HPT and LPT (which were considered operating in choking condition) run close to the design condition.

Therefore, based on the above discussion, the following four design modifications to the existing GT-24 gas turbine have been examined:

Case 1A ICRH cycle gas turbine obtained by using a newly designed LPC and modifying the existing compressor into HPC (for layout, see Fig. 5(a));

Case 2A Recuperated ICRH cycle gas turbine obtained by using a newly designed LPC and modifying the existing compressor into HPC (for layout, see Fig. 5(b));

Case 1B ICRH cycle gas turbine obtained by modifying the existing compressor into LPC and using a newly designed HPC (for layout, see Fig. 5(c));

Case 2B Recuperated ICRH cycle gas turbine obtained by modifying the existing compressor into LPC and using a newly designed HPC (for layout, see Fig. 5(d)).

Thermodynamic Performance of the Modified GT-24 Gas Turbine. In Table 2, the main results of aerothermodynamic analysis obtained for the modified GT-24 gas turbine (nonrecuperated and recuperated ICRH gas turbines) are summarized. For comparison purpose, data of the existing GT-24 gas turbine are also included in Table 2.

The advantages of approach (B), compared to the approach (A), in terms of improved performance are evident from Table 2. In particular, the recuperated ICRH cycle with new HP compressor (Case 2B) permits to obtain values of electric efficiency and power output much greater than that of the existing machine (increase of about 26% and 21% for electric efficiency and power output, respectively).

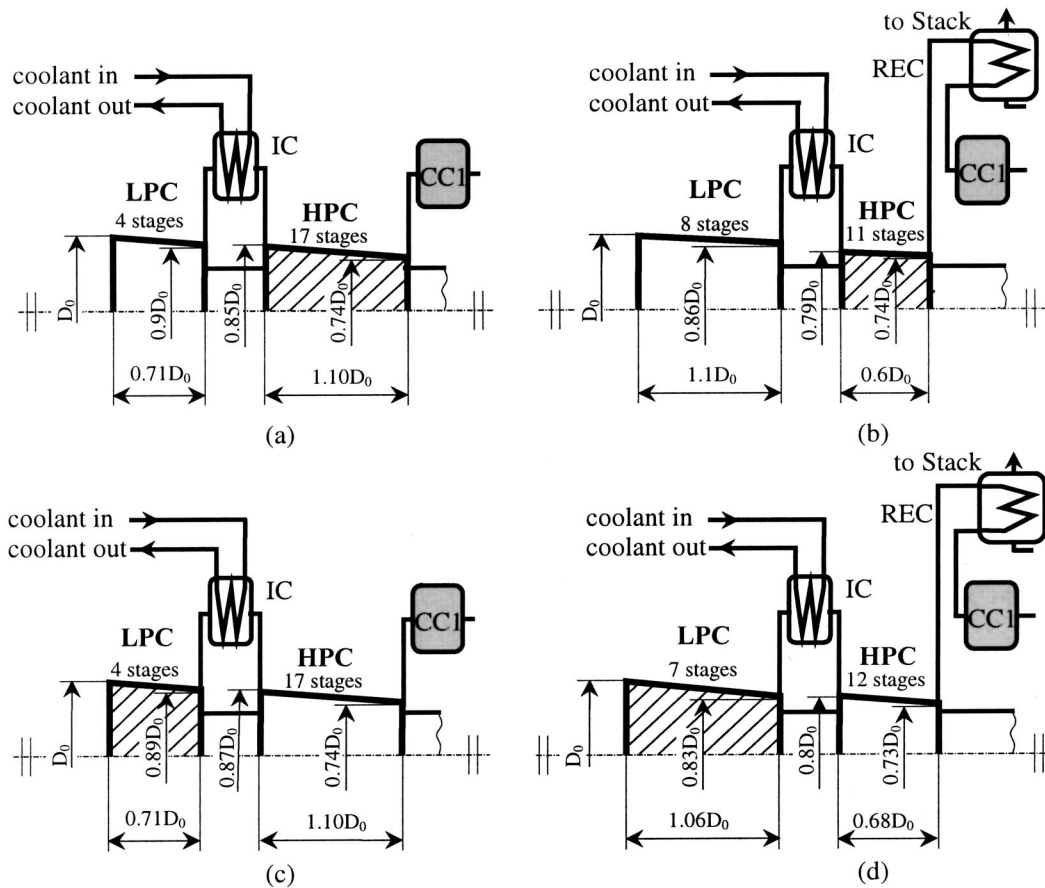


Fig. 5 Schematic layouts of modified Alstom GT-24 compressors: (a) nonrecuperated case with LPC redesigned; (b) recuperated case with LPC redesigned; (c) nonrecuperated case with HPC redesigned; (d) recuperated case with HPC redesigned

Furthermore, the HP compressor for Case 2A operated at a corrected speed ratio of 1.33, resulting in a reduction of about 10% in compressor efficiency compared to its design value (existing machine) and problems related to sonic conditions.

For Case 1B, nonrecuperated ICRH cycle obtained by using new HP compressor, the values of cycle efficiency and power output increase by approximately 2% and 12%, respectively.

In Table 2, values of the equivalent water consumption in the IC, $m_{H_2O}^*$, and the intercooler surface area are also included. The first parameter may be useful to compare the cooling water required by different energy systems. It must be noted that the equivalent water consumption is lower in comparison to the cooling water required (approximately, 200 [(kg/s) $^{\circ}$ C/MW]) for

Table 2 Calculated performance summary of the modified GT-24 recuperated ICRH gas turbines

Examined Cases	P_{el} (MW)	η_{el} (%)	T_{out} ($^{\circ}$ C)	m_{out} (kg/s)	π_{TOT}	π_{LPC}	π_{HPT}	S_{IC} (m^2)	$m_{H_2O}^*$ (kg/s) $^{\circ}$ C/MW)	S_{REC} (m^2)	C_S/C_{S0}
GT-24 RH cycle	179.0	37.5	630	391	30.0	...	2.00	...	...	...	1.00
(Case 1A) GT-24 ICRH cycle (new LPC)	194.0	37.4	618	395	29.7	2.50	1.97	11880	29.00	...	1.14
(Case 2A) GT-24 REC ICRH cycle (new LPC)	186.4	43.1	396	381	28.6	4.90	1.94	16866	65.81	39010	1.33
(Case 1B) GT-24 ICRH cycle (new HPC)	200.5	38.2	619	392	29.4	2.54	1.96	11969	28.28	...	1.00
(Case 2B) GT-24 REC ICRH cycle (new HPC)	216.7	47.4	380	391	31.0	4.48	2.03	16658	53.24	50286	1.00

a same size combined cycle system equipped with Brayton cycle gas turbine.

In Appendix B, the results of the stage-by-stage aerodynamic design analysis of compressor sections for the four cases of the modified GT-24 gas turbine are reported for the first and last compressor stage only.

GE LM-6000 Gas Turbine—The Proposed Design Modifications. This type of gas turbine represents a more complex case with respect to the other two machines examined.

First, it was decided not to modify the high pressure section (HPC-HPT, Fig. 6(a)) of the gas turbine unit and to maintain the same air mass flow rate at the gas turbine inlet as that of the design case. Moreover, the intercooler heat exchanger was positioned downstream of the new LPC (see Figs. 6(a) and (b)), which was completely redesigned to obtain the optimized pressure at the HPC inlet found by the ICRH cycle optimization, [6].

The second (reheat) combustor was inserted between the existing HPT and the new LPT completely redesigned to increase the back pressure at the HPT exit. This last aspect descends from the following considerations:

- It was assumed that the mass flow rate at the gas turbine inlet (and also at the HPT inlet) remains the same as for the design case; so, in the hypothesis that the HPT works in choking condition, the pressure at its inlet (and also at the HPC outlet) does not change;
- A decrease in the temperature (because of the IC presence) and a increase in the pressure (required from the cycle

optimization perspective) at the HPC inlet with respect to the design case, requires that the HPC works with a lower pressure ratio and corrected mass flow than in the design case.

As a consequence, as evident on the compressor performance map (see Fig. 7), the HPC corrected speed must decrease, implying that the HP rotor speed must decrease, since the temperature at the HPC inlet decreases. This is obtained by decreasing the power output of the HPT due to an increase of the back pressure at its exit.

Also, for LM-6000 gas turbine, both nonrecuperated (Fig. 6(a)) and recuperated (Fig. 6(b)) ICRH cycles have been evaluated. In the case of recuperated ICRH cycle, two design conditions for the recuperator have been considered:

- in the first subcase, the recuperator was designed to obtain a T_{out} equal to that of the existing machine;
- in the second subcase, the recuperator was designed to obtain its effectiveness of 0.9.

Therefore, the following four design modifications to the existing machines have been evaluated:

Case 1 ICRH cycle gas turbine obtained by using newly designed LPC and LPT (for layout, see Fig. 6(a));

Case 2 Recuperated ICRH cycle gas turbine obtained by using newly designed LPC and LPT and by designing the recuperator to obtain a T_{out} equal to that of the existing machine (for layout, see Fig. 6(b));

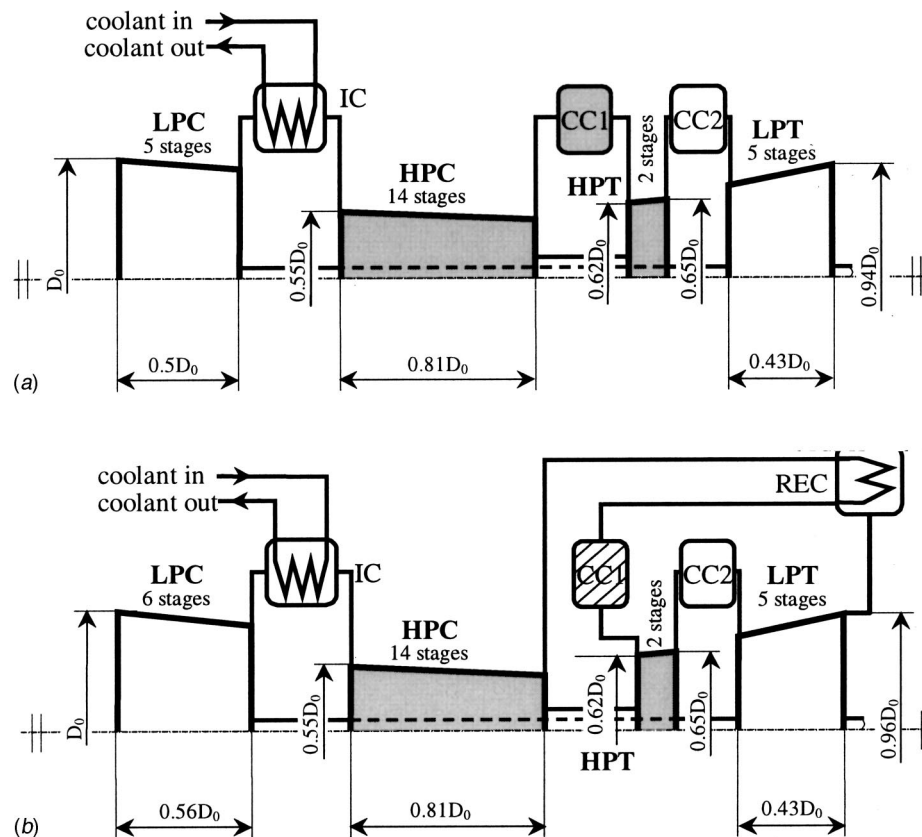


Fig. 6 (a) Schematic layout of the modified GE LM-6000 GT (nonrecuperated ICRH cycle); (b) Schematic layout of the modified GE LM-6000 GT (recuperated ICRH cycle)

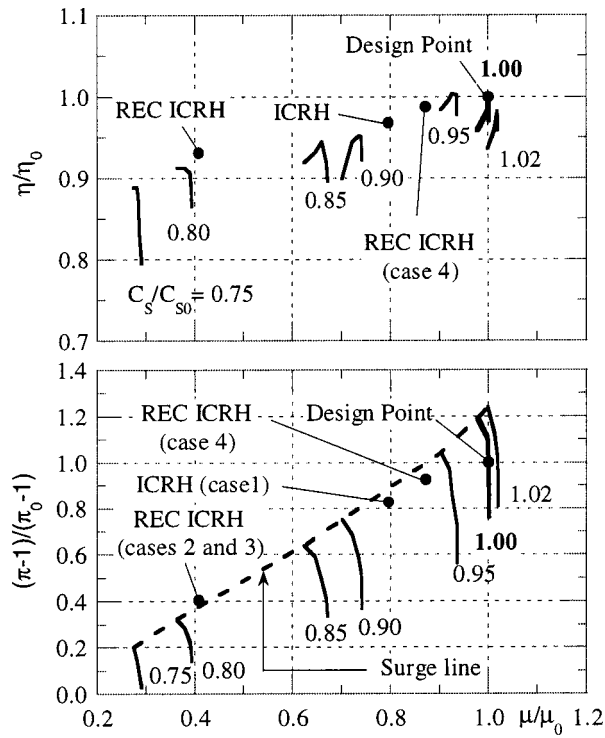


Fig. 7 LM-6000 HP compressor maps and modification of its working point, [9]

Case 3 The Case 3 is similar to Case 2 in all the modifications incorporated except that the recuperator was designed to have an effectiveness of 0.9.

Case 4 The Case-4 is similar to Case-3, except that the LP compressor was designed to obtain a lower pressure ratio than that used in Case-3 to avoid surge condition in the HP compressor (see Fig. 7) as explained below.

Thermodynamic Performance of the Modified LM-6000 Gas Turbine. In Table 3, the main aerothermodynamic performance results obtained for the modified LM-6000 (nonrecuperated and recuperated ICRH cycle) gas turbine are summarized and compared to the existing gas turbine data.

Figure 7 shows the working points for the HP compressor on the compressor performance map, [9], for the four cases considered, including the design case (existing machine). As can be seen in Fig. 7, the HPC operates beyond the surge line for Case 2 and Case 3, and, therefore, these two cases are not viable.

For this reason, the results for Case 2 and Case 3 are not included in Table 3. Case 4 was investigated in which LPC pressure ratio was adjusted until an acceptable HPC working point was obtained. It must be noted that the value of LPC pressure ratio for Case 4 is lower in comparison to the value obtained with the optimum overall cycle efficiency (Case 3).

Furthermore, this change in LPC pressure ratio resulted approximately 3% reduction in the overall cycle efficiency compared to the Case 3.

The above discussion also shows importance of a careful aerothermodynamic analysis in the design of a gas turbine and a need for the compromise in the value of optimum overall cycle efficiency.

In Appendix B the stage-by-stage aerodynamic analysis results for LPC and LPT, for Case 1 and Case 4, are reported for the first and last stages of the modified components.

The poor performance of the nonrecuperated ICRH LM-6000 (modified gas turbine Case (1)) can be attributed to the poor HPC performance in addition to the reheating effect. The increased power output for Case (1), by approximately 66% as a result of the design modification (mainly attributable to incorporation of the reheat effect) is evident from Table 3. Also, by adding recuperator, modified gas turbine with higher cycle efficiency (by approximately 9%) compared to the corresponding existing gas turbine is achievable (see Table 3, Case 4).

In Table 3 equivalent cooling water mass flow rate values are reported. They are lower compared to the cooling water required for a same-size combined cycle system equipped with Brayton cycle gas turbine (that are greater than about 150 [(kg/s)°C/MW]).

Mitsubishi-Rolls-Royce WR-21 Gas Turbine—the Proposed Design Modifications. Two different strategies were adopted for modifying the existing WR-21 gas turbine:

Case A In this case, no modifications to the LPC-LPT and HPC-HPT sections, as shown in Fig. 3, have been required. However, the following two design modifications have been considered: (i) the reheat combustor was inserted between the LPT and PT; and (ii) a newly designed PT was used to account for the augmented inlet gas temperature (see Figs. 8(a) and 8(b)). Therefore, for this strategy, the following three cases have been evaluated:

Case 1A Recuperated ICRH cycle gas turbine obtained by using a newly designed PT and maintaining the existing recuperator (for layout, see Fig. 8(a));

Table 3 Calculated performance summary of the modified LM-6000 recuperated ICRH gas turbines

Examined Cases	P_{el} (MW)	η_{el} (%)	T_{out} (°C)	m_{out} (kg/s)	π_{TOT}	π_{LPC}	π_{HPT}	S_{IC} (m ²)	$m_{H_2O}^*$ (kg/s)°C/MW)	S_{REC} (m ²)	C_s/C_{s0}	n/n_0
Existing LM-6000 Brayton cycle	43.4	41.8	438	127	29.1	2.10	4.24	...	...	...	1.00	1.00
(Case 1) LM-6000 ICRH cycle	71.6	38.4	678	129	29.1	2.50	3.32	3900	25.63	...	0.92	0.85
(Case 4) LM-6000 REC ICRH cycle (π_{LPC} not optimized, $\epsilon_{REC}=0.9$)	84.5	45.6	495	149	34.1	2.65	3.44	4749	27.79	15279	0.94	0.87

Case 2A Recuperated ICRH cycle gas turbine obtained by using a newly designed PT and redesigning the recuperator to obtain the same value of T_{out} as that of the existing gas turbine (for layout, see Fig. 8(b));

Case 3A Recuperated ICRH cycle gas turbine obtained by using a newly designed PT and redesigning the recuperator with an effectiveness equal to 0.9 (for layout, see Fig. 8(b)).

Case B In this case, the following design changes have been considered: (i) no modifications were made to the LPC, HPC, and HPT sections of the gas generator of the existing WR-21 gas turbine; (ii) the reheat combustor was inserted between HPT and LPT sections of the gas generator; and (iii) newly designed LPT and PT were used to account for the augmented inlet gas temperature (see Figs. 9(a) and 9(b)). Therefore, for the strategy (B), the following three design modifications have been evaluated (which are similar to Case 1A to Case 3A, respectively):

Case 1B Recuperated ICRH cycle gas turbine obtained by using newly designed LPT and PT and maintaining the existing recuperator (for layout, see Fig. 9(a));

Case 2B Recuperated ICRH cycle gas turbine obtained by using newly designed LPT and PT and redesigning the recuperator to obtain the same value of T_{out} as that of the existing gas turbine (for layout, see Fig. 9(b));

Case 3B Recuperated ICRH cycle gas turbine obtained by using newly designed LPT and PT and redesigning the recuperator with its effectiveness of 0.9 (for layout, see Fig. 9(b)). It must be noted that no change was implemented to the intercooler heat exchanger.

Thermodynamic Performance of the Modified WR-21 Gas Turbine. The main aerothermodynamic performance results obtained for the modified WR-21 gas turbine (recuperated ICRH cycle gas turbines) are reported and compared to the corresponding existing gas turbine data in Table 4.

The loss in cycle efficiency for Case 1A and Case 1B in the amount of approximately 7.3% and 2.2%, respectively, can be associated with the poor effectiveness of the recuperator under modified flow conditions (see Table 4). An increase in power output by approximately 25% and 17% for Case 1A and Case 1B, respectively mainly attributable to the incorporation of reheat in the design modification, is evident from Table 4. With proper

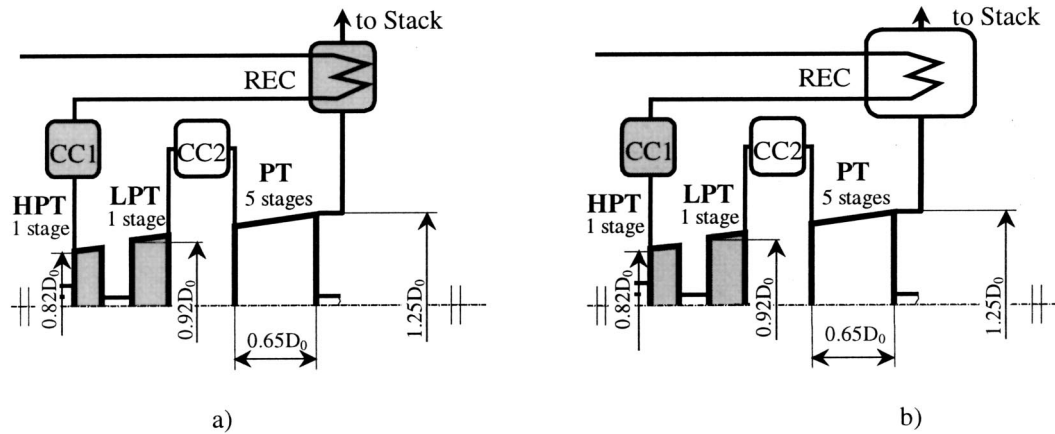


Fig. 8 Schematic layouts of modified Mitsubishi-Rolls-Royce WR-21 with reheat combustor inserted between LPT and PT

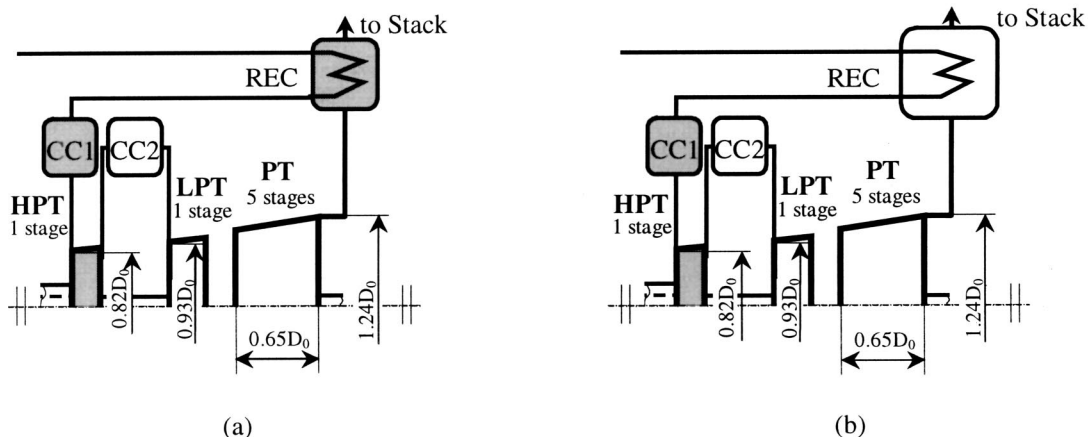


Fig. 9 Schematic layouts of modified Mitsubishi-Rolls-Royce WR-21 with reheat combustor inserted between HPT and LPT

Table 4 Calculated performance summary of the modified WR-21 recuperated ICRH gas turbines

Examined Cases	P_{el} (MW)	η_{el} (%)	T_{out} (°C)	m_{out} (kg/s)	π_{TOT}	π_{LPC}	π_{HPT}	π_{LPT}	S_{IC} (m ²)	$m_{H_2O}^*$ (kg/s)°C/MW	S_{REC} (m ²)
WR-21 REC IC cycle	25.3	41.2	356	73.0	16.2	3.30	1.97	1.64	2763	64.21	831
(Case 1A) WR-21 REC ICRH cycle (new PT)	31.6	38.2	502	73.5	16.2	3.30	1.97	1.64	2763	51.36	831
(Case 2A) WR-21 REC ICRH cycle (new PT and REC) ($T_{out} = T_{out0}$)	31.2	44.7	356	73.2	16.2	3.30	1.97	1.64	2763	52.02	4754
(Case 3A) WR-21 REC ICRH cycle (new PT and REC) ($\epsilon_{REC} = 0.9$)	31.0	47.3	307	73.1	16.2	3.30	1.97	1.64	2763	52.25	11915
(Case 1B) WR-21 REC ICRH cycle (new LPT and PT)	29.7	40.3	434	73.3	16.2	3.30	1.97	1.54	2763	54.54	831
(Case 2B) WR-21 REC ICRH cycle (new LPT, PT, and REC) ($T_{out} = T_{out0}$)	29.5	44.2	356	73.1	16.2	3.30	1.97	1.54	2763	54.92	2793
(Case 3B) WR-21 REC ICRH cycle (new LPT, PT, and REC) ($\epsilon_{REC} = 0.9$)	29.4	48.0	290	73.0	16.2	3.30	1.97	1.54	2763	55.23	12048

design of modified components, higher cycle efficiency was, however, achievable as is evident from the other cases examined (See Table 4).

The incorporation of the reheat combustor between HPT and LPT (Case B) seems to be not economically convenient, due to the larger modification required on the existing WR-21 gas turbine without a significant increase in the cycle efficiency. In fact, a comparison of Case 3B with Case 3A shows an efficiency increase of less than 1.5% with a reduction in power output of about 5% (see Table 4). In Appendix B, the results of PT design for Case 1A to Case 3A and of LPT and PT design for Case 1B to Case 3B are summarized.

Looking at Table 4, the Case 3A, that permits the highest efficiency increase with the lower amount of design modifications, has also a lower equivalent water consumption in the IC than the original case. Moreover, the equivalent water consumption for the IC for Case 3A is lower compared to the cooling water required (greater than about 150 [(kg/s)°C/MW]) for a same size combined cycle system equipped with Brayton cycle gas turbine. On the contrary, the recuperator surface area is much greater than the original design and that will impact the total equipment cost.

Moreover, the cycle efficiency of the modified machine (Case 3A) is found comparable to that of a same power rating combined cycle system, making the modified WR-21 economically attractive.

A Comparative Performance of the Best Modified Gas Turbines. A comparative performance of the three selected gas turbines with their modified counterparts having the best performance is summarized in Table 5. The following main observations can be made from the performance data presented in Table 5:

- The recuperated ICRH cycle derived from GT-24 machine achieved an efficiency and power of 47.4% and 216.7 MW, respectively (an improvement of 26% and 21% in cycle efficiency and net power output, respectively in comparison to the existing machine).
- The recuperated ICRH cycle derived from LM-6000 machine obtained an efficiency of 45.6% and power of 84.5 MW, re-

spectively (9% and 95% increase in the values of cycle efficiency and net power output, respectively with respect to the original machine).

- The recuperated ICRH cycle derived from WR-21 machine attained an efficiency and power of 47.3% and 31 MW, respectively (15% and 23% increase in the values of cycle efficiency and net power output, respectively compared to the existing machine).
- The equivalent cooling water flow rate, a useful parameter to compare the amount of cooling water required by different type of energy systems, is very low (less than 60 [(kg/s)°C/MW]) compared to a value of 150 [(kg/s)°C/MW] or more for a combined cycle plant using Brayton cycle as the topping cycle and with power rating similar to those of the present study.
- The value of exhaust temperature for modified gas turbines is not particularly high, mainly due to the presence of a recuperator, except for the modified LM-6000, which has an exhaust gas temperature of 495°C, higher than that of the original machine. For the modified GT-24 and WR-21 the exhaust gas temperatures are 380°C, and 307°C, respectively, as given in Table 5.
- Based on this study, it can be stated that the improvements in net power output and the overall cycle efficiency for the selected gas turbines can offset the increased equipment cost. Of course, a more detailed evaluation of the proposed design modifications, including changes in mechanical design, will be necessary.

Concluding Remarks

A step-by-step aerothermodynamic design and performance evaluation methodology for converting an existing gas turbine into a recuperated ICRH cycle gas turbine has been presented in the paper. The discussed methodology was implemented to the three existing gas turbines with varying degree of design complexities.

Table 5 Comparison of the best modified machines with respective existing gas turbines

	GT-24 RH cycle	(2B) GT-24 REC ICRH cycle	LM-6000 Brayton cycle	(4) LM-6000 REC ICRH cycle	WR-21 REC IC cycle	(3A) WR-21 REC ICRH cycle
P_{el} (MW)	179.0	216.7	43.4	84.5	25.3	31.0
η_{el} (%)	37.5	47.4	41.8	45.6	41.2	47.3
T_{out} (°C)	630	380	438	495	356	307
m_{out} (kg/s)	391	391	127	149	73	73
π_{TOT}	30.0	31.0	29.1	34.1	16.2	16.2
$m_{H_2O}^*$ ((kg/s)°C/MW)	...	53.24	...	27.79	64.21	52.25

The aerothermodynamic performance results, obtained for the three selected gas turbines, clearly show that a modified gas turbine with higher thermodynamic performance (high cycle efficiency and increased power output) can be achieved with the use of the proposed design modifications.

In all the range of power rating examined (from 25 MW to 180 MW) for the existing gas turbines, it was possible to modify the selected machines to achieve cycle efficiency greater than 45%. In particular, for the WR 21 modified machine, it is possible to attain an efficiency very close to that of a same size combined cycle system equipped with a Brayton cycle gas turbine, rendering this case particularly attractive (also from the economic point of view).

It must be stated that only the commonly available OEMs data relating to the design and performance have been utilized in the present study and, therefore, the presented results should be cautiously used. Nevertheless, the performance results are encouraging and do provide promising developments worthy of considerations by OEM's for more in-depth analysis employing their in-house knowhow and advanced analysis capabilities.

Acknowledgment

The authors would like to thank Prof. Giorgio Negri di Montenegro for his valuable comments and Marco Serra for his help during calculations. The first author would like to also thank the management of Universal Ensco, for their continued encouragement and financial support for his professional growth.

Nomenclature

- A = annulus area
- $C_s = n/\sqrt{T}$ corrected rotational speed
- D = mean diameter
- D_0 = characteristic geometrical dimension
- DHN = W_2/W_1 "de Haller" number
- HR = heat rate
- h = enthalpy, blade height
- m = mass flow rate
- $m_{H_2O}^*$ = $m_{H_2O} \cdot \Delta T / P_{el}$ equivalent cooling water mass flow rate. Mass flow rate for unit of power output required when the cooling water temperature rise is equal to 1°C
- n = rotational speed
- NTU = number of heat exchanger heat transfer unit
- p = pressure
- Q = exchanged thermal power
- R = mean radius
- $\mathcal{R}$ = degree of reaction

- S = heat exchanger surface area
- T = temperature
- P = power output
- U = blade speed at the mean radius
- V_a = axial component of absolute flow velocity
- V_u = tangential component of absolute flow velocity
- W = relative flow velocity
- $Y_c = (p_{02} - p_{03} / p_{03} - p_3)_c$ total pressure loss due to blade tip clearance
- α = absolute flow angle
- β = relative flow angle
- ε = clearance, heat exchanger effectiveness
- $\phi = V_a / U$ flow factor
- η = efficiency
- $\mu = m \sqrt{T/p}$ corrected mass flow
- π = pressure ratio
- ρ = density
- $\Psi = \Delta h_0 / U^2$ stage loading factor

Subscript

- 0 = design case, total physical state
- 1 = compressor rotor or turbine stator entry section (stage entry section)
- 2 = compressor rotor or turbine stator exit section
- 3 = compressor stator or turbine rotor exit section (stage exit section)
- el = electric
- f = fuel
- in = inlet section
- it = inlet turbine
- p = polytropic
- out = gas turbine discharge (downstream the recuperator, if present)

Acronyms

- GT = gas turbine
- HP = high pressure
- HPC = high pressure compressor
- HPT = high pressure turbine
- IC = intercooler
- ICRH = intercooled reheat cycle
- ISO = international standard organization
- LP = low pressure
- LPC = low pressure compressor
- LPT = low pressure turbine
- PT = power turbine
- REC = recuperator
- TOT = overall

Phase (a)

Design GT data

P_{el} [MW]	HR [kJ/kWh]	η_{el} [%]	π_{TOT}	m_{out} [kg/s]	n [RPM]	T_{it} [°C]	T_{out} [°C]
43.400	8613	41.8	29.1	127	3600	1260	438

Performance analysis based on design GT data

LPC			HPC			CC ₁				HPT		LPT	
m_{in} [kg/s]	η_{LPC}	η_{pLPC}	m_{in} [kg/s]	η_{HPC}	η_{pHPC}	m_{in} [kg/s]	m_f [kg/s]	T_{in} [°C]	η_{CC1}	η_{pHPT}	η_{HPT}	η_{pLPT}	η_{LPT}
124.8	2.1	0.9	124.8	13.86	0.9	109.5	2.2	538	0.995	0.89	4.24	0.883	6.75

Modification to obtain a recuperated ICRH cycle

- Re-design LPC to obtain a $\eta_{LPC} = 2.65$
- Adding a intercooler between LPC and HPC
- Adding a recuperator between HPC and CC₁
- Adding a combustor between HPT and LPT
- Re-design LPT to increase the back pressure at the HPT exit

Phase (b)

New Compressors Design

St.	A_1 [m ²]	R_1 [m]	U_1 [m/s]	$(h/D)_1$	A_3 [m ²]	R_3 [m]	U_3 [m/s]	$(h/D)_3$	$\beta_1 - \beta_3$ [°]	DHN	$\mathcal{R}$	Ψ	ϕ	h [kJ/kg]	T_{03} [°C]	p_{03} [bar]	π	$V_{a2} - V_{a1}$ [m/s]
1	2.51	0.94	353	0.228	2.22	0.94	353	0.202	1.2	0.866	0.93	0.13	0.14	16.7	31.5	1.21	1.19	48.2
⋮																		
6	1.41	0.94	353	0.128	1.28	0.94	353	0.117	4.5	0.752	0.50	0.13	0.14	16.7	119.1	2.69	1.15	53.7

New Turbine Design

St.	R_1 [m]	U [m/s]	A_1 [m ²]	$(h/D)_1$	A_2 [m ²]	$(h/D)_2$	A_3 [m ²]	LPT $(h/D)_3$	$\mathcal{R}$	Ψ	ϕ	T_{01} [°C]	p_{01} [bar]	T_{03} [°C]	p_{03} [bar]	π	h [kJ/kg]
1	0.903	340.3	0.371	0.036	0.464	0.045	0.525	0.052	0.30	1.40	0.54	1231	9.552	1107	6.217	1.54	162.15
⋮																	
5	0.914	344.5	1.552	0.150	1.834	0.177	2.237	0.216	0.50	1.00	0.54	763	1.639	666	1.035	1.58	115.82

New Heat Exchangers Design

IC		REC			
m_{in} [kg/s]	T_{in} [°C]	Q_{IC} [kW]	$\dot{m}_{H_2O}$ [kg/s °C / MW]	S_{IC} [m²]	ϵ_{IC}
144.8	119	11409	27.79	4749	0.749
Side COLD		Side HOT			
m_{in} [kg/s]	T_{in} [°C]	m_{in} [kg/s]	T_{in} [°C]	Q_{REC} [kW]	S_{REC} [m²]
123.8	426	148.7	656	28363	15279
					0.9

New Combustor Design

Phase (c)

Performance evaluation of the obtained recuperated ICRH cycle

LPC			HPC					CC ₁						
m_{in} [kg/s]	η_{LPC}		T_{in} [°C]	η_{HPC}	$(C_p/C_{p0})_{HPC}$	$(n/n_0)_{HPC}$	$\mathcal{A}_{HPC}$	m_{in} [kg/s]	m_f [kg/s]	T_{in} [°C]	η_{cc1}			
144.8	0.9		41	0.889	0.936	0.87	12.88	123.8	2.2	633	0.995			
HPT			CC ₂				LPT			SYSTEM				
T_{in} [°C]	η_{HPT}	$\mathcal{A}_{HPT}$	m_{in} [kg/s]	m_f [kg/s]	T_{in} [°C]	η_{cc1}	T_{in} [°C]	η_{LPT}	$\mathcal{A}_{LPT}$	m_{out} [kg/s]	T_{out} [°C]	P_{el} [MW]	η_{el} [%]	HR [kJ/kWh]
1260	0.874	3.44	136.6	1.7	876	0.9	1260	0.89	9.23	148.7	495	84.5	45.6	7896.5

Check of the most important turbomachinery parameters

IF some parameters are out of range or correspond to not a feasible component **THEN** changing of some input parameters and repeat the calculations

ELSE

END

Appendix B

ALSTOM GT-24

New Compressors Design

LPC (case (1A))																		
St.	A_1 [m ²]	R_1 [m]	U_1 [m/s]	$(h/D)_1$	A_3 [m ²]	R_3 [m]	U_3 [m/s]	$(h/D)_3$	$\beta_1 - \beta_2$ [°]	DHN	$\mathcal{R}$	Ψ	ϕ	h [kJ/kg]	T_{03} [°C]	p_{03} [bar]	π	$V_{u2} - V_{u1}$ [m/s]
1	3.41	1.00	378	0.269	2.88	1.00	378	0.228	2.6	0.848	0.92	0.16	0.25	22.8	38	1.290	1.27	61.7
4	2.07	1.00	378	0.163	1.80	1.00	378	0.143	8.5	0.748	0.50	0.16	0.25	22.8	110	2.520	1.21	68.7

LPC (case (2A))																		
St.	A_1 [m ²]	R_1 [m]	U_1 [m/s]	$(h/D)_1$	A_3 [m ²]	R_3 [m]	U_3 [m/s]	$(h/D)_3$	$\beta_1 - \beta_2$ [°]	DHN	$\mathcal{R}$	Ψ	ϕ	h [kJ/kg]	T_{03} [°C]	p_{03} [bar]	π	$V_{u2} - V_{u1}$ [m/s]
1	3.34	1.01	381	0.261	2.84	1.01	381	0.222	2.5	0.853	0.92	0.15	0.25	22.3	37	1.283	1.27	59.8
8	1.23	1.01	381	0.096	1.10	1.01	381	0.086	8.2	0.755	0.50	0.15	0.25	22.3	199	4.966	1.17	66.6

HPC (case (1B))																		
St.	A_1 [m ²]	R_1 [m]	U_1 [m/s]	$(h/D)_1$	A_3 [m ²]	R_3 [m]	U_3 [m/s]	$(h/D)_3$	$\beta_1 - \beta_2$ [°]	DHN	$\mathcal{R}$	Ψ	ϕ	h [kJ/kg]	T_{03} [°C]	p_{03} [bar]	π	$V_{u2} - V_{u1}$ [m/s]
1	1.36	1.00	379	0.108	1.19	0.99	374	0.096	2.6	0.860	0.93	0.14	0.28	20.4	334	3.08	1.22	55.0
17	0.26	0.92	346	0.025	0.24	0.92	346	0.023	10.0	0.754	0.50	0.17	0.30	20.4	665	29.16	1.11	66.9

HPC (case (2B))																		
St.	A_1 [m ²]	R_1 [m]	U_1 [m/s]	$(h/D)_1$	A_3 [m ²]	R_3 [m]	U_3 [m/s]	$(h/D)_3$	$\beta_1 - \beta_2$ [°]	DHN	$\mathcal{R}$	Ψ	ϕ	h [kJ/kg]	T_{03} [°C]	p_{03} [bar]	π	$V_{u2} - V_{u1}$ [m/s]
1	0.77	0.96	362	0.066	0.66	0.95	359	0.058	3.1	0.844	0.92	0.16	0.29	21.3	62	5.47	1.23	60.0
12	0.21	0.92	345	0.020	0.19	0.91	344	0.018	10.6	0.743	0.50	0.18	0.31	21.3	302	31.59	1.14	70.1

GE LM-6000

New Compressors Design

LPC (case (1))																		
St.	A_1 [m ²]	R_1 [m]	U_1 [m/s]	$(h/D)_1$	A_3 [m ²]	R_3 [m]	U_3 [m/s]	$(h/D)_3$	$\beta_1 - \beta_2$ [°]	DHN	$\mathcal{R}$	Ψ	ϕ	h [kJ/kg]	T_{03} [°C]	p_{03} [bar]	π	$V_{u2} - V_{u1}$ [m/s]
1	2.16	0.97	367	0.182	1.89	0.97	367	0.159	1.2	0.861	0.93	0.14	0.13	18.6	33.5	1.233	1.22	51.9
5	1.28	0.97	367	0.108	1.15	0.97	367	0.097	4.5	0.743	0.50	0.14	0.13	18.6	111.9	2.534	1.17	57.8

LPC (case (4))																		
St.	A_1 [m ²]	R_1 [m]	U_1 [m/s]	$(h/D)_1$	A_3 [m ²]	R_3 [m]	U_3 [m/s]	$(h/D)_3$	$\beta_1 - \beta_2$ [°]	DHN	$\mathcal{R}$	Ψ	ϕ	h [kJ/kg]	T_{03} [°C]	p_{03} [bar]	π	$V_{u2} - V_{u1}$ [m/s]
1	2.51	0.94	353	0.228	2.22	0.94	353	0.202	1.2	0.866	0.93	0.13	0.14	16.7	31.5	1.21	1.19	48.2
6	1.41	0.94	353	0.128	1.28	0.94	353	0.117	4.5	0.752	0.50	0.13	0.14	16.7	119.1	2.69	1.15	53.7

New Turbine Design

LPT (case (1))																	
St.	R_1 [m]	U [m/s]	A_1 [m ²]	$(h/D)_1$	A_2 [m ²]	$(h/D)_2$	A_3 [m ²]	$(h/D)_3$	$\mathcal{R}$	Ψ	ϕ	T_{01} [°C]	p_{01} [bar]	T_{03} [°C]	p_{03} [bar]	π	h [kJ/kg]
1	0.89	335.5	0.362	0.037	0.449	0.045	0.506	0.051	0.30	1.4	0.54	1231	8.620	1112	5.691	1.52	157.57
5	0.89	335.5	1.436	0.143	1.684	0.167	2.033	0.202	0.50	1.0	0.54	781	1.585	688	1.025	1.55	112.55

LPT (case (4))																	
St.	R_1 [m]	U [m/s]	A_1 [m ²]	$(h/D)_1$	A_2 [m ²]	$(h/D)_2$	A_3 [m ²]	$(h/D)_3$	$\mathcal{R}$	Ψ	ϕ	T_{01} [°C]	p_{01} [bar]	T_{03} [°C]	p_{03} [bar]	π	h [kJ/kg]
1	0.903	340.3	0.371	0.036	0.464	0.045	0.525	0.052	0.30	1.4	0.54	1231	9.552	1107	6.217	1.54	162.15
5	0.903	340.3	1.552	0.150	1.834	0.177	2.237	0.216	0.50	1.0	0.54	763	1.639	666	1.035	1.58	115.82

MITSUBISHI-ROLLS-ROYCE WR-21

New Turbine Design

PT (case (1A), (2A) and (3A))																	
St.	R_1 [m]	U [m/s]	A_1 [m ²]	$(h/D)_1$	A_2 [m ²]	$(h/D)_2$	A_3 [m ²]	$(h/D)_3$	$\mathcal{R}$	Ψ	ϕ	T_{01} [°C]	p_{01} [bar]	T_{03} [°C]	p_{03} [bar]	π	h [kJ/kg]
1	0.714	269.2	0.378	0.059	0.446	0.069	0.489	0.076	0.30	1.4	0.54	982	4.965	890	3.592	1.38	101.42
5	0.714	269.2	1.037	0.162	1.158	0.181	1.317	0.206	0.50	1.0	0.54	676	1.388	615	1.025	1.35	72.45

LPT (case (1B), (2B) and (3B))																	
St.	R_1 [m]	U [m/s]	A_1 [m ²]	$(h/D)_1$	A_2 [m ²]	$(h/D)_2$	A_3 [m ²]	$(h/D)_3$	$\mathcal{R}$	Ψ	ϕ	T_{01} [°C]	p_{01} [bar]	T_{03} [°C]	p_{03} [bar]	π	h [kJ/kg]
1	0.566	321	0.464	0.116	0.567	0.141	0.653	0.163	0.35	1.3	0.22	971	8.299	864	5.396	1.54	133.52

PT (case (1B), (2B) and (3B))																	
St.	R_1 [m]	U [m/s]	A_1 [m ²]	$(h/D)_1$	A_2 [m ²]	$(h/D)_2$	A_3 [m ²]	$(h/D)_3$	$\mathcal{R}$	Ψ	ϕ	T_{01} [°C]	p_{01} [bar]	T_{03} [°C]	p_{03} [bar]	π	h [kJ/kg]
1	0.697	262.7	0.326	0.054	0.388	0.064	0.426	0.070	0.30	1.4	0.54	864	5.396	785	3.849	1.40	96.58
5	0.697	262.7	0.939	0.156	1.055	0.173	1.209	0.198	0.50	1.0	0.54	579	1.414	519	1.025	1.38	68.99

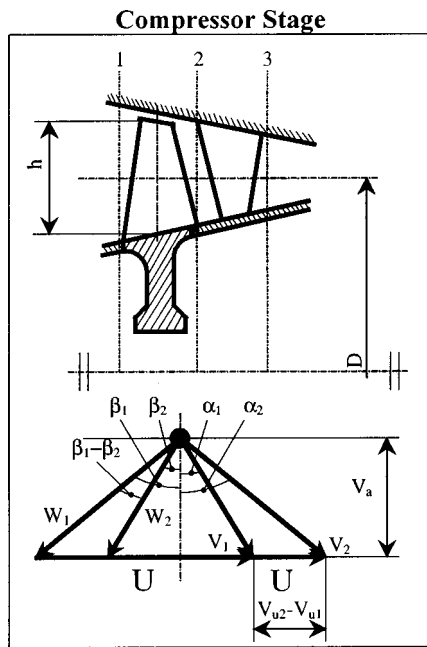


Fig. C1 Compressor stage schematic layout and corresponding velocity triangles

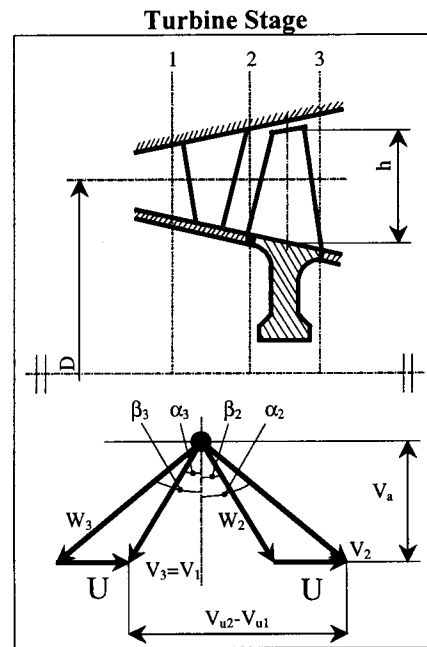


Fig. C2 Turbine stage schematic layout and corresponding velocity triangles

References

- [1] Oganowski, G., 1990, "GE LM-6000 Development of the First 40% Thermal Efficiency Gas Turbine," GE Marine & Industrial Engines Publication GE817.
- [2] Imwinkelried, B., 1994, "ABB's Advanced Gas Turbines GT-24/GT-26: How the Most Efficient Gas Turbine Family was Tested and Validated in a 4-Year Program," Technical Paper Presented at PowerGen '94 AMERICAS, Dec. 7-9, Orlando, FL.
- [3] Michael, V., 1995, "A Turbine for Tomorrow's Navy," Mech. Eng. (Am. Soc. Mech. Eng.), Sept. pp. 70-73.
- [4] El-Masri, M. A., 1987, "Thermodynamics and Performance Projections for Intercooled/Reheat/Recuperated Gas Turbine Systems," ASME Paper No. 87-GT-108.
- [5] Macchi, E., Bombarda, P., Chiesa, P., Consonni, S., and Lozza, G., 1991, "Gas Turbine Based Advanced Cycles for Power Generation. Part B: Performance Analysis of Selected Configurations," International Gas Turbine Congress, Yokohama, Oct.
- [6] Bhargava, R., Bianchi, M., Negri di Montenegro, G., and Peretto, A., 2000, "Thermo-Economic Analysis of an Intercooled, Reheat and Recuperated Gas Turbine for Cogenerative Applications: Part I—Base Load Operation," ASME Paper No. 2000-GT-0316.
- [7] Bhargava, R., Negri di Montenegro, G., and Peretto, A., 2001, "Thermo-economic Analysis of an Intercooled, Reheat and Recuperated Gas Turbine for Cogeneration Applications—Part II: Part-Load," ASME Paper No. 2001-GT-0207.
- [8] Bhargava, R., and Peretto, A., 2001, "A Unique Approach for Thermo-Economic Optimization of an Intercooled, Reheat and Recuperated Gas Turbine for Cogeneration Applications," ASME Paper No. 2001-GT-0206.
- [9] Thermoflex 6 Release 2, 2001, Thermoflow, Inc., Sudbury, MA.
- [10] Saravanamuttoo, H. I. H., and Mac Isaac, B. D., 1983, "Thermodynamic Models for Pipeline Gas Turbine Diagnostics," ASME J. Eng. Gas Turbines Power, **105**, pp. 875-884.
- [11] Kurzke, J., and Riegler, C., 2000, "A New Map Scaling Procedure for Preliminary Conceptual Design of Gas Turbines," ASME Paper No. 2000-GT-0006.
- [12] Cumpsty, N. A., 1989, *Compressor Aerodynamics*, Longman, London.
- [13] Cohen, H., Rogers, G. F. C., and Saravanamuttoo, H. I. H., 1999, *Gas Turbine Theory*, 4th Ed., Longman, London.
- [14] Freeman, C., and Dawson, R. E., 1983, "Core Compressor Developments for Large Civil Jet Engines," Proceedings, International Gas Turbine Congress, Tokyo, Japan, Oct., Paper No. 83-Tokyo-IGTC-46.
- [15] Wisler, D. C., 1988, "Advanced Compressor and Fan Systems," GE Aircraft Engines, Cincinnati, OH.
- [16] Howell, A. R., and Bonham, R. P., 1950, "Overall and Stage Characteristics of Axial Flow Compressors," Proc. Inst. Mech. Eng., **163**, pp. 235-248.
- [17] Benvenuti, E., Bettocchi, R., Cantore, G., Negri di Montenegro, G., and Spina, P. R., 1993, "Gas Turbine Cycle Modeling Oriented to Component Performance Evaluation From Limited Design or Test Data," Proceedings, 7th ASME COGEN-TURBO, Bournemouth, UK, IGTI, Atlanta, **8**, pp. 327-337.
- [18] Benvenuti, E., and Gusso, R., 1985, "Design and Development of the PGT10 Heavy-Duty Advanced Gas Turbine," ASME Paper No. 85-IGT-21.
- [19] Bianchi, D., Noccioni, P., and Silvestri, C. J., 2000, "The New PGT5B/2, a State-of-the-Art 6MW Two-Shaft Gas Turbine," ASME Paper No. 2000-GT-0561.
- [20] Smith, S. F., 1965, "A Simple Correlation of Turbine Efficiency," J. R. Aeronaut. Soc., **69**, pp. 467-470.
- [21] Ainley, D. G., and Mathieson, G. C. R., 1951, "A Method of Performance Estimation for Axial Flow Turbines," Aeronautical Research Council, R&M 2974.

Mustapha Chaker
Director,
Research and Development

Cyrus B. Meher-Homji
Chief Engineer

Thomas Mee III
Chairman and CEO

Gas Turbine Division,
Mee Industries, Inc.,
204 West Pomona Avenue,
Monrovia, CA 91016

Inlet Fogging of Gas Turbine Engines—Part I: Fog Droplet Thermodynamics, Heat Transfer, and Practical Considerations

The inlet fogging of gas turbine engines for power augmentation has seen increasing application over the past decade yet not a single technical paper treating the physics and engineering of the fogging process, droplet size measurement, droplet kinetics, or the duct behavior of droplets, from a gas turbine perspective, is available. This paper provides the results of extensive experimental and theoretical studies conducted over several years coupled with practical aspects learned in the implementation of nearly 500 inlet fogging systems on gas turbines ranging in power from 5 to 250 MW. Part I of the paper covers the underlying theory of droplet thermodynamics and heat transfer, and provides several practical pointers relating to the implementation and application of inlet fogging to gas turbine engines. [DOI: 10.1115/1.1712981]

Introduction and Background

Gas turbine output is a strong function of ambient air temperature, with power output dropping by 0.54–0.90% for every 1°C rise in ambient temperature. This loss in output presents a significant problem to utilities, cogenerators and merchant power plants when electric demands are high during the hot months. One way to counter this drop in output is to cool the inlet air.

Over the past decade and especially over the past five years, the application of inlet fogging for the power augmentation of gas turbines has become increasingly popular. It is estimated that approximately 700 gas turbines have been fitted with fogging systems at this time including many modern F class gas turbines.

In the rapidly deregulating power generation market, the structure of supply agreements, and the dynamics of an open market, usually mean that power producers are paid significantly more for power generated during high demand periods (typically hot summer peaks). This creates an incentive to attempt to overcome the inherent loss of gas turbine power output during periods of high ambient temperature. Peaking power plants need to augment power during high demand periods. Coupled with the need for power augmentation is the requirement to accomplish this with a minimum capital cost for the incremental power generated. High-pressure inlet fogging fits this niche. The concept is a simple one in which a direct evaporation effect is derived by the use of fog generated by high-pressure pumps and atomizing nozzles. The fog evaporates in the inlet duct and cools the air down to the wet bulb temperature.

Fog intercooling,¹ which has been applied from the early days of gas turbine and jet engine technology is a technique that consists of spraying more fog than will evaporate under the given ambient temperature and humidity conditions so that liquid water droplets enter the compressor. The desired quantum of unevaporated fog is carried with the air stream into the compressor where it evaporates and produces an intercooling effect. The resulting reduction in the work of compression can result in a significant additional power boost and an improved heat rate.

¹Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30562. Manuscript received by IGTI, December 2001, final revision, March 2002. Associate Editor: E. Benvenuti.

²Also known as overspray or wet compression.

A review of the technology may be found in Meher-Homji and Mee [1,2]. A detailed parametric study of inlet fogging in the context of gas turbine design parameters has been conducted by Bhargava and Meher-Homji [3]. Early papers on fog intercooling and wet compression started to appear in the late 1940s including Kleinschmidt² [4], and Wilcox and Trout [5]. Other references include Hill [6] Arsen'ev and Berkovich [7], Nolan and Twombly³ [8], and Utamura et al. [9].

To this point, however, *not a single technical paper exists comprehensively covering the physics and engineering of the fogging process, droplet measurement methods, droplet kinetics, and the duct behavior of droplets, from a gas turbine perspective.*

This paper provides the results of extensive experimental and theoretical studies conducted over several years, coupled with practical aspects learned in the design and implementation of nearly 500 inlet fogging systems on gas turbines ranging from 5 to 250 MW. Part I of the paper covers the underlying theory of droplet thermodynamics, and heat transfer and provides several practical pointers relating to the application of inlet fogging to gas turbine engines. Part II covers details on nozzle technology and droplet measurement, putting the topic in the context of gas turbine fogging. Finally, Part III covers fog behavior in inlet ducts and provides results of several wind tunnel experiments.

Components of a High-Pressure Fogging System

A typical high pressure fogging system consists of

- a high-pressure pump skid with associated pumps and controls as shown in Fig. 1
- a set of fog nozzles located in the intake duct after the filters as shown in Fig. 2. Each impaction pin nozzle in the array produces billions of droplets per second creating a fog as shown in Fig. 3
- a PLC-based control system.

Fundamental Research Efforts Conducted

In spite of fogging being a popular power augmentation strategy there is a scarcity of solid technical data covering the theoretical and practical aspects of the fogging process. This paper, along with Parts II and III, will be the first time that a solid set of

²This paper coined the term “wet compression.”

³Nolan and Twombly covered an application of fog intercooling on GE Frame 5 engines.



Fig. 1 Typical gas turbine inlet fogging skid

experimental and empirical data are provided with information that can help users understand the issues and complexities involved in the fogging process. The paper is based on several years of fundamental and applied research programs conducted in the area which include:

- droplet kinetics and thermodynamics,
- droplet collision, coalescence and shattering dynamics,
- droplet/airflow interface dynamics,
- computational fluid dynamics (CFD) studies of gas turbine inlet ducts,
- theoretical and experimental studies of droplet trajectories both in still air and in a moving airflow,
- optimization studies relating to nozzle array placement in gas turbine inlet air ducts,
- comparative experimental analysis of nozzle performance and standardization approaches for droplet measurement for gas turbine applications,
- studies of drains, silencer and trash screen interactions, and
- development of design tools for ensuring the structural integrity of the fogging nozzle manifolds in intake ducts.

Much of the work presented here has been validated by the use of the wind tunnel that is shown in Fig. 4. In addition to experimental verifications, considerable field experience and lessons



Fig. 2 Fog nozzle manifold operating in the inlet duct of a GE-7EA gas turbine



Fig. 3 Fog from an impaction pin-fogging nozzle operating at 2000 psig (138 barg)

learned are also embedded in this paper. The authors hope that turbine operators will find the information in this paper helpful when specifying equipment and evaluating specific fog system designs.

Issues of Importance in Inlet Fogging

From a gas turbine operator's perspective, there are several issues that are of importance when using, or considering the use of, inlet fogging systems. These include

- an understanding of what actually happens with the evaporation of water in the inlet duct and droplet thermodynamics that result in the cooling of the inlet air
- an understanding of the physics of droplet sizing, and the different terminology used, and the significance to gas turbine operation. This is of particular importance because of the wide



Fig. 4 Experimental wind tunnel (10.5 m long, 25 m/sec) used to study droplet kinetics and thermodynamics. This wind tunnel provides dynamic similarity to gas turbine inlet ducts.

range of diameter characterizations used. This is dealt with in this paper briefly; Part II will cover the topic in greater detail.

- An understanding of the different nozzle types offered and the proper testing and evaluation of inlet air fogging nozzles. With the growing popularity of gas turbine inlet air fogging it is becoming increasingly obvious that a standard for measuring droplet sizes generated by fog systems is needed so that gas turbine operators can make informed comparisons of different systems. Fog system manufacturers make various claims based on different testing methods and there is little agreement from one manufacturer to another. The result is confusing and contradictory information that makes it impossible for the gas turbine user to understand the relative performance of competing systems.

- The behavior of fog in gas turbine inlet ducts in terms of rates of evaporation, velocity profiles, and uniformity of distribution.

- A close understanding of the climatic conditions and the co-incident wet bulb and dry bulb temperature history for a particular location. While this may seem to be a trivial topic, there is very little data available in an easy to use form to enable gas turbine users to estimate the amount of power boost that can be attained through the year, or during typical peaking periods, at a particular site. Chaker et al. [10,11] have provided comprehensive data of evaporative cooling potential in numerous locations in the USA and internationally, using the concept of evaporative cooling degree hours (ECDH). Tabulations provided in these papers allow users to quickly estimate the cooling potential available at their site and to estimate the power boost available for their turbines.

- Numerous practical aspects relating to gas turbine operation including issues such as droplet collision and agglomeration, duct wetting, duct drainage, icing considerations, avoidance of foreign object damage (FOD), corrosion considerations in the duct, compressor inlet distortion (uneven temperature and pressure profiles at the compressor inlet which can cause blade vibration in the axial flow compressor), etc. This set of papers covers these important practical issues.

Issues and Complexities Relating to Droplet Behavior

The study of the sizing and the thermo-kinetics behavior of fog droplets is a complex subject. Care has to be taken when studying the continuous changes that occur in both space and time with the billions of droplets that are emitted every second by each nozzle.

Small changes in any test parameter, the measurement methodology, environmental conditions, and the measurement device itself, can result in significant differences in the characterization of nozzles.

The reason for injecting water droplets is to increase the relative humidity of the airflow by evaporation of the droplets, which in turn leads to a decrease in the temperature resulting in an increase in the air mass flow⁴ and more power for the gas turbine. Compressor work is also reduced during inlet fogging as work of compression is proportional to compressor inlet temperature, this also results in more output power. While this may intuitively seem to be a simple matter, there are several interacting factors that define the success and efficiency of the process.

An analysis of droplet sizes produced by inlet air fogging nozzles must take many factors into account and it is therefore important to have a complete understanding of the atomization process. Results can vary dramatically depending on several key parameters, such as the velocity of the air stream in which the nozzle is located, the properties of the water itself (including temperature, dissolved mineral content, etc.), the pressure applied on the liquid, the geometry of the nozzle, and consequently the spray angle and even the location in the plume where the measurement is taken. Some of these factors are highly critical, as will be discussed ahead.

Droplet size itself is a critical factor for the inlet air fogging process. The fog droplets quickly take the velocity of the airflow

(within a few centimeters from the nozzle orifice) and they typically spend just one to two seconds in the airflow before they reach the compressor inlet. The droplets must be small enough that they evaporate, or substantially evaporate, before reaching the compressor inlet. There are many potential drawbacks to using large droplets for the inlet fogging process. These include

- large droplets may not completely evaporate leading to under-saturated air at the compressor inlet (i.e., less than 100% evaporative efficiency)
- large droplets are more likely to distress compressor blades and/or blade coatings
- large droplets are more likely to be removed from the air stream by gravity or by impaction on obstructions in the inlet ducts, such as silencers, trash screens, etc. (Small droplets more readily follow the airflow around obstructions)
- more fallout can lead to excessive pooling of water on the duct floor, thus requiring more extensive drainage systems and leading to more potential for corrosion of the duct surfaces
- excessive pooling at the compressor inlet can result in large slugs or sheets of water being suctioned off the duct floor and ingested by the compressor. This can lead to blade erosion and/or coating wear and, in the most severe cases, may cause compressor blade tip deformation.

Smaller droplets, in the range of 5 to 15 microns, have many advantages for gas turbine inlet air fogging applications. Experience has shown that droplets in this size range do not cause erosion of compressor blading or wear of blade coatings (provided the nozzles are installed so that excessive pooling is avoided). Small droplets are also more likely to follow the airflow around obstructions in the duct and less likely to fall out (i.e., they cause less pooling in the duct) and they provide more evaporative surface area. Evaporation rate is a strong function of the exposed surface area because evaporation occurs only at the water/air interface.

To put it in perspective, a ten-micron droplet has only about 13% the mass of a 20-micron droplet and a given mass of water divided into 10-micron droplets results in four times more surface area than the same mass of water divided into 20-micron droplets. Obviously a 10-micron fog will evaporate much faster, and is much less likely to impact on obstructions or fall out on the duct floor, than a 20-micron fog.

There are several factors that influence droplet size distribution:

- type of nozzle—impaction-pin and swirl-jet nozzles are the types currently offered by GT inlet air fogging suppliers. These are described in more detail ahead
- operating pressure—typically ranges from 1000 to 3000 psig (70 to 210 barg)
- nozzle capacity—typically ranges from about 2 gph to about 5 gph (7 to 18 l/h).

Nozzle flow rate is an important issue. Nozzles with relatively high flow rates have some advantages (fewer nozzles to install and service, etc.) but also have disadvantages (fewer points of emission can mean poor distribution of fog in the inlet air stream, resulting in poor air/fog mixing, uneven cooling or incomplete cooling of the air stream). Also, other factors being the same, a larger flow rate nozzle will, as a general rule, produce larger droplets than a small flow nozzle. An exception to this general rule is discussed in Part II of this paper—swirl-jet nozzles have higher internal frictional losses than impaction-pin nozzles so that the latter can produce smaller droplets at slightly higher flow rates.

Droplet Diameter Nomenclature

The spray from any water-atomizing nozzle contains a range of droplet sizes and many statistical techniques are employed to characterize sprays. The more obvious, but least useful, is the average droplet size. Taking the arithmetic average of all the drop-

⁴By increasing the air density.

lets found in a spray leads to a deceptively small diameter number because, as discussed above, the larger droplets offer much less surface area and carry far more mass.

Therefore, several statistical methods have been developed for defining the range of droplet diameters found in a spray. These statistical numbers give a fictional droplet diameter, which “characterizes” the spray in a meaningful way. Most often these diameters are given in microns.

Two of these methods are most applicable to gas turbine inlet fogging applications. They are

- **Sauter Mean Diameter (SMD or D32)** is the diameter of a hypothetical droplet whose ratio of volume to surface area is equal to that of the entire spray. Since it deals with surface area, Sauter mean diameter is a good way to describe a spray that is used for processes involving evaporation. To enhance droplet evaporation one has to maximize the active surface area and minimize the internal volume of the droplet, thus the lower the Sauter mean diameter, the more rapid the evaporation process.

- **Dv90** is a diameter for which 90% of the water volume in the spray is less than or equal to. A small value of this number indicates that a very small number of larger droplets are present. A small Dv90 minimizes the potential for impaction on obstructions and droplet fallout due to gravity (both of which reduce water pooling on the duct floor) and reduces the potential for compressor blade distress.

Droplet Testing Standards

As mentioned above, there are no existing testing standards for fog sprays in the gas turbine inlet air fogging industry. Different fog system suppliers measure droplets under different conditions, at different locations in the spray plume and with various kinds of droplet measuring instruments.

This has resulted in a situation in which fogging system users find it impossible to meaningfully compare competing nozzle designs. Statements relating to droplet sizes without a solid analytical description of what specifically was measured and how readings were taken make no engineering sense and must be qualified carefully before a gas turbine user can make an intelligent decision.

For example, if a fog system supplier reports droplet sizes that were taken at a position 2 to 3 inches (50 to 75 mm) in front of the nozzle orifice in a high-velocity air stream, this results in extremely misleading information because it ignores the significant part of the spray plume. Furthermore, all atomizing nozzles currently in use for gas turbine inlet fogging applications have a conical spray plume. The jet action of the spray creates a low-pressure area in the middle of the conical plume and the smallest droplets are readily pulled into this area, because they more quickly follow the airflow, while all of the larger droplets remain in the edge of the plume, outside of the area where measurements were taken. Even if the nozzle is tested in a high-velocity airflow, most of the large droplets never enter the measurement area.

For these reasons, it is important that an industry standard be developed to provide a uniform and repeatable approach that makes meaningful comparison possible. Part II of the paper covers one such proposed standard.

Factors Affecting Droplet Size Measurements

Several key factors have a significant impact on droplet size measurements:

- airflow velocity during the measurement
- location in the nozzle spray plume where the measurements are taken (i.e., center or edge)
- ambient temperature and humidity conditions around the nozzle
- the difference in liquid characteristics may also account for differences in size distribution but, since inlet fogging always

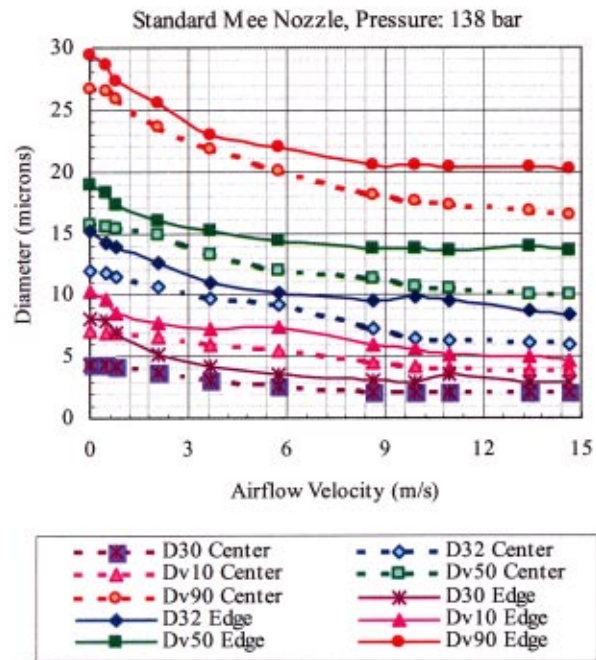


Fig. 5 Effect of air velocity, measurement locations, and diameter definition on droplet size

deals with ambient temperature demineralized water, differences in liquid surface tension and viscosity can be largely ignored.

To illustrate the effect of measurement location and flow velocity on droplet size, diameter curves of an impaction-pin nozzle made by our company, for gas turbine inlet fogging applications is shown in Fig. 5. This data was developed from measurements taken in the aforementioned wind tunnel and provides data for a variety of measurement locations over a range of velocities. This figure clearly indicates the problem faced by a gas turbine user when he is told, for example, that a nozzle produces a 20 microns droplet without also being told how and where and under what conditions the measurements were taken.

Note for instance the red curve at the top of the chart in Fig. 5. The Dv90 diameter, with zero airflow velocity, measured at the edge of the plume is 30 microns, while the same Dv90 diameter for the same nozzle operating at the same pressure is about 16 microns when measured with high airflow velocity in the middle of the plume (second curve from the top, right side of the chart) velocity of 9 m/s (~1800 ft/min). It's easy to see how someone trying to evaluate one nozzle against another could reach a totally false conclusion unless they have an understanding of the subject and the information on how, where and under what conditions the measurements were taken. Measurement location and air velocity alone can account for a very large variation in the reported diameter of a droplet. The reason for the decrease in droplet size at higher airflow rates has to do with the fact that airflow moves the smaller droplets out of the path of the larger droplets and, thus, reduces collision and agglomeration. This is discussed in more detail in Part II of the paper.

Droplet Heat and Mass Transfer Model

Prior to the study of the complex properties and behavior of the whole mass of fog droplets emitted from the nozzle, it is helpful to study the properties and behavior of individual droplets, and then extend this study to the interaction between a single droplet and the other droplets and the interaction with the carrying phase (air). The study of an isolated droplet is first treated using the classical assumption of spherical symmetry for both the liquid

droplet and the surrounding air. The behavior of a single droplet injected into the airflow will be analyzed then a study of the effect of the velocity on droplet thermodynamic and trajectory will be done.

The study of big number of droplets is currently being analyzed using the Lagrange/Euler approach (computational fluid dynamics (CFD) analysis) and will be reported on in a later paper.

Model for an Isolated Droplet. In order to cover the underlying basis of the model developed ahead, the basic theory and equations used are provided here. Heat and mass transfer occur at the interface of the droplet and the surrounding air, mainly by convection and diffusion. This transfer depends on the thermal characteristics of the air (hygrometry, total pressure, and temperature) and the droplet (volume, liquid characteristics, and temperature).

A model has been developed below to study the evaporation process of the droplets. The main object of this model is to show the lifetime of a droplet in a limited volume as it reaches saturation within this volume. Details of the model can be found in Chaker [12].

In this model we take into account the fact that the thermokinetic behavior of the water droplets, which represents the dispersed phase will be influenced by that of the air, which represents the continuous phase.

The heat quantity stored in a droplet between instant t and instant $t + \Delta t$ is given by

$$\Phi = m_d C p_d \frac{(T_{d(t+\Delta t)} - T_{d_t})}{\Delta t}. \quad (1)$$

Therefore, the droplet temperature at the instant $t + \Delta t$ is

$$T_{d(t+\Delta t)} = T_{d_t} + \frac{\Delta t \times \Phi}{m_d C p_d}. \quad (2)$$

The thermal equilibrium between the droplet and the surrounding air occurs when the quantity of heat received by the droplet Φ , in a given time, is equal to the flux exchanged by convection and latent heat

$$\Phi = \Phi_{\text{conv}} + \Phi_{\text{lat}}. \quad (3)$$

Convective Exchange. The rate exchanged by thermal convection between the droplet and the air is given by

$$\Phi_{\text{conv}} = h_{cv} \times S_d \times (T_a - T_d). \quad (4)$$

h_{cv} is the coefficient of thermal convective exchange ($\text{W.m}^{-2}.\text{K}^{-1}$), and is derived from the Nusselt number

$$\text{Nu} = \frac{h_{cv} D_d}{\lambda_a}. \quad (5)$$

λ_a is the thermal conductivity of the air ($\text{W.m}^{-1}.\text{K}^{-1}$). Using a linear regression of the data shown in the Handbook of Chemistry [13], for T_a varying between (253 K < T_a < 373 K), we find

$$\lambda_a = (46.766 + 0.7143 T_a) \times 10^{-4}. \quad (6)$$

For natural convection, the Nusselt number is function of the thermal Grashof and Prandtl numbers and is given by the relation, [14],

$$\text{Nu} = 2 + 0.6 \text{Gr}_t^{0.25} \times \text{Pr}^{0.33} \quad (7)$$

where Pr is the Prandtl number, given by

$$\text{Pr} = \frac{\mu_a \times C p_d}{\lambda_a}. \quad (8)$$

μ_a is the dynamic viscosity of the air ($\text{kg.m}^{-1}.\text{s}^{-1}$). For 273 K < T_a < 373 K it is given by, [13],

$$\mu_a = (0.004823 \times T_a + 0.3976) \times 10^{-5}. \quad (9)$$

The thermal Grashof number Gr_t is given by

$$\text{Gr}_t = \frac{\rho_a^2 \cdot g \cdot \beta \cdot (T_a - T_d) \cdot (D_d)^3}{\mu_a^2} \quad (10)$$

where g is the acceleration of gravity (9.81 m.s^{-2}).

β is the thermal dilation coefficient, and (for a perfect gas) is equal to

$$\beta = \frac{1}{\rho_a} \left(\frac{\partial \rho_a}{\partial T} \right)_{C,p} = \frac{1}{T_a}. \quad (11)$$

By combining Eqs. (5), (7), (10), (11), the coefficient of convective exchange becomes

$$h_{cv} = \frac{\lambda_a}{D_d} \left[2 + 2.6 \times \left(\frac{\rho_a^2 \cdot g \cdot (T_a - T_d) \cdot (D_d)^3}{\mu_a^2 \cdot T_a} \right)^{0.25} \left(\frac{\mu_a}{\rho_a D f_a} \right)^{0.33} \right]. \quad (12)$$

Latent Heat Exchange. The latent heat exchange is equal to

$$\Phi_{\text{lat}} = \frac{\Delta m_d L_v}{\Delta t}. \quad (13)$$

L_v is the latent heat of water vapor (J.kg^{-1}) given by

$$L_v = 1000(2498 - 2.413 \times T_d). \quad (14)$$

The mass variation of the droplet (Δm_d) which occurs due to the flux exchanged by evaporation between the droplet surface and the surrounding air during the interval time Δt is given by

$$\Delta m_d = -\Delta t \times S_d \times \phi_{\text{evap}} \quad (15)$$

where S_d is the exchange surface of the droplet with the surrounding air (m^2).

We assume that the droplet and its environment make an isolated system with their temperatures being uniform and designated as T_d and T_a , respectively, [15]. When the ambient air is not saturated with water vapor, the droplet evaporates naturally.

The mass flux may then be written as

$$\phi_{\text{evap}} = D f_{\text{mass}} \cdot \Delta C_{\text{mass}} \quad (16)$$

where $D f_{\text{mass}}$ is the coefficient of mass diffusion (m.s^{-1}).

ΔC_{mass} is the difference of mass concentrations of water vapor between the droplet-air interface and the air (freestream) (kg.m^{-3}) and is given by

$$\Delta C_{\text{mass}} = C_{\text{mass}_d} - C_{\text{mass}_a}. \quad (17)$$

By assuming the air around the droplet is a perfect gas, the concentration C_{mass_i} may be written as

$$C_{\text{mass}_i} = \frac{\mathcal{M} \cdot p_i}{R \cdot T_i} \quad (18)$$

Therefore, ΔC_{mass} may be written as

$$\Delta C_{\text{mass}} = \frac{\mathcal{M}}{R} \cdot \left(\frac{p_{v_d}}{T_d} - \frac{p_{\text{vap}}}{T_a} \right) \quad (19)$$

where $\mathcal{M}$ is the molar mass (kg.mole^{-1}), and R is the universal gas constant ($8.32 \text{ J.mole}^{-1}.\text{K}^{-1}$) and p_{vap} and p_{v_d} are the vapor pressure of the air and the interface air-droplet (Pa), respectively.

The exchange coefficient by mass diffusion is given by the relation

$$D f_{\text{mass}} = \frac{\text{Sh} \cdot D f_a}{D_d} \quad (20)$$

$D f_a$ is the mass coefficient of diffusion, given by

$$D f_a = 2,26 \cdot 10^{-5} \cdot \left(\frac{101325}{P_a} \right) \cdot \left(\frac{T_a}{273.15} \right) \quad (21)$$

where, P_a is the total pressure (Pa) and T_a is the temperature of the air (K).

ρ_a is the density of the humid air ($\text{kg}\cdot\text{m}^{-3}$) and for $T_a \leq 373$ K. It is given by the relation

$$\rho_a = \left(\frac{353.15}{T_a} \right) \left[1 - 0.377 \times \frac{P_{\text{vap}}}{P_a} \right]. \quad (22)$$

P_{vap} is the partial pressure of the water vapor. P_a is the total pressure of the air, which depends on the air temperature. P_{vap} reaches its maximum (saturation) because the quantity of the vapor in the air cannot be higher than the maximum value corresponding to the equilibrium between the water at the vapor phase and the liquid phase. By considering air to be a perfect gas, it is given by the following formula, [16],

$$P_a = e^{77.3417 - 8.2 \times \ln(T_a - 0.15) + 5.7114 \times 10^{-3} \times T_a - 7235.46/T_a - 0.15}. \quad (23)$$

The Sherwood number Sh is a function of the Grashof Gr_m mass number and the Schmidt number Sc for natural mass transfer, as show by the following empirical relation, [14],

$$Sh = 2 + 0.6 Gr_m^{0.25} Sc^{0.33}. \quad (24)$$

The Sc is Schmidt number is given by the equation

$$Sc = \frac{\mu_a}{\rho_a D_{fa}}. \quad (25)$$

The Mass Grashof number is equal to

$$Gr_{\text{mass}} = \frac{\rho_a^2 \cdot g \cdot \beta^* \cdot \Delta C_{\text{mol}} \cdot (D_d)^3}{\mu_a^2}. \quad (26)$$

ΔC_{mol} is the difference in molar concentration of vapor in the ambient air and at the interface droplet air ($\text{mole}\cdot\text{m}^{-3}$). It is equal to

$$\Delta C_{\text{mol}} = \frac{1}{R} \cdot \left(\frac{P_{\text{vap}}}{T_a} - \frac{P_d}{T_d} \right) \quad (27)$$

β^* is the coefficient of mass dilatation ($\text{m}^3\cdot\text{mole}^{-1}$), and equal to

$$\beta^* = - \frac{1}{\rho_a} \cdot \left(\frac{\partial \rho_a}{\partial C} \right)_{T_a, P_a} = - \frac{\mathcal{M}}{\rho_a}. \quad (28)$$

By substituting β^* in Eq. (16), the Grashof mass number becomes

$$Gr_m = \frac{\rho_a^2 \cdot g \cdot (D_d)^3}{\mu_a^2} \left(\frac{P_d \cdot T_a}{P_{\text{vap}} \cdot T_d} - 1 \right). \quad (29)$$

Finally, the evaporative flux may be written in the form

$$\phi_{\text{evap}} = \frac{\mathcal{M} \cdot Sh \cdot D_{fa}}{R \cdot (D_d)} \left(\frac{P_{v_d}}{T_d} - \frac{P_{\text{vap}}}{T_a} \right). \quad (30)$$

Final Equations to Calculate the Temperature of the Droplet and the Air. By combining Eqs. (4) and (30), the heat quantity Φ defined in Eq. (3) becomes

$$\Phi = S_d \left[h_{cv} (T_a - T_d) - \frac{\Delta t \cdot \mathcal{M} \cdot Sh \cdot D_{fa}}{R \cdot D_d} \left(\frac{P_{v_d}}{T_d} - \frac{P_{\text{vap}}}{T_a} \right) \right]. \quad (31)$$

The equation of the droplet temperature at the instant $t + \Delta t$ may be written as follows, by replacing the heat quantity Φ (31) by its value in Eq. (2),

$$T_{d(t+\Delta t)} = T_{d_t} + \frac{\Delta t \cdot S_d}{m_d C_p} [h_{cv} \cdot (T_{a_t} - T_{d_t}) - L_v \phi_{\text{evap}}]. \quad (32)$$

Knowing the droplet temperature at the instant $t + \Delta t$, we may now calculate the air temperature around the droplet by establishing a thermal balance with the air

$$m_a C_p (T_{a(t+\Delta t)} - T_{a_t}) = h_{cv} \cdot S_d \cdot (T_{d(t+\Delta t)} - T_{a_t}) \quad (33)$$

where C_p is the specific heat of the air ($\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$), and is given for $273 \text{ K} \leq T_a \leq 373 \text{ K}$ by

$$C_p = 981 + 0.08 \cdot T_a. \quad (34)$$

Finally, the air temperature is equal to

$$T_{a(t+\Delta t)} = T_{a_t} + \frac{h_{cv} \cdot S_d}{m_a \cdot C_p} (T_{d(t+\Delta t)} - T_{a_t}). \quad (35)$$

Droplet in the Airflow (Forced Convection). The above analysis was for an isolated droplet in still air. However, in gas turbine fogging applications, the droplets are injected into a moving airstream at a different velocity. Due to the small size of the droplets and low Reynolds number (< 1), the drag forces are very large and the droplets attain the air stream velocity in a few milliseconds.⁵

Depending on the droplet relative velocity, either natural or forced convection will occur. Natural convection occurs when the relative velocity of a droplet compared to the surrounding air is zero, and forced convection occurs when a relative velocity differential exists.

With forced convection, the Nusselt number is given by the Ranz and Marshall [14] equation

$$Nu = 2 + 0.6 \times Re_d^{0.5} Pr^{0.33}. \quad (36)$$

And for forced mass transfer, the Sherwood number is given by, [14],

$$Sh = 2 + 0.6 Re_d^{0.5} Sc^{0.33} \quad (37)$$

where Re_d is the Reynolds number of the droplet and given by

$$Re_d = \frac{D_d \times \Delta U \times \rho_a}{\mu_a}. \quad (38)$$

The effect of velocity is nonsignificant in our conditions (the effect is less than 10% for 50 microns droplet and further decreases when the droplet size diminishes) because droplets are very small and the velocity response time is lower than 10 ms. This means that the use of isolated droplet model gives a good estimation of the behavior of the droplets in the duct. This has been verified by visual observations taken in a wind tunnel experimental setup.

Computational Model

Based on the equations derived above, a computational model was developed. The model works in iterative manner and provides the transient behavior of fog droplets in terms of the droplet diameter, change in relative humidity and temperature and time to attain saturation. The iterations stop ($I = I_{\text{max}}$) when the air in the volume around the droplet within active radius (RA) becomes saturated or when the droplet evaporates completely. The steps followed by the model shown in Fig. 6.

Model Results Relating to Fog Behavior

From the model developed above, a quantification can be made of fog droplet behavior in gas turbine inlet ducts. As soon as a droplet touches the air, and if the vapor pressure near the droplet surface is higher than the vapor pressure of the air far from the droplet (i.e., the droplet is in unsaturated air), evaporation of the droplet starts to occur. In order to balance the evaporation (mass transfer), the droplet has to loose a quantity of energy (heat transfer), which reduces the temperature of the droplet and of the air surrounding it.

An application of the model developed in the earlier section is provided here. Figures 7 and 8 show the time related behavior of droplets for two droplet sizes. Figure 7 depicts the behavior of a

⁵Less than 10 ms for the largest droplets developed in the MeeFog nozzles.

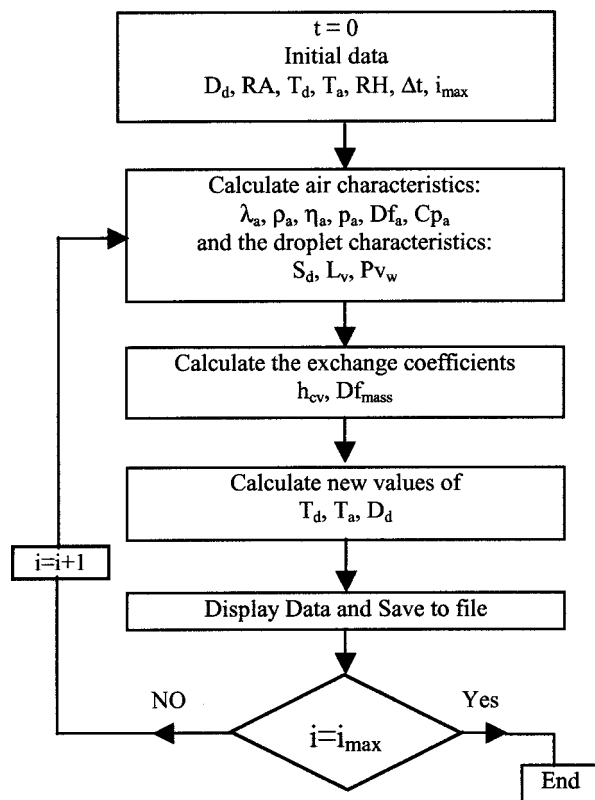


Fig. 6 Computational model for droplet evaporation

14-microns droplet and Fig. 8 for a 30-microns droplet. The dry bulb temperature of the air around the droplet is 35°C (94°F), the RH is 30% and consequently the wet bulb temperature is 21.4°C (70.5°F).

In these examples, we used two different temperatures for the water droplet. In the first the temperature of the droplet was taken as equal to 20°C (68°F) and in the second to as 35°C (94°F).

Analysis of the figures indicate the following:

- the temperature of the droplet in the two cases converges very quickly to 21.4°C (70.5°F), i.e., to the WBT
- the 14-micron droplet evaporated, and brought the surrounding air to the wet bulb temperature, in about 1.5 seconds, while the 30-micron droplet took around 5 seconds. The importance of droplet diameter to affect rapid evaporation is evident from this observation
- the smaller droplet reached a diameter of 3 microns in 1.5 seconds (the residence time in a typical inlet air duct), while the larger droplet was still almost 15 microns in the same time frame
- the effect of the temperature of the water droplet is negligible during the evaporation process. The higher temperature lowers the surface tension of the droplet, and increases the vapor pressure at the droplet surface, and therefore the evaporation is marginally but not significantly quicker.

Figure 9 shows data derived from duct measurements corroborating the general pattern of the temperature fall and relative humidity rise as predicted by model. The abscissa (i.e., the time scale) is a function of droplet size and starting air conditions. However, experience with gas turbines has indicated that temperature reduction to the wet bulb temperature can be obtained with a residence time from 0.8 to 2.0 seconds.

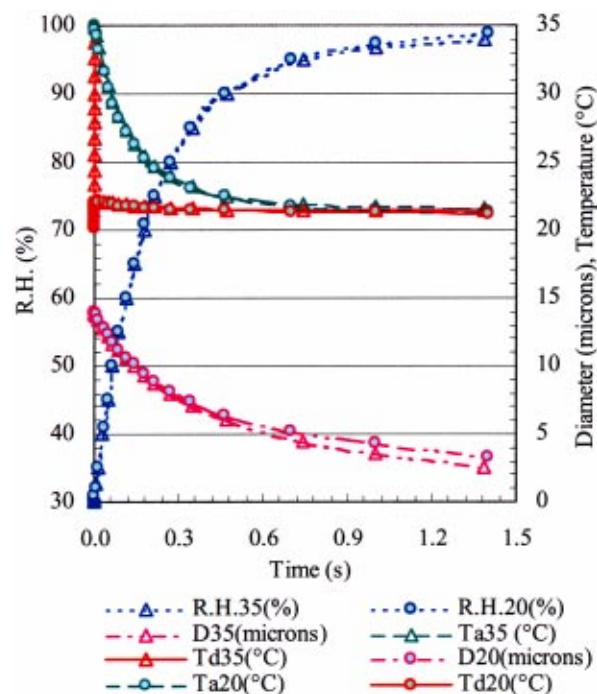


Fig. 7 Transient behavior of a two 14-micron droplets, one at 35°C (94°F) and one at 20°C (68°F); starting air conditions are 35°C and 30% RH

Use of the Model to Show the Interaction of Droplet With the Surrounding Air

In reality, the fog evaporative cooling process for gas turbines involves trillions (10^{12}) of droplets⁶ interacting with the ambient air as they progress down the duct. Therefore, considering only one droplet in isolation can be highly misleading.

To examine the real-world situation more closely we use the concept of active radius (RA), which defines the volume around the droplet for which saturation conditions are calculated. Varying RA can give one an idea of how the evaporation process changes as a function of the how well the fog is mixed with the inlet air. Water vapor will diffuse rapidly across the entire RA volume so that a large droplet will evaporate faster in a larger RA volume. In the real world, droplets are not evenly distributed in the airflow. This is especially true when larger orifice nozzles (with high flow rates) are used because the spacing between the nozzles is greater, leaving large sections of unfogged air.

The model calculates the temperature and relative humidity conditions over time for different RA values, providing results as shown in Figs. 10 and 11. Figure 10 shows results of the evaporation of a droplet of 50 microns at 20% relative humidity (Fig. 10(a, b, and c) in the left column) and 60% (Fig. 10(d, e, and f) in the right column), both at an initial ambient temperature of 30°C. The effect of RA size on droplet evaporation rate can be seen by comparing Fig. 10(a) (RA 81) and 10(c) (RA 500). Note that with the larger RA the droplet size decreases rapidly but the temperature and humidity remain nearly constant. With actual, real-world conditions RA tends to be smaller, because the droplets are in reality very close together, particularly inside the initial spray plumes. A smaller RA means that the temperature and humidity in the RA volume change quickly but the droplet size remains quite large. For example, in the time frame allowed in real gas turbine

⁶One impaction pin nozzle operating at 138 barg can create over three billion droplets per second and a large gas turbine may have as many as one thousand fog nozzles.

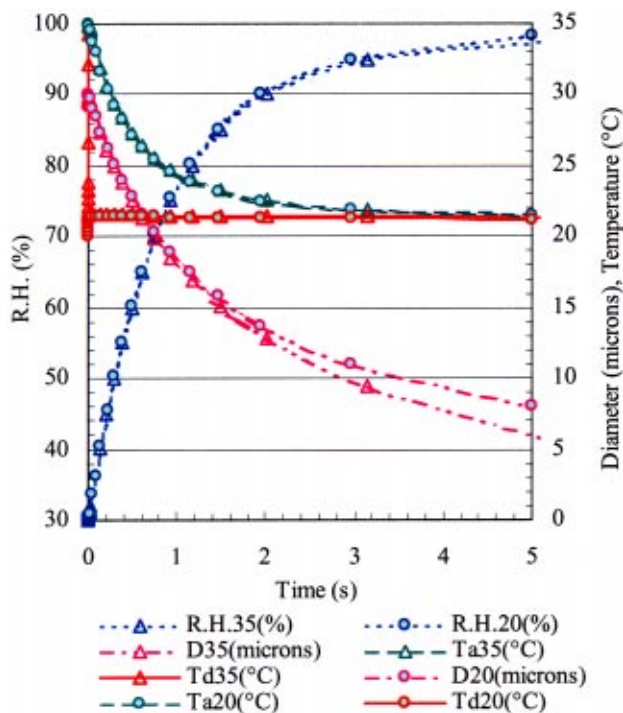


Fig. 8 Transient behavior of two 30-micron droplets, one at 35°C and one at 20°C; starting air conditions are 35°C and 30% RH

ducts (about two seconds), the 50-micron droplet would be approximately 30 microns when it reached the compressor inlet.

The situation is further exacerbated by the fact that, in real-world installations, the fog-nozzle plumes do not completely cover the duct cross section, so that a substantial portion of the air passes the nozzle manifolds without mixing into the fog plumes (especially true for large-orifice high-flow nozzles). Water vapor from the spray plumes must diffuse into the space between the spray plumes and, more significantly, heat must flow from the uncooled spaces between the plumes to the cool air inside the plumes. Neither of these processes are very rapid. The information presented here shows that the real-world evaporation process, where droplet density is very high, is far slower than when a droplet is considered in isolation, with a large RA.

It also points out another key factor, if there are a mix of droplet sizes present, with some droplets above 30 microns, the smaller droplets will evaporate first increasing the local relative humidity and so making it even more difficult for the larger droplets to evaporate. Therefore the large droplets pose an even bigger risk for erosion should they be ingested into the compressor and are more likely to impact or fall out and cause excessive wetting of the duct floor.

In a gas turbine, the residence time available is 1–2 seconds, and hence droplets larger than about 30 microns will not evaporate and will, in fact, be quite large when they enter the compressor.

Practical Considerations

When specifying or evaluating an inlet fog system, the following engineering factors must be considered:

- methods for avoiding FOD (foreign object damage)
- methods for avoiding Icing at the inlet
- the existence and proper design of duct drains
- potential for compressor surge and methods to mitigate risks (applies more to compressors that are being fog intercooled)

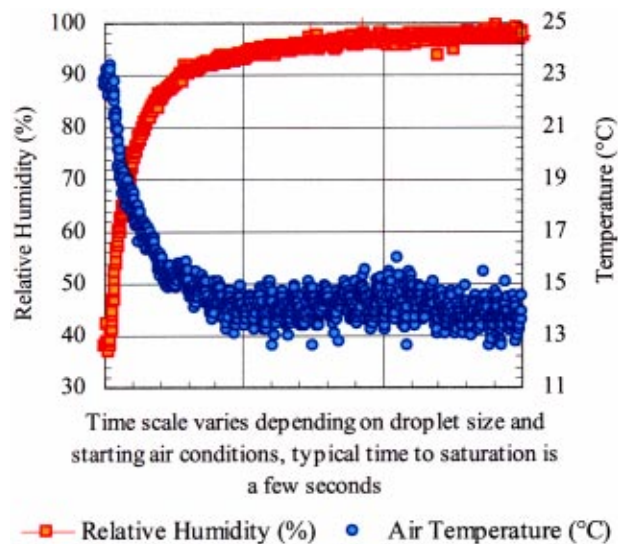


Fig. 9 Qualitative behavior measured in a duct showing the increase in relative humidity from a starting condition of 35% RH and the corresponding drop in air temperature in the duct

- methods for ensuring intake temperature uniformity to avoid potential stall or blade vibration
- fouling and gas turbine performance deterioration
- droplet size of fog produced and the potential for compressor erosion or blade coating distress.

Foreign Object Damage (FOD). As the nozzle manifolds exist in the air stream, care must be taken to avoid any chance of foreign object damage (FOD) that might result from loose nozzles or nozzle array components. It should be noted that wind tunnel tests for MeeFog nozzles have shown that even at airflow velocities of 3000 fpm (15.2 m/sec), the nozzle will fall to the floor very quickly, although normally the nozzles are located in the much lower velocity area after the air filtration. Safety wiring of nozzles (Fig. 12) and analysis of the fog nozzle array for airflow-induced vibration ensures that the structure is strong and cannot fatigue. Details of the airflow induced vibration design of the nozzle array along with a chart that users can utilize to evaluate their designs have been provided in Part III of this paper.

Gas Turbine Inlet Icing. The fog control system should automatically terminate fogging whenever there is any chance of inlet icing due to the static temperature depression that occurs in the bellmouth, which is caused by the acceleration of the air to Mach numbers of about 0.5 for heavy-duty gas turbines and 0.8 for aeroderivative machines.

The MeeFog system control software includes a user-definable set point called “inlet temp minimum.” This parameter can be set between 10°C and 27°C (50°F and 80°F) but it is set at the factory to 15.5°C (60°F). The fog system is staged such that the calculated after-fogging temperature will not fall below the value of the inlet temperature minimum parameter. Furthermore, once the after-fog temperature reaches this value any and all active over-fogging component of the system is disabled.

Several OEMs publish a combination of relative humidity and temperatures at which anti-icing measures are turned on. With fogging applications where the ending relative humidity is close to 100%, temperatures as low as 10°C can be utilized.⁷

Duct Drainage. This is an important subject and there are a

⁷There are several considerations other than just calculating the intake temperature static depression caused by air acceleration to Mach numbers of 0.5 to 0.8. There is also some heating (on the order of 1°C) due to the condensation that occurs and also due to heat transfer from the number 1 bearing, etc.

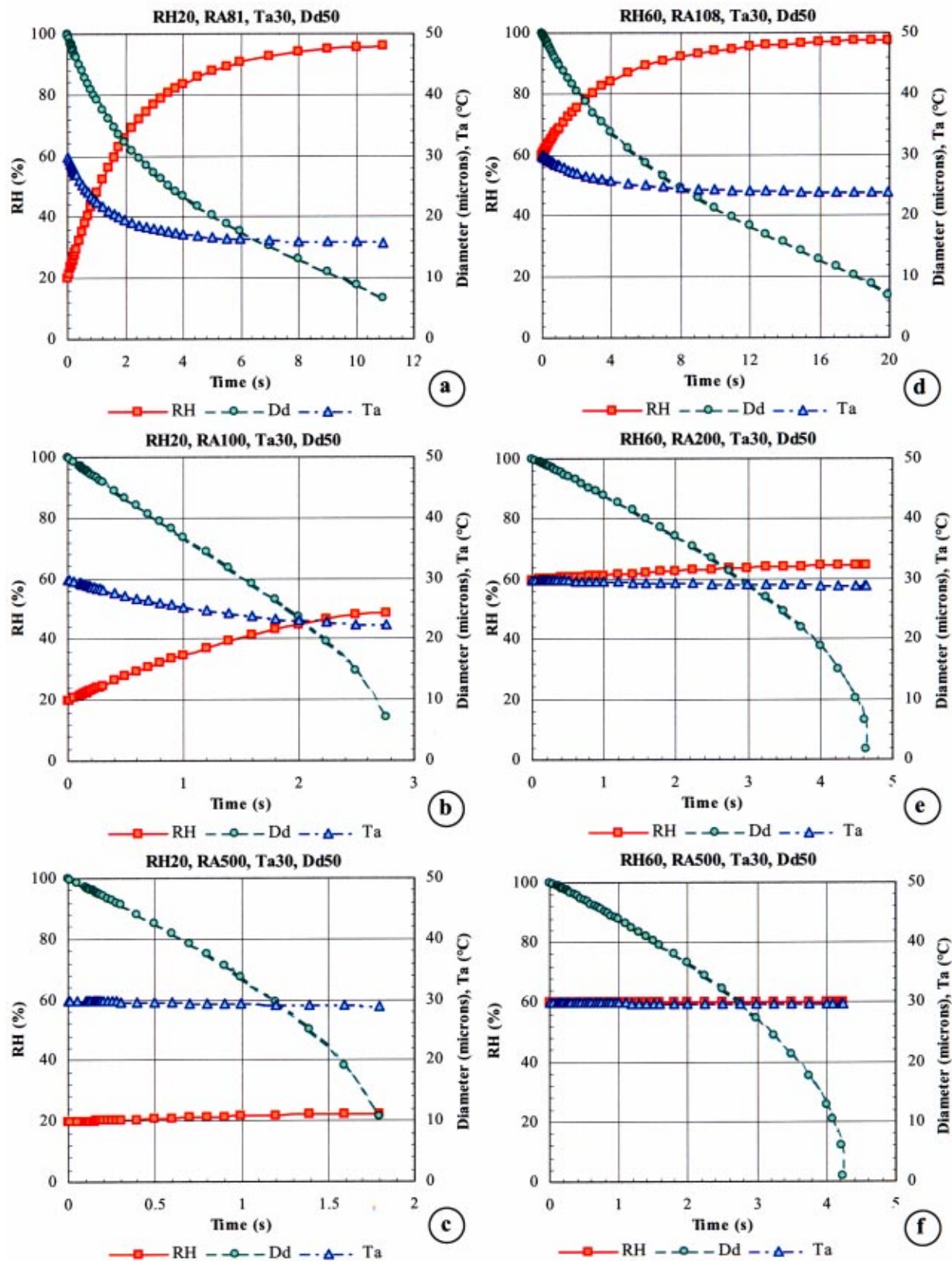


Fig. 10 Interaction of droplet to surrounding air conditions; droplet diameter; 50 microns

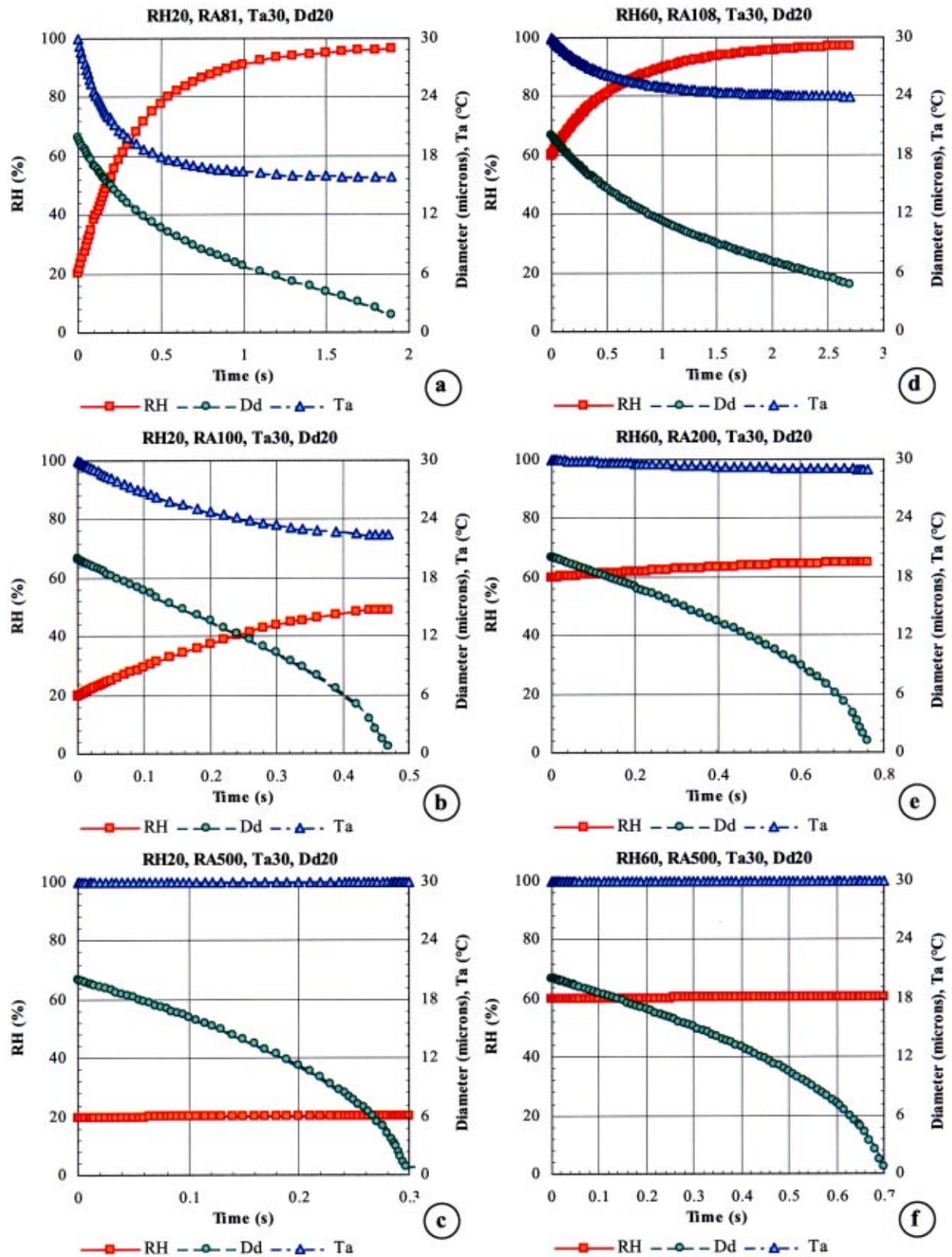


Fig. 11 Interaction of droplet to surrounding air conditions; droplet diameter: 20 microns



Fig. 12 Lock wire arrangements

lot of practical issues involved that a fog supplier should have learned from experience. Drains should be strategically located both near the nozzle manifolds and also in the intake bell mouth region. Drains should be carefully designed and must be continuously operating. The number of drains should be determined based on experience, and the configuration of the duct and obstructions that might result in water collection. Special shaped channel sections can be located on the duct walls and floors to channel collected water to the drains. One such channel is shown in Fig. 13. Appropriate sealing systems must be used to prevent the flow of ambient air into the duct. In the case of *P* trap seals, it is imperative that water be supplied to these to ensure that they do not run dry due to evaporation, which will allow the ingress of air. We have found greater success with stainless steel swing-check valves. Experience has shown that drain flows are typically 2–6% of the total water flow, though in some cases with restrictive duct configurations, as high as 8% may be attained under full flow conditions.

Drain flow should be monitored and logged as a function of ambient conditions and the number of stages. This can be done simply by measuring the flow for 5 minutes and then calculating the flow rate as a percent of the overall water flow.

A leak in the high-pressure lines inside the duct would cause an increase in drain flow rate but a well-designed fog system has pressure switches on the outlet of the pumps, which immediately terminate water flow in the event of a leak. More details relating to drains are provided in Part III of this paper.

Compressor Surge. This is mostly only a concern on turbines that are being fog intercooled (i.e., those in which a quantity of overspray is allowed). The effect of overspray is to cause the operating points on the compressor map to move towards the surge line. Ingistov [17] has reported on four GT-7EA Units that

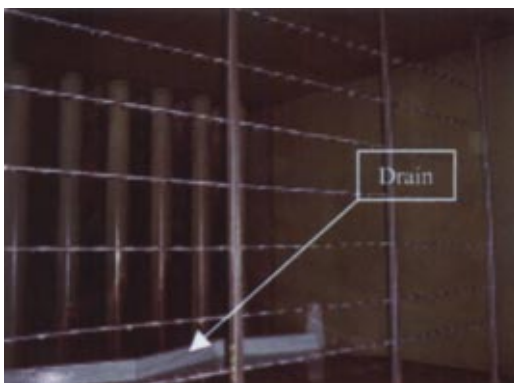


Fig. 13 Special channel system for water drain

have been operating with overspray for over two years. No surge related problems have been reported with over-fogging levels as high as 0.6% to 1% percent of the air mass flow.

Compressor Intake Temperature (CIT) Uniformity. Axial compressors have stringent inlet temperature and pressure uniformity criteria. This issue relates to blading vibration that can be induced due to extensive variations in air temperature at the face of the compressor. Details of blading issues may be found in Meher-Homji [18–21].

Fogging systems are designed in multiple stages and each stage has manifolds distributed within the inlet duct to provide a relatively uniform intake temperature. This is an important consideration as the susceptibility of the compressor to stall or surge could also be affected by severe temperature distortion. Even though an attempt is made to evenly distribute the nozzles on each stage, it may be desirable to avoid operating a fogging system at low staging levels for extended periods of time.

The accurate measurement of temperature distortion requires specialized instrumentation. However, the following equation provides an idea of the criteria for a typical high Mach number aeroderivative machine. Different machines would have different criteria.

$$\text{Distortion} = \frac{\Delta T_{\max}}{T_{\text{avg}}} = \frac{(T_{\text{avg,max}})60^\circ - (T_{\text{avg,min}})60^\circ}{T_{\text{avg,face}}} \leq 0.03 \quad (39)$$

where

- $T_{\text{avg,max}}$ = maximum area weighted average total temperature ($^\circ\text{R}$) in the warmest 60° sector of the annulus,
- $T_{\text{avg,min}}$ = minimum area weighted average total temperature ($^\circ\text{R}$) in the coldest 60° sector of the annulus, and
- $T_{\text{avg,face}}$ = average area weighted average total temperature ($^\circ\text{R}$) over the full face of the annulus.

Local temperatures within the sectors must be within 20% of the face average. There are also criteria relating to rapid changes in the inlet temperature but fogging cannot exceed these criteria. Even in the event of an emergency shutdown of the fog system, the residual fog in the duct will take some time to evaporate, so the result is never an instantaneous shock temperature change. The location of gas turbine CIT sensors, typically do not provide a means to evaluate temperature distortion. In several engines with complicated ducts, considerable temperature and pressure distortions exist even without fogging and it is not uncommon to find temperature difference between the CIT sensors of as much as 2°C (3.6°F). In some gas turbines, the CIT sensors may be located in the intake duct in a location that is partially starved of airflow. These sorts of pre-existing distortions can be made more evident by inlet fogging. It is a good idea to map the compressor inlet sensor readings with different ambient conditions and operating parameters and different numbers of fog stages in operation. This can form a useful baseline for analysis.

Intake Duct Considerations. There are several issues relating to the intake duct itself that should be considered. For instance, when installing nozzle manifolds on existing turbines, there is often not an ideal amount of space in the inlet ducts, hence careful compromises need to be made. Items such as bleed heating pipes (often present for DLN machines where air flow is bled for controlling the combustor conditions under certain operating regimes) or silencers may present obstructions, which the fog can impact on, resulting in excessive pooling of water on the duct floors. Duct supports as shown in Fig. 14 must also be considered in fogging system design. Further, most engines utilize trash screens and these will act as fog agglomerators causing some larger droplets. For the intake bell region of cold end drives that have intake cones, cone wetting may occur and some water may progress towards the compressor. However, due to the Weber number effect, the high air velocities result in droplet shattering



Fig. 14 Duct support structures like those shown here can collect fog and result in excessive pooling of water. Such obstructions must be taken into account when designing the nozzle manifolds.

and consequently the extent of this problem is minimized. The fog system supplier should be knowledgeable of these practical considerations and design to minimize any problems. Much of the choice of selecting and designing a system is an art and involves CFD analysis coupled with past experience and visual observations.

Axial Compressor Fouling. It is important to distinguish between the problems of natural climatic fog and the fog generated by the fogging system.

High *natural* humidity and climatic fog, as often occurs during nights and early mornings, can cause high filter delta P trips and sometimes the heavily fouled filters tend to unload (leaching effect through the filter), thus causing compressor fouling. However, if the air filtration system is working well, the increased humidity caused by the fogging system does not inherently increase fouling. Fouling is so site specific that it is very difficult to predict. If the No. 1 bearing is leaking oil this may combine with the high humidity (caused by inlet fogging) to create some fouling.

An important issue with inlet fogging is to wash the silencers and duct surfaces thoroughly to avoid dirt that has been accumulated here being washed into the compressor by the fogging system. This is particularly important when fogging is being installed as a retrofit on older machines. On retrofit applications, it may be necessary to perform several crank washes before the problem resolves. More details on compressor fouling may be found in Meher-Homji [22] and Meher-Homji et al. [23].

Compressor Erosion. For pure evaporative cooling systems, system parameters can be adjusted to ensure evaporation of fog prior to the compressor inlet. Yet, it is always possible that some droplets do enter the compressor. For droplet sizes that are relatively small (less than 15–20 microns) CFD studies have shown that the flow will tend to follow the air stream. There is the pos-

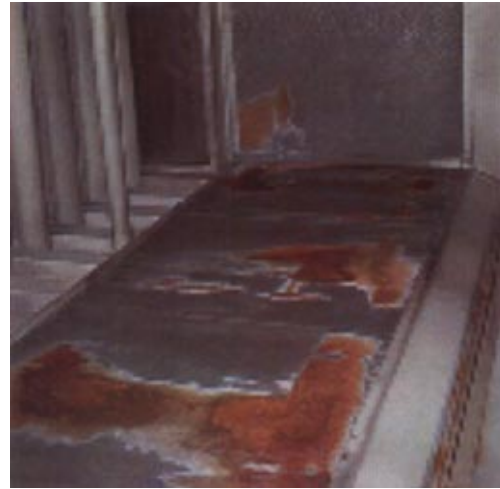


Fig. 15 Corroded inlet air duct floor

sibility of larger water particles forming on the trash screen and inlet cone of the gas turbine but with proper design and drainage approaches this can be minimized.

Corrosion in the Inlet Duct. The use of demineralized water can further deteriorate inlet ducts that are already in a deteriorated state. The increased humidity, as well as water pooling on the duct floor, are clearly corrosion factors. With proper maintenance and painting this problem can be mitigated. The use of stainless steel 316L as the duct material is gaining in popularity as life cycle studies have indicated that while the first cost is a little higher, the life cycle costs are significantly lower. It is important to note that ducts, the world over, often operate in a distressed condition such as seen in Fig. 15. It is important that such problems are addressed and corrected prior to the implementation of fogging.

Compressor Coating Distress. Some gas turbines that have undergone overspray have experienced coating distress in the first few stages of the axial flow compressor. In most cases, this can be minimized by using nozzles with the smallest possible droplet size, careful location of fogging nozzles, avoidance of excessive water accumulation on ducts walls and floors and on the inlet cone and by several other proprietary approaches. Corrosive ambient conditions such as chlorides or even trace amounts of HCl will cause acidity and hence coating damage. Airborne contaminants in even the ppb range can often create very acidic environments for the axial compressor blading. This is an issue that must be resolved by *proper inlet air filtration*, especially in aggressive industrial environments. Special coatings can help protect blading in areas with high levels of ambient contaminants. For overspray application, the situation should be studied on a case-by-case basis by carefully evaluating the blading material and coating technologies available. Haskell [24] has described the criticality of ambient air quality.

In spite of these precautions, overspray may, over time, create some coating distress but the benefit of increased output far outweighs the cost of more frequent recoating of blading.

In a few rare cases, there have been some reports of coating distress on units that are undergoing evaporative fogging only. There can be several causes of this:

- improper settings of fog system control parameters resulting in overspray conditions
- improper orientation or location of fog nozzles resulting in fog impaction on obstructions in the duct and the creation of very large droplets, or of excessive pooling on the duct floor
- lack of drains or lack of continuously operating drains or inappropriately located drains

- corrosive ambient air contaminants that cause acidity and hence coating damage. This is an issue that may be resolved by *proper inlet air filtration*, especially in aggressive industrial environments, or by selecting corrosion resistant blade coatings
- in some cases, if some leading edge coating distress already exists prior to fogging this might progress faster with fogging due to the reasons mentioned above.

Practical Considerations During System Design and Implementation. Some important practical considerations to be made in the implementation of any fogging system are provided below.

- Check vendors design calculations with regards to water flow requirements, particularly under off design conditions, and evaluate the design under different climatic conditions.
- Ensure that you provide the fogging system vendor with detailed sketches and photos of the inlet system. OEM's drawings may not show the exact configuration of duct internals, so up-to-date photographs are the best way of ensuring proper nozzle manifold design and location.
- Evaluate the amount of overspray required (i.e., intercooling effect) and ensure that the compressor can accommodate this. Surge margin should be evaluated.
- Evaluate generator capacity and lube oil coolers, etc., to ensure that the balance-of-plant equipment can tolerate higher turbine output on hot days. Often fog can be applied to cool these.
- Specify all stainless steel wetted components. Even the water supply lines should be stainless steel or some other noncorroding material. Iron or galvanized supply water lines can result in excessive plugging of inlet filters, or even plugging of the nozzles themselves. The presence of even trace amounts of certain minerals in the supply water can result in the growth of anaerobic bacteria, which can be a source of frequent nozzle plugging.
- In the event of a problem, does the design permit rapid isolation so operation can proceed without the fogging system?
- Does the design optimize the location of the fogging manifolds with respect to air filters and inlet systems. Insist on design justification from suppliers.
- Check that sequential fogging capability (cooling stages) is adequate to meet your demand profile and turndown profile. Uncertainties in the actual air mass flow of the axial compressor can be on the order of several degrees Celsius. Therefore, unless these uncertainties are mitigated in some way, very small stage increments are not meaningful.
- What design features have the vendor provided to avoid potential FOD?
- Check shedding frequency and rigidity of the manifolds to avoid flow-induced vibration. Vendor calculations should be reviewed.
- Review vendors proposed manifold design for structural rigidity and strength.
- Review proposed tie-in of the fog pump skid PLC (programmable logic controller) with plant DCS (distributed control system).
- Ensure that the design does not impose a large pressure drop.
- Evaluate the proposed installation for maintainability and accessibility.
- Evaluate the fog droplet size. Close approximations of droplet size can be made using empirical formulas, given the type of nozzle and the system operating pressure.
- Evaluate the number of fog nozzles. Nozzles that flow more water, at a given pressure, generally make bigger droplets. Fewer nozzles may have a benefit in reduced maintenance or upfront cost but, as discussed above, they have the disadvantages of creating larger droplets and of providing fewer points of emission and, therefore, less coverage of the inlet airflow.
- Ensure that appropriate drain lines will be installed in the inlet system.
- Installation of the weather sensor can be at the skid location,

if this is representative of inlet conditions to the gas turbine. Ensure that there is no secondary effect causing wrong measurements (e.g. close proximity to an air cooled heat exchanger or source of radiant heat).

- Demineralized water supply lines must be thoroughly flushed to ensure that no dirt has accumulated during installation.
- Thoroughly wash the inlet duct and silencers. Dirt in the inlet system can be carried into the compressor causing axial compressor fouling and creating performance deterioration or compressor damage. This is of special importance on retrofit applications.

Closure

This paper has provided a comprehensive analysis of the thermodynamics and heat transfer phenomena for gas turbine inlet fogging and has presented a model that can be used to evaluate the behavior of fog droplets in inlet ducts. The importance of the correct use of droplet diameters and measurement approaches has been covered and the important effect of flow velocity on fog droplet behavior has been described. Several practical aspects relating to inlet fogging have been covered. Chaker et al. [24,25] provide further details on droplet sizing and behavior in inlet ducts.

Acknowledgments

The lead author would like to acknowledge the contributions of Allen Reinholtz of Mee Industries' controls group for his work in helping set up the wind tunnel measurement systems and Conrad Klemzak, MeeFog R&D technician, for his help with the experimental setups. We also acknowledge and thank the large number of MeeFog system users whose technical inputs and support has been most valuable.

Nomenclature

Cp_a	= specific heat of air ($J.kg^{-1}.K^{-1}$)
Cp_d	= specific heat of water ($J.kg^{-1}.K^{-1}$)
D_d	= Droplet diameter (m)
Df_a	= mass coefficient of diffusion for air ($m^2.s^{-1}$)
Df_{mass}	= coefficient of mass diffusion ($m.s^{-1}$)
DBT	= dry bulb temperature ($^{\circ}C$)
ECDH	= equivalent cooling degree hours ($^{\circ}C-hr$)
g	= gravitational acceleration ($m.s^{-2}$)
Gr_m	= mass Grashof number
Gr_t	= heat Grashof number
hcv	= coefficient of thermal convective exchange ($W.m^{-1}.K^{-1}$)
Lv	= latent heat of vaporization of water ($J.Kg^{-1}$)
M	= molecular weight ($kg.mole^{-1}$)
m_d	= mass of the droplet (kg)
Nu	= Nusselt number
P_a	= atmospheric pressure (P_a)
Pr	= Prandtl number
Pvd	= vapor pressure at the interface droplet air (Pa)
P_{vap}	= partial pressure of the vapor (Pa)
Re	= Reynolds number
R	= universal gas constant ($8.31 J.mol^{-1}.K^{-1}$)
RA	= active radius
RH	= relative humidity (%)
S_d	= droplet surface (m^2)
Sc	= Schmidt number
Sh	= Sherwood number
T	= absolute temperature (K)
T_a	= temperature of air (K)
T_d	= droplet temperature (K)
We	= Weber number
WBT	= wet bulb temperature
ΔC_{mass}	= different of mass concentrations of water vapor at the interface droplet air and the air ($kg.m^{-3}$)
Δm_d	= mass variation of the droplet (kg)

V_{rel} = droplet relative velocity (m.s^{-1})
 Δt = time step (s)

Greek

β = coefficient of thermal dilatation (K^{-1})
 Φ_{conv} = flux exchanged by thermal convection (W)
 Φ_{lat} = flux exchanged by latent heat (W)
 γ_w = water surface tension (N.m^{-1})
 λ_a = thermal conductivity, air ($\text{W.m}^{-1}\text{K}^{-1}$)
 μ_a = dynamic viscosity, air ($\text{kg.m}^{-1}\text{s}^{-1}$)
 ν_a = kinetic viscosity, air ($\text{m}^2.\text{s}^{-1}$)
 ρ_a = density, air (kg.m^{-3})

Subscripts

a = air
 d = droplet
 t = time

References

- [1] Meher-Homji, C. B., and Mee, T. R., 1999, "Gas Turbine Power Augmentation by Fogging of Inlet Air," *Proceedings of the 28th Turbomachinery Symposium*, Houston, TX, Sept.
- [2] Meher-Homji, C. B., and Mee, T. R., 2000, "Inlet Fogging of Gas Turbine Engines—Part II: Practical Considerations, Control and O&M Aspects," ASME Paper No. 2000-GT-0308.
- [3] Bhargava, R., and Meher-Homji, C. B., 2002, "Parametric Analysis of Existing Gas Turbines With Inlet Evaporative and Overspray Fogging," ASME Paper No. 2002-GT-30560.
- [4] Kleinschmidt, R. V., 1946, "The Value of Wet Compression in Gas Turbine Cycles," Annual Meeting of the ASME, Dec. 2–6.
- [5] Wilcox, E. C., and Trout, A. M., 1951, "Analysis of Thrust Augmentation of Turbojet Engines by Water Injection at the Compressor Inlet Including Charts for Calculation Compression Processes With Water Injection," NACA Report No. 1006.
- [6] Hill, P. G., 1963, "Aerodynamic and Thermodynamic Effects of Coolant Injection on Axial Flow Compressors," *Aeronaut. Q.*, Feb. pp. 333–348.
- [7] Arsen'ev, L. V., and Berkovich, A. L., 1996, "The Parameters of Gas Turbine Units With Water Injected Into the Compressor," *Therm. Eng.*, **43**(6), pp. 461–465.
- [8] Nolan, J. P., and Twombly, V. J., 1990, "Gas Turbine Performance Improvement Direct Mixing Evaporative Cooling System," ASME Paper No. 90-GT-368.
- [9] Utamura, M., Kuwahara, T., Murata, H., and Horii, N., 1999, "Effects of Intensive Evaporative Cooling on Performance Characteristics of Land-Based Gas Turbine," *Proceedings of the ASME International Joint Power Generation Conference*, ASME, New York.
- [10] Chaker, M., Meher-Homji, C. B., Mee, T., and Nicolson, A., 2001, "Inlet Fogging of Gas Turbine Engines—Detailed Climatic Analysis of Gas Turbine Evaporative Cooling Potential," ASME Paper No. 2001-GT-526.
- [11] Chaker, M., and Meher-Homji, C. B., 2002, "Inlet Fogging of Gas Turbine Engines—Detailed Climatic Analysis of Gas Turbine Evaporative Cooling Potential for International Locations," ASME Paper No. 2002-GT-30559.
- [12] Chaker, M., 1995, "Etudes de la Desintegration Electrohydrodynamique de Gouttes Fortement Chargees," doctoral thesis, University of Nice, Sophia Antipolis, France.
- [13] *Handbook Of Chemistry*, 59th Ed., CRC Press, Boca Raton, FL.
- [14] Ranz, W. E., and Marshall, W. R., 1952, "Evaporation From Drops, Parts I and II," *Chem. Eng. Prog.*, **48**(3).
- [15] Pelletret, R. Y., 1984, "Contribution a l'etude des transferts couples de masse et de chaleur entre une gouttelette d'un liquide hydroscopique et l'air. Conception d'echangeurs par pulverisation d'un liquide dans un flux d'air," doctoral thesis, University of Valenciennes, France.
- [16] Wexler, A., and Greenspan, L., 1971, "Vapor Pressure Equation in the Range 0–100°C," *J. Res. Natl. Bur. Stand., Sect. A*, **75A**, p. 213.
- [17] Ingistov, S., 2000, "Fog System Performance in Power Augmentation of Heavy Duty Power Generating Gas Turbines GE Frame 7EA," ASME Paper No. 2000-GT-305.
- [18] Meher-Homji, C. B., 1995, "Blading Vibration and Failures in Gas Turbines—Part I: Blading Dynamics and the Operating Environment," ASME Paper No. 95-GT-418.
- [19] Meher-Homji, C.-B., 1995, "Blading Vibration and Failures in Gas Turbines—Part II: Compressor and Turbine Airfoil Distress," ASME Paper No. 95-GT-419.
- [20] Meher-Homji, C.-B., 1995, "Blading Vibration and Failures in Gas Turbines—Part III: Detection and Troubleshooting," ASME Paper No. 95-GT-420.
- [21] Meher-Homji, C.-B., 1995, "Blading Vibration and Failures in Gas Turbines—Part IV: Case Studies," ASME Paper No. 95-GT-421.
- [22] Meher-Homji, C. B., 1990, "Gas Turbine Axial Compressor Fouling—A Unified Treatment of Its Effects, Detection & Control," *Int. J. Turbo. Jet Engines*, **9**(4).
- [23] Meher-Homji, C. B., Chaker, M., and Motiwala, H., 2001, "Gas Turbine Performance Deterioration," *Proceedings of the 30th Turbomachinery Symposium*, Turbomachinery Laboratory, Texas A&M University, Houston, TX, Sept. 17–20.
- [24] Chaker, M., Meher-Homji, C. B., and Mee T. R., III, 2002, "Inlet Fogging of Gas Turbine Engines—Part II: Fog Droplet Sizing Analysis, Nozzle Types, Measurement and Testing," **126**, pp. 559–570.
- [25] Chaker, M., Meher-Homji, C. B., and Mee T. R., III, 2002, "Inlet Fogging of Gas Turbine Engines—Part III: Fog Behavior in Inlet Ducts, CFD Analysis and Wind Tunnel Experiments," **126**, pp. 571–580.

Mustapha Chaker
Director, Research and Development
Cyrus B. Meher-Homji
Chief Engineer

Thomas Mee III
Chairman and CEO,

Gas Turbine Division,
Mee Industries, Inc.,
204 West Pomona Avenue,
Monrovia, CA 91016

Inlet Fogging of Gas Turbine Engines—Part II: Fog Droplet Sizing Analysis, Nozzle Types, Measurement, and Testing

The inlet fogging of gas turbine engines for power augmentation has seen increasing application over the past decade yet not a single technical paper treating the physics and engineering of the fogging process, droplet size measurement, droplet kinetics, or the duct behavior of droplets, from a gas turbine perspective, is available. This paper provides the results of extensive experimental and theoretical studies conducted over several years, coupled with practical aspects learned in the implementation of nearly 500 inlet fogging systems on gas turbines ranging in power from 5 to 250 MW. Part II of the paper treats the practical aspects of fog nozzle droplet sizing, measurement and testing presenting the information from a gas turbine fogging perspective. This paper describes the different measurement techniques available, covers design aspects of nozzles, provides experimental data on different nozzles, and provides recommendations for a standardized nozzle testing method for gas turbine inlet air fogging. [DOI: 10.1115/1.1712982]

Introduction

Over the past decade and especially over the past five years, the application of inlet fogging for the power augmentation of gas turbines has become increasingly popular. It is estimated that approximately 700 gas turbines worldwide have fogging systems at this time including several new F-class gas turbines. To this point, however, not a single technical paper exists comprehensively covering the physics and engineering of the fogging process, droplet measurement, kinetics, duct behavior of droplets from a gas turbine perspective.

This paper provides the results of extensive experimental and theoretical studies conducted over several years, and also mentions practical aspects learned in the implementation of nearly 500 inlet fogging systems on gas turbines. A major problem faced by gas turbine users considering the utilization of inlet fogging is that different fog nozzle manufacturers and suppliers present data in very different formats and under different operating conditions. In this paper we point out the key operating parameters that are pivotal in a gas turbine fogging application and also provide data on a large number of nozzle tests made in our experimental wind tunnel.

Measurement Approaches to Determine Fog Droplet Size

The importance of the determination of accurate droplet size for gas turbine inlet fogging applications was covered in Part I of this paper. The atomization process of any type of fog nozzle consists of the conversion of the kinetic energy of water under pressure into a spray that contains a wide range of statistically distributed droplet sizes. Different techniques exist to measure particles size. For gas turbine inlet fogging applications, imaging (microscopy and the use of high speed video cameras) and the laser light scattering techniques are commonly used.

There are two techniques that use light scattering principles: the spatial technique and the flux technique.

The spatial technique instantly samples a large number of droplets in a given volume. This is a number-density-weighted technique (Malvern and Imaging Systems are two well known manufacturers of this type of equipment).

The flux or temporal technique samples and counts individual droplets passing through the sampling volume (established by the cross section of two beams of laser light) in a given time interval and is a number-flux-weighted technique. PDPA and PMS are two systems that use this technique.

With the flux measurement technique, ignoring a droplet of 100 microns, is the same as ignoring 1000 droplets of 10 micron (since volume is a cube function of the diameter). In gas turbine fogging applications therefore, ignoring a relatively significant number of small droplets does not significantly affect the distribution; however, the distribution is drastically altered if one ignores droplets of a larger diameter. In the measurement of fog droplet size, an important source of error derives from the conversion: if the error in the measurement of the mean diameter is 5%, an error of 125% will be propagated when this result is converted to the volume, since the volume or mass mean diameter is a cubic function of the mean diameter. This problem occurs when converting from a measured diameter to a calculated volume as is done with a flux type system.

Due to the difference in the droplet's velocity at the sampling region, the two techniques may produce different results as stated by Dodge [1]. In examining measurement approaches for gas turbine fogging applications, it is clear that a spatial sizing technique (working with volumes) and analyzing a huge number of droplets instantly is more appropriate as we are dealing with billions of droplets.

Teske et al. [2] compared droplet sizes measured using five different measurement techniques: Malvern, particle measurement system, phase Doppler particle analyzer, hot wire, and video imaging and observed a similarity between the measured data from Malvern, PDPA and Imaging Video Systems. He found that, the agreement between several Malvern instruments was superior to the agreement between several PDPA instruments.

Similar studies conducted by the British Crop Protection Council [3] and others, [4–8], confirmed the preciseness, repeatability,

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30563. Manuscript received by IGTI, December 2001, final revision, March 2002. Associate Editor: E. Benvenuti.

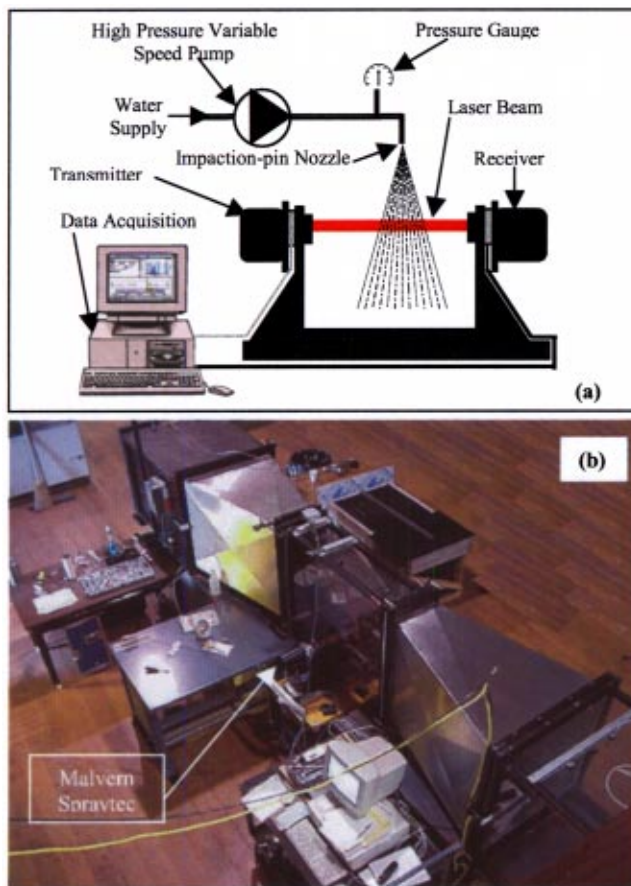


Fig. 1 (a) Experimental setup, in still air and (b) wind tunnel section showing Malvern Spraytec measurement system for nozzle testing under airflow conditions

and measurement stability of the Malvern system compared to other measurement systems when measuring the droplet size distribution of spray atomized using different types of nozzles.

Based on our extensive experience in the water atomization and droplet measurement fields, and on results confirmed by many researchers, we have standardized on the use of a Malvern Spraytec system to characterize gas turbine inlet air fogging nozzles.

The Malvern system is a non-intrusive system. It instantly samples a large number of droplets occupying a given volume. This laser diffraction system, along with an algorithm allowing us to correct for multiple scattering errors, was used to measure the size distribution of the droplets throughout this paper.

A photograph of the experimental setup using a Malvern Spraytec system is shown in Fig. 1.

Droplet Diameter Definitions

It is important for anyone evaluating the relative performance of inlet air fogging nozzles to understand the terminology used to characterize nozzle performance. All atomizing nozzles produce a range of droplet sizes so, in order to characterize a nozzle with a single value, it is necessary to take some statistical function of the droplets measured. Such statistical functions produce a value that refers to a droplet diameter, which in some way describes or characterizes the total spray. The stated diameter is possibly fictitious because none of the droplet measured may actually have the stated diameter.

In reality, inlet air fogging nozzles produce several billion droplets per second. Therefore, in order to ensure reasonable results, it is important that any measurement technique measure a large number of droplets. Furthermore, all nozzles produce, to a limit,

smaller droplets as the pressure is increased. Therefore, to be of practical use any statement of the size of droplets produced should be accompanied by the operating pressure of the nozzle. As was described in Part I of this paper, one has to consider the definition of the various types of droplet diameters as well as the *conditions* under which measurements were taken. Different measurement conditions may produce very different droplet size results. The effect of measurement conditions such as airflow velocity and location of the measurement in the spray plume will be discussed in detail ahead.

Some common diameters (mean diameters and representative diameters) used are presented below.

Mean Diameters

Detailed formulas for the mean diameters definitions are provided in the Appendix.

Arithmetic Mean Diameter (AMD) D_{10} . The simple average diameter of all the droplets in a spray. D_{10} is equal to the sum of the diameter of all the droplets divided by the quantity of droplets.

Surface Area Mean Diameter (SAMD) D_{20} . The droplet diameter is measured directly using an imaging system. The SAMD diameter is then calculated on the basis of the droplets surface area. By taking the square of the diameter of each droplet, it is possible to calculate its surface area:

$$A_d = \pi \times D_d^2 \quad (1)$$

An average surface area is then calculated using the sum of the surface areas of all the droplets divided by the number of droplets. The square root of the average surface area is then divided by π to get the SAMD. The SAMD value characterizes the spray by giving the diameter of a hypothetical droplet that has a surface area equal to the average surface area of all the measured droplets.

Volume Mean Diameter (VMD) D_{30} . This characteristic diameter is calculated on the basis of the droplet's volume. The volume of a droplet is given by

$$V_d = \frac{\pi \times D_d^3}{6} \quad (2)$$

Using this equation, it is possible to calculate the volume of each droplet, then the average volume by taking the sum of the volume of all the droplets and dividing it by their number, and then to finally take the cubic root of the averaged volume using Eq. (2) to get the VMD. The VMD value characterizes the spray by giving the diameter of a hypothetical droplet that has a volume equal to the average volume of all the measured droplets.

Sauter Mean Diameter (SMD) D_{32} . This diameter is calculated using the concept of the volume to surface area ratio. It is equal to the sum of the cube of all diameters divided by the sum of the square of all diameters. This yields a characteristic droplet diameter that has a volume-to-surface-area ratio equal to the volume-to-surface-area ratio of the entire spray. This diameter is particularly important in gas turbine evaporative fogging system applications because the mass transfer happens at the interface of the droplets and the surrounding air (i.e., at the droplet surface). To enhance the evaporation of a population of droplets, one has to maximize the active surface areas and minimize the internal volumes.

Absorption Diameter (AD) D_{21} . This diameter is calculated using the surface to diameter ratio concept. It is equal to the square root of the sum of the square of all the droplets diameters divided their straight sum.

Evaporative Diameter (ED) D_{31} . This diameter is calculated using the volume to diameter concept. It is equal to the sum of the cube of all the droplets diameters divided by their straight sum.

Representative Diameters

There are other representative diameters, which can be easily measured from cumulative distribution¹ curves. These are defined follows.

Dv01 (Also Known as Dv10). Is a representative diameter where 10% of the total volume of the liquid sprayed is made up of droplets with diameters smaller or equal to the stated value.

Dv05 (Also Known as Dv50). This is the same as the volume median diameter or mass median diameter (MMD), assuming water. This is the representative diameter where 50% of the total volume of the liquid sprayed is made up of droplets larger than the stated value and 50% is made up of droplets with diameters smaller than the stated value.

Dv09 (Also Known as Dv90). This is the representative diameter where 90% of the total volume of the liquid sprayed is made up of droplets with diameters smaller than or equal to the stated value. This representative diameter is commonly used to characterize gas turbine inlet air fogging nozzles because there is concern that the ingestion of large droplets by the axial flow compressor might cause blade erosion or blade coating wear.

Other median diameters may be used, depending on the application, such as the number median diameter D_{N50} and the surface median diameter D_{S50} .

RSF Relative Span Factor. This is a dimensionless parameter indicative of the uniformity of the drop size distribution. It is given by

$$RSF = \frac{D_{v09} - D_{v01}}{D_{v05}} \quad (3)$$

Fog Spray Histograms

Figure 2 shows two histograms ((a) and (b)) which are output from the Malvern laser particle analyzer software. The vertical, blue-colored bars represent “bins” of droplet sizes, which can be read off the abscissa. The volume frequency (as a percent of the total volume) can be read off the axis on the right side. Examining Fig. 2(a), we see that the droplets in the 10-micron bin represent about 4% of the total volume of the spray.

The red-colored s-shaped curve shows the cumulative volume across the size bins. Examining Fig. 2(a) again, and reading from the cumulative volume axis (on the left) we see that 50% of the volume of the droplets are about 18 microns or smaller (i.e., the Dv50, or volume/mass mean diameter is 18 microns).

The charts also give:

- *Transmission*, which is the percent light transmission through the spray. If this value is below a certain limit, the spray density may be too high and multiple-scattering errors may result.
- *CV*, which is the concentration volume given in parts per million. *CV* also gives an indication of liquid water content (1 CV is equal to one cubic centimeter of liquid water per cubic meter of air). *CV* plays an important role in our proposed standard characterization method for gas turbine inlet air fogging systems, given in the last section of this paper.
- *Specific surface area*, which is the surface area of the spray, expressed as square meters per cubic centimeters of water.
- Dv10, Dv50, Dv90, and D_{32} (SMD), all of which are defined above.
- D_{43} , which is the mean mass or De Brouckere's or Harden's diameter used in combustion equilibrium applications.
- *Span*, which is the relative span factor as defined above.

¹Droplet size distribution in the spray is characterized by Rosin-Rammler expression and function of two parameters, which are the measure of the range of droplet sizes and a representative diameter.

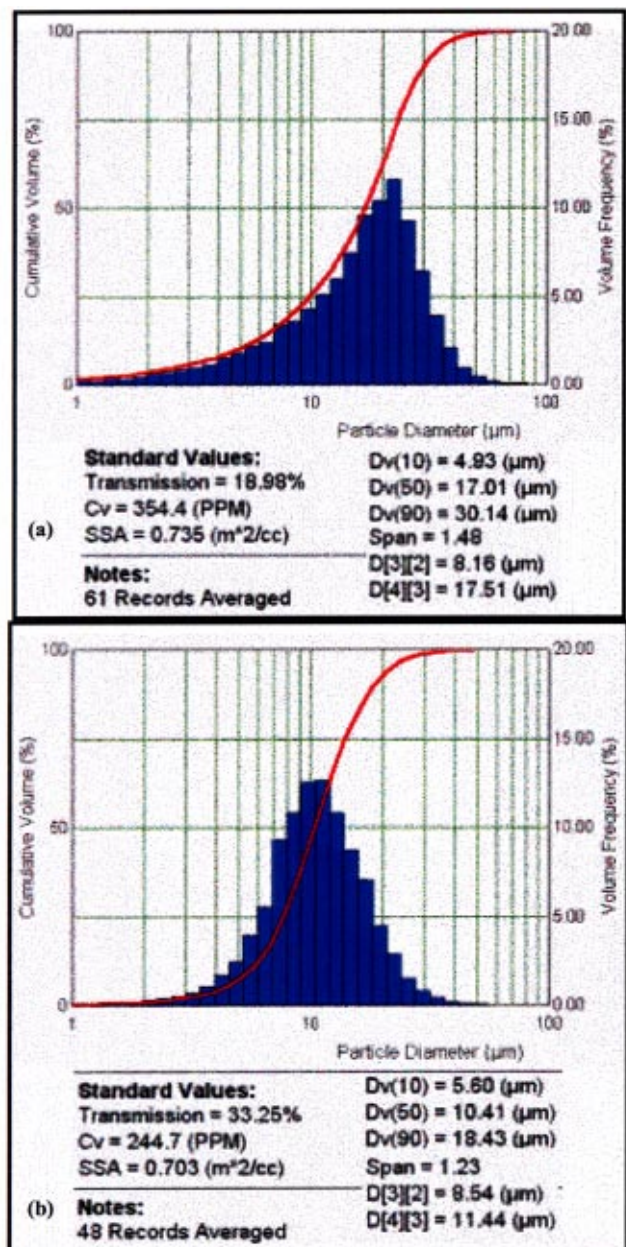


Fig. 2 Two droplet measurement histograms showing the importance of the use of both SMD and Dv90. Data is from actual experiments done on nozzles

Appropriate Diameter Numbers for Characterizing and Comparing Inlet Air Fogging Nozzles

The most important quantitative factors for inlet air fogging are the surface area of water exposed for evaporation (affects the evaporative cooling efficiency of the spray) and the size of the largest droplets (affects the potential for compressor blade distress and the amount of water that falls out in the duct). Given the above, the SMD and Dv90 numbers are of primary significance for gas turbine inlet-air cooling.

Evaporation rate is a strong function of the surface area of water exposed to the air. SMD expresses the total surface area of water exposed for a given volume of water sprayed. Looking again at Fig. 2, we see that there is about a 5% difference in SMD (given on the chart as $D_{[2,3]}$) and, as would be expected, there

is also about a 5% difference in specific surface area—square meters of surface per cubic cm of water atomized (SSA on the chart).

The importance of knowing both SMD and Dv90 can be seen in Fig. 2. As noted above, the SMD in both histograms is approximately the same (about 8 microns). However, in the first histogram (Fig. 2(a)), there are a larger number of big droplets (Dv90 about 30 microns) while in the second histogram (Fig. 2(b)) Dv90 is about 18 microns—nearly half that of the first. Thus, in order to meaningfully evaluate nozzles, one *has* to know the Dv90 number as well as the SMD.

In conclusion we feel that for gas turbine fogging applications, the SMD and Dv90 numbers are of significance.

Further to this, the *location* of the measurement in the spray plume, and other measurement conditions such as airflow velocity, are also very important. Measurements taken in different locations can yield very different results. In the sections ahead we provide a recommended standard for the measurement of droplets.

It is interesting to note that measurements of droplet diameters at distances progressively further downstream from the fogging nozzles (in an air stream that is under-saturated) results in progressively larger diameters. This might lead one to assume that droplet coalescence was occurring. In fact, what is occurring is that the smallest droplets evaporate resulting in the average diameter of the remaining population increasing.

Consider for instance a spray consisting of just one 10-micron water droplet and one 50-micron droplet. The larger droplet has about 125 times the mass and about 25 times the surface area of the 10-micron droplet. Now imagine removing, by evaporation, the mass of one 10-micron droplet from both droplets. The smaller droplet disappears but the diameter of the larger droplet reduces from 50 microns to just 49.9 microns.

Evaporation rate is, of course, a strong function of surface area and throughout the evaporation process, the 50 micron droplet will always have more surface area than the ten micron droplet as it started with 25 times more surface area. Even if 25 times more mass is removed from the larger droplet, it would still be more than 46 microns in diameter. With the removal of 100 times more mass the larger droplet would still be almost 30 microns in diameter.

To visualize the impact of this effect on a large population of droplets, consider a spray of billions of droplets, half at 10 microns and half at 50 microns. The average droplet is 30 microns. Now evaporate away the 10-micron droplets, and reduce the mass of the larger droplets, by 25 times the mass of a 10-micron droplet, and we are left with a spray having billions of droplets larger than 46 microns. The average droplet size of the spray increased from 30 to 46 microns yet no coalescence occurred.

This phenomenon was proved in the wind tunnel, described in Part III of this paper, by showing that agglomeration was not occurring. The airflow in the wind tunnel was saturated, so that evaporation would not occur, and droplet size was measured at various points downstream of the nozzles. The phenomenon can also be shown using the empirically validated evaporation rate model, also given in Part I of this paper.

As mentioned above, the *location* of the measurement in the spray plume and other measurement conditions such as airflow velocity, are also very important. Figure 2 demonstrates this fact, because the two histograms are in fact the same nozzle operating at the same pressure but represent measurements take at different airflow velocities and at different locations in the spray plume. In the sections ahead we provide a recommended standard for the measurement of droplets for gas turbine inlet air fogging systems. The standard makes it possible to make a meaningful comparison of different fogging nozzles.

A Suggested Representative Diameter. The importance of considering both SMD and Dv90 when comparing different fogging nozzles suggests the possibility of adopting a single representative diameter that would take both factors into account and



Fig. 3 Swirl and impaction-pin nozzles

would thus be more appropriate for quantifying gas turbine inlet air fogging sprays. One very simple approach would be to take an average of the SMD and Dv90 values for a given spray. For the sprays depicted in histograms of Figs. 2(a) and 2(b) this simplistic approach yields diameters of 19.15 microns and 13.49 microns, respectively. The result is a single representative diameter that gives a good indication as to which nozzle is most suitable for gas turbine inlet air fogging. We are working on statistical approaches to refine and validate this parameter and a more detailed study of a suitable representative diameter will be given in a later paper.

Nozzle Design

For gas turbine inlet air fogging applications, the goal of atomization is to disintegrate the water into the smallest droplet sizes possible in order to maximize the surface area so as to increase the heat transfer between the droplet surface and the air. The underlying principle of any pressure-atomizing nozzle is to convert the applied pressure into kinetic energy by the use of a small orifice.

When selecting fog nozzles for inlet-air cooling some engineering compromises have to be made. There may be a cost and maintenance benefit associated with using fewer nozzles with higher flow rates but, on the downside, higher flow rate nozzles have to be operated at higher pressures (additional cost) or they make larger droplets (less effective cooling, potential for blade stress) and fewer nozzles means fewer points of fog emission (potential for uneven cooling at the compressor face and less effective cooling of the entire air mass flow).

The most important factors for nozzle selection are droplet size and the distribution of fog in the duct cross-section, as these affect both the efficiency of the fog cooling process and the safe operation of the turbine.

If the nozzle pattern at the manifolds allows a significant portion of air to pass by without mixing with the fog spray plumes, the result will be less evaporation before the air reaches the compressor inlet.

There are two types of nozzles which are in general use for gas turbine fogging applications: impaction-pin nozzles and swirl-jet nozzles, although impaction-pin designs predominate. A photograph of the two types of nozzles is shown in Fig. 3. It is very important to note that the performance of any nozzle is a function of its specific design and therefore any comparative testing must be done with the specific design contemplated for use. A general discussion of the two design philosophies is presented in this section. Later, comparative tests will be shown with specific nozzles that have been used for inlet air fogging applications.

Impaction-Pin Nozzles. With impaction-pin nozzles, water is forced through a smooth orifice and the high-velocity jet is directed at an impaction pin located above the orifice. The velocity of the jet, and the size of the droplets produced, depends on the applied pressure. The impaction of the water jet on the pin results in a thin sheet of water in a conical shape. As the conical sheet extends away from the nozzle orifice the surface area of the sheet expands so that the water sheet becomes increasingly thinner.

Eventually the surface tension forces eventually cause the sheet to break down into “fingers” of water and aerodynamic instabilities cause the fingers to break into ligaments and then into droplets. The thickness of the sheet of water depends on the orifice hole diameter, the impaction-pin geometry and the operating pressure. Increasing the pressure results in an increase in velocity causing the sheet to extend and the thickness to decrease resulting in a decrease in droplet size as well as an increase in the flow rate of the nozzle.²

A nozzle with a smooth and shallow orifice will have low frictional losses and will, therefore, be more efficient at converting the energy of the applied pressure to axial velocity and hence will make a thinner sheet and smaller droplets. It is important that nozzles be manufactured with minimal variations as a small difference in orifice diameter, orifice smoothness or depth, will result in different flow rates and different droplet sizes. It is therefore important to measure the droplet size of many nozzles from any given manufacturer and to examine the statistical spread.

The velocity of the water jet produced by any direct-pressure nozzle is a function of the orifice size geometry and the fluid pressure. Jet velocity is also a function of the viscosity and density of the pumped fluid and the point at which the sheet disintegrates into ligaments and then to droplet (and therefore the size of the droplet produce) is a function of both jet velocity and surface tension of the fluid. However, for purposes of comparing inlet-fogging nozzles, the authors have largely ignored density, surface tension, and viscosity effects, since the pumped fluid is always demineralized water essentially at ambient temperature.

Due to the high velocity of the droplets very close to the nozzle tip, the Weber number is high and this leads to an important secondary break up of the larger droplets. The result of the collision between droplets in this area is a shattering due to the high Weber number. Details of high Weber number droplet-shattering effect are provided ahead.

It is helpful to consider three different locations (zones) in the fog plume when considering the question of the probability of droplet collision and the resulting separation, coalescence, and bouncing:

1. the reflexive and stretching separation zone: Very near the nozzle orifice droplet velocities are very high (resulting in Weber numbers as high as 50) and droplet density is very high. Collision of droplets is highly probable but mostly results in stretching and reflexive separation.
2. the coalescence and bouncing zone: Moving further from the orifice, the droplets move a bit slower (lower Weber number) but droplet density is still high. Collision of droplets is somewhat less probable but still occurs and the result of collision is coalescence or agglomeration.
3. the noncollision zone: Moving still farther from the orifice, the droplets move much slower and the Weber number is very low. As the plume has spread out, the droplet density is low and the likelihood of collision is almost nil. Further, droplet collision at low relative velocity would result in bouncing, and not coalescence.

The relative size of these zones depends on the specific nozzle design, the fluid pressure and the velocity of the air that the nozzle is spraying into.

Nozzle designs should be optimized for the specific application. Figure 4 shows visual plume shape of different makes of impaction-pin nozzles. The nozzle in the center shows a wider plume (better coverage in the duct cross section) and a more homogeneous distribution.

²There are practical design optimization issues involved with increasing the operating pressure. These issues include, the thickness of the orifice plate that would then increase frictional losses, and also droplet coalescence agglomeration effects (increasing pressure also increases the density of droplets near the orifice and, hence, increases the probability of collision and coalescence). There are also the practical limitations of pumping and piping issues when very high pressures are considered.



Fig. 4 Visual plume shape of different makes of impaction-pin nozzles at an operating pressure of 138 barg (2000 psig). Mee nozzle (152 micron orifice, 316 stainless steel construction) in the center

The orifice diameter and operating differential pressure of an impaction-pin nozzle governs the flow rate. Impaction-pin nozzles with different orifice diameters and their resultant plumes are shown in Fig. 5. Diameters are 5.8 mils (147 microns) on the left through 18 mils (457 microns) on the right. Optimal design, from a flow rate, droplet size, and plume dimension standpoint for gas turbine applications, was found to be 6 mil (152 micron) diameter (second nozzle from the left).

Impaction-pin nozzles exhibit what is often called the “sprinkle” effect. By this we mean that some droplets tend to hit the pin and form larger droplets. However, these are either broken up (due to the high Weber number) or fall out immediately and, in any case, they represent a very small portion of the total volume of the spray.

The distribution of the droplets across the cross section of the plume is complicated. Due to the presence of the impaction pin, the shape of the plume is not symmetrical as can be seen in Fig. 6



Fig. 5 Various impaction-pin nozzles, from 147-micron (on right) to 457-micron orifice diameter (left), operating pressure 138 barg



Fig. 6 Impaction-pin nozzle showing small shading effect from impaction pin

(view from the impaction-pin side). The view from the other side is shown in Fig. 7. As can be seen in these figures, the shading effect of the impaction pin is very local to the nozzle tips and therefore has little effect on the overall fog plume.

Secondary Breakup—Weber Number Effect. The secondary breakup of liquid droplets occurs if the surface tension force, which acts to keep the droplet spherical, is no longer able to balance the opposing aerodynamic forces due to the high relative velocity between the droplets and the surrounding airflow. The ratio of the aerodynamic and surface tension forces is the dimensionless Weber number given by this equation:

$$We = \frac{\rho_a \cdot v_{rel}^2 \cdot D_d}{\gamma_w} \quad (4)$$

For water, with a critical Weber number value of 6 taken over large interval of Ohnesorge number [9], secondary breakup of water droplets may occur. Details regarding secondary breakup may be found in Pilch and Erdman [9]. This reference provides the use of breakup time data and velocity history data to predict the maximum size of stable fragments for acceleration-induced breakup of liquid droplet. It is important to note that the Weber number effect applies to both impaction-pin and swirl nozzles but has a greater significance with impaction-pin nozzles due to the



Fig. 7 Opposite view of the nozzle shown in Fig. 6

higher relative velocities of droplets under similar operating conditions (i.e., fluid pressure and air velocity). The Weber number also has other applications for understanding the shattering of droplets in intake ducts as will be discussed in Part III of this paper.

Swirl-Jet Nozzles. With swirl-jet nozzles, the fluid is forced to enter tangentially into the swirl chamber taking a helical path before discharging through a cylindrical hole concentric to the swirl chamber. This leads to the formation of an air core with a larger diameter in the swirl chamber, as compared to the one in the orifice. The swirling jet results in the formation of an axisymmetrical, thin conical water sheet. The conical sheet breaks down into ligaments and then small droplets. Because of the swirling nature of the flow, there are both axial and radial forces involved. Unlike the impaction-pin nozzle the ligaments are formed perpendicular to the axis of the nozzle (due to the swirling action of the conical sheet). As the operating pressure is increased, both the radial and axial forces increase. This phenomenon occurs up to a point after which an increase of the operating pressure causes the axial velocity to dominate and the radial forces drop causing the nozzle to operate as an “open orifice” nozzle, i.e., with increasing pressures the cone angle decreases and eventually disappears. The maximum cone angle depends on the geometrical characteristics³ of the swirl nozzle. This result is in accord with the findings of Dupouy et al. [10]. It is also important to note that the swirl action creates tangential forces, which induces frictional losses. This results in a reduced flow efficiency, so that a swirl-jet nozzle with the same flow rate as an impaction-pin nozzle must have a larger orifice and will make larger droplets.

With a swirl-jet nozzle, continuously increasing the operating pressure results in a decrease in droplet size only to the point that the swirl phenomenon breaks down (i.e., the nozzle begins to act like an open orifice). An impaction-pin nozzle, however, makes smaller droplets as the pressure is increased.

Droplet Size Measurement

Location of Measurement. A large variation in droplet size, and size distribution, is found depending on the location in the plume where the measurements are taken. The droplet size at the center of the plume is smaller than the size at the edge. As the droplets leave the tip of the nozzle at high velocity, the induced airflow creates a draft toward the center of the plume. This draft carries the smallest droplets to the center of the plume, since smaller droplets follow the velocity and the direction of the airflow—droplets under 10 microns immediately follow the trajectory of the airflow. Consequently, measuring the size at the edge of the plume and at the center, gives a completely different droplet size and size distribution.

Figure 8 shows the distribution of the droplet size inside the plume in still air and in an airflow velocity field for an impaction-pin and a swirl-jet nozzle.⁴ Figure 8 is for an impaction-pin nozzle in still air (Figs. 8(a) and 8(b)) and under airflow velocity (Figs. 8(c) and 8(d)). Results for a swirl-jet nozzle are shown for still air in Figs. 8(e) and 8(f). These plots show both the Dv90 diameters (on the right) and the SMD (D32) diameters (on the left). The plot uses a color continuum to represent droplet sizes. In this plot, the smallest droplet sizes are represented by blue and the largest by red. The different color dots⁵ show the size heterogeneity in the distribution of the droplets, in the R (radial from the orifice) and Z (axial from the orifice) directions.

The charts in Fig. 8 clearly show that:

- droplet size is a strong function of the measurement location.

³Ratio of swirl-chamber diameter to final orifice diameter and L/D ratio of the swirl chamber, and the L/D ratio of the final orifice.

⁴Swirl-jet Nozzle: 152 micron (0.06 inch) flow rate of 7.5 l/h (0.033 gpm) at 138 barg (2000 psig). Impaction pin nozzle: 152 micron (0.006 inch) orifice diameter, flow rate of 10.2 l/h (0.045 gpm) at 138 barg (2000 psig).

⁵The physical sizes of the dots on this plot are also proportional to the diameters.

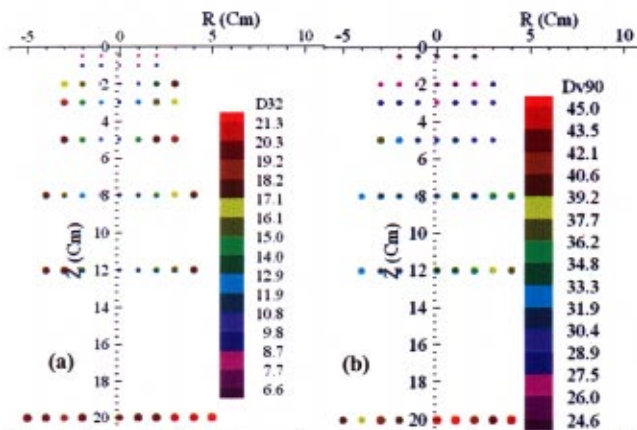


Figure 8-a and 8-b. Impaction-pin droplet size experimental results for still air, operating pressure 2000 psig (138 barg)

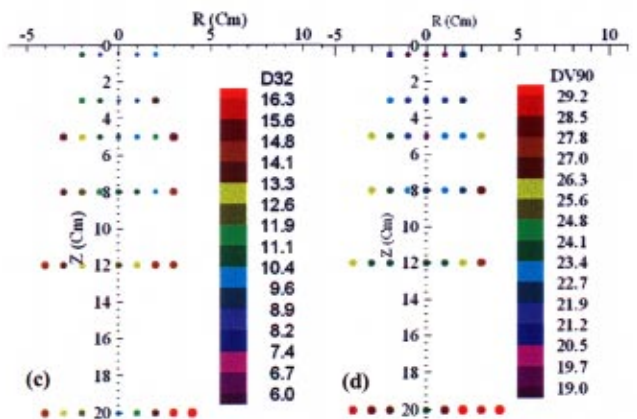


Figure 8-c and 8-d. Impaction-pin droplet size experimental results for airflow velocity of 900 fpm (4.6 m/sec), operating pressure 2000 psig (138 barg)

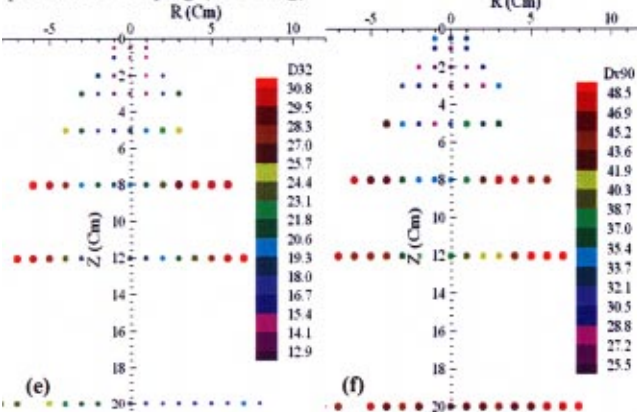


Fig. 8 Impaction-pin droplet size experimental results for still air, operating pressure 2000 psig (138 barg) (a, b); impaction-pin droplet size experimental results for airflow velocity of 900 fpm (4.6 m/sec) operating pressure 2000 psig (138 barg) (c, d); swirl-jet nozzle droplet size experimental results for still air, operating pressure 2000 psig (138 barg) (e, f)

- smallest droplet sizes are found in the center of the plume near the exit of the nozzle.
- the smaller drops evaporate quickly, leaving only the larger droplets as axial distance increases.
- in the radial direction, droplet size increases as we move to

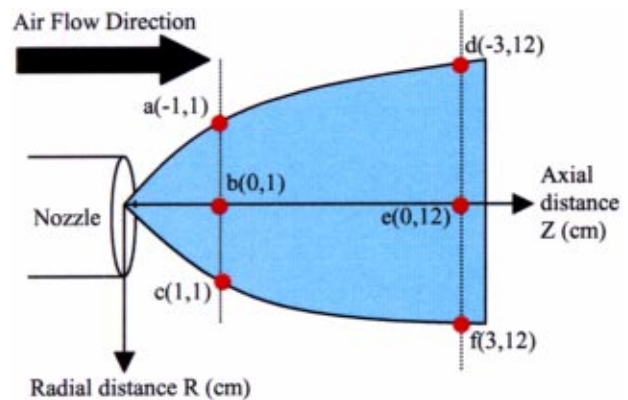


Fig. 9 Spatial location for experimental testing

the edge of the plume because the small droplets quickly follow the airflow and move to the center while the larger droplets maintain their initial trajectories and stay in the edge of the plume.

Size is affected by air velocity as can be seen by comparing Figs. 8(a) and 8(c). For example, the maximum diameter decreased from 21.35 microns to 16.54 microns. This is due to the fact that airflow separates the small droplets and large droplets into different trajectories, so the larger droplets, which initially have a higher velocity, do not collide and coalesce with the smaller droplets. All droplets take the same velocity within several centimeters of the orifice, and the density of droplets is much less as the plume expands, so that there is little probability of coalescence further from the orifice.

Comparing the droplet size between impaction-pin and swirl-jet nozzles (Figs. 8(a) and 8(b) with 8(e) and 8(f)) tested in still air, we can see that both D32 and Dv90 sizes are much smaller in the case of impaction-pin nozzle, even though the impaction-pin nozzle has a flow rate approximately 25% higher than the swirl-jet nozzle.

Effect of Velocity on Droplet Size

In order to explain the effect of velocity on droplet size, experimental tests were conducted in our wind tunnel with impaction-pin nozzles operating at 138 barg. The spatial location definitions of the measurement points Z and R are shown in Fig. 9. Figure 10 shows the results of tests when wind tunnel velocities are varied between 0 and 2500 ft/min (0–12.7 m/sec).

Readings were taken at different radial and axial positions in the fog plume. Each graph in Fig. 10 has an indicator such as “R-1Z0.5.” This means that the measurement point is taken at a location of 1 cm above the plume centerline and at an axial distance of 0.5 cm from the nozzle. In Fig. 10, Z has been fixed at 0.5 and R is at -1, 0 and +1 cm respectively for Figs. 10(a, b and c). Thus Figs. 10(a, b, and c) represent measurements near the nozzle at different radial locations. The set of Figs. 10(d, e, and f) are at an axial distance of 12 cm ($Z12$) and radial locations of -3, 0, and +3 cm. Thus Figs. 10(d, e, and f) represent measurements at some distance from the nozzle. In each of the figures, the abscissa is the droplet diameter in microns. The axis to the left is the cumulative volume in % (the S-shaped curves relate to this) while the axis on the right is the volume Frequency (the bell shaped curves relate to this). The different color lines all relate to measurements taken at velocities of 0 fpm, 1000 fpm, and 2500 fpm (0 m/sec, 5.1 m/sec, and 12.7 m/sec).

In order to understand how these diagrams can be used, we provide the following examples. In Fig. 10(a), if we enter the

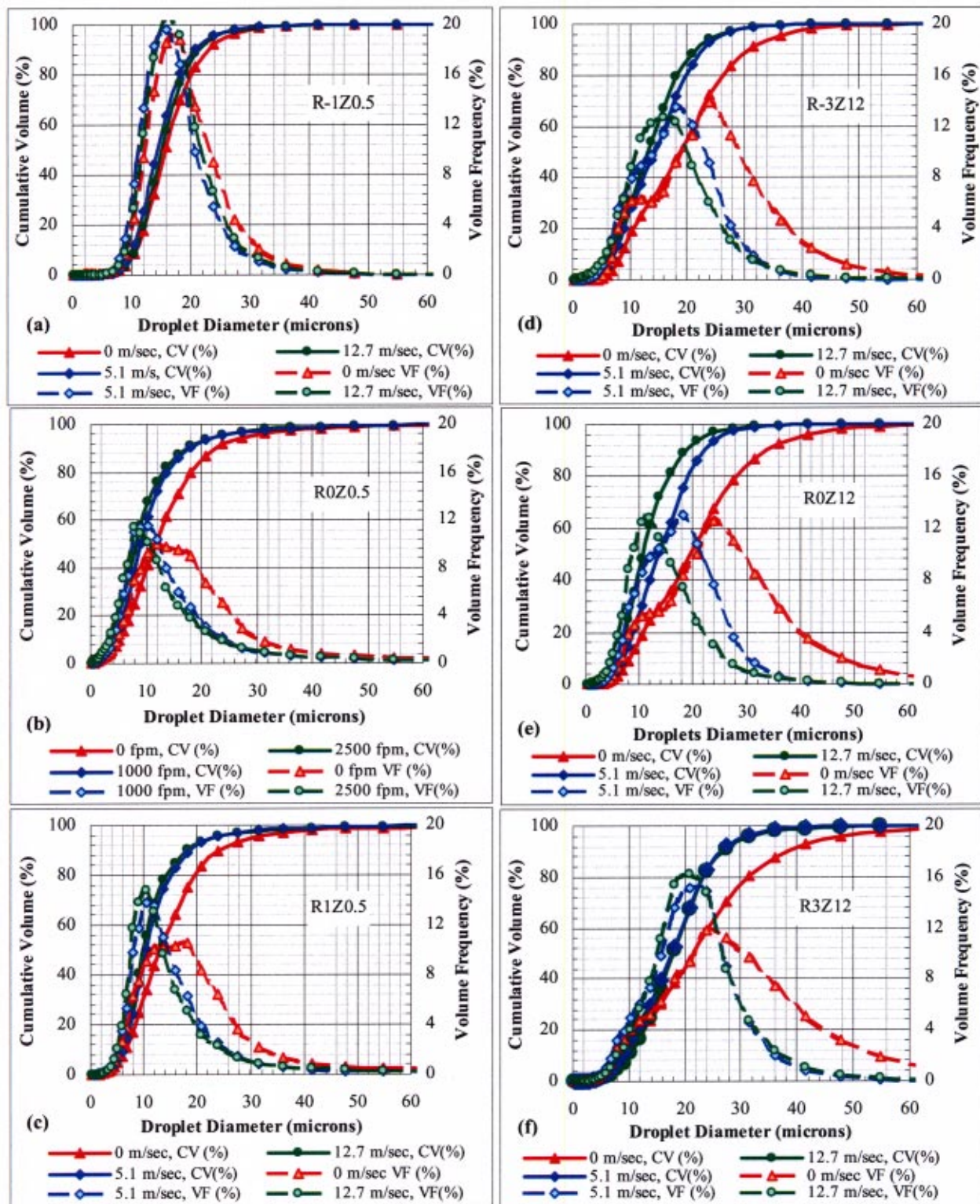


Fig. 10 Experimental measurements showing influence of flow velocities and measurement distances on droplet sizes, operating pressure: 138 barg

chart at a cumulative volume of 90 (on the left side), and look at the red S-shaped curve (velocity of 0 fpm) we can read from the abscissa a diameter of 20 microns. This means that 90% of the spray volume is contained in droplets with diameters less than 20 microns or, in other words, the Dv_{90} diameter is 20 microns.

Similarly the Dv_{50} (mass median diameter) can be read to be approximately 15 microns.

In Fig. 10(e), if we enter the volume frequency bell shaped curve for 0 fpm, we see that approximately 10% of the volume of the spray

Table 1 Sauter mean diameters (microns) for the different locations and at different flow velocities for the nozzle tested in Fig. 10

Velocity Position	0 fpm 0 m/s	1000 fpm 5.1 m/s	2500 fpm 12.7 m/s
R-1Z0.5	10.29	9.4	7.94
R0Z0.5	8.45	6.96	5.99
R1Z0.5	10.82	9.19	9.24
R-3Z12	15.38	11.77	9.7
R0Z12	15.18	12.58	10.88
R3Z12	16.36	14.41	12.83

(or of the mass, since we are in all cases dealing with demineralized water) is droplets with a diameter of about 30 microns.⁶

Table 1, shows the Sauter mean diameters for the different velocities and for the different locations tested in Fig. 10.

Observations on the Effect of Location and Velocity on Droplet Size. Based on Fig. 10, the following important observations can be made:

- The cumulative volume (the s-shaped curves) exhibit very small variations for changes in air flow velocity when measurements are made near the nozzle as can be seen in Fig. 10(a).
- This difference increases as the axial distance Z increases as can be seen in Figs. 10(d, e, and f).
- When the Z value is high, i.e., when measurements are made far from the nozzle in the axial direction, the bell shaped curves tend to move to the right, i.e. the smaller droplets have evaporated and only larger diameters remain.
- In examining Fig. 10(d), it can be seen that with increasing airflow velocities the droplet size distribution move to the left (decrease) and so the cumulative volume (S-curves) shifts to the left as expected.
- As the velocities increase, the droplet diameters drop. For example, in Fig. 10(e), the $Dv90$ sizes for flow velocities of 0, 5.1, and 12.7 m.s⁻¹ are 30, 22, and 18 microns, respectively. This occurs due to fewer collisions, as described above.
- Some impaction-pin nozzles exhibit bimodal distribution but only at low velocities. For example, in Fig. 10(d), the red curve which represents a flow velocity of 0 m.s⁻¹, shows a bimodal pattern and only at larger Z locations. This is probably due to droplets recirculation. As velocity increases, the distribution becomes unimodal (see the blue line representing 5 m.s⁻¹ in Fig. 10(d)).
- The effect of gravity can be seen. For example, in Figs. 10(a), 10(b), and 10(c), the $Dv90$ numbers are 22, 23, and 24 microns as the measurement location is moved from the top ($R = -1$) to the bottom ($R = 1$) of the plume. When we examine the behavior 12 cm from the nozzle (Figs. 10(d), 10(e), and 10(f)), the sizes are 32, 35, and 40 microns a larger change than was observed at $R = 0.5$ cm. As would be expected because more time has passed so the droplets have fallen further. Table 1 shows gravity effect on $D32$.
- In examining Fig. 10(f) it is interesting to note that the volume frequency bell curve shifts to the left as the velocity increases. Again, this is due to the fact that as the velocity increases, the droplets follow the trajectory of the airflow and the probability of collision is lowered, resulting in smaller measured sizes. This effect can also be seen but to a lesser extent in Fig. 10(c).
- Form these curves we see that from a droplet-size standpoint, increasing the airflow velocity above 1000 fpm (5.1 m.s⁻¹) does not reduce droplet sizes significantly. In most cases, gas turbine fogging nozzles are installed in a location in the inlet duct that has about this velocity. Locating the nozzles in a higher velocity region therefore would have little effect on droplet size.

⁶The droplets are in a bin or class of 29–30 microns.

Experimental Study of Impaction-Pin and Swirl-Jet Nozzles

An extensive experimental study was conducted to examine the comparative performance of swirl-jet and impaction-pin fog nozzles in their application to gas turbine inlet air fogging. The study included both still-air tests and also wind tunnel experiments at different airflow velocities and for different nozzle-operating pressures. Some of the results are provided here along with comments.

Still-Air Experimental Analysis—Swirl-Jet and Impaction-Pin Nozzles. The experiment was done in still air, at an operating pressure of 2000 psig (137 barg) and a distance from the nozzle of 2 inches (5.08 cm). The two nozzles selected had similar orifice sizes (about 150 microns) but because of the greater losses in the swirl-jet nozzle, the flow rate for the impaction-pin nozzles was slightly higher (0.045 gpm, 0.170 liter/min for the impaction-pin and 0.036 gpm, 0.136 liter/min for the swirl jet, both at 138 barg, 2000 psig).

The objective of these tests was to compare the general characteristics of two nozzle types.⁷ Specific nozzle designs may differ slightly in performance details but the differences between a nozzle of a given design (i.e., swirl or impaction pin) with a given orifice diameter were not found to be significant. The droplet size (both SMD and $Dv90$) and the concentration volume (CV in ppm) were measured. The concentration volume quantifies the amount of water in the measurement volume. The experimental results are presented in Fig. 11(a) (swirl nozzle) and in Fig. 11(b) (impaction nozzle). The following observations may be made from Fig. 11.

- For both types of nozzle, the droplet diameters are smaller in the center of the plume. As stated earlier, this occurs due to the airflow ingress effect as explained by Obakata and Long [11]. This clearly shows that results obtained from measurements taken only at the center of the spray plume are of little value.
- In examining the volume concentration (CV) one can see that for the swirl-jet (Fig. 11(a)) the concentration is much less in the center due to the hollow cone effect. With the impaction pin, there is more uniformity⁸ as is seen in Fig. 11(b). This is due to the smaller droplet size produced. With a swirl-jet volume concentrations will become more homogeneous as the axial distance increases. (This effect can also be seen in Fig. 8.)

The impaction pin makes smaller droplets than the swirl jet; on average 6 micron smaller SMD and 7 micron smaller $Dv90$. This is primarily due to the fact that the straight-through orifice of the impaction pin has less frictional loss than the swirl-jet nozzle, so jet velocity is higher and the resulting conical sheet is thinner, hence smaller droplets.

Wind Tunnel Experimental Analysis—Swirl Jet and Impaction Pin. To compare the performance of the two types of nozzles under airflow conditions, wind tunnel experiments were conducted. The airflow velocity was fixed at 15.2 m/sec (3000 feet per minute). The nozzle operating pressure was varied between 21 and 207 barg (300 and 3000 psig). Measurements were taken in two positions in the plume. The first measurement was in the center of the plume 7.6 cm (3 inches) from the nozzle ($R = 0$ cm, $Z = 7.6$ cm). The second measurement location was also 7.6 cm from the nozzle orifice but at the edge of the plume ($R = -2.5$ cm, $Z = 7.6$ cm).

In examining Figs. 12(a) and 12(b) for impaction pin and swirl nozzles, respectively, it is clear that impaction-pin droplet sizes make significantly smaller droplets even at higher airflow velocities. This is because the initial droplet sizes are smaller.

⁷These observations are based on numerous tests that were conducted on a wide range of makes and designs of swirl-jet and impaction-pin nozzles.

⁸There will be a small nonuniformity if measurements are made in line with the impaction pin. However, this spatial effect is very small.

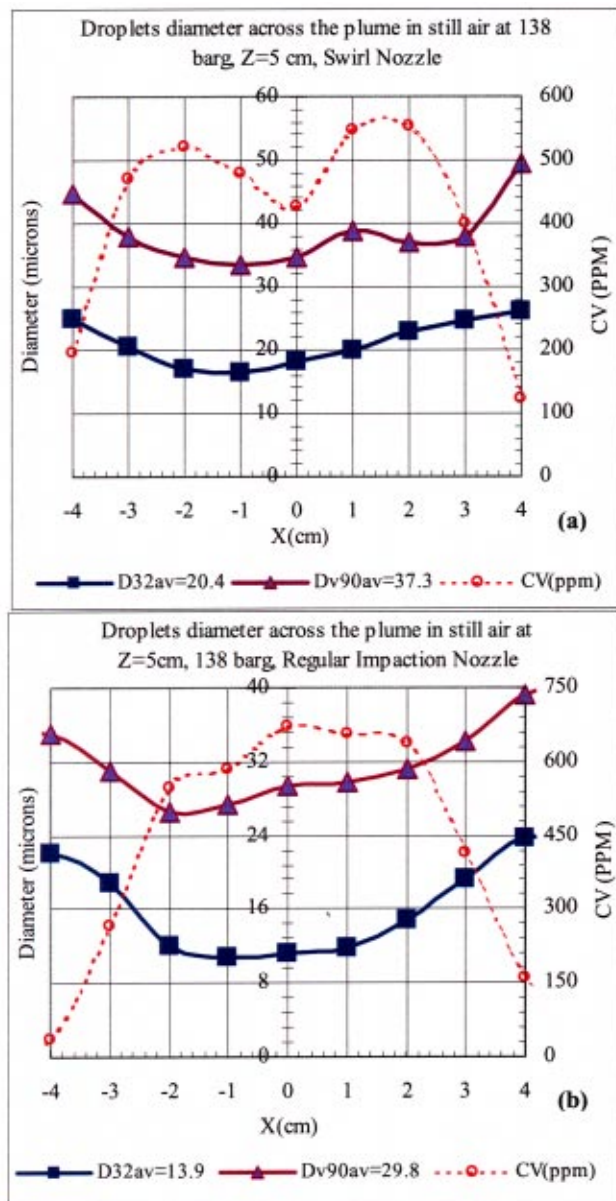


Fig. 11 (a, b) Experimental results of droplet size distribution in the plume for comparable swirl and impaction-pin nozzles

The following observations can be made from Fig. 12:

- Droplet size decreases with an increase in pressure. While this holds true for both types of nozzle, the effect is more prevalent for swirl-jet nozzles until a pressure of about 1500 (103 barg) to 2000 psig (138 barg).
- For both nozzles, the curve of decreasing droplet size, with increasing pressure, begins to flatten out. This is due to the fact that there is an increase in droplet density and velocity near the nozzle orifice, which results in an increase in the probability of collision and coalescence. The effect is more pronounced with swirl-jet nozzles because the swirl cone begins to collapse with increasing pressure, resulting in both a thicker conical sheet (bigger droplets) and a higher droplet density and velocity (increased collision probability).
- For both nozzles, increasing the operating pressure above about 137 barg (2000 psig) does not result in significantly smaller droplets.

Even looking at the corresponding plumes visually (Fig. 13)

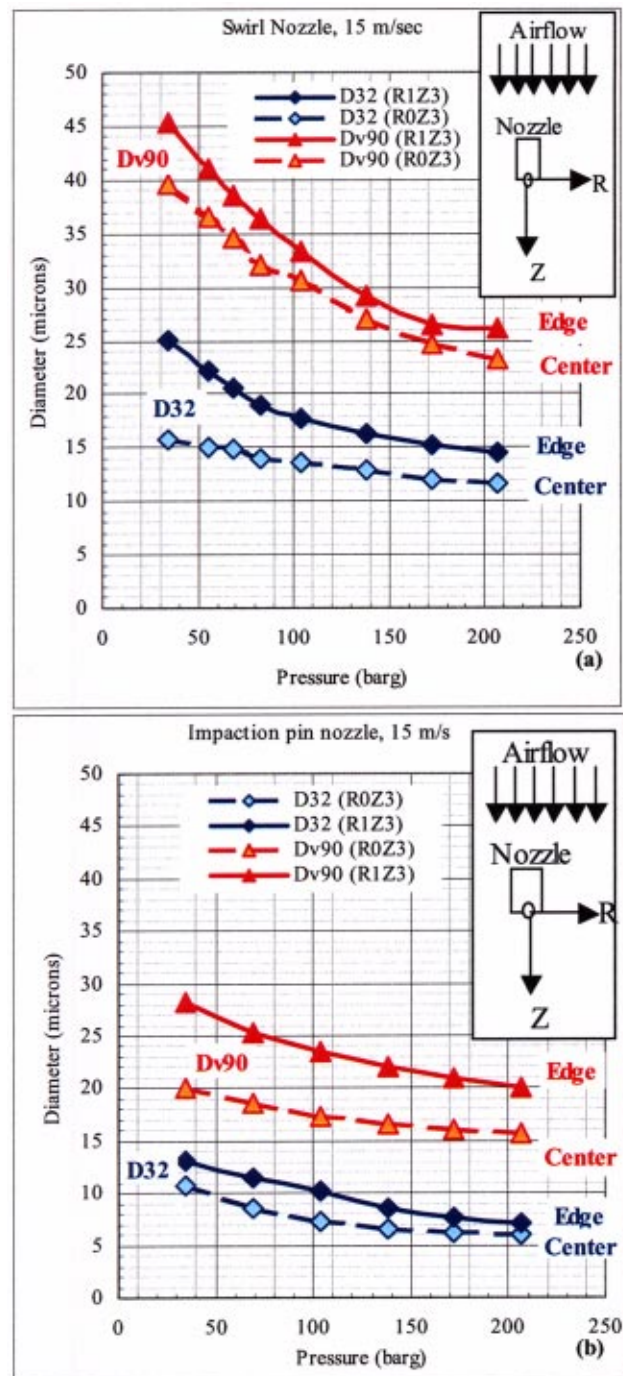


Fig. 12 Experimental comparisons between impaction and swirl-jet nozzles at varying pressures. Measurements of both Dv90 and D32 have been taken at different locations in the plume. The graph shows that the impaction-pin nozzle provides smaller diameters regardless of where measured

one can see that, for a swirl-jet nozzle, the cone shape can be distinctly seen, because the large droplets have higher penetration velocities. In examining Fig. 13(a), the smoke-like nature of the fog can be clearly seen. This provides a visual confirmation of the above data.

Standardization of Measurement Methods and Data Interpretation for Fogging Nozzles for Gas Turbine Applications

There is no extant standard of droplet size measurement for gas turbine inlet air fogging applications. ASTM has provided overall

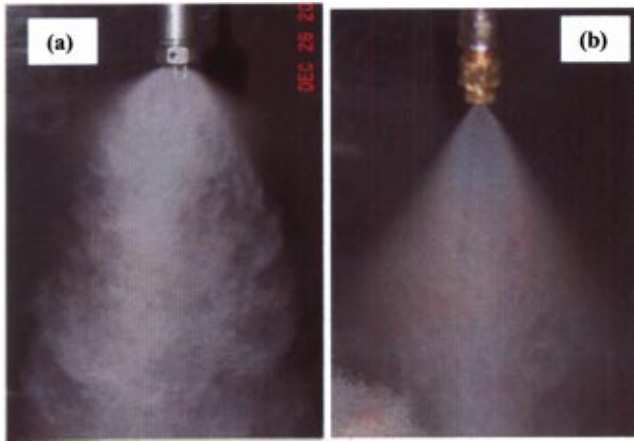


Fig. 13 Plume characteristics of impactation-pin (left) and swirl-jet type nozzles (right) at operating pressure of 137 barg. The smoke-like nature of the impactation-pin nozzle is evident. The straight edge of the swirl-jet nozzle is indicative of the high momentum of the larger droplets, implying a much larger droplet size at the edges as also indicated by the graphs above

guidelines but no definitive approaches have been used in the fogging industry. The result is that the user is confronted by data that is impossible to reconcile. To overcome this problem, a new measurement standard has been developed and is proposed here. The measurement standard incorporates the following:

- defines the instrumentation to be used,
- defines locations where measurements are to be made,
- defines the airflow velocities under which the measurements should be made,
- defines the averaging approach and representative diameters to be used, and
- defines the final report format to be used.

Proposed Standard of Droplet Size Measurement for Gas Turbine Fogging Applications. We propose the following standard be adopted for the characterization of gas turbine inlet air fogging nozzles:

- Use the Malvern Spraytec system, or equivalent type instrument, for measurement of droplet size.
- Measurements should be made at flow velocity of 4.6 m/sec (15 ft/sec).⁹
- All tests should be conducted on a minimum of 5 randomly selected nozzles within error bars of 10%.
- Measure *across the spray plume* at a distance of 2 inches¹⁰ (5 cm) from the orifice. Take as many readings as are required to cover the entire cross section of the plume.
- The measurement duration for each portion measured should be a least one minute, to ensure measurement repeatability.
- Calculation of the diameter should be a weighted average performed by using the volume concentrations (ppm) derived at each measurement point. This calculations should be done on all representative diameters.
- Final report should include the Dv90 and D32 (Sauter mean) diameters as these are the most significant for gas turbine operations.
- A single representative diameter may be derived from Dv90 and D32 values.

⁹This velocity has been chosen as a reasonable velocity for typical location of inlet fogging manifolds in gas turbine ducts.

¹⁰The distance of 2 inches (50.8 mm) was been selected, as it is far enough away to allow for the measurement of any coalescence yet close enough to minimize the effect of evaporation.

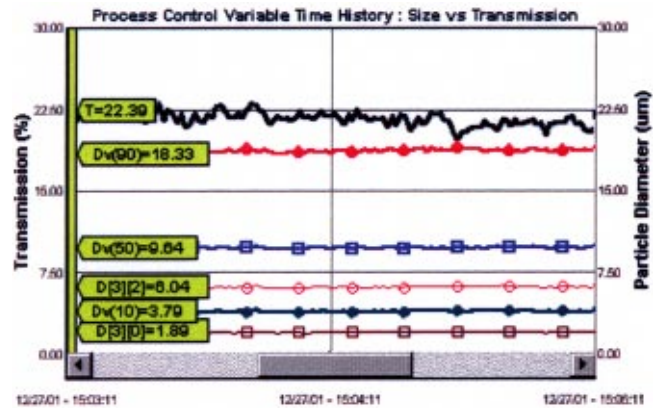


Fig. 14 Data check for repeatability showing consistency in diameters

The variation of the droplet size over a period of 5 minutes, for a given position in the spray, with a sampling frequency of 100 Hz is shown in Fig. 14. The figure shows a relative variation of less than 5% in the droplet size, which indicates that the Malvern unit offers good repeatability. This sort of figure should be show for any droplet test to assure repeatability.

Further work is required to develop a comprehensive specification and testing procedure but the standard given is a good start. This standard was developed by and has been extensively used by Mee Industries for all its analytical work. In our opinion, the most detailed and accurate method for measuring and reporting droplet sizes for inlet fogging nozzles and will make it possible for turbine operators to make reasonable comparisons of different fog nozzles.

Closure

This paper has provided a comprehensive analysis of droplet sizing methodology, design aspects of fog nozzles and measurement approaches as they apply to gas turbine inlet air fogging. The importance of the correct use of representative droplet diameter definitions and proper measurement approaches have been covered, and the effect of airflow velocity on fog droplet behavior, and the resulting effect on final droplet sizes, has been described. Details of experimental studies comparing swirl-jet and impactation-pin nozzles have been provided. Extensive testing has shown that under all operating conditions, the droplet sizes of impactation-pin nozzles are smaller than swirl-jet nozzles with the same flow rate. Finally, a standard procedure for the characterization of fog droplets in the context of gas turbine inlet air fogging has been defined and proposed. Chaker et al. [12,13] provide further details on theoretical and practical considerations and fog behavior in gas turbine inlet air ducts.

Acknowledgments

The lead author would like to acknowledge the contributions of Allen Reinholtz of Mee Industries' controls group for his work in helping set up the wind tunnel measurement systems and Conrad Klemzak, MeeFog R&D technician, for his help with the experimental setups. We also acknowledge and thank the large number of Meefog system users who's technical inputs and support has been most valuable.

Nomenclature

- A_d = surface area of the droplet (m^2)
- AD or D21 = absorption diameter (m)
- AMD or D10 = arithmetic mean diameter (m)
- ASTM = American Society for Testing and Materials
- CV = concentration volume (ppm)

D_d = droplet diameter (m)
 ED or D31 = evaporative diameter (m)
 fpm = feet per minute
 MMD or Dv50 = mass median diameter (m)
 PDPA = phase Doppler particle analyzer
 PMS = particle measurement system
 RSF = relative span factor
 SAMD = surface area mean diameter (m)
 SMD or D32 = Sauter mean diameter (m)
 V_d = droplet volume (m^3)
 VMD or D30 = volume mean diameter (m)
 V_{rel} = droplet relative velocity (m.s^{-1})
 We = Weber number
 γ_w = surface tension of the water (N.m^{-1})
 ρ_a = density of the air (kg.m^{-3})

Appendix

References

- [1] Dodge, L. G., 1992, "TESS: Tool for Spray Studies," Tech Today.
- [2] Teske, M. E., Thistle, H. W., Hewitt, A. J., and Kirk, I. W., 2000, "Conversion Of Droplet Size Distributions From PMS Optical Array Probe To Malvern Laser Diffraction," Eighth International Conference on Liquid Atomization And Spray Systems, Pasadena, CA, July.
- [3] Doble, S. J., Matthews, G. A., Rutherford, I., and E. S. E., Southcombe, 1985, "A System for Classifying Hydraulic Nozzles and Other Atomizers into Categories of Spray Quality," *Proceedings of the 1985 British Crop Protection Conference: Weeds*, vol. 9A-5, pp. 1125–1133.
- [4] Arnold, A. C., 1990, "A Comparative Study of Drop Sizing Equipment for Agricultural Fan-Spray Atomizers," *Aeronaut. Sci. Technol.*, **12**, pp. 431–445.
- [5] Young, B. W., and Bachalo, W. B., 1987, "The Direct Comparison of Three 'In-Flight' Droplet Sizing Techniques for Pesticide Spray Research," International Symposium on Optical Particle Sizing: Theory and Practice, Rouen, France.
- [6] Dodge, L. G., 1987, "Comparison of Performance of Drop-Sizing Instruments," *Appl. Opt.*, **27**, pp. 1328–1341.
- [7] Arnold, A. C., 1987, "The Drop Size of the Spray From Agricultural Fan Spray Atomizers as Determined by a Malvern and the Particle Measuring System (PMS) Instrument," *Atomization Spray Technol.*, **3**, pp. 155–167.
- [8] Le Coz, J. F., 1998, "Comparison Of Different Drop Sizing Techniques On Direct Injection Gasoline Sprays," 9th International Symposium On Application Of Laser Techniques To Fluid Mechanics, Lisbon, July 13–16.
- [9] Pilch, M., and Erdman, C. A., 1987, "The Use of Breakup Time Data and Velocity History Data to Predict the Maximum Size of Stable Fragments for Acceleration-Induced Breakup of Liquid Drop," *Int. J. Multiphase Flow*, **13**(6), pp. 741–757.
- [10] Dupouy, D., Flores, B., Lisiecki, D., and Dumouchel, C., 1994, "Behavior Of Swirl Atomizers Of Small Dimensions," *ICLASS 94*, Rouen, France, July, pp. 374–381.
- [11] Obokata, T., and Long, W. Q., 1994, "LDA/PDA Characterization of Conical Spray for Diesel Engine," *ICLASS 94* Rouen, France, July, pp. 278–285.
- [12] Chaker, M., Meher-Homji, C. B., and Mee T. R., III, 2002, "Inlet Fogging of Gas Turbine Engines—Part I: Fog Droplet Thermodynamics, Heat Transfer and Practical Considerations," **126**, pp. 545–558.
- [13] Chaker, M., Meher-Homji, C. B., and Mee T. R., III, 2002, "Inlet Fogging of Gas Turbine Engines—Part III: Fog Behavior in Inlet Ducts, CFD Analysis and Wind Tunnel Experiments," **126**, pp. 571–580.

Mustapha Chaker
Director,
Research and Development
Cyrus B. Meher-Homji
Chief Engineer
Thomas Mee III
Chairman and CEO

Gas Turbine Division,
Mee Industries, Inc.,
204 West Pomona Avenue,
Monrovia, CA 91016

Inlet Fogging of Gas Turbine Engines—Part III: Fog Behavior in Inlet Ducts, Computational Fluid Dynamics Analysis, and Wind Tunnel Experiments

The inlet fogging of gas turbine engines for power augmentation has seen increasing application over the past decade yet not a single technical paper treating the physics and engineering of the fogging process, droplet size measurement, droplet kinetics, or the duct behavior of droplets, from a gas turbine perspective, is available. This paper along with Parts I and II provides the results of extensive experimental and theoretical studies conducted over several years coupled with practical aspects learned in the implementation of nearly 500 inlet fogging systems on gas turbines ranging in power from 5 to 250 MW. In Part III of this paper, the complex behavior of fog droplets in the inlet duct is addressed and experimental results from several wind tunnel studies are covered.
[DOI: 10.1115/1.1712983]

Introduction

Over the past decade, the application of inlet fogging for the power augmentation of gas turbines has become increasingly popular. It is estimated that approximately 700 gas turbines have fogging systems installed at this time including many modern F class machines. Part III of this paper focuses on fog behavior in inlet ducts while Parts I and II, Chaker et al. [1,2], cover the area of modeling droplet evaporation, practical considerations, and nozzle testing.

Due to the wide range of duct configurations and arrangements, there are several issues that become of cardinal importance in designing and implementing fogging systems. Some of these issues include

- ducts with complex geometry. For example, some ducts have multiple 90 deg bends
- ducts with short configurations that minimize fog residence time
- duct obstructions, blow in door interference, trash screens, silencer bull noses, etc.
- wall and floor wetting considerations
- drainage issues
- silencer issues and fog interaction
- optimal selection of fog nozzle manifold pitches.

In order to examine and understand several of these complex issues, a carefully designed and controlled testing facility was needed. For this reason a special variable speed wind tunnel was constructed to simulate different aspects of gas turbine intake ducts. The wind tunnel allows the real-world testing of nozzles, the development of optimal nozzle location and orientation schemes, and the testing of special drain and water removal approaches. The wind tunnel is shown in Fig. 1.

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30564. Manuscript received by IGTI, December 2001, final revision, March 2002. Associate Editor: E. Benvenuti.

Experimental Setup Description

Wind Tunnel. The wind tunnel consists of a multisectioned duct approximately 11 meters in length (36 feet). It has the following sections which can be configured in various ways (dimensions are mm and inches):

- A and B: Two (2) long duct sections with clear polycarbonate walls; 2440×860×860 mm (96×34×34 inches)
- C: One short duct section with clear polycarbonate walls and floors; 1220×860×305 mm (48×34×12 inches)
- D and E: Two reduction sections from stainless steel sheet metal; 1220 mm long, one end 860×860 mm, other end 305×860 mm (48 inches long, 34×34 inches and 34×12 inches)
- F: One short section with clear polycarbonate walls, which can serve as a 90 deg bend; 860×860×860 mm (34×34×34 inches)
- G: One fan section, from waterproof plywood; 1220×860×860 mm (48×34×34 inches).

Sections A, B, and C are built from clear, 1/4 inch thick, polycarbonate sheet. This allows visual observation of the trajectories of the fog droplets in the wind tunnel. Section F can be used either as a 90 deg elbow section in the duct or to increase the length of the duct. This is useful for examining the flow of fog through bends that typical of intake ducts.

Another section that simulates the silencer in a gas turbine was built, also in clear polycarbonate. This section, with dimensions 860×860×127 mm (34×34×5 inches), was built to be used in the small Section C. The air velocity in the duct can reach 5.1 m/s (1000 feet per minute) in the big sections A, B, etc. and 15.2 m/s (3000 feet per minute) in the small Section C without the silencer or 25.4 m/s (5,000 feet per minute) with the silencer section in place.

The reduction Sections D and E are built from stainless steel sheet. Section G is built from wood and made waterproof using a special paint. Two trash screens were built, one at the beginning and one at the end of Section G to protect the variable speed fan from foreign objects and to simulate and measure the size and trajectories of the droplets emitted from trash screens in typical gas turbine ducts. The size of the mesh is 13 mm×13 mm (1/2



Fig. 1 Experimental wind tunnel used for studying fog behavior in gas turbine inlet air ducts: 11 meter (36 foot) long, capable of up to 25 m/s (5000 fpm) airflow velocity

inch by 1/2 inch) and the thickness of the wire is 16 mm (1/16 inch). The second trash screen used is stainless steel with a 51 mm×51 mm (2 inch by 2 inch) mesh dimension and 3.2 mm (1/8 inch) wire thickness.

High Pressure Water Atomizing System. The high-pressure atomizing system, which generates the fog droplets, uses a variable speed drive Cat Pump that is capable of developing 207 barg (3000 psig). The atomizing nozzles used for testing purposes are either impaction-pin or swirl-type nozzles. The water is filtered and demineralized and the flow is measured with a flow meter before being injected into the duct.

The high-pressure water passes through a manifold and is divided into a number of lines each of which contains a number of nozzles. The configuration allows for modification of the distance between the nozzles, the distance between lines, and also the angle of the nozzles compared to the airflow direction. Single nozzles can also be tested as required.

Data Acquisition System. The laboratory is equipped with a high-speed data acquisition system, which consists of a DAS 1200 board with up to 16 input channels and 2 output channels. Daisy Lab software is used to manage the data acquisition. The DAS monitors

- relative humidity in several locations,
- air velocities,
- fog water flow rate, and
- pressure drop in the duct.

The setup is flexible to allow for a wide variety of gas turbine fogging related experiments. The position of the sensors in the duct is shown in Fig. 2.

All of the sensors were calibrated before conducting experiments.

Temperature and Relative Humidity. All temperature and relative humidity measurements are taken using VAISALA HMD60U/YO sensors. Accuracies are $\pm 1\%$ for the temperature and $\pm 2\%$ for the relative humidity.

Water Flow Meter. The water flow coming to the nozzles is measured using an Omega flow meter FLR 1011. The accuracy of this model is about $\pm 3\%$. The sensors use a Pelton-type turbine wheel and an electro-optical detection device to convert flow rates into a linear 0 to 5 VDC signal.

Air Flow Velocity. The sensor measuring the velocity of the air in the duct is an Omega FMA-905-I air velocity transducer. Additional airflow measurement using a handheld turbine meter is also possible.

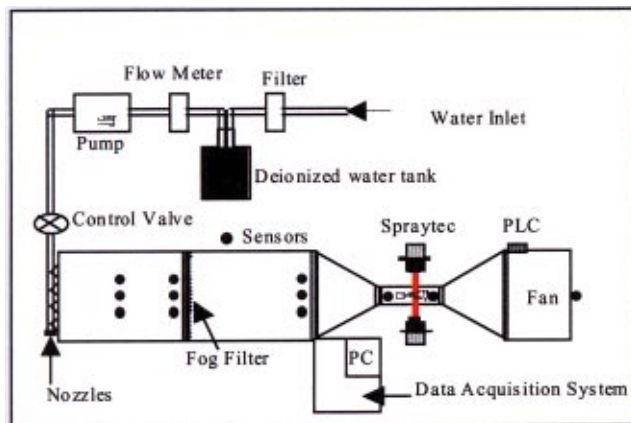


Fig. 2 Instrumentation layout in the experimental wind tunnel

Duct Pressure Drop. The sensors measuring the pressure drop in the duct are differential pressure transducers, Omega type PX-274. The measured values correlated well when compared to calculated numbers.

Malvern Spraytec Laser Light Scattering Droplet Measurement System. A sophisticated Malvern Spraytec droplet measurement system is used along with associated software to characterize and measure droplet distributions and sizes. The laser measurement system is shown in Fig. 3 along with a nozzle undergoing tests. Further details are provided in Part II of the paper.

Experimental Work Using the Wind Tunnel

Nozzle Location. Gas turbine inlet systems come in a wide variety of configurations and shapes. Some of the complexities that have to be addressed include:

- multiple side entry configurations—two or three-side entry configurations where considerable care has to be taken to arrange nozzle manifolds in such a way has to have uniform coverage.
- configurations with steeply curved roofs that require progressive changes in the nozzle spray angle to avoid roof wetting and impaction on nearby nozzle manifolds.
- short duct configurations where residence time is minimal. In these situations special patterns of the fog nozzles may have to be used to optimize the temperature distribution across the airflow. In most cases, horizontal lines are used but in some cases lines may

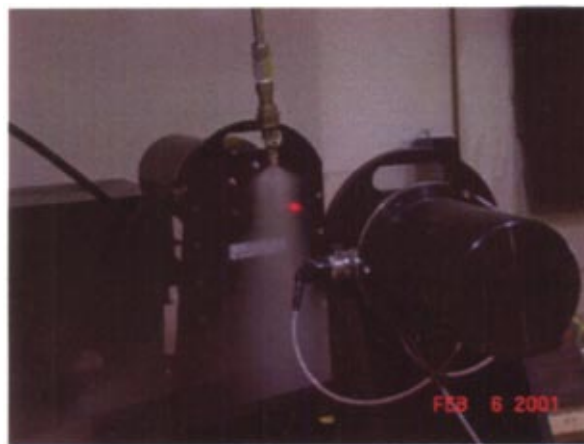


Fig. 3 Malvern spraytec laser light scattering droplet measurement system



Fig. 4 V-shaped duct configuration requiring a special nozzle array configuration

have to be vertical (three-side entry filter systems, for example) to allow nozzle angular changes to be made in a vertical plane.

- complexities relating to unusual duct obstructions, such as blow-in doors, reheat manifolds, duct support trusses, and even generator cooling ducts, which in some turbines pass through the main inlet air duct.

It is important to note that the guidelines presented ahead are based on Mee Fog nozzles, which generate a fog with a $Dv90$ less than 20 microns¹ and, due to the wide range of intake duct configurations, the optimization of nozzle locations and spray angles is often based on experience. When the situation warrants, computational fluid dynamics (CFD) studies can be performed but even CFD studies must be modified by experience. Analytical tools are useful but lessons learned from *actual installations* contribute immeasurably to a successful final selection of the nozzle array configuration. Figure 4 shows a configuration for a V-shaped air filter house that required a nozzle array as shown. A steep duct roof is shown in Fig. 5 and the nozzle angular orientation is shown in Fig. 6.

The decision to locate the nozzle array either before or after the silencer depends on several factors. A residence time of 1 to 2 seconds of unobstructed flow is ideal but rarely exists unless a special duct modification is made or it is a new gas turbine installation where extra duct sections can be incorporated at the design stage.

Residence time is an important consideration that must be carefully evaluated. As shown in Fig. 7, droplets attain the airflow velocity in a few milliseconds due to the large drag forces.² This figure shows the response time for a droplet to attain air stream velocity as a function of droplet size.

Even very small droplets can collide with items in the ducts (Fig. 8), including silencers, walls at duct bends, structural supports, blow-in doors and trash screens, of course the problem is much worse with nozzles that produce larger droplets. Such collisions can lead to pooling of water on duct floors or the creation of larger droplets if fog water collects on surfaces and is then

¹As commercially available nozzles made by other makers may produce larger droplets, the guidelines presented here may not apply.

²As droplet Reynolds number is very small, the coefficient of drag is exceedingly large and the droplet accelerates very rapidly.

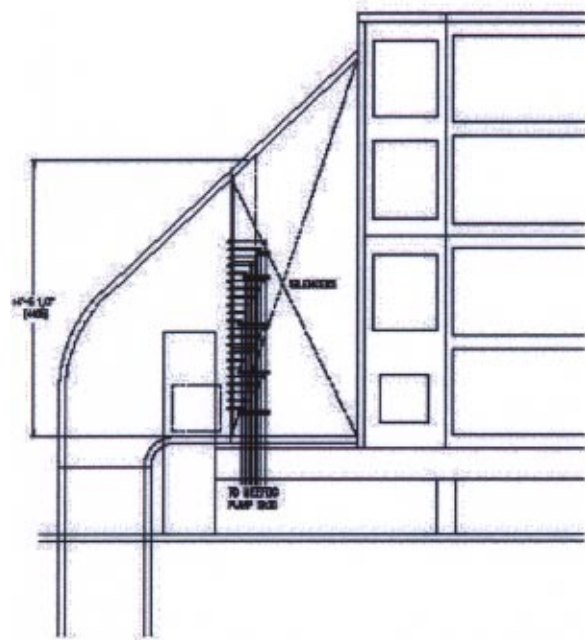


Fig. 5 Nozzle array configuration for a steep-roofed duct

re-entrained into the high-velocity airflow. These factors make the proper design of the nozzle arrays, and the water collection and drainage systems, a very important aspect of proper fog system design.

Nozzle Orientation and Fog Distribution. Once a suitable location for the nozzle arrays has been established, one must consider the questions of nozzle orientation and distribution, i.e., the spacing of the nozzles on the nozzle manifolds. The fundamental factor driving the design of nozzle orientation and distribution is to obtain uniform droplet distribution in the intake duct, and to thereby ensure that the airflow is evenly cooled. Uneven distribution of nozzles, or large gaps between the nozzle manifold tubes, can result in poor mixing of the fog with the inlet airflow. This leads to longer evaporation times and temperature distortions at the compressor inlet.

The nozzle orientation angle chosen depends on

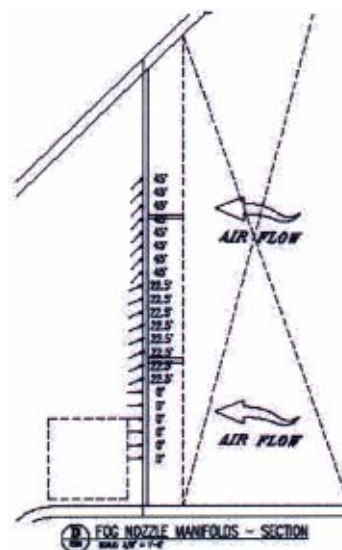


Fig. 6 Optimization of nozzle angular orientation

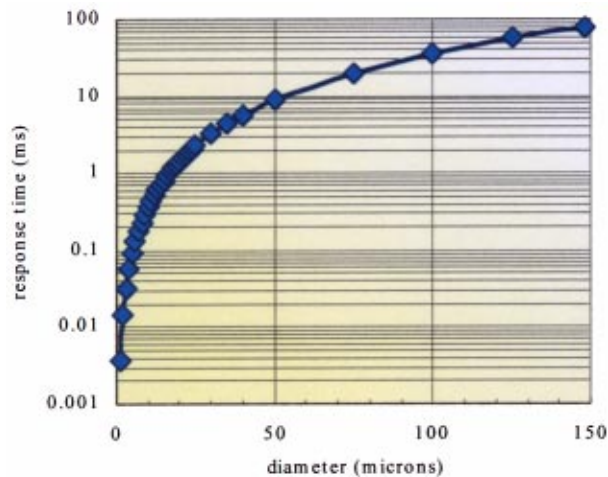


Fig. 7 Response time for droplets to attain air stream velocity as a function of droplet size

- airflow velocity,
- distance between nozzle lines,
- operating pressure and spray pattern of the nozzles,
- duct wall and roof shape constraints, and
- overall duct geometry.

Several studies have been done by CFD modeling and by empirical testing (conducted using the wind tunnel and in actual installations) to verify and optimize nozzle layouts. The nozzle angle with respect to the inlet airflow may vary between 0 deg (co-flow) and 90 deg (perpendicular to the flow). Nozzle orientation perpendicular to the flow can be used if the plume does not interact with the duct walls or the other nozzle lines. Orienting the nozzles into the airflow generally results in droplet impaction on the nozzle manifold tubes themselves, and is therefore not recommended. At 90 deg, the increased angle gives a marginally longer residence time, as compared to the co-flow position. However, due to practical considerations, 90 deg is rarely used. Angles varying from 0 to 60 deg have been successfully employed.

The effect of nozzle angle with respect to flow is shown in Figs. 9, 10, and 11. Figure 9 shows the shape of the plume in a co-flow position (i.e., the nozzle spraying in the direction of the airflow). The plume diameter stays relatively constant in the axial length. This is an advantage when the distance between the nozzles is not too large as it provides a homogenous pattern across the duct. However, in cases where there is a large spacing between nozzles, the 90 deg orientation may be advantageous because, as can be

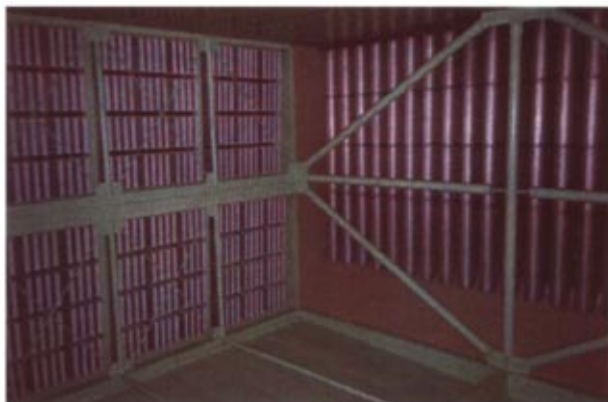


Fig. 8 Typical obstruction found in an inlet air duct



Fig. 9 Co-flow nozzle orientation of a fog nozzle in wind tunnel; airflow velocity is 4 m/s (800 ft/min), operating pressure is 138 barg (2000 psig)



Fig. 10 90 deg orientation of a fog nozzle in the wind tunnel; airflow velocity is 4 m/s (800 ft/min), operating pressure is 138 barg (2000 psig)

seen in Fig. 10, the plume diameter is much larger and consequently covers more of the duct cross section. The counterflow position is shown in Fig. 11. The counter flow position is never used for the reason stated above.

With respect to fog distribution, Fig. 12 shows two lines with multiple nozzles in the wind tunnel, operating at 4 m/s (800 fpm). The distance between the nozzle lines is 20.3 cm (8 inches). It can be seen that the plumes make contact approximately at an axial distance equal to two times the line pitch. Figure 13 shows a different view of the same setup. The distance between nozzles is 4 inches (10.1 cm). In this case we see that the point of plume contact is approximately equal to the nozzle pitch.

It is important to note that if swirl-jet nozzles were installed in this same way, the conical shape of the swirl jet plume and the existence of much larger droplets in the edges of the plume, may result in coalescence where the cones intersect.³ In some situations, where larger pitches are required, it may be beneficial to use a triangular configuration as shown in Fig. 14.

This configuration results in a very well distributed fog pattern. Several experiments were performed to measure the possibility of

³Part II of this paper discusses this subject at some length.



Fig. 11 Counterflow orientation of a fog nozzle in the wind tunnel; airflow velocity is 4 m/s (800 ft/min), operating pressure is 138 barg (2000 psig)



Fig. 13 Face view of nozzle array; airflow velocity is 4 m/s (800 ft/min), operating pressure is 138 barg (2000 psig)

droplet collision and coalescence in the intersection of two or more fog nozzle plumes. These experiments show that there is no measurable coalescence of droplets in this region. Numerical analysis confirms that the statistical probability of coalescence in this area is very small. As discussed in Part II of this paper, collision and coalescence do play a roll in droplet size near the nozzle orifice, where both relative velocities and droplet density are much higher.

It is important to understand the velocity and pressure profiles in the intake duct and to recognize that certain areas will have accelerated flow and pressure gradients due to duct bends and turns. Nozzles may have to be specially oriented to accommodate these patterns. In real-world fogging applications, ideal duct configurations hardly ever occur, so proper design of fog nozzle orientation and manifold location is a critical factor for fog system design.

Number of Nozzles and Nozzle Pattern. The number of nozzles, and their spray pattern, should be appropriately designed so they provide uniform fogging of the gas turbine inlet duct. There is a tradeoff between nozzle flow rate, spray pattern, number of nozzles and pressure drop across the nozzle manifolds. Having a fewer number of higher flow rate nozzles can result in

larger spaces between the plumes and less homogeneity of the fog distribution. Furthermore, larger flow nozzles, as a general rule, produce larger droplets and fewer nozzles lead to uneven temperature distribution, especially during part-load fogging.

The nozzle pattern itself should be such as to cover the maximum cross-sectional area of the airflow.

Fog Droplet Behavior at Silencers. In practice, nozzle arrays are often best located upstream of the silencers as this loca-



Fig. 12 Side view of nozzle array; airflow velocity is 4 m/s (800 ft/min) operating pressure is 138 barg (2000 psig)

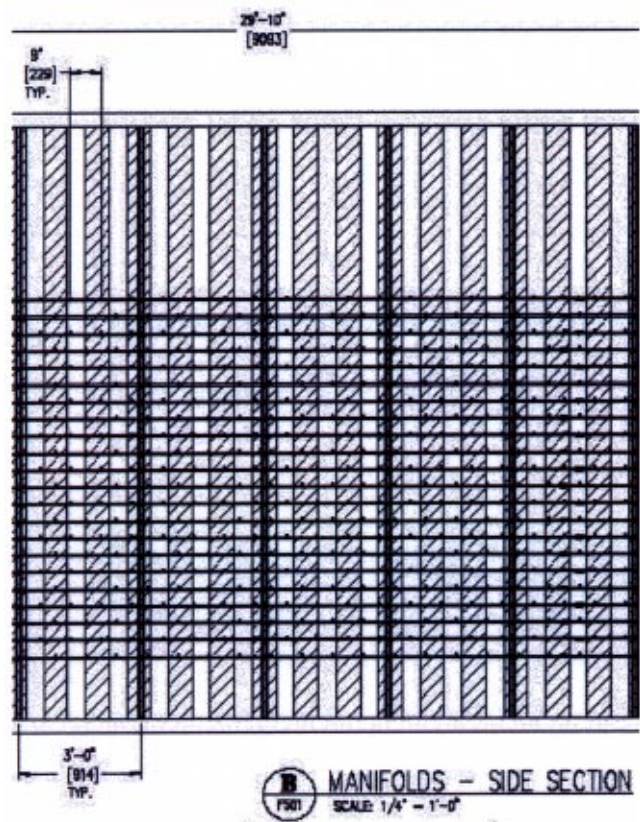


Fig. 14 Triangular pattern nozzle arrangement

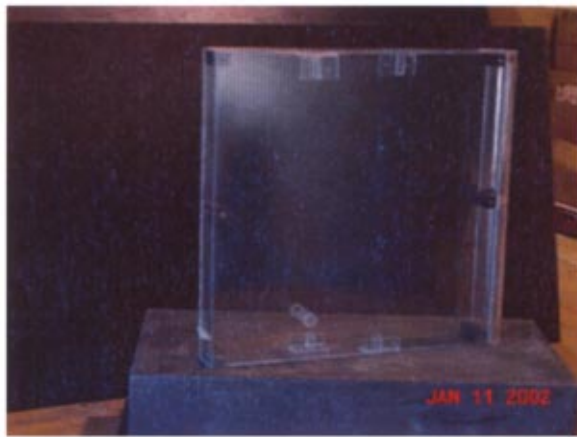


Fig. 15 Silencer section model used for wind tunnel studies

tion gives the longest residence time and best possible mixing. Therefore it is important to understand by actual experimentation and observation

- how the fog interacts with the flow field around the silencers,
- wetting effects on the silencer,
- ideal distance between fog nozzles and silencers,
- special approaches such as locating the nozzles between the silencer baffles (this is rarely done but has worked successfully in some cases), and
- effect of different silencer nose profiles on fog droplet behavior and wetting.

Several studies were been done using the instrumented wind tunnel. A transparent Polycarbonate model of a silencer section was built and tests were conducted under varying fog and air velocity conditions. Figure 15 shows the silencer section model. The silencer model was also modified with different nose profiles to examine the affect on flow behavior.

The flow through silencers was also modeled by CFD techniques as presented in the section ahead and experimental verification was conducted with the wind tunnel. This was done by locating the model at different positions near the wall of the droplet measurement section of the wind tunnel. This allowed visual observation of the flow of fog and also measurement of the droplet sizes at the various silencer locations and under various fogging conditions. Drain water flow from the silencer nose, with different nose profiles, was also measured. These tests resulted in the development of beneficial design tools that allow fairly accurate predictions of quantity fog impaction that can be expected on silencer noses when fog droplets are very small, and under favorable airflow velocities, most of the fog droplets will follow the flow lines around the nose of the silencer.

Fog Interaction With Trash Screens. As most fogging systems have to operate upstream of a trash screen, it is important to experimentally study the large droplet creation effects from the trash screen. This was done experimentally using the wind tunnel.

Figure 16 shows a photograph of trash screen sections used in the wind tunnel.

Trash screens are particularly good collectors of small fog droplets because the cross section of the screen wires is so small that droplets cannot easily follow the air stream around them. Experimental results indicate that the droplet sizes formed from fog collection on trash screen wires were found to be proportional to the wire diameter and in some cases droplets as large as 2 mm (0.08 inches) were noted. This may be of critical importance when trash screens are very close to the compressor inlet. However, when the

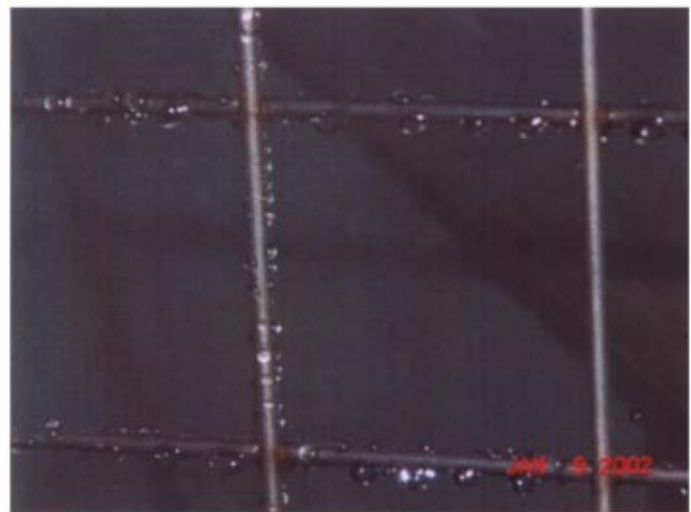


Fig. 16 Fog collection on a trash-screen resulting in larger droplet formation

trash screens are located near the filter house, prior to the elbow section, the problem is less severe as good drainage systems can collect any large droplets.

Fog Collection on Intake Cones. It is also possible that the intake cones of cold-end drive machines will collect some moisture and some of this may then progress inwards to the compressor, covering the intake struts. Observations and calculations have shown however that the Weber number effect will come into play here, as the relative velocity differences between a droplet on the cone and the flow as it accelerates into the bellmouth will cause the droplet to shatter. More details are provided in the section ahead on the Weber number effect. Figure 17 shows the intake cone of a large gas turbine.

Droplet Eliminator Application and Testing. Due to the requirements of certain retrofit applications, several studies have been conducted in the wind tunnel to study droplet eliminator behavior under fogging conditions and to optimize the angular location of the eliminator. An experimental setup is shown in Fig. 18.

The droplet eliminator induces a large pressure drop that is, as expected a strong function of the flow velocity as shown in Fig. 19. Further, as can be seen in the figure, the pressure drop increases significantly when the eliminator gets wet and saturated



Fig. 17 Intake cone and struts at an axial compressor inlet

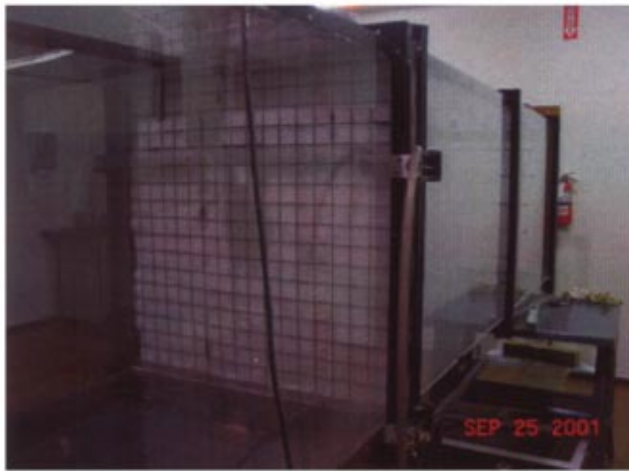


Fig. 18 Typical fog droplet eliminator (as used in an HVAC humidification application)

with water. Our experience with numerous installations has indicated that the use of droplet eliminators is not beneficial for gas turbine applications because of the inlet pressure drop penalty, which detracts from performance year round.

Significance of Weber Number for Inlet Fogging

The Weber number was covered in Part II of the paper with respect to droplet behavior near the fog nozzle. The Weber number is the ratio of aerodynamic forces to surface tension forces and is given by the equation

$$We = \frac{\rho_a V_{rel}^2 D_d}{\gamma_w} \quad (1)$$

Studies have shown that shattering of droplets occurs when the inertial forces overcome the surface tension forces, which happens when the Weber number is less than 13, [3]. Details on aerodynamic breakup of liquid droplets are provided in Suzuki and Mitachi [4] and Samenfink et al. [5].

Due to the high relative velocities that would occur in the event that water droplets collected on the intake cone or on the inlet guide vanes (IGVs), the Weber number effect is of some importance.

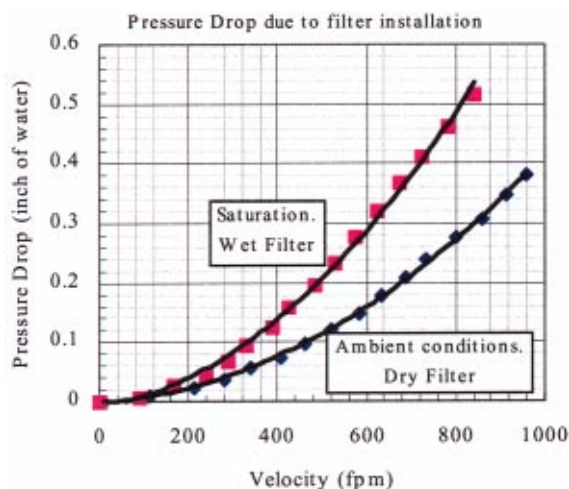


Fig. 19 Pressure drop of a droplet eliminator under both dry and wet conditions

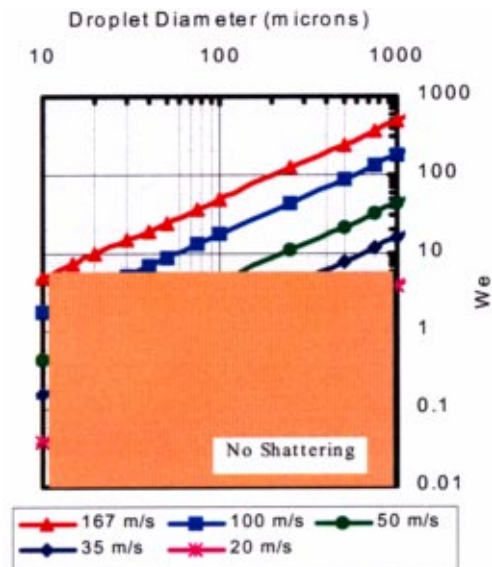


Fig. 20 Graph showing the Weber number for varying droplet sizes. Velocities shown are relative velocities between the droplet and the airstreams

Figure 20 shows a graph that shows what air velocities and droplet sizes would result in droplet shattering. The velocities indicated are relative velocities between the droplet and the airflow. For the cases where we have droplets adhering to stationary surfaces, we can assume that the droplets themselves are at a very low velocity and so the *relative* velocity becomes high, almost equal to the air stream velocity. It is important to note that this figure should only be applied to larger agglomerated droplets that may be present near the compressor.

CFD Analysis for Fog Flow in Gas Turbine Inlet Ducts

CFD analysis has been extensively used for

- examinations of flow and pressure fields in intake ducts,
- the study of droplet behavior around obstructions,
- to optimize nozzle array locations and the arrangement and orientation of nozzles on the arrays, and
- to evaluate droplet trajectories for different droplet diameters and airflow velocities so that the potential for collision and coalescence of droplets can be studied.

The CFD model uses a Lagrange-Euler framework for two-phase flows, which is appropriate for the modeling of fog behavior. Flow patterns for the fog for different types of ducts have been examined. Many configurations that have been modeled by CFD studies have been experimentally verified using wind tunnel and real-world studies. In this way, the accuracy of the models can be constantly tuned and validated. Some typical applications are depicted ahead.

The trajectories of fog droplets emitted from a nozzle in an elbow section in the duct are shown in Fig. 21. This has been done for relatively small droplet sizes. It is possible to vary the airflow and to see how the droplet trajectory will follow the duct bends. Figure 22 shows the velocity distribution in the duct over silencer sections in two dimensions. Figure 23 shows the distribution of the velocity in the duct in three dimensions. Figure 24 shows the velocity profile in a duct bend in two dimensions.

CFD analysis can be applied to establish the best possible location for fog nozzle manifolds in both retrofit situations and new units. The ideal location for nozzle manifolds is the location that gives the maximum residence time in the duct for the fog droplets prior to the airflow entering the compressor (to give maximum evaporative efficiency and ensure a minimum size for any droplets

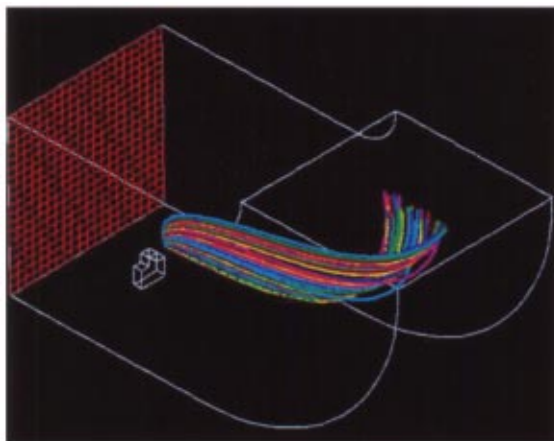


Fig. 21 CFD representation of fog flow streamlines in a duct bend

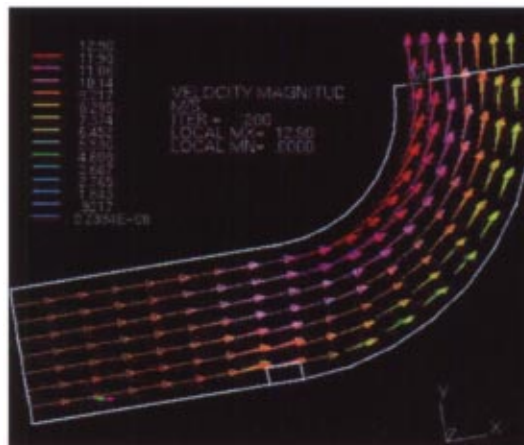


Fig. 24 Velocity flow vectors in turning duct flow

entering the compressor) and that results in the least amount of fog impaction and collection on obstructions in the duct.

As mentioned before, establishing the best location for nozzle manifolds is as much an art as a technology and requires extensive experience with real-world fog applications. Duct configurations and the types of obstructions present in ducts vary widely and it

is, therefore, important that operators who are considering installing an inlet fog system provide detailed drawings and, where possible, photographs of the inside of the air filter house and the inlet ducts to the fog system supplier. A detailed design justification should also be requested from the fog system supplier. The design justification should include a detailed description of the proposed design and information about the design, including:

- the design justification for the chosen fog nozzle manifold location with information about residence time and expected size of any droplets that might enter the compressor
- information about potential for fog impaction on obstructions, silencers, duct walls and floors, etc., and methods proposed for mitigation of excessive pooling
- information about drains, drain channels, and other water collection devices
- calculations for nozzle array strength and methods for avoidance of airflow-induced vibration and FOD.

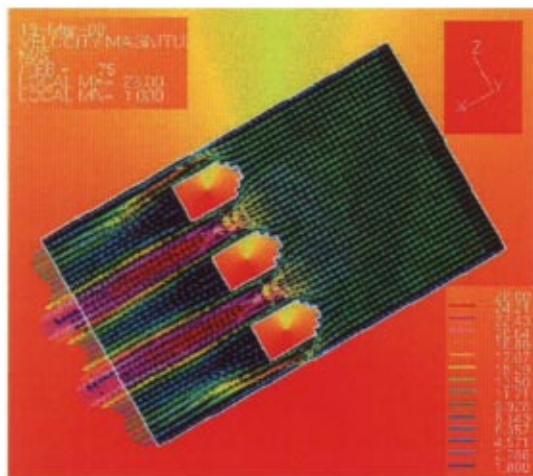


Fig. 22 Flow pattern velocity field around silencer section

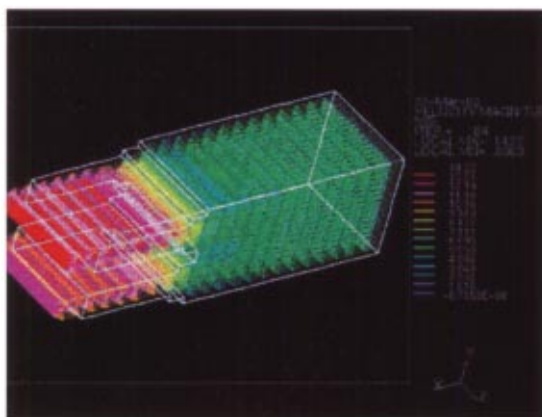


Fig. 23 Three-dimensional visualization of flow velocity field over a silencer section

Mechanical Design of the Fog Nozzle Manifold

Vortex Shedding and Vibration Considerations. Fog nozzle manifolds are located downstream of the filter systems so proper care has to be taken to ensure their mechanical integrity. The nozzle manifolds and their supporting structures will exhibit resonant oscillations under certain velocity conditions, as a consequence of vortex shedding. Bluff bodies in an air stream shed vortices at an oscillating rate that is a function of the flow velocity and the shape of the structure. If the vortex shedding frequency is equal to, or a harmonic function of, the natural resonant frequency of the manifold tube or support structure, the oscillations can reach an amplitude sufficient to cause catastrophic failure of the array. There are several design approaches relating to strengthening of the structure and the use of vibration absorbing clamps but it is important to carefully evaluate the natural frequency behavior of the nozzle array. The array should be studied under the range of velocities that can occur for the given turbine. The compressor ingests a constant volume of air (assuming a fixed-speed single-shaft design) so that changing atmospheric conditions result in changes in the air velocity in the duct. Furthermore, if the turbine is operated off-design for extended periods of time, the airflow velocity in the duct must be considered under these conditions. Nozzle manifolds must be evaluated under both the wet and dry conditions as water in the tubes will change their natural frequency.

To avoid the occurrence of resonance, calculations must be made of the natural frequencies of the tubes and the struts using analytical formulas. The forcing frequencies (vortex shedding frequencies) due to the airflow past the cylindrical or bar-shaped

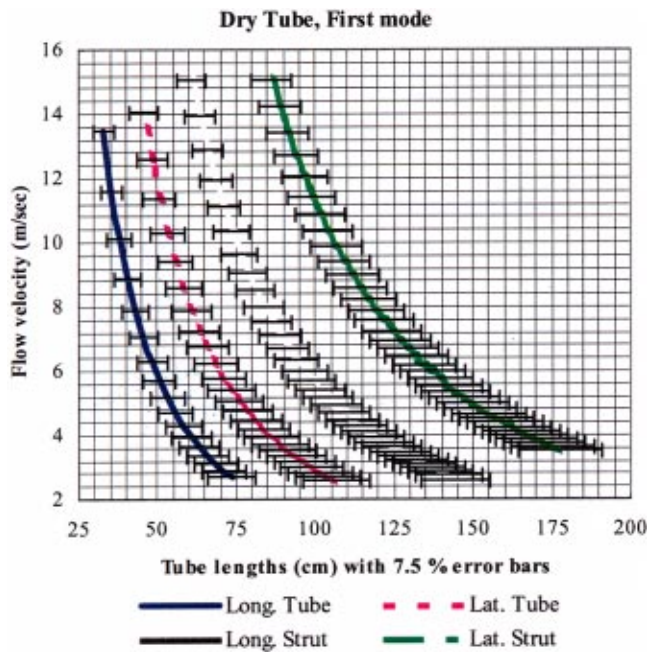


Fig. 25 Resonant frequency interference diagram for dry operation of nozzle array tubes

(tube or strut) are calculated using formulas and dimensionless numbers—the Reynolds (Re) and Strouhal (St) numbers. The resonant frequencies of tubes and struts depend on the material of construction and dimensions. Knowing the range of possible air-flow velocities in the intake duct, it is possible to select the right nozzle line configuration in order to ensure the avoidance of resonance. Details on flow-induced vibration may be found in Thomson [6], Barbi et al. [7], Stansby [8], and Ongoren et al. [9].

Based on our standard tubes and struts, a design guide has been developed to evaluate the structural properties of nozzle arrays. Figure 25 provides design envelope guidelines based on the first mode of vibrations for tubes running dry for a range of velocities. Figure 26 provides the same guideline for tubes running wet. These curves are plotted with error bars of $\pm 7.5\%$. They show the interaction for only the first mode of oscillation, the second and the third mode only occur at a tube length of more than one meter (39 inches).

Pressure Drop Considerations. The air in the duct behaves as a compressible fluid when it goes through an orifice; the flow expands adiabatically as the pressure in the air falls. In this case the downstream static pressure will be less than that measured upstream. The equation to calculate the pressure drop in this condition is

$$\Delta P = \frac{(1 - \beta^4) \times U^2 \times \rho}{2 \times Y^2 \times C^2} \quad (2)$$

where ΔP is the pressure drop, β is the ratio of the open area to the total area: $\beta = A_{\text{open}}/A_{\text{tot}}$. U is the flow velocity through the orifice throat, ρ is the fluid density measured at the upstream position, C is the discharge coefficient, it is equal to 0.6 when the Reynolds number is higher than 10,000, which is always the case in our conditions, and Y is the expansion factor, which depends only on the ratio of specific heat at constant pressure (cp) to that at constant volume (γ).

The pressure drop through the typical nozzle manifold, as calculated and corroborated by measurement, are exceedingly small (on the order of <2.5 mm water gauge, <0.1 inch water gauge). It

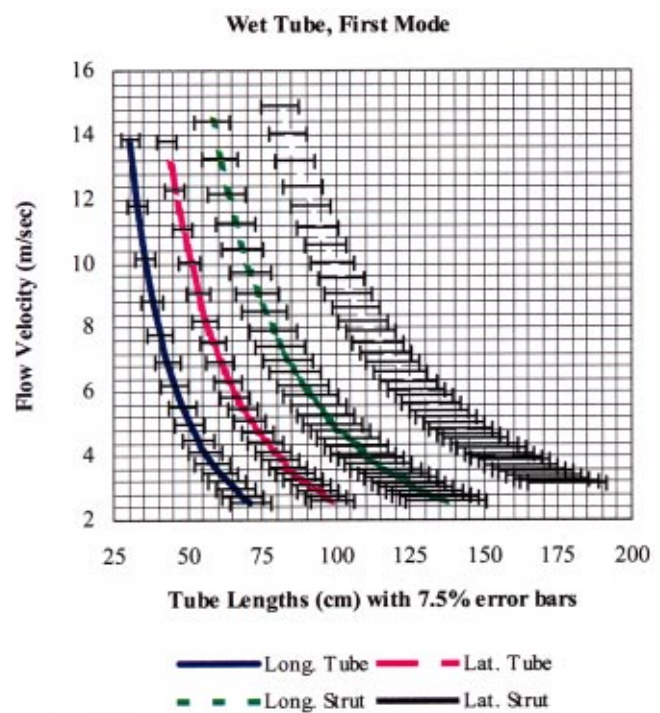


Fig. 26 Resonant frequency interference diagram for wet operation of nozzle array tubes

is in fact so small that it is difficult to measure.^{4iv} To understand the reason for the small pressure drop, we note that β is relatively big for a typical inlet air fogging array and U is also small in the location just after the filters.

Duct Drain Studies and Recommendations

The importance of proper and efficient drainage has been covered in Part I of this paper. Drains are often required at several locations in the inlet duct as well as at the bellmouth region. It is inevitable that some water runoff will occur due to wetting of duct walls and floors, and the presence of bends and other obstructions. Drains should be either P trap designs or flapper-style check valves. All drains provided must be open and operating during periods of inlet fogging. P trap arrangements are not preferred because of the risk of the water seal evaporating causing the trap to run dry and allowing ingestion of untreated air into the engine. P traps must also be drained during freezing conditions and fitted with a valve that can be closed during extended shutdown of the fog system, when the trap will be empty.

Several experimental and in-the-field tests have been conducted on drain systems to study:

- optimizing drain locations,
- optimizing the drain sizes,
- optimizing guttering that can be used to channel water into the drains, and
- development of special wall-drain gutters.

Figure 27 shows an experimental setup developed to actually quantify how much water could be evacuated by different line sizes under different negative pressure conditions. While this can be evaluated theoretically, we felt that it was important to actually test different drain configurations under realistic conditions. In this test apparatus, a viewing window was located to examine the flow behavior of the water that was being evacuated.

^{4iv}We refer only to the pressure drop due to the nozzle manifold. The overall filter house pressure drop is a function of the filter differential pressure that is a square function of the flow velocity.



Fig. 27 Test rig for drain studies

This sort of study ensures that the drains installed have the capability of accommodating the total flow under array leak conditions that might result in a large amount of water being dumped. However, it is important to note that, with a well-designed fog system, drains handle only 2–5% runoff flow under normal conditions. Furthermore, even very small leak in the nozzle system would be immediately detected by a properly designed fogging control system and an immediate shutdown would occur.

The use of viewing windows is recommended both at the nozzle array location and at the inlet bellmouth location. This is valuable in evaluating the fogging system performance and for tuning purposes. A viewing window installed on a large heavy-duty gas turbine located at the intake section to the axial compressor is shown in Fig. 28.

Closure

This paper has provided details of wind tunnel testing and CFD analysis for inlet air fogging in gas turbine ducts. In this paper along with Parts I and II, only some of the key issues could be briefly covered, in order to help gas turbine users in implementing fogging systems. Further papers will treat specific topics in greater detail. For an understanding of fogging behavior, it is imperative that theoretical analyses be validated by laboratory and when possible, by in-the-field testing.

Acknowledgments

The lead author would like to acknowledge the contributions of Allen Reinholtz of Mee Industries' controls group for his work in helping set up the wind tunnel measurement systems and Conrad Klemzak, MeeFog R&D technician, for his help with the experimental setups. We also acknowledge and thank the large number of MeeFog system users who's technical inputs and support has been most valuable.

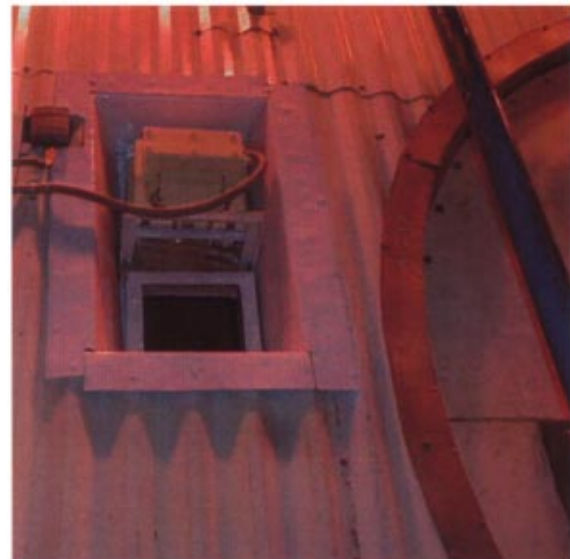


Fig. 28 Viewing window at the compressor inlet section of large gas turbine. Upper window is for lighting arrangement.

Nomenclature

C	= discharge coefficient
CFD	= computational fluid dynamics
D_d	= droplet diameter (m)
IGV	= inlet guide vanes
U	= flow velocity (m.s^{-1})
V_{rel}	= relative velocity of the droplet (m.s^{-1})
We	= Weber Number
Y	= expansion factor
β	= ratio of open area to total area
ΔP	= pressure drop in the duct (Wg)
γ_w	= surface tension of the water (N.m^{-1})
ρ_a	= density of the air (kg.m^{-3})

References

- [1] Chaker, M., Meher-Homji, C. B., and Mee, T. R., III, 2002, "Inlet Fogging of Gas Turbine Engines—Part I: Fog Droplet Thermodynamics, Heat Transfer and Practical Considerations," **126**, pp. 545–558.
- [2] Chaker, M., Meher-Homji, C. B., and Mee, T. R., III, 2002, "Inlet Fogging of Gas Turbine Engines—Part II: Fog Droplet Sizing Analysis, Nozzle Types, Measurement and Testing," **126**, pp. 559–570.
- [3] Faeth, G. M., Hsing, L. P., and Wu, P. K., 1995, "Structure and Breakup Properties of Sprays," *Int. J. Multiphase Flow*, **21**, p. 99.
- [4] Suzuki, T., and Mitachi, K., 2000, "Experimental Study on Aerodynamic Breakup of Liquid Droplets in Time Dependent Relative Velocity Fields," *Eight International Conference on Liquid Atomization and Spray Systems*, Pasadena, CA, July.
- [5] Samenfink, W., Hallmann, M., Elsaber, A., and Eittig, S., 1994, "Secondary Breakup of Liquid Droplets: Experimental Investigation for a Numerical Description," *ICLASS-94*, Rouen, France July.
- [6] Thomson, W. T., 1998, *Theory Of Vibration With Applications*, 3rd Ed., Prentice-Hall, Englewood Cliffs, NJ, p. 223.
- [7] Barbi, C., Favier, D. P., Maresca, C. A., and Telionis, D. P., 1986, "Vortex Shedding and Lock-On of a Circular Cylinder in Oscillatory Flow," *J. Fluid Mech.*, **179**, p. 527.
- [8] Stansby, P. K., 1976, "The Locking-On of Vortex Shedding due to the Cross-Stream Vibration of Circular Cylinders in Uniform and Shear Flows," *J. Fluid Mech.*, **74**, p. 641.
- [9] Ongoren, A., and Rockwell, D., 1988, "Flow Structure from an Oscillating Cylinder—Part 1: Mechanisms of Phase Shift and Recovery in the Near Wake," *J. Fluid Mech.*, **191**, p. 197.

Stefano Campanari
Assistant Professor
e-mail: stefano.campanari@polimi.it

Ennio Macchi
Full Professor of Energy Conversion
e-mail: ennio.macchi@polimi.it

Energetics Department,
Politecnico di Milano,
Piazza Leonardo da Vinci, 32,
20133 Milano, Italy

Technical and Tariff Scenarios Effect on Microturbine Trigenerative Applications

The paper considers the use of gas fired micro turbine generators (MTG) for trigeneration (combined production of electricity, heating, and cooling) applications in tertiary buildings. The importance of the adopted MTG technology is investigated, showing that the high electrical efficiency levels achievable by future advanced ceramic MTGs would improve dramatically the economic competitiveness of the application, as well as the primary energy savings and environmental benefits. Calculations are performed by the simulation code TRIGEN, capable of optimizing the plant operating mode in each time step and integrating the results over the entire year. The requirement of a "target" energy saving index on the optimization procedure is also addressed. [DOI: 10.1115/1.1762904]

1 Introduction

Small-scale distributed cogeneration or trigeneration (combined production of electricity, heating and cooling) is generally considered as an energy saving and environmentally benign solution, [1–3]. Gas fired microturbines (MTGs) are a promising candidate for the application to this market, thanks to their projected low ownership cost and low emissions; recent works, [4,5], have also shown how their integration with heat pumps and/or absorption chillers yields further relevant economic and energetic advantages. However, the competition is hard between these plants and conventional solutions using electricity generated by modern large-scale natural-gas fired combined cycles, [6], both by the point of view of energy saving and carbon dioxide emissions as well as economic competitiveness. Recent works have shown how the application of state-of-the-art MTG-based cogeneration plant to variable loads does not easily yield energy saving with respect to centralized electricity generation solutions, at least when the MTG is dimensioned to fulfill the peak demand, [7]. This paper investigates how the technical evolution of microturbines from present commercially available units to future advanced ceramic units will improve the economics and the energy balances of MTG cogenerative applications. It addresses also the task of the optimized choice of the MTG size (plant output vs. demand power ratio) in presence of variable loads and the influence of gas and electricity market conditions.

2 The Test Case

All calculations carried out in the paper refer to the case of a tertiary building, with specified heating, cooling and electric load hourly variation during the year (featuring climatic and load changes). While the specific load profiles are kept constant, the building size is varied during the analysis.

2.1 Plant Configuration. A cogenerative plant scheme is considered, where a microturbine is operated with a heat recovery unit, joined to an absorption chiller and to an inverse-cycle heat pump/chiller system (ERC/EHP), with the aim of creating a trigeneration (heating, cooling, and electricity) plant.

As shown in a previous work, [5], the addition of the inverse cycle adds great flexibility to the system, while the investment costs related to absorption chiller (AC) are more than counter-

balanced by the energy savings allowed by free heat recovery. The plant is integrated with the possibility of employing auxiliary boilers for backup and peak-load heat generation and connected to the electric grid. An example of plant configuration is given in Fig. 1. In all cases:

- the system is connected to the electric grid (EG) for purchase, backup or sale of the excess electricity, when economically viable;
- heating and cooling demands (H_d, C_d) are fully covered, using the natural gas boiler (NGB) when necessary. The NGB and ERC/EHP capacity is always set in order to match the peak load. A redundancy in the number of ERC/EHP units¹ is selected in order to guarantee the cooling peak load also in case of failure of one compressor;
- heat to AC can be supplied either by the MTG recovery unit or by the NGB.

2.2 Load Profiles. The selected load profiles are intended representative for a tertiary building located in central Europe. For a total volume of 10,000 m³, the peak electric, heating, and cooling demand are, respectively, 65, 148, and 201 kW.

The annual load is defined with a combination of 21 day-types accounting for seasonal climate variations (winter and summer days with medium/extremely hot/extremely cold temperatures, half-season days) and for working, half-working, and weekend day conditions, [5]. Loads are given specific to the building unit volume. The yearly distribution of the three loads is shown in Fig. 2, together with the resulting equivalent hours. It can be seen that cooling demand has the largest peak, but the involved annual energy is relatively small (about one third of the heating energy). For a large fraction of the year, there is co-presence of heating and cooling demands.

3 Components Characterization

3.1 MTG Simulation. The microturbine model is based on the simulation code described by [8]. The model has been calibrated with the assumptions of Table 1 to simulate the performances of two commercial units and a future ceramic microturbine.

- a Capstone 30 microturbine (25.6 kW_{el} net electric power) featuring a 23.3% net electric efficiency;

¹Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Atlanta, GA, June 16–19, 2003, Paper No. 2003-GT-38275. Manuscript received by IGTI, October 2002, final revision, March 2003. Associate Editor: H. R. Simmons.

¹ERC/EHP capacity is split among a number of units which depends on the MTG versus building size ratio (more units when heat recovered from MTG is low with respect to H_d and C_d), with a minimum of three units at 50% capacity.

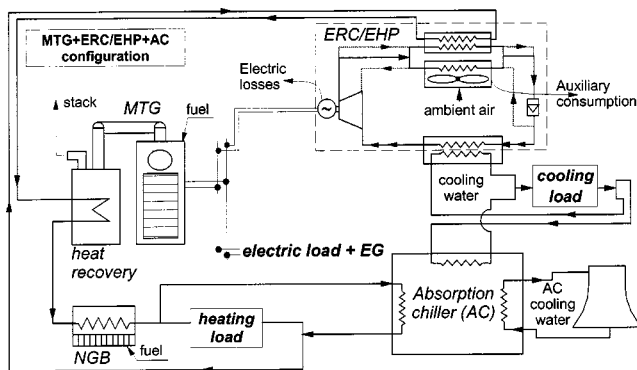


Fig. 1 Layout configuration for the trigeneration plant. The ERC/EHP unit may operate with contemporary generation of heating and cooling with a summer configuration (ERC, blue lines) or winter (EHP, red lines, dashed) configuration.

- a Turbec T100 microturbine (100 kW_{el} electric power) with 30% net electric efficiency;
- a future ceramic microturbine (300 kW_{el} net electric power) based on the prototype experiences of the Japanese research “New Sunshine Program” (models CGT301-302 as by [9–11]).

All MTGs are based on a recuperated cycle configuration. The first two features a single-stage centrifugal compressor and a single-stage, uncooled, and metallic radial turbine. The third MTG features an advanced centrifugal supersonic compressor and, differently from the other cases, a two-stage ceramic axial turbine of silicon nitride (Si₃N₄). According to the simulation code, the quoted net electrical efficiency of 40% is achieved with a turbine inlet temperature (TIT) of 1350°C, provided that the sum of recuperator leakages and eventual cooling flows (e.g., turbine disks cooling flows) is limited to 1% of inlet air flow.

For the first two microturbines, partial load performance is calibrated with available data (experimental data from [12] and from

EPRI characterization tests [13] for MTG#1 and manufacturer data [14] for MTG#2). Simulation is performed at variable speed using compressor and turbine maps given in [8] and with the assumption of a constant TIT regulation.

For the ceramic unit, given the lack of available information on part-load performance, we have decided to assume the same partial load behavior of the MTG#2. Figure 3 exhibits the calculated net electrical efficiency of the three MTGs as a function of power output at ISO ambient conditions. The simulation model provides also information about the ambient temperature effect on MTG performance, not discussed here.

Even if Fig. 3 addresses a wider operating range, emission containment² has suggested to limit MTG operation by switching off the units below 60% of the nominal power output.

Figure 3 shows also MTG first law efficiency in cogenerating mode (or total efficiency) defined as

$$\eta_I = \frac{P_{el} + P_{th,rec}}{m_f \times LHV_f} \quad (1)$$

for two different levels of stack temperature after heat recovery (70 and 120°C). The first stack temperature describes the operation mode with all the recovered heat used for heating purposes (winter operation); the second one is compatible with the absorption chiller feeding and holds in presence of a cooling demand. It should be pointed out that relevant differences occur between the two resulting efficiencies, due to the MTGs low heat input vs. exhaust gases mass flow rate, as typical for recuperated cycles with relatively low TIT. As expected, MTGs with better electrical efficiency exhibit also higher first law efficiency, owing to their lower stack losses.

Finally, the lower part of Fig. 3 shows also a parameter related to the MTG primary energy saving, frequently used in this paper, the EES (“engine energy saving”), expressed by

$$EES = \left(1 - \frac{E_{f,MTG}}{\frac{E_{el,MTG}}{\eta_{el,ref} \times \eta_{grid,ref}} + \frac{Q_{rec,MTG}}{\eta_{th,ref}}} \right) \times 100. \quad (2)$$

EES compares MTG primary energy consumption³ with the one related to the separated generation of the same amount of heating and electricity by “reference” NGB and centralized electric power plants (including distribution grid losses). Recovered heat includes heat used by the absorption chiller.

Reference values assumed for centralized plant efficiency are shown in Table 2; the terms of comparison are advanced large-scale combined cycles for electricity production and well-maintained boilers for heating generation.

Definition of EES and of reference efficiencies follows the most recent recommendations proposed by the European Union for the qualification of cogeneration plants [16]. The recommendation considers “high efficiency cogeneration” the cases of new installations with EES > 10% (> 5% for existing plants), with the exception of small-scale plants (with an installed capacity below 1 MW_{el}, as considered in this work) where a less stringent performance (EES > 0) is required.

The results of Fig. 3 indicate clearly the great influence on EES of the MTG technology: while for MTG#1 positive EES can be reached only adopting low stack temperatures (not allowed by AC), MTG#3 can reach EES as high as 15–20%, depending on stack temperature.

Evaluation of the performance of the overall trigeneration plant, supplying all the energy demands, is made with a global primary energy saving (PES) index, defined as the difference between the

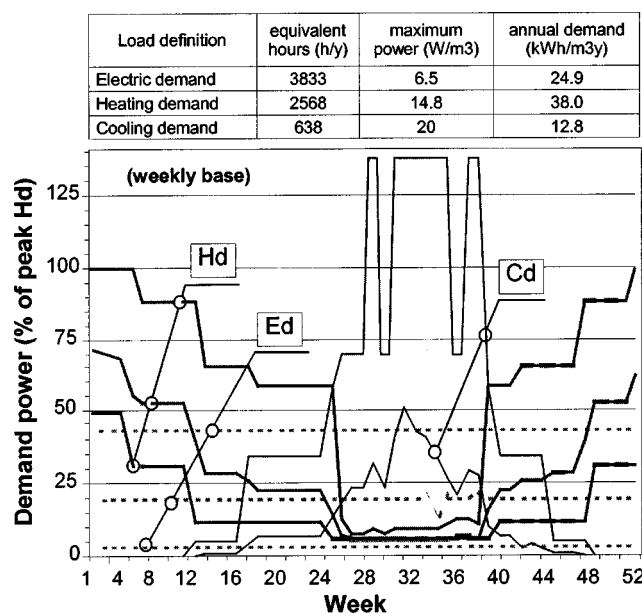


Fig. 2 Yearly distribution of the electric, heating and cooling loads. Lines represent weekly minimum, average, and maximum power, referred to the peak of heat demand (with the exception of the minimum cooling demand which is equal to zero).

²The existence of a low emission operating range (reflecting lean premix combustion [15]) has been confirmed by experimental tests carried out on MTG#1 [5,12].

³EES can be defined either with reference to instantaneous power values (as in Fig. 3) or to time integrated energy values (as in the yearly balances discussed in the economic analysis).

Table 1 Assumptions on components performance and cycle operating parameters for simulating the performance of the three selected MTGs at nominal conditions (ISO ambient conditions). For MTG#3, performance improvements are stipulated for most of the components, including turbomachines, recuperator, and electric components.

MTG Cycle Calculation Assumptions			
MTG#	1	2	3
Inlet air flow (kg/s)	0.27	0.66	0.87
Compression ratio β	3.4	4.5	7.5
Turbine inlet temperature TIT, °C	820	950	1350
Δp air filter		1000 Pa	
$\Delta p/p$ recuperator hot/cold side, %	5/3	4/2	3/1.5
$\Delta p/p$ combustor, %		3	
$\Delta p/p$ fuel injection, %		50	
Combustion efficiency		0.995	
Recuperator heat loss (% of exchanged thermal power)		2	
$\Delta T_{\min}$ recuperator, °C	95	85	80
Compressor polytropic efficiency	0.85	0.855	0.86
Turbine polytropic efficiency	0.84	0.845	0.87
Overall mechanical efficiency (referred to shaft power)		0.98	
Electric generator efficiency		0.94	0.95
Power conditioning efficiency		0.93	0.94
Fuel compressor isentropic efficiency	0.65	0.67	0.7
Fuel compressor mechanical+electrical efficiency	0.6	0.65	0.7
Supply fuel (NG) pressure (bar)		1.05	
Fuel lower heating value (kJ/kg)		49,070	

MTG Simulated Nominal Performances			
MTG#	1	2	3
Net electric efficiency, η_{el} (%)	23.3	30.0	40.0
First-law efficiency, η_1 (% , T stack 70°C)	80.4	81.6	84.5
First-law efficiency, η_1 (% , T stack 120°C)	66.8	69.6	72.8
Recuperator effectiveness	0.78	0.81	0.85
Net specific power (kJ/kg)	96	151	344
Net power output, P_{el} (kW)	26	100	300
References for model calibration	[12,13]	[14,17]	[9–11]

primary energy yearly consumption of the conventional solution (PEC_{conv}) and that of the cogenerative solution (PEC_{cog}):

$$PES = (PEC_{conv} - PEC_{cog}) / PEC_{conv} \quad (3)$$

PEC_{cog} is by turn the sum of the system (MTG+NGB) fuel consumption and the fuel consumption of the reference centralized power stations required to fulfill the electric balance:

$$PEC_{cog} = E_{f,MTG+NGB} + (E_{el,s} - E_{el,p}) / (\eta_{el,ref} \times \eta_{grid,ref}) \quad (4)$$

The conventional solution covers E_d by the electric grid and H_d , C_d based on a NGB and/or an ERC/EHP unit; primary energy consumption of the conventional solution is defined by

$$PEC_{conv} = \frac{E_{el}}{\eta_{el,ref} \times \eta_{grid,ref}} + \frac{E_{h,NGB}}{\eta_{th,ref}} \quad (5)$$

where E_{el} is the electricity consumption of the electric demand and of the ERC/EHP unit, and E_h is the heating output of NGB. The calculation model optimizes the repartition of heating output between EHP and NGB.

3.2 Other Components. ERC/EHP performance is calculated by means of a coefficient of performance (COP), based on an iterative procedure involving calculation of:

- actual refrigerant evaporating and condensing temperatures, as a function of hourly load demand, external ambient temperature and internal building temperature,
- “mechanical” cooling cycle performance, by means of the second-law efficiency, and
- number of EHP/ERC units in operation and net electrical COP, including auxiliary consumption (fans).⁴

⁴Auxiliary losses and costs of the heating/cooling distribution network are not accounted for, since they are assumed equal for the cogenerative and the conventional plant.

Input data include:

- number of EHP/ERC units, working fluid, type of compressor (piston, scroll, screw types corresponding to the increasing plant sizes related here to MTG#1, 2, 3),
- nominal unit operating conditions (cooling or heating power, ambient air temperature, water inlet/outlet temperatures), and
- nominal load and water temperatures of the heating/cooling distribution system.

Calculation involves the simulation of heat exchangers operating temperature differences, as discussed in detail in [5].

An efficiency versus load curve for the natural gas boiler has been assumed, accounting also for losses related to on/off cycles, as proposed in [5].

AC efficiency variation versus load is computed according manufacturers’ data; the average absorption chiller COP is 0.7⁵.

4 Calculation Model

Calculations are performed based on a specifically developed simulation tool (TRIGEN), already described in a previous paper, [5]. The code requires the assignment of a number of input data for the definition of economic assumptions, components and plant performance, load profiles etc., providing detailed energy, economic, and emissions yearly balances. The model optimizes the operating conditions of the trigeneration plant for each of the 8760 hours of the year. The assumption is made that the plant control system is smart enough to optimize the plant-operating mode to achieve the most cost-effective situation in each time step (one hour). In all cases treated in this paper, the final choice is made on an economic basis, but other optimizing strategies, say best energy

⁵Single-stage AC are considered, as required by the relatively low temperature of the available heat source.

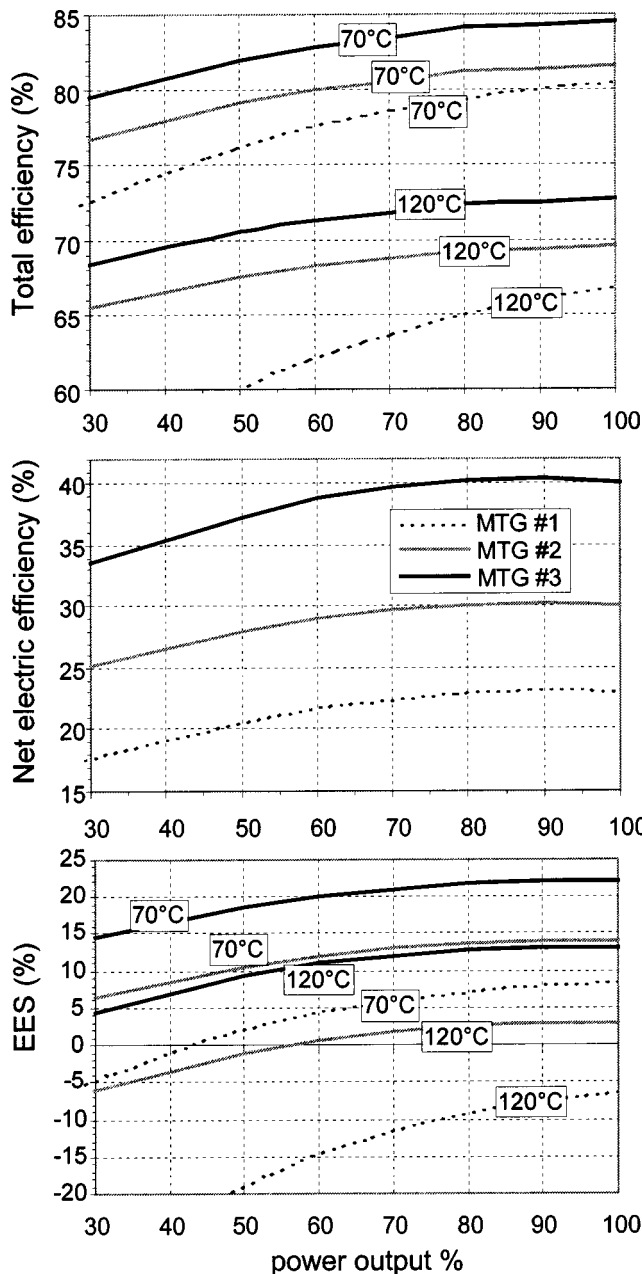


Fig. 3 MTG part-load performance. The actual operating range considered in the simulation is limited to 60–100% due to specific emissions limitation.

saving, or minimum pollutant emissions, or any combination of economic, energetic, and environmental achievement could be easily specified.

The model has the possibility of splitting plant generation capacity onto several MTG units. A multiple MTG set offers higher part-load efficiency due to the possibility of switching off some units at reduced load and operate the remaining MTGs at higher load. As already shown in [5], this advantage becomes relevant only when a single MTG would operate at part load for a significant amount of time.

Table 2 Reference values for the definition of energy savings

Electric efficiency $\eta_{el,ref}$	0.55
Heating thermal efficiency $\eta_{th,ref}$	0.90
Grid efficiency $\eta_{grid,ref}$	0.925

cant amount of time. This possibility is not considered in this work, both for simplicity and for the following reasons:

1. optimized MTG sizing and management lead to relatively infrequent partial load operation and
2. part-load performance of the MTGs are quite good, particularly in the operating range (60–100% of nominal load) considered.

The code selects the option, which features the minimum instant cost among different possibilities. The instant cost is defined as

$$C = C_{el,p} + C_{MTG,f,p} + C_{NGB,f,p} + C_{O\&M} - V_{el,s} \quad (6)$$

where the first four terms represent the hourly cost related to: (i) purchase of electricity, (ii) purchase of natural gas for the MTG, and (iii) for the NGB⁶ and (iv) the cost for MTG operation and maintenance (assumed proportional to the electricity generated⁷). The last term represents the value of the electricity sold to the grid⁸.

The two most common operating options of the system are:

(A) Load-following

The system follows all three loads (electric, heating, cooling), by operating at partial load the MTG and trying to approach zero electricity exchange with the electric grid. MTG works at the power output necessary for covering the electric load, including ERC/EHP; heating is generated by MTG and, if necessary, by the EHP; cooling is covered primarily by the AC, followed by the ERC. NGB (for heating generation or for the AC heat consumption) may be used to balance the heating demand when contemporary heating and cooling demand is present.

(B) Full-power output

MTG operates at full power output (base load); cogenerated heat is used to satisfy heating demand or to feed AC, wasting the excess. Electricity is used to cover the electric demand (Ed) and the eventual surplus is used by the EHP/ERC for the fraction of the heating and/or cooling demand not covered by cogeneration and AC. Remaining excess electricity is sold to the grid.

The model also considers the possibility of operating with the additional plant configurations obtained by alternatively switching off the AC, the ERC/EHP, and the MTG components.

For instance, if the ERC/EHP is switched off, the plant may operate as a “heat following” (MTG+NGB), or “cooling following” (MTG+AC+NGB), in both cases with electricity balance by EG. If the MTG is switched off, the plant may operate only with the ERC/EHP unit, purchasing electricity from EG, and/or generating heat by operating NGB. The calculation is carried out for all the applicable operating options, aiming to investigate the most extensive as possible range of alternatives. More details about the simulation methodology are given in Ref. [5].

4.1 Tariffs and Investment Cost Assumptions. Tariffs for natural gas and electricity purchase are assumed as presented in Table 3. Natural gas costs is considered in the range 2.5–5 €/kWh, with a reference value of 3.5 €/kWh.

Electricity tariff reflects, with some simplification, the Italian situation.⁹ It can be seen that the electricity cost can be as high as four or five times the NG cost in peak hours, but the tariff is much lower during low charge hours and for the electricity sold to the grid. The “reverse metering factor” RMF, representing the electricity sale/purchase cost ratio, for the purpose of this simulation, ranges from 0.5 to 1 with a basic value of 0.75.

⁶MTG and NGB fuels may be different or have different taxation rules.

⁷According to manufacturers, a full-maintenance contract based on generated electricity is the most likely choice for MTGs users.

⁸This term is present only if excess electricity is generated by MTG.

⁹Fixed costs of electricity are unimportant since full coverage of electric demand is required to the grid.

Table 3 NG and electricity tariff assumptions

Natural Gas Purchase		Basic Case	Range Investigated
Specific cost (€/kWh _{LHV})	gas to MTG	3.5	2.5 ÷ 5
	gas to NGB	5	...
Electricity Purchase and Sale			
Reverse metering factor RMF		0.75	0.5 ÷ 1.0
Specific cost (€/kWh)	Peak hours	15	...
	Low charge hours ⁽¹⁾	7.5	...

⁽¹⁾Night hours (21 p.m.–7 a.m.) and weekend hours.

Table 4 Cost and investment assumptions

Capital Costs		
	MTG#1, 2	MTG#3
Gas turbine (€/kW _{el,nom})	600	600÷900
EHP/ERC (€/kW _c)	200	
Absorption chiller (€/kW _c)	300	
Natural gas boiler (€/kW _h)	15	
Size effect on components cost	(100/P _{el,nom,MTG} [kW]) ^{0.2}	
Balance of Plant cost (including project, contingencies, insurance)	30% of total component cost	
	× (100/P _{el,nom,MTG} [kW]) ^{0.4}	
Other Assumptions		
MTG O&M cost (€/kWh)	0.5	
MTG maintenance outage	2 days/year	
Operating life and interest	10 years, 7%	
Inflation rate	2.7%	
Taxes on gross profit	0 (basic case)÷40%	

The most relevant assumptions related to the cost of components and plant and to the parameters involved in the financial analysis are indicated in Table 4. For all components, we stipulate the same simplified rule of cost variation versus size. For MTG#2, a “target”—lower than actual—specific cost of 600 €/kW [15], is assumed. For the MTG#1, specific cost increases, due to the stipulated size effect, not accounting for differences in component quality and performance.

For the MTG#3, we have investigated specific costs in the range between 600 and 900 €/kW, due to uncertainties on special components (e.g., ceramic blades) costs and presence of more costly, two-stage axial turbine.

Costs of the other components are derived by a market survey.

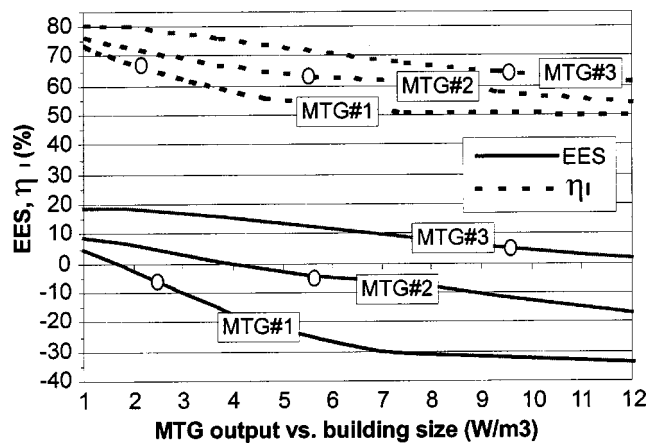
5 Effect of MTG Efficiency

The different performance of the three considered MTGs greatly influences the system energy savings and economics. All the following results refer to energy and economic balances obtained by yearly simulations.

5.1 Optimization of MTG Versus Building Size. The first issue addressed here is the influence of the size of the building where a MTG is installed. Let's first consider the energy consequences of this choice (lower part of Fig. 4): when the MTG is installed in a very large building (left side of the figure¹⁰), the EES parameter approaches the value obtained at nominal load with full heat recovery.¹¹ The building electric, heating and cooling demand is so large, that the MTG operates most of the time at full load, with full heat recovery. When the building size decreases, also EES decreases, for two main reasons: (i) Ed decreases, so that

¹⁰Of course, the same situation would occur when a smaller turbine (with the same efficiency) would be applied to a fixed building. Large turbines are on the right side of Figs. 4–5.

¹¹As already pointed out, EES strongly depends on stack temperature, that varies between summer and winter operation.

**Fig. 4 Effects of MTG efficiency on the optimization of MTG versus building size, by the point of view of first-law efficiency and EES**

sometimes the engine operates at part load and (ii) Hd (and Cd) decreases, so that sometimes part of the cogenerated heat is wasted. This situation is presented in Fig. 4, where the large building area is on the graphics left side. Progressive wasting of cogenerated heat is well represented by the decrease of the yearly average first-law system total efficiency (dashed lines in Fig. 4).

Depending on the MTG efficiency, the plant vs. building size corresponding to zero energy savings varies: for MTG#2, when the MTG output exceeds 65% of building electric peak demand, EES becomes negative, i.e. cogeneration is no more an energy saving solution. The same situation is close to 250% of building electric peak demand for MTG#3, thanks to its higher efficiency.

As it could be anticipated by the curves in Fig. 3, due to the severe terms of comparison adopted in Table 2, MTG#1 reaches a positive EES only when the MTG output is below 20% of the building electric peak demand.

By the point of view of economics (Fig. 5), the situation is somewhat different: as a general result, the solutions that achieve the best economical conditions yield EES quite lower than optimum. All three MTGs allow significant savings on the total energy bill (NG+electricity annual consumption), ranging from about 15% for MTG#1 to fractions as high as 50% for MTG#3. The economic comparison is carried out with respect to a “conventional” case where Ed is covered by the grid, a centralized

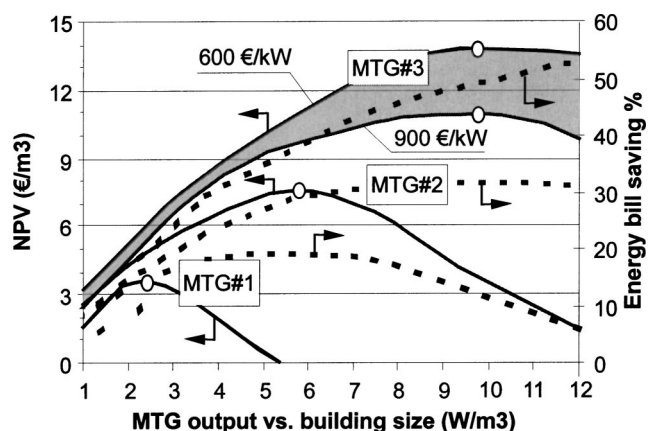
**Fig. 5 Effects of MTG efficiency on the optimization of MTG versus building size, by the point of view of NPV and economic savings on the energy bill**

Table 5 Effects of MTG efficiency on economic optimization. All parameters are calculated at NPV optimum.

MTG	#1	#2	#3
MTG specific cost (Table 4, €/kW _{el})	600	600	600
NPV _{opt} (€/m ³)	3.42	7.52	13.59
EES (%)	-5.83	-4.40	5.80
MTG output versus building size (W/m ³)	2.5	5.6	9.3
MTG nominal power/Ed peak power ratio	0.44	0.84	1.46
IRR (%)	21.0	31.2	42.0
Trigeneration plant investment cost (k€)	136	233	376
MTG investment cost (%)	14.9	26.8	38.5
Conventional plant investment cost (k€)	104	150	200
Differential investment cost (k€)	32	83	176
Annual energy bill saving (k€/year)	8	29	78
Building size (m ³)	9,090	18,690	32,160
ERC/EHP compressors trigeneration/ conventional plant (number and type)	3+1/4+1 piston	3+1/4+ 1 scroll	2+1/3+1 screw
Average COP _c	2.9	3.3	3.6
Average COP _h	3.5	4.0	4.3
MTG fuel input (MWh)	395.5	1153.1	3234.3
MTG first law efficiency	0.675	0.636	0.647
E _{el} generated (MWh)	92.8	346.1	1099.3
E _{el,s} - E _{el,p} (MWh)	-217.5	-268.7	66.6
PECcog (Eq. (4), MWh)	824.7	1681.4	2602.5
PES (Eq. (3), %)	-18.3	-22.7	-14.3
PES2 (%) ⁽¹⁾	8.5	8.3	17.0

⁽¹⁾PES2 is calculated by Eqs. (3)–(5) by assuming that in the reference case all the heating demand is generated by NGB and cooling demand by ERC working with COP_c given in the table.

ERC/EHP plant (same units of the MTG case) generates Cd and NGB or EHP generates either Hd according to economics.¹²

It is generally convenient to increase the MTG versus building size ratio, so that the MTG covers a larger portion of the energy demands and yields a larger saving on the building energy bill (dashed lines of Fig. 5). Of course larger MTG output yields higher capital costs and limits the convenience of installing too large turbines (or installing the turbine into too small buildings).

An optimized investment NPV (white dots in Figs. 4 and 5) is reached for a MTG nominal output of 0.44–1.46 peak electric demand, depending on the engine efficiency.¹³ The results obtained at the matching MTG output/building size ratio that yields the optimum NPV are summarized in Table 5.

MTG#1 reaches best specific NPV values close to 3.4 €/m³; the situation is much more favorable for the other MTGs, for the combined effect of lower plant specific cost (due to scale effect) and larger energy savings (due to better MTG performance).

MTG#3 yields best NPV all over the investigated range of specific cost, with a maximum between 10.9 and 13.6 €/m³; however, the situation is different for the internal rate of return, which ranks below the case of MTG#2 (26.3 versus 31.2%) when the highest specific cost is stipulated. This means that a 50% increase of specific cost cannot be justified by the better thermodynamic performance.

As far as the plant primary energy saving is concerned, all MTGs suffer the effect of an economic optimization at lower than maximum first-law efficiency (Fig. 4) and the comparison with the highly energy-saving heat-pump system used in the conventional solution, which features the same COP indicated in Table 5. PES resulting from economic optimization is never positive, indicating that cogeneration from MTG plants is hardly competitive by the point of view of energy saving with a centralized heat pump system with average COP close to 4, when electricity is generated in modern, high-efficiency power stations (Table 2).

Better PES could be reached with smaller (but not economically optimal) MTG versus building size ratios, as well as with different optimization approaches aiming to address energy saving

rather than economics, as discussed in the last chapter. Besides, the situation would be different if the conventional solution would be based on heat generation by NGB (a solution far more diffused in common practice); in this case, as shown by the last values of Table 5 (PES2), MTG system yields relevant energy savings.¹⁴

Further inspection on the effects of MTG efficiency can be drawn by Fig. 6, representing the system EES and energy bill saving for each week of the year. It is possible to see how MTG#1 yields energy savings only with winter heat recovery; while during summer, lower electric efficiency and high stack temperature heat recovery for the absorption chiller prevent the possibility of reaching positive EES and yield also limited economic savings.

The higher efficiency of #3 yields significant energy savings also during summer, while half seasons feature the most difficult conditions, due to limited heat recovery.

The situation is similar for the energy bill savings, with MTG#3 always in the range of a 40–50% bill saving. The negative peak during winter is situated in the week where MTG maintenance is scheduled (see Table 4). The first week features also a majority of nonworking days and low charge hours, where the MTG is switched off, yielding low savings.

6 Tariff and Investment Cost Variation

The above-presented economic outlook of the three MTGs changes when different assumptions on tariffs and taxation are assumed.

6.1 Natural Gas Cost Influence. As shown in Fig. 7, if the natural gas cost decreases to 2.5 €/kWh (a value which could represent a tariff incentive), IRR increases up to 40–60%. The system operates for more hours per year, exporting electricity to the grid and switching on the MTG during a large fraction of low charge hours.

Oppositely, if the natural gas cost increases up to 5 €/kWh, only MTG#3 keeps a reasonable investment internal return rate. Yearly operation is limited to 1400 h in winter for MTG#1, while MTG#2 and #3 keep convenient operation during peak hours all over the year for the investigated range of natural gas prices.

¹²With the operating conditions and the tariffs assumed, it is generally more convenient to generate heat by EHP rather than NGB.

¹³The 23% efficient MTG reaches best NPV at building sizes larger than what has been found in a previous work, [5], where lower fuel and investment costs were assumed.

¹⁴Values are slightly better for MTG#1 than for MTG#2 due to the different effect of the economic optimization (see total efficiency curves in Fig. 4).

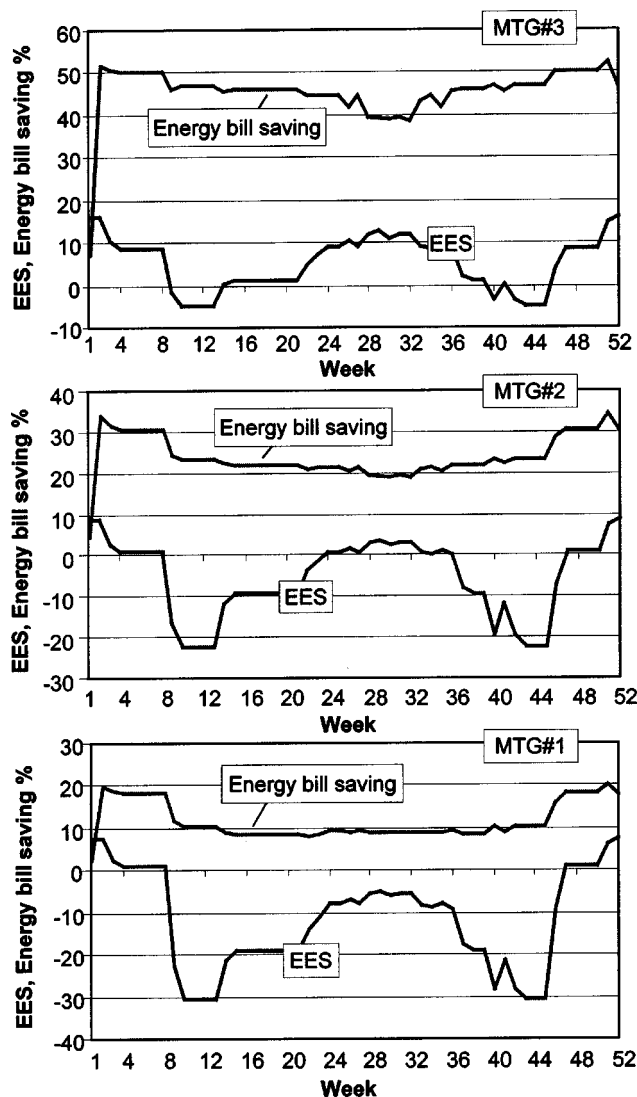


Fig. 6 Weekly analysis of plant EES and economic savings on the energy bill

6.2 Reverse Metering Factor Influence. Figure 8 deals with the effect of the “reverse metering factor,” defined as the ratio between purchase cost and sale price of electricity. Allowing the sale of electricity with the same price of purchase ($\text{RMF}=1$) yields of course more favorable IRR and progressively increases the quantity of electricity sold to the grid. The effect is particularly significant for the more efficient MTGs, while the situation of MTG#1 is less influenced and the quantity of electricity sold to the grid remains below 5% of the generated electricity.

6.3 Taxes Influence. Figure 9 shows the effect of taxation on investment IRR. Depending on country-specific fiscal regulations, taxation influences the investment IRR (Fig. 9). The situation with zero taxes could represent the case of special incentives or the case of a private customer, simply saving costs on the energy bill.

7 Operation at Optimized EES

The discussion carried out above is based on the economic optimization of the plant operation, as by the calculation model with instant cost minimization discussed in Chapter 3.

The situation changes if a constraint is imposed on the system

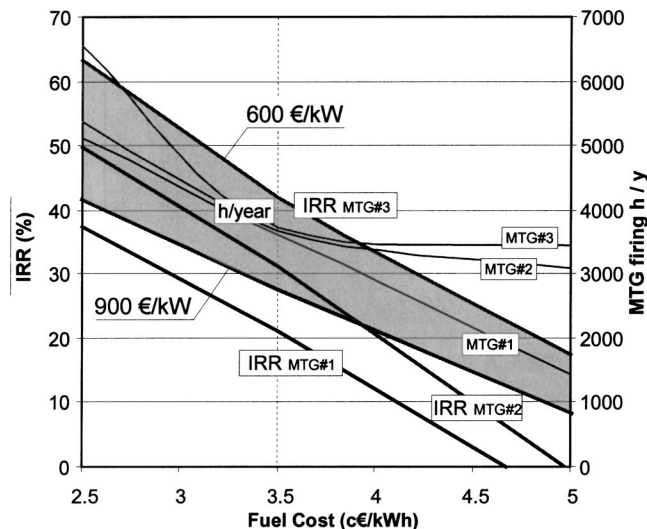


Fig. 7 Effect of fuel cost on system IRR and MTG firing hours per year

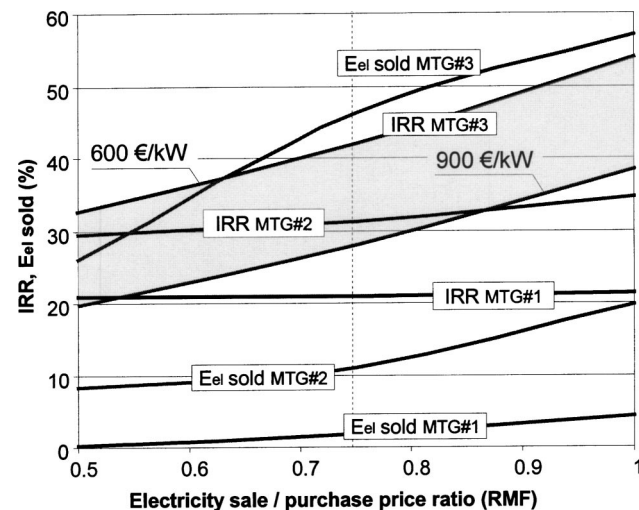


Fig. 8 Influence of electricity purchase/sale cost ratio (RMF) on system IRR and on the fraction of electricity sold/electricity generated

yearly EES, as it could happen for instance when legislation requires to reach a qualification to gain access to some kind of incentive.

Aiming to investigate this issue, it is considered here to manage the plant with a strategy choosing for each hour the mode of operation that yields the best economics, only among the modes characterized by EES above a fixed limit. When applied to the entire year, this strategy yields a higher EES than a simple economic optimization, of course losing something on the economic side.¹⁵

Figure 10 shows what happens when the MTG plant is asked to reach progressively higher EES levels: IRR decreases, while system total efficiency increases. Some of the effects are

¹⁵It is not guaranteed that this approach yields the best possible yearly economic balance for a given yearly EES, as the optimization is calculated hour by hour. However, this strategy could be easily implemented in a real control system, provided that a preliminary simulation as the one carried out here has given the more effective limit to be applied to the hourly EES.

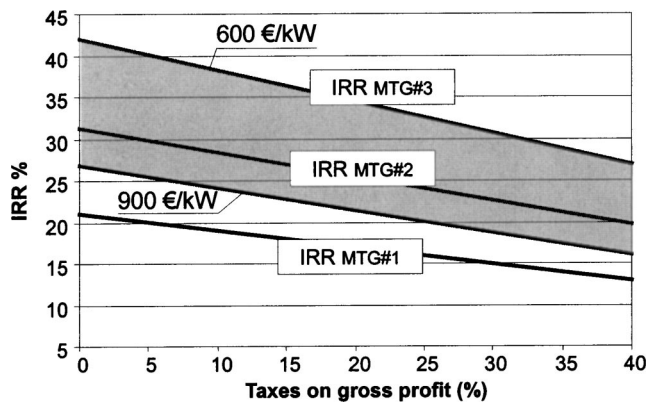


Fig. 9 Effect of taxation on investment IRR

1. progressive substitution of hours of economically convenient operation and significant waste of heat (e.g., during peak hours) with hours of heat-following operation (e.g., instead of electricity-following operation) or MTG switched off;
2. during low charge periods, addition of hours of cogeneration instead of hours of MTG switched off and heating generation by EHP.

Operation at higher EES yield also better system PES than found by economic optimization; however, best PES for MTG#3 at maximum considered EES (15% in Fig. 10, with yearly average first law efficiency close to 75%) remains negative when reference is made to an EHP/ERC conventional system (-7.5% versus the original -14.3%).

8 Conclusions

The economic viability of tri-generation systems built around gas fired MTGs is strongly related to many factors, including the cost of natural gas and the electric tariff scenario. However, the calculations carried out in this paper demonstrate that the most

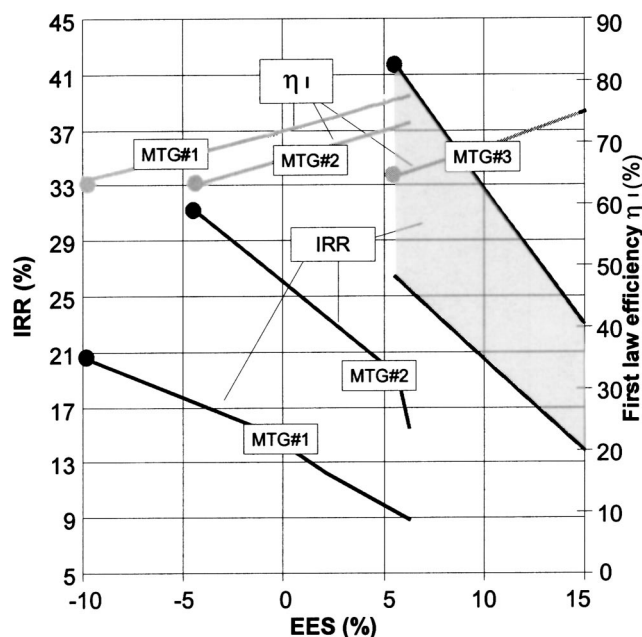


Fig. 10 Effects of operation with target EES as constraint on IRR and first-law efficiency. Dots indicate the configurations resulting from economic optimization.

significant factor for the success of this application is the micro-turbine technology. If advances on turbine inlet temperature, recuperator effectiveness and compressor performance would permit net electrical efficiencies in the range of 35–40%, the application becomes very promising in any reasonable tariff scenario. Primary energy saving and environmental benefits obtainable by the combination of high efficiency MTGs, heat pumps and absorption machines would become very significant.

Acknowledgments

This work has been partly funded by MIUR (Italian Ministry for education, university and scientific research) COFIN program, fiscal year 2001.

Nomenclature

- AC = absorption chiller
 C = instant cost ($\text{€}/\text{kWh}$), (Eq. (3))
 C_d = specific cooling demand (% or W/m^3)
COG = cogenerator (MTG+heat recovery boiler)
COP = coefficient of performance
 E = energy (kWh)
 E_d = specific electricity demand (% or W/m^3)
EG = electric grid
EHP = electric heat pump
ERC = electric refrigerating compressor
EES = engine primary energy saving (%), Eq. (2)
 H_d = specific heating demand (% or W/m^3)
IRR = investment internal return rate (%)
LHV = fuel lower heating value (kJ/kg_f)
 m = mass flow rate (kg/s)
MTG = micro turbine generator
NG = natural gas
NGB = natural gas boiler
NPV = specific net present value ($\text{€}/\text{m}^3$)
 P_{el} = net electric power (alternating current, kW)
 P_{th} = thermal power (kW)
 Q_{rec} = MTG cogenerated heat (kWh)
RMF = reverse metering factor, electricity sale/purchase cost ratio
 T = temperature ($^{\circ}\text{C}$ or K)
TIT = turbine inlet temperature ($^{\circ}\text{C}$)
 β = pressure ratio
 η_1 = first law efficiency, Eq. (1)
 η_{el} = net electric efficiency $= P_{el} / (m_f \times \text{LHV})$
 η_{grid} = distribution grid efficiency
 η_{th} = thermal efficiency $= P_{th} / (m_f \times \text{LHV})$

Subscripts

- c = cooling
cog = cogeneration plant
conv = conventional solution (NGB+EG+ERC)
 el = electric
 f = fuel (natural gas)
 h = heating
nom = nominal
 p = purchased
rec = recovered
ref = reference (see Table 2)
 s = sold

References

- [1] Green, S., 2001, "Distributed Generation—A New Wave," *Power Engineering International*, PennWell, Mar.
- [2] Gliddon-Bush, C., 2001, "Micro Size, Maximum Efficiency," *Power Engineering International*, PennWell, July.
- [3] Malrup, L., 2002, "Gaining Experience With Microturbine CHP," *Cogeneration and On-Site Power Production*, James and James, eds., 3, May, pp. 60–63.
- [4] Anon., 2002, "Landmark Installation for Micro-CHP," *Modern Power Systems*, Wilmington Publishing, May.

- [5] Campanari, S., Boncompagni, L., and Macchi, E., 2002, "Microturbines and Trigeneration: Optimization Strategies and Multiple Engine Configuration Effects," ASME Paper No. 2002-GT-30417, to be published on ASME J. Eng. Gas Turbines Power.
- [6] Roncato, J. P., and Macchi, E., 2000, "Report of Study Group 7.2: Comparison of Medium or Large Scale CHP and Combined Cycles, in Various Countries," Woc7 Report, *Proc. of World Gas Conference 2000*, Nice, June, pp. 55–82.
- [7] Campanari, S., and Macchi, E., 2002, "Future Potentials of MTGs: Hybrid Cycles and Tri-Generation," in *Micro Turbine Generators*, M. J. Moore, ed., Professional Engineering Publishing Ltd., IMechE, London, Chap. 4, pp. 43–66.
- [8] Campanari, S., 2000, "Full Load and Part-Load Performance Prediction for Integrated SOFC and Microturbine Systems," ASME J. Eng. Gas Turbines Power, **122**, pp. 239–246.
- [9] Kobayashi, H. et al., 1998, "Current Status of Ceramic Gas Turbine (GT302)," ASME Paper 98-GT-501.
- [10] Ohashi, I., and Arakawa, S., 1995, "Development of 300 kW Class Ceramic Gas Turbine (CGT 303)," ASME J. Eng. Gas Turbines Power, **117**, p. 777.
- [11] Taoka, T. et al., 1998, "Current Status of the CGT301, Ceramic Gas Turbine," ASME Paper 98-GT-288.
- [12] Campanari, S., and Boncompagni, L., 2001, "Experimental Acquisition of Emission Data From a Commercial Microturbine," Internal Technical Note, Department of Energetics, Politecnico di Milano, Sept.
- [13] Anon., 1999, "Performance and Electrical Characterization Tests on a Microturbine Commercial Prototype," EPRI-TR 114270, Dec.
- [14] Anon., 2002, "T100 Microturbine CHP System Technical Description," D10293 Ver. 5.0, Turbec AB, Sweden.
- [15] De Biasi, V., 2001, "DOE Developing Technology Base for Advanced Microturbine Designs," *Gas Turbine World*, Pequot Fairfield, CT, **31**(4).
- [16] Anon., 2002, "Methodology for Determining the Efficiency of Cogeneration Production," Annex III of "Proposal for a Directive of the European Parliament and of the Council on the promotion of cogeneration"—2002/0185 (COD), Brussels, July.
- [17] Carnö, J., Cavani, A., and Liinanki, L., 1998, "Micro Gas Turbine for Combined Heat and Power in Distributed Generation," ASME Paper 98-GT-309.

High-Temperature Fatigue Properties of Single Crystal Superalloys in Air and Hydrogen

N. K. Arakere

Associate Professor,
Mechanical Engineering Department,
University of Florida,
Gainesville, FL 32611-6300
e-mail: nagaraj@ufl.edu

Hot section components in high-performance aircraft and rocket engines are increasingly being made of single crystal nickel superalloys such as PWA1480, PWA1484, CMSX-4, and Rene N-4 as these materials provide superior creep, stress rupture, melt resistance, and thermomechanical fatigue capabilities over their polycrystalline counterparts. Fatigue failures in PWA1480 single crystal nickel-base superalloy turbine blades used in the space shuttle main engine fuel turbopump are discussed. During testing many turbine blades experienced stage II noncrystallographic fatigue cracks with multiple origins at the core leading edge radius and extending down the airfoil span along the core surface. The longer cracks transitioned from stage II fatigue to crystallographic stage I fatigue propagation, on octahedral planes. An investigation of crack depths on the population of blades as a function of secondary crystallographic orientation (β) revealed that for $\beta=45^\circ \pm 15^\circ$ deg tip cracks arrested after some growth or did not initiate at all. Finite element analysis of stress response at the blade tip, as a function of primary and secondary crystal orientation, revealed that there are preferential β orientations for which crack growth is minimized at the blade tip. To assess blade fatigue life and durability extensive testing of uniaxial single crystal specimens with different orientations has been tested over a wide temperature range in air and hydrogen. A detailed analysis of the experimentally determined low cycle fatigue properties for PWA1480 and SC 7-14-6 single crystal materials as a function of specimen crystallographic orientation is presented at high temperature (75°F–1800°F) in high-pressure hydrogen and air. Fatigue failure parameters are investigated for low cycle fatigue data of single crystal material based on the shear stress amplitudes on the 24 octahedral and 6 cube slip systems for FCC single crystals. The max shear stress amplitude [$\Delta\tau_{max}$] on the slip planes reduces the scatter in the low cycle fatigue data and is found to be a good fatigue damage parameter, especially at elevated temperatures. The parameter $\Delta\tau_{max}$ did not characterize the room temperature low cycle fatigue data in high-pressure hydrogen well because of the noncrystallographic eutectic failure mechanism activated by hydrogen at room temperature. Fatigue life equations are developed for various temperature ranges and environmental conditions based on power-law curve fits of the failure parameter with low cycle fatigue test data. These curve fits can be used for assessing blade fatigue life. [DOI: 10.1115/1.1501075]

1 Introduction

High cycle fatigue is currently the primary cause of component failures in gas turbine aircraft engines ([1]). Currently the most widely used single crystal nickel-base turbine blade superalloys are PWA 1480, PWA 1484, RENE' N-5, and CMSX-4. PWA 1480 will be used in the NASA SSME alternate turbopump. These alloys play an important role in commercial, military, and space propulsion systems ([2]). Fatigue crack initiation in single crystals can be crystallographic or noncrystallographic depending on the interaction between the point-source defect species (carbides, micropores, and eutectics) with the effects of environment, temperature, and stress ([2]). Single crystal materials have highly orthotropic properties making the position of the crystal lattice relative to the part geometry a significant factor in the overall analysis. Study of failure modes of single crystal turbine blades therefore has to account for material anisotropy and variations in crystal orientation. Estimation of blade fatigue life, therefore, represents an important aspect of durability assessment.

Turbine blades and vanes, used in aircraft and rocket engines,

are probably the most demanding structural applications for high-temperature materials due to the combination of high operating temperature, corrosive environment, high monotonic and cyclic stresses, long expected component lifetimes, and the enormous consequence of structural failure. Directional solidification is used to produce a single crystal ([3]) with the $\langle 001 \rangle$ low modulus orientation parallel to the growth direction. The secondary direction normal to the growth direction, which is typically referenced to a line parallel to the blade attachment, is random if a grain selector is used to form the single crystal. Both the primary and secondary crystal orientations, though, can be selected if seeds are used to generate the single crystal. However, in most cases, grain selectors are used to produce the desired $\langle 001 \rangle$ growth direction. In this case, the secondary orientations of the single crystal components are determined but not controlled. Initially, control of the secondary orientation was not considered necessary ([4]). However, recent reviews of space shuttle main engine (SSME) turbine blade lifetime data have indicated that secondary orientation has a significant impact on high cycle fatigue resistance ([5–7]).

This paper presents fatigue failures in the space shuttle main engine (SSME) alternate advanced high-pressure fuel turbo pump (AHPFTP/AT) single crystal turbine blades. Analysis of fatigue stresses in the turbine airfoil region based on finite element modeling of single crystal turbine blades, is presented. Stress intensity factor variation at the crack tip as a function of secondary crystallographic orientation is examined. Analysis of fatigue stresses

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, New Orleans, LA, June 4–7, 2001; Paper 01-GT-0585. Manuscript received by IGTI, December 2000; final revision, March 2001. Associate Editor: R. Natole.

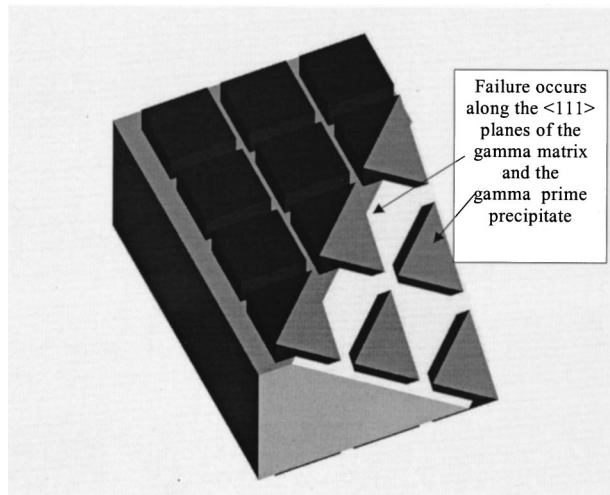


Fig. 1 Stage I type global octahedral failure mode ([2,8,9])

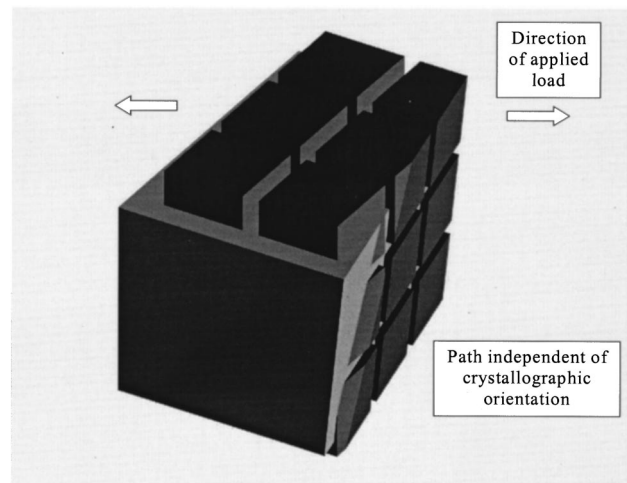


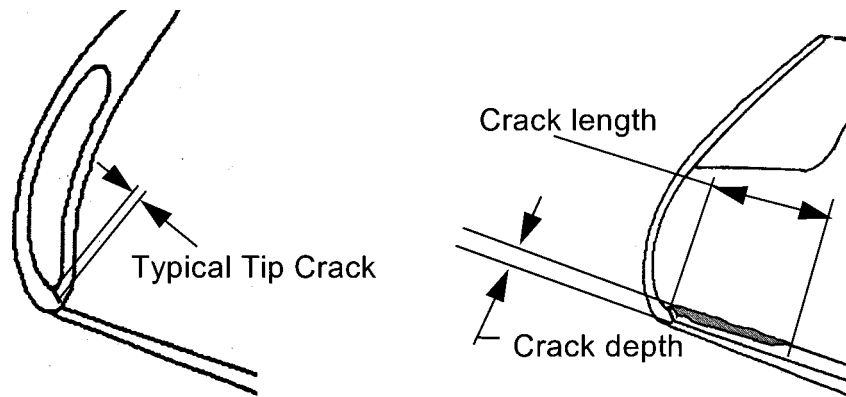
Fig. 2 Noncrystallographic stage II type of failure ([2,8,9])

in the blade tip and the blade-disk attachment regions show that control of secondary and primary crystallographic orientation has the potential to optimize blade design by increasing its resistance to fatigue crack growth. Experimentally determined LCF data for PWA1480/1493 and SC 7-14-6, two single-crystal nickel-base turbine blade superalloys, are presented for a wide range of temperatures, environmental conditions, and specimen orientations. The maximum shear stress amplitude $\Delta\tau_{\max}$ on the 30 slip systems was found to be an effective fatigue failure parameter, based on the curve fit between $\Delta\tau_{\max}$ and cycles to failure. The parameter $[\Delta\tau_{\max}]$ did not characterize the room temperature low cycle fatigue data in high-pressure hydrogen well because of the noncrystallographic eutectic failure mechanism activated by hydrogen at room temperature. Low cycle fatigue data in high-pressure hydrogen at 1400°F and 1600°F was characterized well by the $\Delta\tau_{\max}$ failure parameter.

2 Fatigue Crack Initiation and Crack Growth in Nickel-Base Superalloys

Crack initiation in Ni-base superalloys fall into two broad classifications, crystallographic and noncrystallographic. The interaction between the effects of environment, temperature, and stress intensity determines which point-source defect species initiates a crystallographic or noncrystallographic fatigue crack ([2]). At low temperature (<427 C) and stress conditions crystallographic initiation appears to be the most prevalent mode. At moderately high temperature (above 593C) initiation at TaC carbides predominates over occasional (111) eutectic origins.

The fatigue crack growth behavior of single crystal nickel superalloys is governed by a complex interaction between the operative deformation mechanism, stress intensity, and environmental conditions. As a result of the two-phase microstructure present in single crystal nickel alloys a complex set of fracture mode



XXX FPI results for 1st stage blade population.

Unit 6-5D	22 starts	12972 sec	23 of 50 blades with tip crack
Unit 3-4B	21 starts	12617 sec	34 of 50 blades with tip crack
Unit 7-3B	19 starts	10216 sec	31 of 50 blades with tip crack
Unit 7-1	14 starts	7201 sec	0 of 26 blades with tip crack
Unit 9-1	11 starts	6309 sec	1 of 50 blades with tip crack
Unit 5-4	10 starts	6022 sec	9 of 50 blades with tip crack
Unit 8-1A	10 starts	5782 sec	19 of 50 blades with tip crack
Unit 3-1	12 starts	3251 sec	0 of 39 blades with tip crack *
Unit 5-3	21 starts	2296 sec	2 of 49 blades with tip crack *

* Full blade set not available

Fig. 3 Blade LE tip cracks discovered during inspection in AHPFTP units ([5,11])

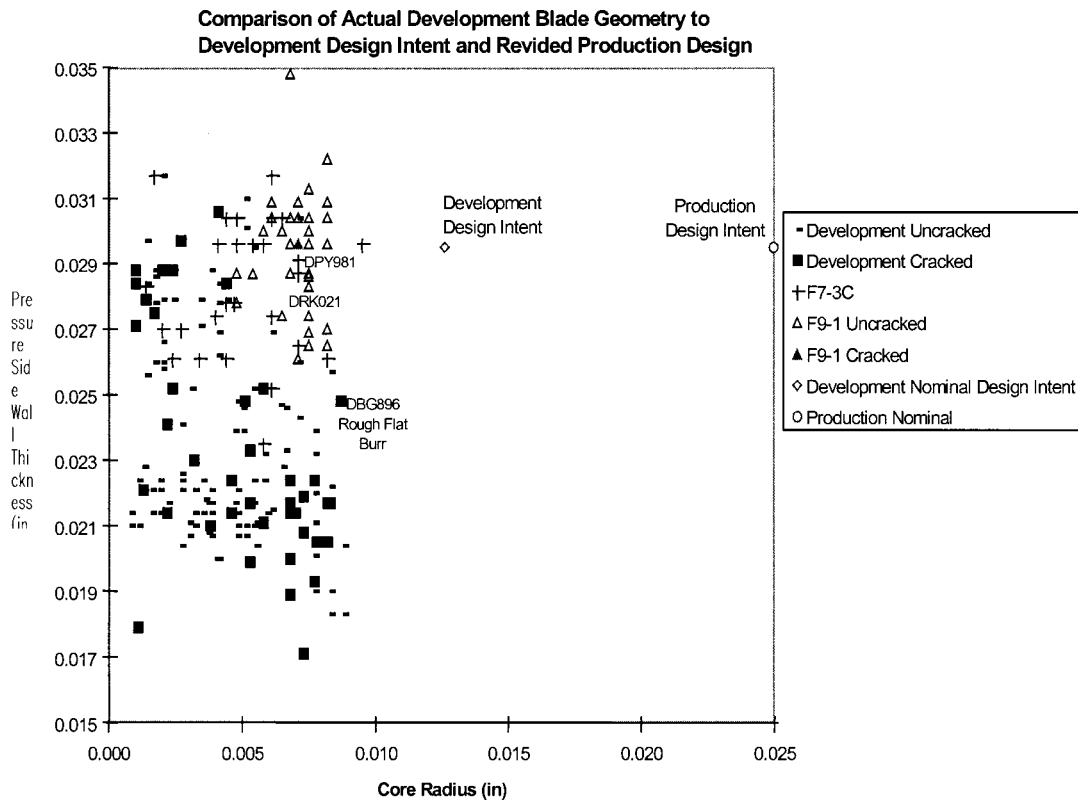


Fig. 4 Population of cracked blades relative to geometry ([5,11])

exists based on the dislocation motion in the matrix (γ) and precipitate phase (γ'). Fractography performed on failed fatigue test specimens has revealed the presence of three basic fracture modes for single crystal super alloys. Two are stage I type failure modes with crystallographic crack propagation while the third is a stage II noncrystallographic failure mode ([8,9]). Each is characterized by the manner in which the crack propagates through the crystal

structure. Temperature and ΔK have been shown to be the primary drivers in determining which failure mode are observed ([9]).

Crystallographic fracture by cleavage in the $\langle 001 \rangle$ plane is one of the two stage I type modes of fatigue crack growth ([8,10]). This case is represented by the crack propagating along the $\langle 001 \rangle$ plane normal to the applied load along one of several potential

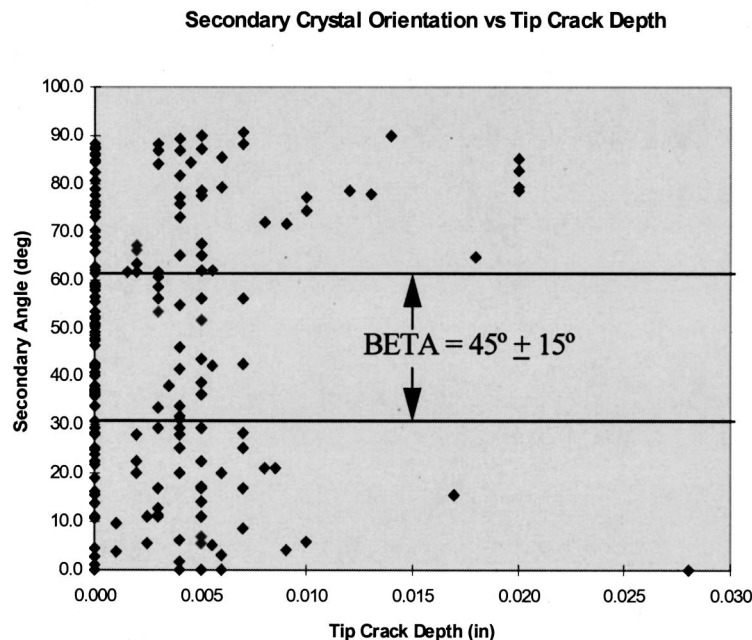


Fig. 5 Tip crack depth versus secondary orientation (β) for AHPFTP first blades ([5,11])

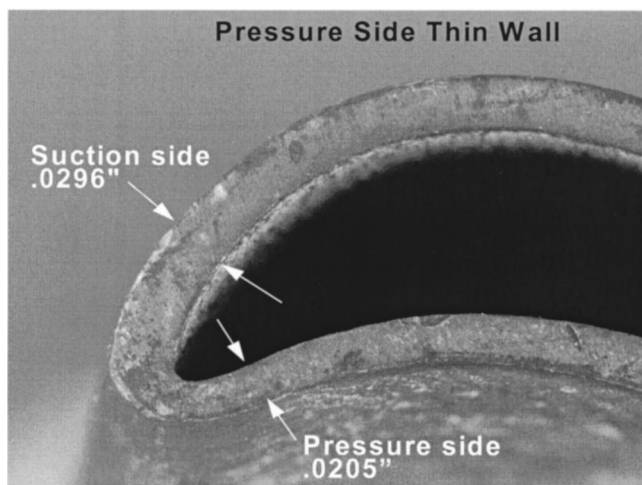


Fig. 6 Variation in pressure and suction side wall thickness unbalance in the AHPFTP first stage blade ([5,11])

paths through the crystal microstructure. Propagation through the γ matrix along $\langle 111 \rangle$ planes is typical of this mode. Another common path is along the microscopic $\langle 111 \rangle$ planes of the γ matrix and the γ' precipitates while remaining in the global $\langle 001 \rangle$ crystallographic plane. Mode I stress intensity calculations are valid for this type of crack growth given that the overall crack path remains in the $\langle 001 \rangle$ plane normal to the direction of the applied loading. This mode of failure occurs in turbine blades at low temperatures and stresses. Global octahedral fracture is the second stage I type failure mode. Crack propagation for this type of failure occurs through the crystal lattice along the global $\langle 111 \rangle$ planes as shown in Fig. 1.

Orientation of these failure planes can vary significantly from the plane normal to the applied load. Mode I and mode II type loading (mixed mode) must be considered in determining the

equivalent stress intensity factor at the crack tip for this type of propagation. Telesman and Ghosn [10] have formulated a method for obtaining an equivalent or octahedral stress intensity factor, ΔK_{EQ} or ΔK_{OCT} , devised to better represent the conditions at the crack face as shown in Eq. (1):

$$\Delta K_{OCT} = \{[K_{RSS}]^2 + [\Delta K_{ION(111)}]^2\}^{1/2}. \quad (1)$$

This method utilizes K_{RSS} that represents the stress intensity factor at the crack tip due to the resolved shear stresses on the octahedral $\langle 111 \rangle$ planes. In addition $\Delta K_{ION(111)}$ represents the Mode I stress intensity factor using the components of the applied load normal to the $\langle 111 \rangle$ plane. This has been shown to be effective at modeling the octahedral failure mode ([10]).

Macroscopic octahedral-type crack propagation is seen to occur at all temperature ranges. Typically it is seen at higher testing temperatures when a stage II type crack reaches a sufficient length and transitions to a stage I type propagation. Experience in gas turbine engines has shown that cracks typically propagate in a stage II type mode from initiation. Transition to stage I propagation does not take place until the crack has reached a significant length. Preventing the propagation of stage II cracks will preclude the transition to a stage I failure mode.

Noncrystallographic stage II type propagation is also typically seen to occur in single-crystal gas turbine blades ([2,8]). This propagation is along a path normal to the applied load regardless on the crystallographic orientation as shown in Fig. 2. Cracks proceed through the γ matrix and γ' precipitates independent of the microstructure. Mode I stress intensity solution is effective at predicting the state of stress at the crack tip for this type of propagation.

3 Turbine Blade Failures in the Space Shuttle Main Engine Alternate Advanced High-Pressure Fuel Turbo Pump (SSME AHPFTP)

To supply liquid hydrogen to the main injector, the three main engines on the space shuttle are each equipped with a high-pressure fuel turbopump (HPFTP). This pump is driven by a two-

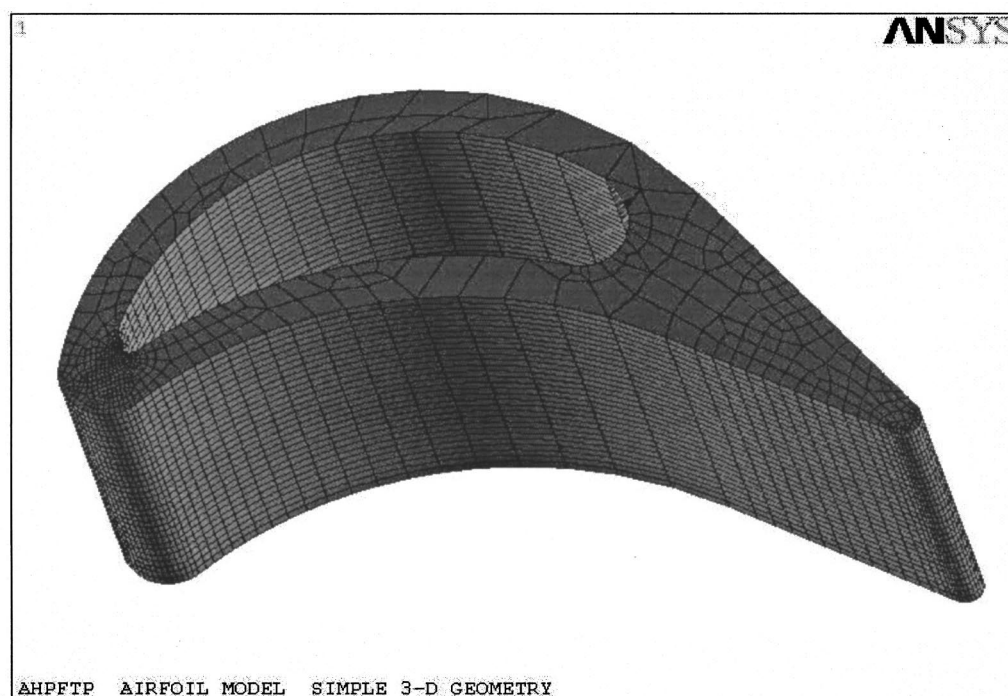
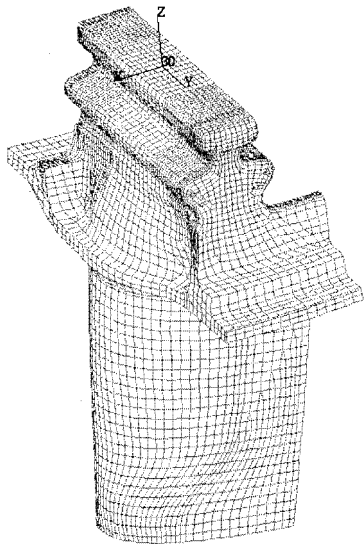


Fig. 7 Simple three-dimensional airfoil model of blade geometry ([5])



First Stage Blade Casting or Material Coordinate System
Z axis along stacking axis pointing radially inward.
X axis pointing away from the pressure side.
Y axis pointing towards the second stage blade.

Fig. 8 First-Stage blade finite element model and casting or material coordinate system ([6])

stage turbine that extracts its power from the combustion products of hydrogen and oxygen generated from a preburner upstream. A three-stage turbine with a similar preburner is utilized to drive the high-pressure oxidizer turbopump (HPOTP), which is tasked with delivering liquid oxygen to the main injector. To supply the liquid hydrogen to the main injector at the required pressure of 6640 psi and a flow rate of 17,000 gal/min the turbine must extract 74,000 hp from the combustion products. Each of the two turbine stages in the AHPFTP utilizes 50 uncooled blades operating at a temperature of 1600°F and a rotational speed of 37,000 rpm. Each blade has an airfoil of 25 mm (1 in) length and a total weight of

113 gm (4 Oz), and extracts an average of 740 hp. Blades are designed for a life between overhauls of 55 shuttle missions or 7.5 hrs of operation ([5,11]).

Turbine blades of the AHPFTP are fabricated from single crystal nickel-base PWA1480 material. Many of these blades have failed during testing due to initiation and propagation of fatigue cracks from an area of high stress concentration at the blade tip leading edge. These failures have been discussed by Arakere and Swanson [6], where effects of crystal orientation on blade fatigue life was investigated. As part of the fatigue failure investigation, an attempt was made to identify a particular characteristic of the blade that might correlate with the known cracked blades. The pressure sidewall thickness and leading edge core radius were both considered primary factors. In addition, primary and secondary crystal orientation, run time, and number of engine starts were all evaluated. There was no single factor that correlated well with crack initiation. However, the combined effect of the core radius and pressure sidewall did show some correlation as shown in Figs. 3 and 4 ([5,11]). The secondary crystallographic orientation did seem to have considerable influence on whether the crack would arrest or continue to propagate.

A review of crack depths on the population of blades involved in the investigation showed an interesting correlation with the secondary crystal orientation, designated as angle β . Blades where cracks had grown in excess of .007 in. or through the pressure sidewall were all in the family of blades with β less than 30 deg or greater than 60 deg. Due to cubic crystal structure β of 30 deg and 60 deg are equivalent for this blade geometry. The population of blades with tip crack depths in the .007 in. range or less was centered on a β of 45 deg. Figure 5 reveals that for $\beta=45\pm15$ deg tip cracks arrested after some growth or did not initiate at all ([5,11]). These results suggest that perhaps there are preferential β orientations for which crack growth is minimized at the blade tip.

3.1 Fatigue Stresses at the Blade Tip Critical Location.

Manufacturing problems resulted in a small leading edge fillet radius and also a disparity between the pressure side and suction side airfoil wall thickness in the turbine blade, as shown in Fig. 6. Pressure sidewall thickness in many blades was found to be 35% below the nominal design intent for most blades in the investigation population. This condition appears to be the result of rotation of the ceramic core during the casting process. Steady or mean stresses at the blade core leading edge was predicted using three-

Table 1 Stress intensity factor values as a function of secondary crystallographic orientation (β) ([5])

ΔK_I and ΔK_{EQ} for various angles β									
Crack Closure 1300 psi				Crack Opening 2850 psi					
β (Deg)	K_I psi $\sqrt{\text{in}}$	K_{II} Psi $\sqrt{\text{in}}$	K_{EQ} psi $\sqrt{\text{in}}$	K_I psi $\sqrt{\text{in}}$	K_{II} Psi $\sqrt{\text{in}}$	K_{EQ} psi $\sqrt{\text{in}}$	β (Deg)	ΔK_I psi $\sqrt{\text{in}}$	ΔK_{EQ} psi $\sqrt{\text{in}}$
-20	3299	362	3319	12132	1331	12205	-20	8833	8886
0	3184	579	3236	11711	2131	11903	0	8527	8667
10	3016	589	3073	11092	2167	11302	10	8076	8229
20	2843	514	2889	10457	1891	10627	20	7614	7738
30	2738	395	2766	10069	1451	10173	30	7331	7407
40	2777	279	2791	10211	1027	10263	40	7434	7472
50	2957	225	2966	10875	829	10907	50	7918	7941
60	3171	259	3182	11662	952	11701	60	8491	8519
70	3299	362	3319	12132	1331	12205	70	8833	8886
80	3295	489	3331	12119	1797	12252	80	8824	8920
90	3184	579	3236	11711	2131	11903	90	8527	8667
110	2843	514	2889	10457	1891	10627	110	7614	7738

Table 2 Strain-controlled low cycle fatigue test data for PWA 1480 at 1200 for four specimen orientations ([6])

Specimen Orientation	Max Test Strain	Min Test Strain	R Ratio	Strain Range	Cycles to Failure
<001>	.01509	.00014	0.01	.01495	1326
<001>	.0174	.0027	0.16	0.0147	1593
<001>	.0112	.0002	0.02	0.011	4414
<001>	.01202	.00008	0.01	0.0119	5673
<001>	.00891	.00018	0.02	.00873	29516
<111>	.01219	-0.006	-0.49	.01819	26
<111>	.0096	.0015	0.16	0.0081	843
<111>	.00809	.00008	0.01	.00801	1016
<111>	.006	0.0	0.0	0.006	3410
<111>	.00291	-0.00284	-0.98	.00575	7101
<111>	.00591	.00015	0.03	.00576	7356
<111>	.01205	0.00625	0.52	0.0058	7904
<213>	.01212	0.0	0.0	.01212	79
<213>	.00795	.00013	0.02	.00782	4175
<213>	.00601	.00005	0.01	.00596	34676
<213>	.006	0.0	0.0	0.006	114789
<011>	.0092	.0004	0.04	0.0088	2672
<011>	.00896	.00013	0.01	.00883	7532
<011>	.00695	.00019	0.03	.00676	30220

dimensional finite element analysis to be 52.8 ksi for the nominal design case. The maximum stress at the leading edge increased to 117.8 ksi for the turbine blade shown in Fig. 6 with nonuniform wall thickness. Many of the cracked blades had a nonuniform wall thickness. This significant increase in steady stress greatly reduces the allowable alternating or dynamic stress capability of the blade.

In addition to the geometric concerns, potential drivers for dynamic stresses at the blade tip were evaluated. The leading edge core area showed sensitivity to a chordwise bending mode at 104% power level occurring at 22,662 Hz (mode 6). Fatigue testing of actual blades with small core radius performed in the lab showed the ability to initiate a crack in the blade tip with a mode 6-excitation frequency. Based on the Goodman diagram a dynamic stress of only 5 ksi peak would put the blade over the fatigue limit of 10^8 cycles with such a high mean stress. Given the inspection data and analysis results, it was concluded that the

small leading edge fillet radius and the nonuniform wall thickness resulted in high mean stress that ultimately lead to the blade tip crack initiation.

4 A Simplified Finite Element Model of the Alternate Advanced High-Pressure Fuel Turbo Pump (AHPFTP) Turbine Blade

Initially, finite element models using simplified geometry and load conditions were used for making a qualitative assessment of the influence of secondary crystallographic orientation. Figure 7 shows the simplified three-dimensional model that focused on representing the stress and deflection fields in the blade by applying simple loads and boundary conditions ([5]). This model produces maximum principal stresses and deflections comparable to those obtained using the complex three-dimensional model that represents the actual geometry of the turbine blades, as seen in Fig. 8 ([5]). Turbine blades in operation are subjected to rotational, aerodynamic, and thermal loads. Based on the deflected shape of the internal cavity produced in actual blades it was determined that an internal pressure applied to these surfaces would do an adequate job of representing the effect of rotational loads. This pressure was adjusted to 2075 psi to attain the maximum predicted principal stress at the blade leading edge of 117.8 ksi predicted by the complex three-dimensional model.

Predictions of peak alternating or dynamic stresses in the critical blade tip region are in the range of 45 ksi for the blade at operating conditions. Analytical models of the blade frequency response indicate that a chordwise bending mode (mode 6) generates significant stresses at the blade tip in the failure region. This chordwise mode loads a crack at the blade tip in a mode I fashion. To include the effect of dynamic stresses the applied pressure was increased in the simple three-dimensional model to 2850 psi to generate an additional 45 ksi maximum principal stress at the leading edge. Maximum principal stresses at the crack tip were established as $\sigma_{\text{MIN}} = 73$ ksi for crack closure and $\sigma_{\text{MAX}} = 163$ ksi for crack opening in assessing the ΔK at the crack tip.

The effect of secondary crystallographic orientation on the stress intensity factor at a crack located in the critical region was investigated. Single crystal materials have highly orthotropic

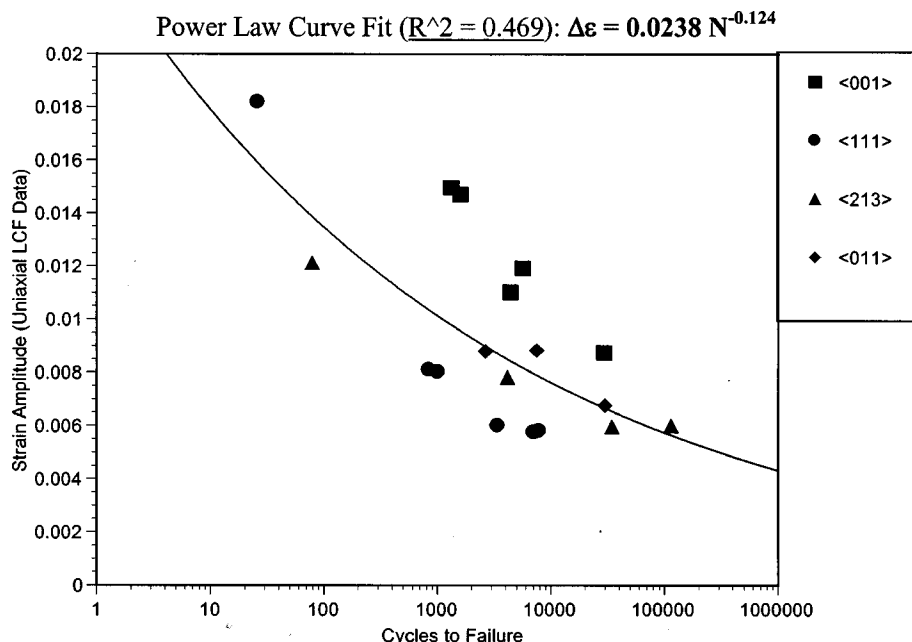


Fig. 9 Strain range versus cycles to failure for low cycle fatigue test data (PWA 1480 at 1200°F)

Table 3 Maximum values of shear stress and shear strain on the slip systems and normal stress and strain values on the same planes ([6,7]). (PWA 1480 at 1200°F in air.)

$\gamma_{\max}$ = Max shear strain of 30 slip systems for max specimen test strain value
 $\gamma_{\min}$ = Max shear strain of 30 slip systems for min specimen test strain value
 $\tau_{\max}$ = Max shear stress of 30 slip systems for max specimen test strain value
 $\tau_{\min}$ = Max shear stress of 30 slip systems for min specimen test strain value

Specimen Orientation	$\gamma_{\max}$	$\gamma_{\min}$	$\Delta\gamma/2$	$\epsilon_{\max}$	$\epsilon_{\min}$	$\Delta\epsilon/2$	$\tau_{\max}$ psi	$\tau_{\min}$ psi	$\Delta\tau$ psi	$\sigma_{\max}$ psi	$\sigma_{\min}$ psi	$\Delta\sigma$ psi	Cycles to Failure
<001> $\tau_{\max} = \tau^{14}$ $\gamma_{\max} = \gamma^{14}$	0.02	0.000185	0.0099075	0.00097	9.25E-06	0.0004804	1.10E+05	1016	1.08E+05	7.75E+04	719	7.68E+04	1326
	0.023	0.0036	0.0097	0.0015	1.78E-04	0.000661	1.26E+05	1.96E+04	1.06E+05	8.93E+04	1.39E+04	7.54E+04	1593
	0.015	2.64E-04	0.007368	7.34E-04	1.32E-05	0.0003604	8.13E+04	1452	7.98E+04	5.75E+04	1027	5.65E+04	4414
	0.016	0	0.008	7.94E-04	0	0.000397	8.73E+04	0	8.73E+04	6.17E+04	0	6.17E+04	5673
	0.012	0	0.006	5.89E-04	0	0.0002945	6.47E+04	0	6.47E+04	4.57E+04	0	4.57E+04	29516
<111> $\tau_{\max} = \tau^{25}$ $\gamma_{\max} = \gamma^{25}$	0.014	-7.06E-03	0.01053	2.05E-03	-1.01E-03	0.00153	2.25E+05	-1.10E+05	3.35E+05	1.59E+05	-7.80E+04	2.37E+05	26
	0.011	0.00176	0.00462	0.0016	0.00025	0.000675	1.77E+05	2.77E+04	1.49E+05	1.25E+05	1.96E+04	1.05E+05	843
	.0095	9.40E-05	0.004703	0.00136	1.34E-05	0.0006733	1.49E+05	1478	1.48E+05	1.06E+05	1045	1.05E+05	1016
	.0076	0	0.0038	0.001	0	0.0005	1.10E+05	0	1.10E+05	7.84E+04	0	7.84E+04	3410
	.0034	-0.0033	0.00335	0.00049	-0.00048	0.000485	5.40E+04	-5.30E+04	1.07E+05	3.80E+04	-3.70E+04	7.50E+04	7101
<213> $\tau_{\max} = \tau^{29}$ $\gamma_{\max} = \gamma^{29}$	.0069	1.76E-04	0.003362	9.90E-04	2.50E-05	0.0004825	1.09E+05	2771	1.06E+05	7.70E+04	1959	7.50E+04	7356
	0.014	0.007	0.0035	0.002	0.001	0.0005	2.25E+05	1.10E+05	1.15E+05	1.60E+05	7.80E+04	8.20E+04	7904
	0.018	0	0.009	0.002	0	0.001	1.60E+05	0	1.60E+05	1.30E+05	0	1.30E+05	79
	0.012	1.90E-04	0.005905	0.0013	2.10E-05	0.0006395	1.06E+05	1732	1.04E+05	8.60E+04	1400	8.46E+04	4175
	.0088	0	0.0044	0.00098	0	0.00049	8.00E+04	0	8.00E+04	6.50E+04	0	6.50E+04	34676
<011> $\tau_{\max} = \tau^{15}$ $\gamma_{\max} = \gamma^{15}$	.0088	0	0.0044	0.00098	0	0.00049	8.00E+04	0	8.00E+04	6.50E+04	0	6.50E+04	114789
	0.015	6.50E-04	0.007175	0.0039	1.68E-04	0.001866	1.23E+05	5333	1.18E+05	1.73E+05	7538	1.65E+05	2672
	0.015	0	0.0075	0.0039	0	0.00195	1.23E+05	0	1.23E+05	1.70E+05	0	1.70E+05	7532
	0.011	3.10E-04	0.005345	0.0029	8.00E-05	0.00141	9.30E+04	2532	9.05E+04	1.31E+05	3581	1.27E+05	30220

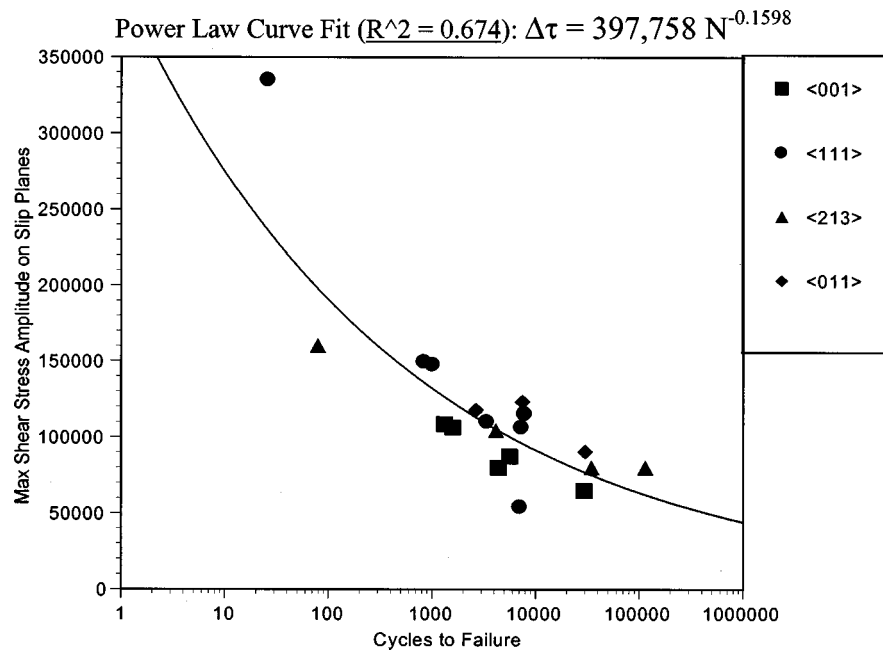


Fig. 10 Shear stress amplitude [$\Delta\tau_{\max}$] versus N

properties making the position of the crystal lattice relative to the part geometry a significant factor in the overall analysis. For example, the elastic or Young's modulus (E) is a strong function of crystal orientation over the standard stereographic triangle, given by Eq. (2) ([13]).

$$E^{-1} = S_{11} - [2(S_{11} - S_{12}) - S_{44}] \times [\cos^2 \phi (\sin^2 \phi - \sin^2 \theta \cos^2 \phi \cos^2 \theta)] \quad (2)$$

where θ is the angle between the growth direction and (001) and ϕ is the angle between the (001)-(110) boundary of the triangle. The terms S_{11} , S_{12} , and S_{44} are the elastic compliances. A two-dimensional model of the airfoil cross section was generated to perform the fracture mechanics assessment of stress intensity factors at the crack tip. Stress intensity solutions were calculated for through the thickness crack at two pressure load conditions and 12 different secondary orientation angles, β . The two pressure loads represent the conditions for $\sigma_{\min}$ and $\sigma_{\max}$ at the leading edge fillet region. From these two results, a ΔK was determined for each secondary orientation and crack orientation. Calculated K results for each of the β angles at the two internal pressures, using Eq. (1), is displayed in Table 1. Note that $\Delta K_{\text{EQ}} = \Delta K_{\text{OCT}}$. Solutions for stress intensity factors are repeated for β angles separated by 90 deg due to the cubic structure of the single crystal.

Stress intensity factors at the crack tip in the AHPFTP blade show a significant dependence on secondary crystallographic orientation. Secondary orientation can influence the magnitude of ΔK at the crack tip by 15–20%. This percentage change in ΔK equates to roughly a 40% change in crack growth rate for PWA 1480 material with an R -value of 0.5 at a temperature of 1100°F ([8]). Changes of 10 times or more are possible for a 20% change in ΔK at certain temperatures and stress ratios ([8]). This implies that there is potential for one secondary orientation to preclude fatigue crack growth while for another the fatigue crack growth continues unchecked. The two-dimensional model predicted the lowest ΔK for a β value of 40 deg and the highest ΔK for $\beta = 0$ deg or 90 deg. A review of the data collected from the AHPFTP experience indicates that no blades with β of 40 deg had cracks propagate past their initiation length. A number of blades with β of 0 deg or 90 deg exhibited cracks that propagated past their initiation size and in several cases airfoil failure resulted. The inference here is that given two blades with identical wall thick-

ness, core radius and initial crack size the ΔK at the crack tip would be 20% lower for a blade with a β of 40 deg versus one with a β of 90 deg.

The simplified modeling techniques utilized did an adequate job at selecting a preferred secondary orientation for the AHPFTP first-stage blade. This orientation will give the maximum resistance to fatigue crack growth for a flaw initiating from the blade tip leading edge.

5 Analysis of Fatigue Test Data of Single Crystal Nickel Superalloys

Low cycle fatigue data were analyzed in detail by Arakere and Swanson [6] by evaluating the maximum shear stress amplitude on the 24 octahedral and 6 cube slip systems. The maximum shear stress amplitude, $\Delta\tau_{\max}$, was found to be an effective parameter for characterizing fatigue test data is presented for PWA1480 and SC 7-14-6 for various temperatures and environmental conditions. PWA1480 strain-controlled low cycle fatigue data are presented for 1200°F in air, and for room temperature, 1400°F and 1600°F

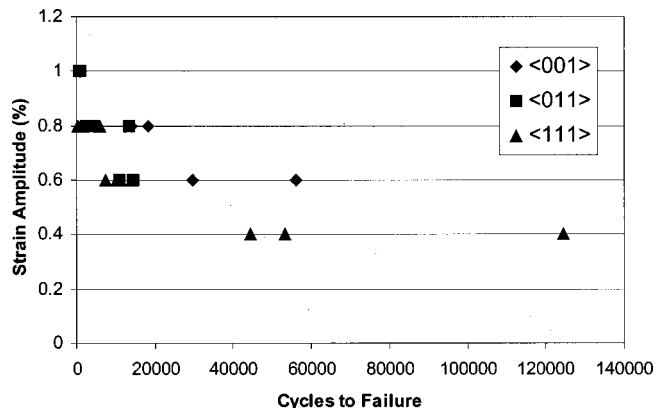


Fig. 11 Low cycle fatigue data for PWA 1493 at room temperature in 5000 psi high-pressure hydrogen: strain amplitude versus cycles to failure

Table 4 PWA 1480 low cycle fatigue high-pressure hydrogen (5000 psi) data at ambient temperature

Specimen Orientation	Max Strain $\epsilon_{\max}$	Min Strain $\epsilon_{\min}$	Strain Ratio $R = \epsilon_{\min}/\epsilon_{\max}$	Strain Range	$\Delta\gamma_{\max}$	$\Delta\epsilon_{\max}$	$\Delta\tau_{\max}$ (psi)	$\Delta\sigma_{\max}$ (psi)	Cycles to Failure	
<001>	0.005	-0.005	-1	0.01	0.01310	0.001	84,853	180,000	693	
	0.005	-0.005	-1	0.01	0.01310	0.001	84,853	180,000	1093	
	$\tau_{\max} = \tau^{15}$	0.004	-0.004	-1	0.008	0.01048	0.008	67,882	144,000	2929
		0.004	-0.004	-1	0.008	0.01048	0.008	67,882	144,000	3340
		0.004	-0.004	-1	0.008	0.01048	0.008	67,882	144,000	13964
		0.004	-0.004	-1	0.008	0.01048	0.008	67,882	144,000	18324
		0.003	-0.003	-1	0.006	0.00786	0.006	50,912	108,000	29551
		0.003	-0.003	-1	0.006	0.00786	0.006	50,912	108,000	56172
<011>	0.005	-0.005	-1	0.01	0.008034	0.005514	115,010	216,820	826	
	0.005	-0.005	-1	0.01	0.008034	0.005514	115,010	216,820	930	
	$\tau_{\max} = \tau^{27}$	0.004	-0.004	-1	0.008	0.006427	0.004416	92,005	173,460	2897
		0.004	-0.004	-1	0.008	0.006427	0.004416	92,005	173,460	3256
		0.004	-0.004	-1	0.008	0.006427	0.004416	92,005	173,460	4234
		0.004	-0.004	-1	0.008	0.006427	0.004416	92,005	173,460	13388
		0.003	-0.003	-1	0.006	0.004820	0.00339	69,004	130,090	10946
		0.003	-0.003	-1	0.006	0.004820	0.00339	69,004	130,090	14465
<111>	0.004	-0.004	-1	0.008	0.00927	0.007998	167,830	355,950	496	
	0.004	-0.004	-1	0.008	0.00927	0.007998	167,830	355,950	985	
	$\tau_{\max} = \tau^{25}$	0.004	-0.004	-1	0.008	0.00927	0.007998	167,830	355,950	5863
		0.003	-0.003	-1	0.006	0.006943	0.00599	125,870	266,970	7410
		0.003	-0.003	-1	0.006	0.006943	0.00599	125,870	266,970	10097
		0.003	-0.003	-1	0.006	0.006943	0.00599	125,870	266,970	14173
		0.002	-0.002	-1	0.004	0.004628	0.00399	83,914	177,980	44440
		0.002	-0.002	-1	0.004	0.004628	0.00399	83,914	177,980	53189
		0.002	-0.002	-1	0.004	0.004628	0.00399	83,914	177,980	124485

in high-pressure hydrogen (5000 psi). SC 7-14-6 strain-controlled low cycle fatigue data is taken from reference [14] and is recorded at 1800°F in air.

Slip in metal crystals often occurs on planes of high atomic density in closely packed directions. Deformation mechanisms operative in high γ' fraction nickel-base superalloys such as PWA 1480/1493 and SC-7-14-6 with FCC crystal structure are divided into three-temperature regions ([2]). In the low-temperature regime (26°C to 427°C, 79°F to 800°F) the principal deformation mechanism is by (111)/(<110>) slip, and hence fractures produced at these temperatures exhibit (111) facets. Above 427°C (800°F) thermally activated cube cross slip is observed which is manifested by an increasing yield strength up to 871°C (1600°F) and a proportionate increase in (111) dislocations that have cross slipped to (001) planes. Thus nickel-based FCC single crystal superalloys slip primarily on the octahedral and cube planes in specific slip directions.

Shear stresses on octahedral and cube slip planes was computed for uniaxial low cycle fatigue test specimens under strain control by Arakere and Swanson [6]. Shear stresses on the 24 octahedral and 6 cube slip systems are denoted by $\tau^1, \tau^2, \dots, \tau^{30}$. The shear stresses on the 24 octahedral slip systems are given by ([6,15])

$$\begin{Bmatrix} \tau^1 \\ \tau^2 \\ \tau^3 \\ \tau^4 \\ \tau^5 \\ \tau^6 \\ \tau^7 \\ \tau^8 \\ \tau^9 \\ \tau^{10} \\ \tau^{11} \\ \tau^{12} \end{Bmatrix} = \frac{1}{\sqrt{6}} \begin{bmatrix} 1 & 0 & -1 & 1 & 0 & -1 \\ 0 & -1 & 1 & -1 & 1 & 0 \\ 1 & -1 & 0 & 0 & 1 & -1 \\ -1 & 0 & 1 & 1 & 0 & -1 \\ -1 & 1 & 0 & 0 & -1 & -1 \\ 0 & 1 & -1 & -1 & -1 & 0 \\ 1 & -1 & 0 & 0 & -1 & -1 \\ 0 & 1 & -1 & -1 & 1 & 0 \\ 1 & 0 & -1 & -1 & 0 & -1 \\ 0 & -1 & 1 & -1 & -1 & 0 \\ -1 & 0 & 1 & -1 & 0 & -1 \\ -1 & 1 & 0 & 0 & 1 & -1 \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{zx} \\ \sigma_{yz} \end{Bmatrix},$$

$$\begin{Bmatrix} \tau^{13} \\ \tau^{14} \\ \tau^{15} \\ \tau^{16} \\ \tau^{17} \\ \tau^{18} \\ \tau^{19} \\ \tau^{20} \\ \tau^{21} \\ \tau^{22} \\ \tau^{23} \\ \tau^{24} \end{Bmatrix} = \frac{1}{3\sqrt{2}} \begin{bmatrix} -1 & 2 & -1 & 1 & -2 & 1 \\ 2 & -1 & -1 & 1 & 1 & -2 \\ -1 & -1 & 2 & -2 & 1 & 1 \\ -1 & 2 & -1 & -1 & -2 & -1 \\ -1 & -1 & 2 & 2 & 1 & -1 \\ 2 & -1 & -1 & -1 & 1 & 2 \\ -1 & -1 & 2 & 2 & -1 & 1 \\ 2 & -1 & -1 & -1 & -1 & -2 \\ -1 & 2 & -1 & -1 & 2 & 1 \\ 2 & -1 & -1 & 1 & -1 & 2 \\ -1 & 2 & -1 & 1 & 2 & -1 \\ -1 & -1 & 2 & -2 & -1 & -1 \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{zx} \\ \sigma_{yz} \end{Bmatrix} \quad (3)$$

and the shear stresses on the 6 cube slip systems are

$$\begin{Bmatrix} \tau^{25} \\ \tau^{26} \\ \tau^{27} \\ \tau^{28} \\ \tau^{29} \\ \tau^{30} \end{Bmatrix} = \frac{1}{\sqrt{2}} \begin{bmatrix} 0 & 0 & 0 & 1 & 1 & 0 \\ 0 & 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & 0 & 1 & 0 & 1 \\ 0 & 0 & 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & 0 & 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{zx} \\ \sigma_{yz} \end{Bmatrix}. \quad (4)$$

5.1 PWA1493 Data at 1200°F in Air. Strain-controlled low cycle fatigue tests conducted at 1200°F in air for PWA1480/1493 uniaxial smooth specimens, for four different orientations, is shown in Table 2 ([6]). Computation of shear stresses and strains on the 24 octahedral and 6 cube slip systems for FCC single crystal fatigue specimens was presented by Arakere and Swanson [6] in detail and hence is not repeated here. The four specimen orientations are <001>, <111>, <213>, and <011>. Figure 9 shows the plot of strain range versus cycles to failure. A wide scatter is observed in the data with poor correlation for a power-law fit.

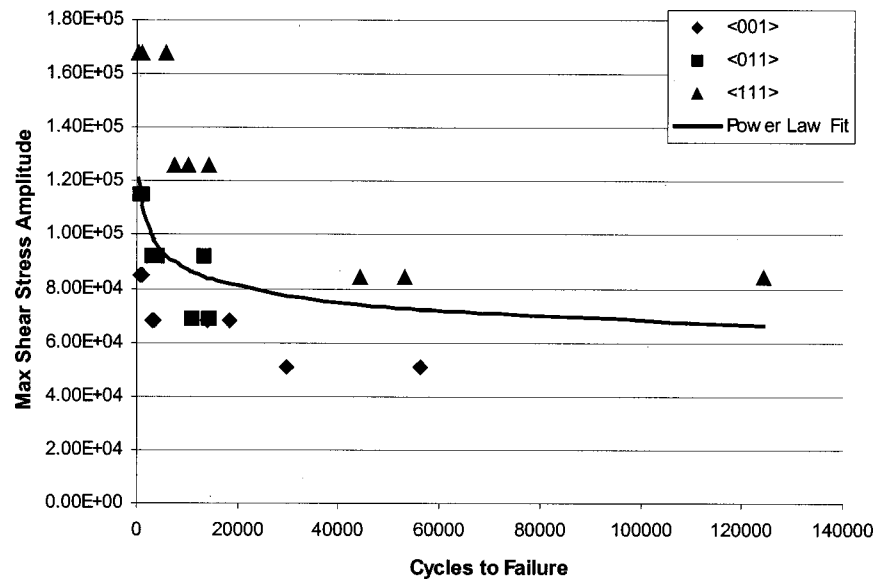


Fig. 12 Shear stress amplitude ($\Delta\tau_{\max}$) versus cycles to failure for PWA 1493 at room temperature in 5000 psi hydrogen power-law curve fit ($R^2=0.246$): $\Delta\tau = 238,349 N^{-0.1095}$

The maximum shear stress and strain for each data point, for min and max test strain values, is selected from the 30 values corresponding to the 30 slip systems. These values are tabulated in Table 3. For the $\langle 001 \rangle$ and $\langle 011 \rangle$ configurations the max shear stress and strain occur on the secondary slip system (τ^{14} , γ^{14} and τ^{15} , γ^{15} , respectively). For the $\langle 111 \rangle$ and $\langle 213 \rangle$ configurations, the max shear stress and strain occur on the cube slip system (τ^{25} , γ^{25} and τ^{29} , γ^{29} , respectively).

The maximum shear stress amplitude [$\Delta\tau_{\max}$] versus cycles to failure is shown in Fig. 10. The power-law curve fit for Fig. 10 is given by Eq. (5).

$$\Delta\tau_{\max} = 397,758 N^{-0.1598} \quad (5)$$

The results suggest that $\Delta\tau_{\max}$ is a good fatigue failure parameter. This conclusion is, perhaps, not very surprising since the deformation mechanisms in single crystals are controlled by the propagation of dislocations driven by shear.

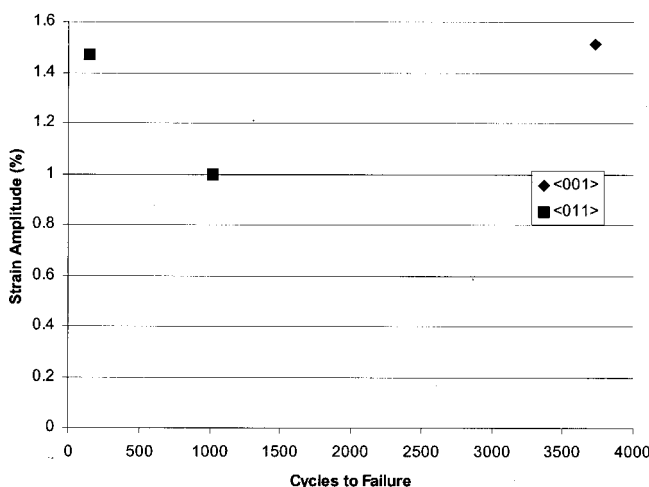


Fig. 13 Low cycle fatigue data for PWA 1493 at 1400°F in 5000 psi high-pressure hydrogen: strain amplitude versus cycles to failure

5.2 PWA1480 Data at Room Temperature (75°F) in High-Pressure Hydrogen. Turbine blades in the space shuttle main engine (SSME) alternate high pressure fuel turbo pump (AHPFTP) are made of PWA1480 single crystal material ($\langle 6,11 \rangle$). The blades are subjected to the effects of high-pressure hydrogen. The effects of high-pressure hydrogen are most detrimental at room temperature, from a fatigue crack nucleation perspective, and its effects are less pronounced at higher temperatures ([2,19]).

At room temperature (26°C), and in high-pressure hydrogen, the eutectic γ/γ' initiates fatigue cracks by an interlaminar (between γ and γ') failure mechanism, resulting in noncrystallographic fracture ([2,16]). In room temperature air, carbides typically initiate crystallographic fracture. Fatigue cracks frequently nucleate at microporosities when tested in air at moderate temperature (above 427°C).

Figure 11 shows the strain amplitude versus cycles to failure low cycle fatigue data for PWA1480 at room temperature (26°C, 75°F) in 5000 psi hydrogen, for three different specimen orientations. Testing was performed under strain control. The data in Fig.

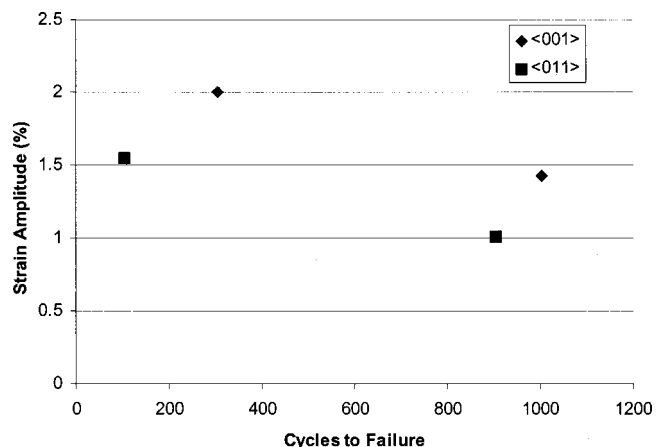


Fig. 14 Low cycle fatigue data for PWA 1493 at 1600°F in 5000 psi high-pressure hydrogen: strain amplitude versus cycles to failure

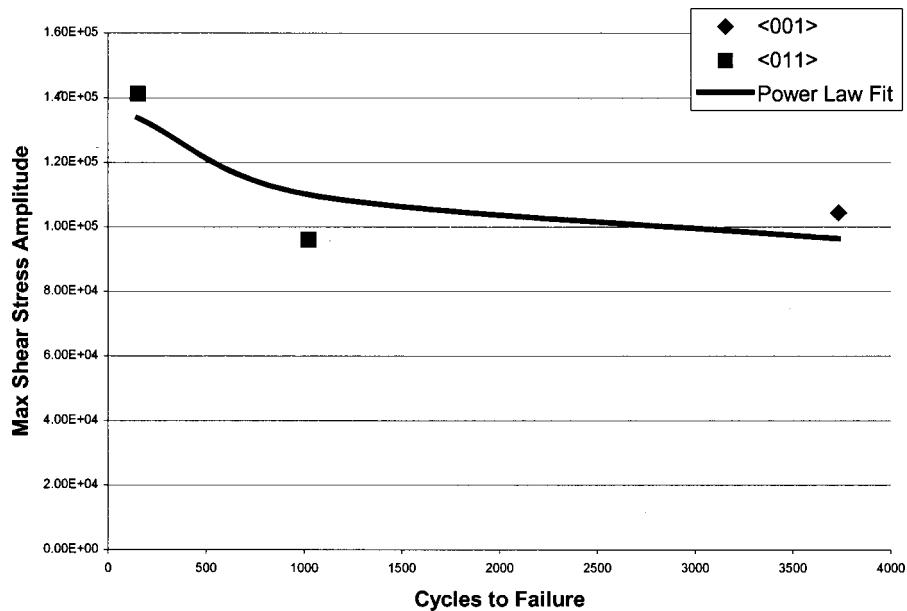


Fig. 15 Shear stress amplitude ($\Delta\tau_{\max}$) versus cycles to failure for PWA 1493 at 1400°F in 5000 psi hydrogen power-law curve fit ($R^2=0.661$): $\Delta\tau=223,516 N^{-0.1023}$

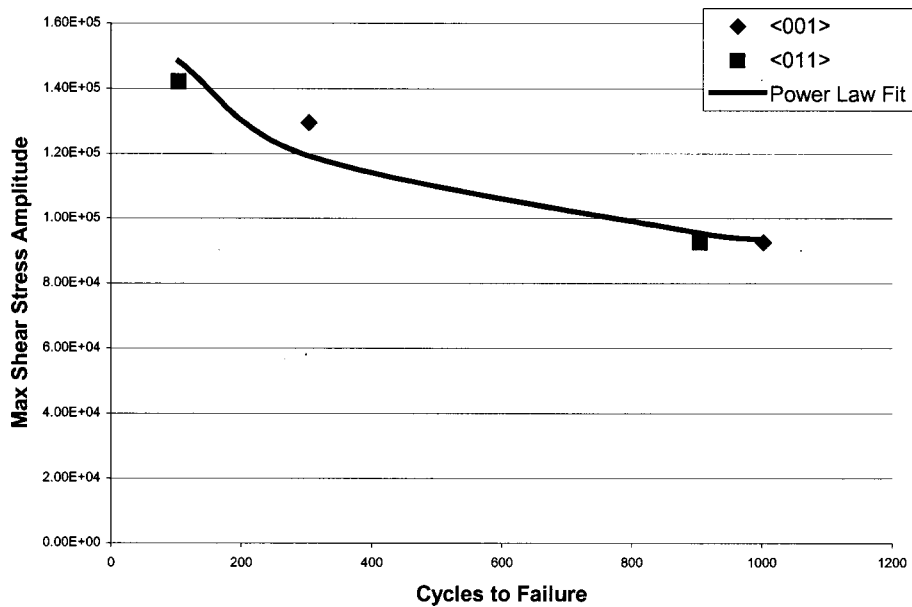


Fig. 16 Shear stress amplitude ($\Delta\tau_{\max}$) versus cycles to failure for PWA 1493 at 1600°F in 5000 psi hydrogen power-law curve fit ($R^2=0.9365$): $\Delta\tau=381,241 N^{-0.2034}$

Table 5 PWA 1493 low cycle fatigue data measured in high-pressure hydrogen (5000 psi) at 1400°F

Specimen Orientation	Max Strain $\epsilon_{\max}$	Min Strain $\epsilon_{\min}$	Strain Ratio $R = \epsilon_{\min}/\epsilon_{\max}$	Strain Range	$\Delta\gamma_{\max}$	$\Delta\epsilon_{\max}$	$\Delta\tau_{\max}$ (psi)	$\Delta\sigma_{\max}$ (psi)	Cycles to Failure
<001> $\tau_{\max} = \tau^{15}$	0.0075	-0.0075	-1	0.0151	0.0199	0.0151	104,420	221,520	3733
<011> $\tau_{\max} = \tau^{27}$	0.00735	-0.00735	-1	0.0147	0.01212	0.0081	141,190	266,190	152
	0.005	-0.005	-1	0.01	0.00824	0.00551	96,051	181,080	1023

Table 6 PWA 1493 low cycle fatigue data taken measured in high-pressure hydrogen (5000 psi) at 1600°F

Specimen Orientation	Max Strain $\epsilon_{\max}$	Min Strain $\epsilon_{\min}$	Strain Ratio $R = \epsilon_{\min}/\epsilon_{\max}$	Strain Range	$\Delta\gamma_{\max}$	$\Delta\epsilon_{\max}$	$\Delta\tau_{\max}$ (psi)	$\Delta\sigma_{\max}$ (psi)	Cycles to Failure
$\langle 001 \rangle$ $\tau_{\max} = \tau^{15}$	0.0071	-0.0071	-1	0.0143	0.01899	0.0143	92,555	196,340	1002
	0.010	-0.010	-1	0.020	0.02657	0.020	129,450	274,600	303
$\langle 011 \rangle$ $\tau_{\max} = \tau^{27}$	0.0077	-0.0077	-1	0.0155	0.01295	0.00865	142,100	267,910	104
	0.005	-0.005	-1	0.0101	0.00843	0.00564	92,597	174,570	905

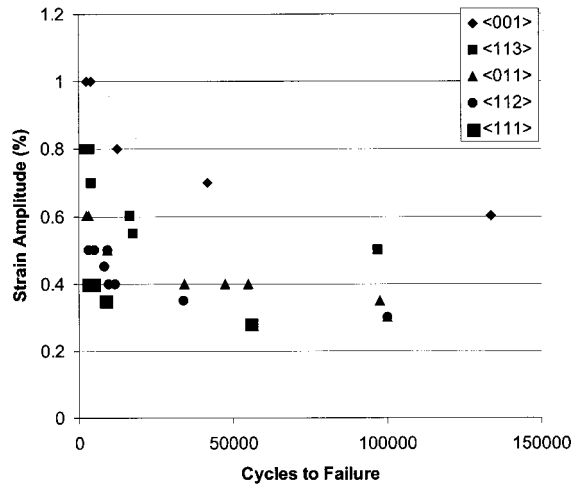


Fig. 17 Low cycle fatigue data for SC 7-14-6 at 1800°F in air: strain amplitude versus cycles to failure ([14])

11 show a fairly wide scatter. Table 4 shows the low cycle fatigue data and other fatigue damage parameters evaluated on the slip planes. Figure 12 shows a plot of $[\Delta\tau_{\max}]$ versus cycles to failure with the power-law curve fit showing a poor correlation. The presence of high-pressure hydrogen at room temperature activates the eutectic and causes noncrystallographic fracture, as explained earlier. This type of noncrystallographic fracture is not captured well by an analysis of shear stresses on slip planes. A failure parameter that can model the interlaminar failure mechanism between the γ and γ' structures has to be developed for better results.

5.3 PWA1480 Data at 1400°F and 1600°F in High-Pressure Hydrogen.

At higher temperatures hydrogen does not activate the eutectic failure mechanism and under these conditions it will be seen that $[\Delta\tau_{\max}]$ is a good failure parameter for modeling low cycle fatigue data. Figures 13 and 14 show the strain amplitude versus cycles to failure for PWA1493 in high-pressure hydrogen (5000 psi) at 1400°F and 1600°F, respectively. These tests were

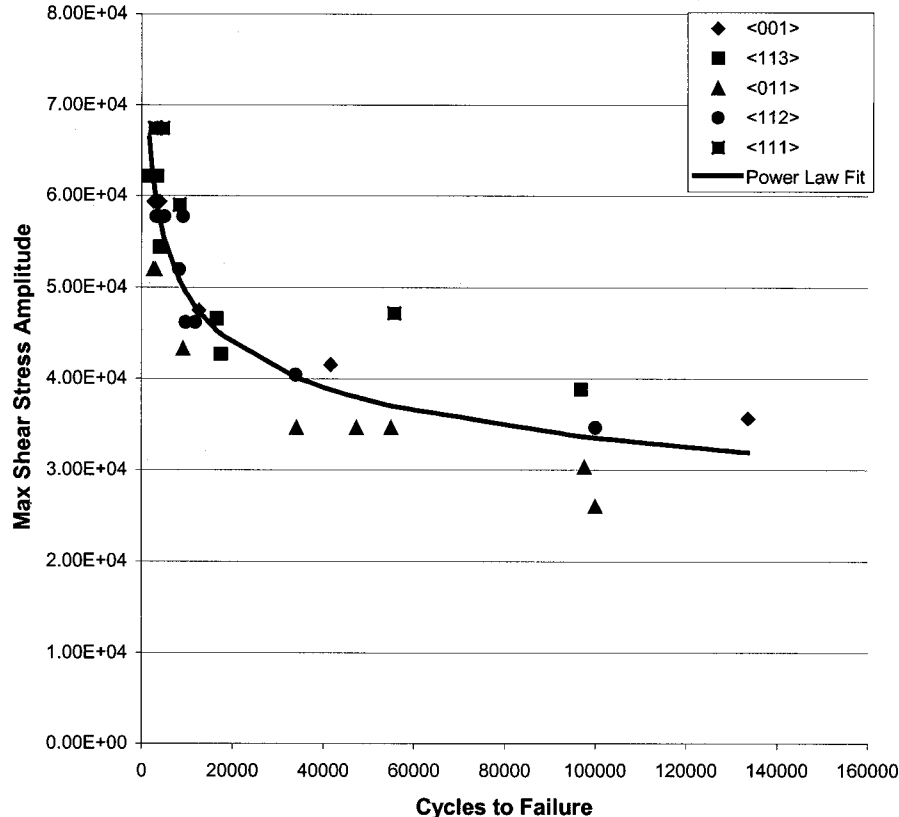


Fig. 18 Shear stress amplitude ($\Delta\tau_{\max}$) versus cycles to failure for SC 7-14-6 at 1800°F in air power-law curve fit ($R^2=0.7931$): $\Delta\tau=230,275 \text{ N}^{-0.1675}$

Table 7 Low cycle fatigue data for single crystal Ni-base superalloy SC 7-14-6 at 1800°F in air ([14])

Specimen Orientation	Strain Range	$\Delta\gamma_{\max}$	$\Delta\epsilon_{\max}$	$\Delta\tau_{\max}$ (psi)	$\Delta\sigma_{\max}$ (psi)	Cycles to Failure	
<001>	0.01	1.3337E-02	1.0000E-02	5.9350E+04	1.2590E+05	3985	
	0.01	1.3337E-02	1.0000E-02	5.9350E+04	1.2590E+05	2649	
	$\tau_{\max} = \tau^{15}$	0.008	1.0670E-02	8.0000E-03	4.7480E+04	12608	
	0.007	9.3359E-03	7.0000E-03	4.1545E+04	8.8130E+04	41616	
	0.006	8.0022E-03	6.0000E-03	3.5610E+04	7.5540E+04	133615	
<113>	0.008	1.2041E-02	8.5518E-03	6.2118E+04	1.1859E+05	3506	
	0.008	1.2041E-02	8.5518E-03	6.2118E+04	1.1859E+05	1698	
	$\tau_{\max} = \tau^{15}$	0.007	1.0536E-02	7.4829E-03	5.4353E+04	1.0377E+05	4042
	0.006	9.0311E-03	6.4139E-03	4.6588E+04	8.8946E+04	16532	
	0.0055	8.2785E-03	5.8794E-03	4.2706E+04	8.1533E+04	17500	
<011>	0.0055	8.2785E-03	5.8794E-03	4.2706E+04	8.1533E+04	17383	
	0.005	7.5259E-03	5.3449E-03	3.8823E+04	7.4121E+04	96847	
	0.006	5.1090E-03	3.4154E-03	5.1940E+04	9.7922E+04	2616	
	0.006	5.1090E-03	3.4154E-03	5.1940E+04	9.7922E+04	3062	
	$\tau_{\max} = \tau^{27}$	0.005	4.2575E-03	2.8462E-03	4.3283E+04	8.1601E+04	9112
<112>	0.004	3.4060E-03	2.2769E-03	3.4627E+04	6.5281E+04	34063	
	0.004	3.4060E-03	2.2769E-03	3.4627E+04	6.5281E+04	54951	
	0.004	3.4060E-03	2.2769E-03	3.4627E+04	6.5281E+04	47292	
	0.0035	2.9802E-03	1.9923E-03	3.0298E+04	5.7121E+04	97593	
	0.003	2.5545E-03	1.7077E-03	2.5970E+04	4.8961E+04	100000	
<111>	0.005	7.1612E-03	5.1387E-03	5.7711E+04	1.0880E+05	3271	
	0.005	7.1612E-03	5.1387E-03	5.7711E+04	1.0880E+05	5024	
	$\tau_{\max} = \tau^{29}$	0.005	7.1612E-03	5.1387E-03	5.7711E+04	1.0880E+05	9112
	0.0045	6.4451E-03	4.6249E-03	5.1940E+04	9.7922E+04	8298	
	0.004	5.7290E-03	4.1110E-03	4.6169E+04	8.7042E+04	9665	
<111>	0.004	5.7290E-03	4.1110E-03	4.6169E+04	8.7042E+04	11812	
	0.0035	5.0129E-03	3.5971E-03	4.0398E+04	7.6161E+04	33882	
	0.003	4.2967E-03	3.0832E-03	3.4627E+04	6.5281E+04	100000	
	0.004	4.7426E-03	6.4648E-04	6.7392E+04	1.4294E+05	2886	
	0.004	4.7426E-03	6.4648E-04	6.7392E+04	1.4294E+05	3075	
$\tau_{\max} = \tau^{25}$	0.004	4.7426E-03	6.4648E-04	6.7392E+04	1.4294E+05	4652	
<111>	0.0035	4.1498E-03	5.6567E-04	5.8968E+04	1.2507E+05	8382	
	0.0028	3.3198E-03	4.5254E-04	4.7175E+04	1.0005E+05	55647	

conducted at the NASA MSFC. There are only three data points at 1400°F and 4 at 1600°F because of the difficulty and expense in performing fatigue tests under these conditions. Figures 15 and 16 show the plots of $[\Delta\tau_{\max}]$ versus cycles to failure for 1400°F and 1600°F temperatures, respectively. The power-law curve fits are seen to have a good correlation because the resulting fractures are crystallographic in nature at these high temperatures. Tables 5 and 6 show the low cycle fatigue data and the fatigue parameters.

5.4 SC 7-14-6 Low Cycle Fatigue Data at 1800°F in Air.

Figure 17 shows the strain amplitude versus cycles to failure low cycle fatigue data for SC 7-14-6 at 1800°F in air for five different specimen orientations (<001>, <113>, <011>, <112>, and <111>) taken from reference [14]. A wide amount of scatter is seen in the plot. Figure 18 shows $[\Delta\tau_{\max}]$ versus cycles to failure plot with an excellent correlation for a power-law fit. Table 7 shows the low cycle fatigue data and the fatigue parameters. The R -values for the test data, unfortunately, is not listed in reference [14].

6 Conclusions

Single crystal turbine blade fatigue crack initiation and crack growth in SSME AHPFTP/AT observed after testing are discussed. Fatigue cracks were initiating from the leading edge radius of the core and extending across the airfoil tip on the pressure side (Fig. 3). Analysis of the fracture surfaces on the blades found evidence of high cycle fatigue. Stage II noncrystallographic fatigue cracks with multiple origins was present at the core leading

edge radius and extending down the airfoil span along the core surface. Typical cracks were in the .007 in. range. However, several blades had cracks extending across the entire pressure sidewall, approximately .025 in., through the wall to the flow path side and had propagated down the airfoil span approximately .100 in. These longer cracks had started to transition from stage II fatigue to crystallographic stage I fatigue propagation. Fracture surfaces on most cracks showed the presence of multiple arrest marks. Reasons that contributed to initiation and growth of tip cracks were the small leading edge fillet radius, the nonuniform wall thickness on pressure and section sides, the high mean stress, and consequently low allowable alternating stress. An investigation of crack depths on the population of blades as a function of secondary crystallographic orientation revealed that for $\beta = 45^\circ \pm 15^\circ$ deg tip cracks arrested after some growth or did not initiate at all. Blades where cracks had grown in excess of .007 in. or through the pressure sidewall were all in the family of blades with β less than 30 deg or greater than 60 deg. These results suggest that there are preferential β orientations for which crack growth is minimized at the blade tip.

Experimentally determined low cycle fatigue data for PWA1480/1493 and SC 7-14-6, two single crystal nickel-base turbine blade superalloys, are presented for a wide range of temperatures, environmental conditions, and specimen orientations. The maximum shear stress amplitude $\Delta\tau_{\max}$ on the 30 slip systems was found to be an effective fatigue failure parameter, based on the curve fit between $\Delta\tau_{\max}$ and cycles to failure. The parameter

$[\Delta\tau_{\max}]$ did not characterize the room temperature low cycle fatigue data in high-pressure hydrogen well because of the noncrystallographic eutectic failure mechanism activated by hydrogen at room temperature. Low cycle fatigue data in high-pressure hydrogen at 1400°F and 1600°F was characterized well by the $\Delta\tau_{\max}$ failure parameter. Since deformation mechanisms in single crystals are controlled by the propagation of dislocations driven by shear, $\Delta\tau_{\max}$ might indeed be a good fatigue failure parameter to use.

Acknowledgments

This work was supported by a grant from the NASA Marshall Space Flight Center (MSFC), Huntsville, AL. NASA MSFC provided the LCF data for PWA1493 in air and high-pressure hydrogen. The author would like to thank Dr. Greg Swanson at the NASA MSFC for supporting this work.

References

- [1] Cowles, B. A., 1996, "High Cycle Fatigue in Aircraft Gas Turbines—An Industry Perspective," *Int. J. Fatigue*, pp. 1–16.
- [2] DeLuca, D., and Annis, C., 1995, "Fatigue in Single Crystal Nickel Superalloys," Office of Naval Research, Department of the Navy FR23800, Aug.
- [3] VerSnyder, F. L., and Guard, R. W., 1960, "Directional Grain Structure for High Temperature Strength," *Trans. ASM*, **52**, p. 485.
- [4] McLean, M., 1983, "Mechanical Behavior: Superalloys," *Directionally Solidified Materials for High Temperature Service*, The Metals Society, London, p. 151.
- [5] Moroso, J., 1999, "Effect of Secondary Orientation on Fatigue Crack Growth in Single Crystal Turbine Blades," M. S. thesis, Mechanical Engineering Department, University of Florida, Gainesville, FL, May.
- [6] Arakere, N. K., and Swanson, G., 2000, "Effect of Crystal Orientation on Fatigue Failure of Single Crystal Nickel Base Turbine Blade Superalloys," presented at the ASME IGTI conference May 8–11, Munich, and accepted for publication in the ASME Journal of Gas Turbines and Power. ASME J. Eng. Gas Turbines Power.
- [7] Swanson, G., and Arakere, N. K., 2000, "Fatigue Failure of Single Crystal Nickel Base Turbine Blade Superalloys," NASA Technical Paper TP-2000-210074.
- [8] Cunningham, S., DeLuca, D., and Haake, F., 1996, "Crack Growth and Life Prediction in Single-Crystal Nickel Superalloys, Vol. 1," Wright Laboratory WL-TR-94-4089, Feb.
- [9] Cunningham, S., DeLuca, D., and Haake, F., 1994, "Crack Growth and Life Prediction in Single-Crystal Nickel Superalloys, Vol. 2," Wright Laboratory WL-TR-96-4048, Aug.
- [10] Telesman, J., and Ghosn, L., 1995, "Fatigue Crack Growth Behavior of PWA 1484 Single Crystal Superalloy at Elevated Temperatures," International Gas Turbine and Aeroengine Congress and Exposition, June.
- [11] Pratt & Whitney, 1998, "SSME-AT HPFTP 1st Stage Blade Failure Investigation Final Report," Pratt & Whitney, West Palm Beach, FL, Feb.
- [12] Pratt & Whitney, 1996, "SSME Alternate Turbopump Development Program HPFTP Critical Design Review," P&W FR24581-1 Dec. 23, NASA Contract NAS8-36801.
- [13] Kear, B. H., and Pearcey, B. J., 1967, "Tensile and Creep Properties of Single Crystals of the Nickel-Base Superalloy Mar-M 200," *Trans. AIME*, **239**, p. 1209.
- [14] Dalal, R. P., Thomas, C. R., and Dardi, L. E., 1984, "The Effect of Crystallographic Orientation on the Physical and Mechanical Properties of an Investment Cast Single Crystal Nickel-Base Superalloy," *Superalloys*, M. Gell, C. S. Kortovich, R. H. Bricknell, W. B. Kent, and J. F. Radavich, eds., TMS-AIME, Warrendale, OH, pp. 185–197.
- [15] Stouffer, D. C., and Dame, L. T., 1996, *Inelastic Deformation of Metals*, John Wiley and Sons, New York.
- [16] DeLuca, D. P., and Cowles, B. A., 1989, "Fatigue and Fracture of Single Crystal Nickel in High Pressure Hydrogen," *Hydrogen Effects on Material Behavior*, N. R. Moody and A. W. Thomson, eds., TMS, Warrendale, PA.

A. El-Shafei
A. S. El-Kabbany
A. A. Younan

Faculty of Engineering,
Department of Mechanical Design and
Production,
Cairo University,
Giza 12316, Egypt

Rotor Balancing Without Trial Weights

The traditional balancing methods using trial or calibration weights are quite effective, yet too many trials may result in a lengthy balancing process. It had been suggested in the literature that it is possible to balance flexible rotors without the use of trial weights, if a rotor model is available. A procedure is developed in this paper to balance flexible rotors using complex modes and complex vibration measurements. It is shown that a complex rotor model is essential for the success of the technique. Moreover, careful calibration of the rotor model is the major cornerstone of the procedure. Experimental results illustrate the success of the procedure. [DOI: 10.1115/1.1762903]

Introduction

Balancing of flexible rotors has acquired the attention of many researchers since the days of Thearle [1]. The main balancing techniques are the influence coefficient method, [2], and the modal balancing method, [3]. Both techniques require measurement of rotor vibration, installing trial weights and measuring vibration after installing the trial weights, and perhaps repetition of this process several times to balance the rotor. There are field balancing methods, e.g., static-couple method, [4], that reduce the balancing process. However, in many cases, the balancing process can be quite expensive, time consuming and necessitates several startups of the machine.

Several attempts have been proposed in the literature to try to reduce this expensive balancing effort, through the use of a rotor model. Perhaps the first such method is the work of Hundal and Harker [5], in which, based on the available natural frequencies and mode shapes, the correction masses are determined by equating the modal component of the unbalance to the corresponding modal components of the correction masses. No damping was considered. Since then several researchers, [6–8], have considered refinements to the problem including adding damping, considering flexural free and constrained modes, and obtaining generalized unbalance by least-squares minimization of data errors. More recently, [9], it was proposed to use a numerically obtained unbalance response to simulate trial runs on a computer rather than in the field, while Wiese [10] proposed a method for balancing of paper rolls below their flexible modes, and used the first calculated flexural mode to balance the rotor.

A good contribution to the subject, [11], was recently published and is based on balancing flexible rotors without trial runs through a set of modal equations. The knowledge of the rotor modes is required, and a structured procedure to balance the rotor at the critical speeds is followed. The results were illustrated on a heavily damped test rig. Three main problems are encountered in this analysis. One is that undamped modes were used to model a heavily damped rotor. The second is that the balancing weights were located based on the analysis of the response at resonance. The third problem is the cumbersome application of a technique to account for transducer orientation.

In this paper, the procedure developed in [11] is developed further to account for complex modes. It is shown that using complex modes and complex vibration measurements, solve all three problems mentioned above, and result in a consistent balancing procedure. However, as illustrated, the cornerstone to apply this

technique is to use a calibrated rotor model. The procedure is applied to a test rig at Cairo University. The methods used to calibrate the rotor model are illustrated, and experimental results for balancing several flexible modes without trial runs illustrate the effectiveness of the method.

Theory

In this section, we follow the method described in [11], with two major deviations. One we use complex vibration measurements, and the other we use complex modes. Consider a flexible rotor, then its response can be represented in terms of its mode shapes as

$$v(z, t) = \sum_{r=1}^n v_r(z, t) = \sum_{r=1}^n q_r(t) \cdot \varphi_r(z) \quad (1)$$

where n is the number of modes affecting the rotor response, $q_r(t)$ represents the r th principal coordinate of the system, and $v_r(z, t)$ is the r th component of the complex vibration at the axial coordinate z

$$v_r(z, t) = x_r(z, t) + i y_r(z, t) \quad (2)$$

where $x_r(z, t)$ and $y_r(z, t)$ are the measured vibration in the horizontal and vertical directions respectively. Thus this vibration at axial location (z), represents the orbit at the plane (z) for the r th mode, and it requires two probes 90 deg apart to measure. By measuring the amplitude and the phase of vibration in each direction, $x_r(z, t)$ and $y_r(z, t)$ are represented by complex numbers.

$\varphi_r(z)$ is the r th complex mode shape of the free response of the rotor

$$\varphi_r(z) = \varphi_{xr}(z) + i \varphi_{yr}(z) \quad (3)$$

where $\varphi_{xr}(z)$ and $\varphi_{yr}(z)$ are the horizontal and the vertical components of the mode shapes, respectively. These mode shapes represent the orbital natural rotor deflection, at a particular axial location, including the phase shift effects in each direction. Thus, each component of the mode shape is represented as a complex number.

Similarly, the eccentricity distribution may be expressed as

$$e(z) = \sum_{r=1}^n \varepsilon_r \varphi_r(z) \quad (4)$$

where $\varepsilon_r \varphi_r(z)$ represents the complex modal component of eccentricity. Each one of these modal components excites a single mode of vibration. Multiplying Eq. (4) by $\rho A(z) \varphi_r(z)$, where $A(z)$ is the area at the particular axial position, and integrating along the shaft, then

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30633. Manuscript received by IGTI, Dec. 2001, final revision, Mar. 2002. Associate Editor: E. Benvenuti.

$$\varepsilon_r = \frac{1}{m_r} \int \rho \cdot e(z) A(z) \varphi_r(z) dz. \quad (5)$$

$\varphi_r(z)$ represents the shape taken by the shaft in the r th mode of vibration. This shape is defined as the ratio between the vibration along the length of the shaft and the vibration at the reference, which is usually the measuring plane $z = z_i$. By this normalization the amplitude of the mode shape at the reference plane will be unity with a phase angle of zero. The r th mode shape factor for the axial position $z = z_j$ would be

$$(\varphi_r)_{ji} = \frac{v_r(z_j, t)}{v_r(z_i, t)}. \quad (6)$$

This normalization is essential to transfer the effects of the single mode to the measuring plane.

The modal mass, m_r , represents the equivalent mass of a single degree-of-freedom system, which would generate a response similar to that observed in the r th mode of vibration. This modal mass is calculated from the following equation:

$$m_{ri} = \sum_{j=1}^m m_j [(\varphi_r)_{ji}]^2 \quad (7)$$

where m_j is the physical mass of the rotor station at the axial location $z = z_j$. It is important to mention that this modal mass is complex because it is calculated using the complex $(\varphi_r)_{ji}$ and would represent the equivalent mass for a single degree-of-freedom system.

The rotor can be considered now as a single degree-of-freedom system, so the vibration at the reference plane (measuring plane) v_{ri} produced by the modal unbalance ε_{ri} is

$$v_{ri} = \frac{\Omega_r^2 \varepsilon_{ri}}{(1 - \Omega_r^2) + i(\eta_r \Omega_r)} = A_r(\omega) \varepsilon_{ri} \quad (8)$$

where Ω_r is the ratio between the rotation speed and the r th natural frequency and η_r is the r th loss factor and $A_r(\omega)$ is a nondimensional amplification factor.

A balancing mass U_{rj} is installed to compensate for the vibration given in Eq. (8), such that

$$U_{rj} \cdot R_j \cdot (\varphi_r)_{ji} = -m_{ri} \cdot \varepsilon_{ri} \quad (9)$$

where U_{rj} is a complex number representing the balancing mass magnitude and phase, m_{ri} represents the r th mode modal mass as seen by an observer located at the measuring plane, and R_j is the radius at which the balancing mass is located at the balancing plane. By substituting into Eq. (9) from Eq. (8), then

$$U_{rj} = -\frac{m_{ri} \cdot \varepsilon_{ri}}{R_j \cdot (\varphi_r)_{ji}} = -\left(\frac{m_{ri}}{R_j \cdot A_r(\omega) \cdot (\varphi_r)_{ji}} \right) \cdot v_{ri}. \quad (10)$$

The term between parentheses is known as the sensitivity factor. It is the reciprocal of the influence coefficient. It describes the relation between the unbalance and the response. Trial runs are usually used to evaluate this term. However, trial runs can be avoided if there is another method to calculate this term. In this paper, a calibrated rotor model is used to calculate this term and no trial runs are needed. The negative sign considers that the correction mass has to be attached in antiphase with the unbalance force.

For modal balancing, the rotation speed is considered equal to the rotor natural frequency, in other words substitute by $\Omega = 1$ in Eq. (10), considering Eq. (8), then

$$U_{ri} = -\left(\frac{i \eta_r \cdot m_{ri}}{R_j \cdot (\varphi_r)_{ji}} \right) \cdot v_{ri}. \quad (11)$$

Usage of a single correction mass excites all modes of vibration. To avoid this effect the single correction mass has to be transformed into a set of masses, which have the same effect

without affecting other modes. This set of correction masses can be calculated using the following system of equations, [11],

$$\sum_{j=1}^n U_r(z_j) \cdot R_j \cdot (\varphi_s)_{ji} = \begin{cases} i \eta_r \cdot m_{ri} \cdot v_{ri} & \text{for } r = s \\ 0 & \text{for } r \neq s \end{cases} \quad (12)$$

where $U_r(z_j)$ is the correction mass required at the balancing plane (z_j), which forms part of the modal mass set required to compensate the unbalance in the r th mode. The number of the correction mass elements n should be equal to the number of balancing planes, which in turn should be equal to the number of vibration modes taken into consideration.

The above theory necessitates the measurement of the orbit at a single plane, and the complete knowledge of the complex orbital mode shapes of the rotor. In this regard, although the derivation is similar to that presented in [11], the above theory is based on completely different assumptions and should be considered fundamentally different. Actually, by using complex vibration measurement, the measured rotor orbit is considered, thus the artificial and cumbersome technique used to account for transducer orientation in [11] is eliminated. Moreover, by using complex mode shapes, damped modes are considered, thus the current analysis is valid both for damped and undamped systems. Finally, since we are considering orbital displacements and orbital mode shapes, then all quantities considered in Eqs. (12), the balancing equations, are phasors, with the result that not only the magnitude of the balancing weight is obtained from Eqs. (12), as in [11], but also the location of the balancing weight. This is very important, since the technique proposed in [11] for locating the balancing weights by considering the phase shift between the force and vibration by 90 deg, would not be easily applied for higher or cumbersome mode shapes.

Test Rig

To verify the applicability of the above theory, a test rig was designed and built. The test rig used in this work is as shown in Fig. 1. It was designed to have two critical speeds in a speed range up to 8000 rpm. The rotor is supported on two rolling element bearings, which provide little damping to the system. The rotor has total length of 1.1 m, main diameter of 31 mm and has 5 equal disks of mass 7 kg each, and is directly coupled to a 2 HP variable speed motor.

Rotor Modeling

The rotor was modeled by a finite element program RIMAP, [12], using the rotor dimensions and a preliminary estimation of the support stiffness. The stiffness that has been used in this preliminary model was 5e7 N/m in both the horizontal and vertical directions. Figure 2 shows the rotor model and mode shapes with the corresponding critical speeds, at 1689 rpm and 6714 rpm, while Fig. 3 shows the calculated unbalance response near the nondrive end support, with the unbalance on the second and fourth disks. To verify the rotor model a coastdown test was performed, as shown in Fig. 4(a), while the polar plot for the first mode is shown in Fig. 4(b). These coastdown data are obtained by proximity probe measurements, with data filtered by a Bently Nevada DVF3 tracking filter, and data collected through IEEE 488 interface to a Pentium II computer with data plotted, and analyzed by LabView software.

It can be seen from Fig. 4(a) that, two critical speeds were passed in this test. This coastdown test shows that the actual values of the critical speeds are different from those calculated by the preliminary rotor model. They were measured to be 1674 rpm and 6102 rpm. Moreover, it can be seen clearly in Fig. 4(b) that the first critical is a split critical, which means that the rotor support is asymmetric, or in other words, the stiffness in the horizontal and vertical directions are not equal. This can also be seen by closer inspection of the data in Fig. 4(a) near the critical speed.

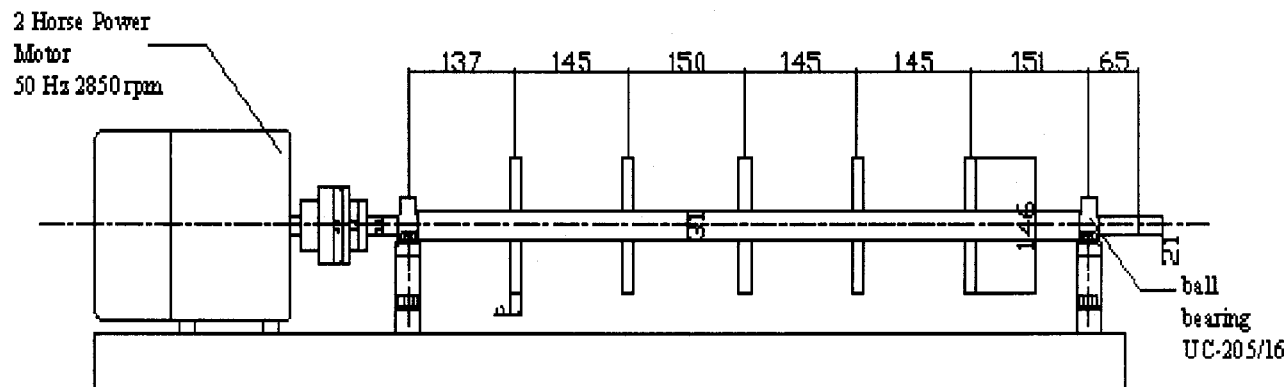


Fig. 1 Test rig

Several values of stiffness in both directions were introduced to the model in order to get the same critical speeds from the model as those obtained from the experimental run. The best results were obtained when the stiffness $2.5e7$ and $5e8$ N/m in the horizontal and vertical directions, respectively.

The hardest parameter to estimate for rotor modeling is the amount of damping in the system. The polar plot of Fig. 4(b), measured on the test rig is a good indicator of the damping in the rotor, and can be used in the model. Several methods are available

in the literature, [13], to estimate the damping from the polar plot. A perceived damping value was estimated from the experimentally measured polar plots, Fig. 5, to obtain consistent damping values in both modes. These values are used to obtain the calibrated rotor model of Fig. 6.

Model Verification

In order to verify the model, the vibration was measured at the drive end side in addition to the measurements taken at the non-drive end side. The polar plot was plotted for each measurement as shown in Fig. 5. It can be seen that the rotor behaves closely to that predicted by the mode shapes shown in Fig. 2. The measured vibration on the drive end and nondrive end are in phase in the first mode, and are 180 deg out-phase in the second mode, as the polar plots of Fig. 5 illustrate.

Comparing the amplitudes of the calibrated rotor model Fig. 6, and the measured coastdown Fig. 4(a), it becomes clear that the amplitudes at resonance are not similar. There can be several reasons for this including that the amplitudes and distribution of unbalance to obtain the computed response in Fig. 6 is totally assumed. Thus its level of excitation for each mode is not quite consistent with the unbalance distribution in the rotor. Another factor that affects the amplitude response at resonance is the damping in the calibrated model. However, since the damping

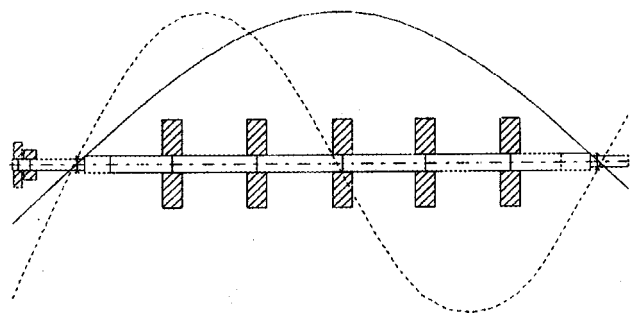


Fig. 2 Calculated mode shapes

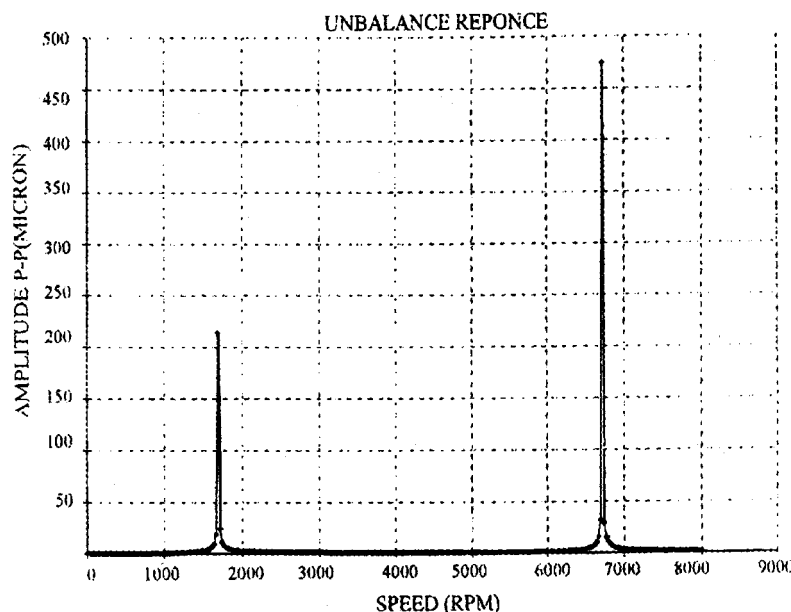
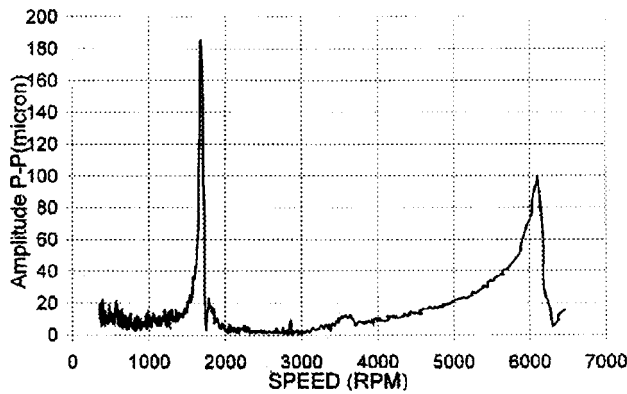
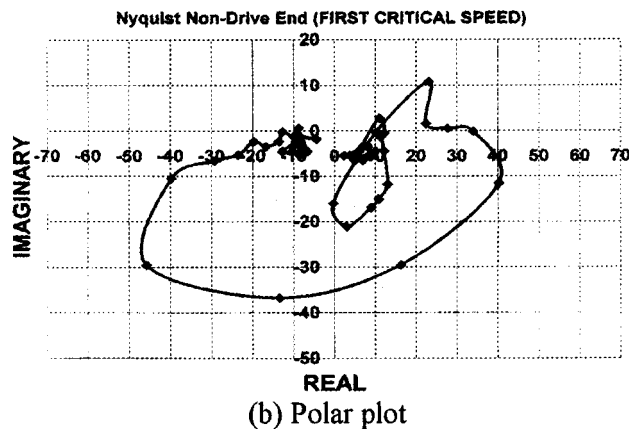


Fig. 3 Bode plot produced by the preliminary model



(a) Measured Bode plot



(b) Polar plot

Fig. 4 Coastdown test

used in the model is estimated from the experimental data, we are confident that the differences of the amplitudes between Figs. 4(a) and 6 are caused by the values and distribution of unbalance in the model. Instead of considering a parametric study that would assume an unbalance distribution producing exactly equal results, which would necessarily constitute a balancing process, we were content with the current calibrated model of Fig. 6, as our final model, since it appears to be producing modal data consistent with our measurements. The amplitude differences can actually be accommodated during the final balancing process.

From all these tests, it can be seen that the model produces results that are very close to the performance of the real rotor, but there are still some deviations in the magnitudes between the experimental and the calculated data.

Although there are more advanced sophisticated techniques for measuring rotor properties based on modal testing, [14], yet it was deemed that the current model is validated with the limited testing presented herein. It can be considered now that the rotor model is acceptable and can be used to calculate the modal parameters needed in Eq. (12), even though there are some amplitude deviations at resonance.

Modal Parameters Estimation

In order to estimate the modal parameters needed in Eq. (12) the complex mode shape for the calibrated model was calculated along the rotor length. The developed calibrated model produces the rotor mode shapes in both the horizontal and vertical directions as complex numbers. These mode shapes were then used to calculate the complex mode shapes ($\phi_r(z)$) according to Eq. (3). These mode shapes were then normalized according to Eq. (6), taking the measuring plane as the reference plane. These normalized complex mode shapes were also used to calculate the complex modal mass (m_{ri}) according to Eq. (7).

The modal loss factor (η_r) was experimentally determined us-

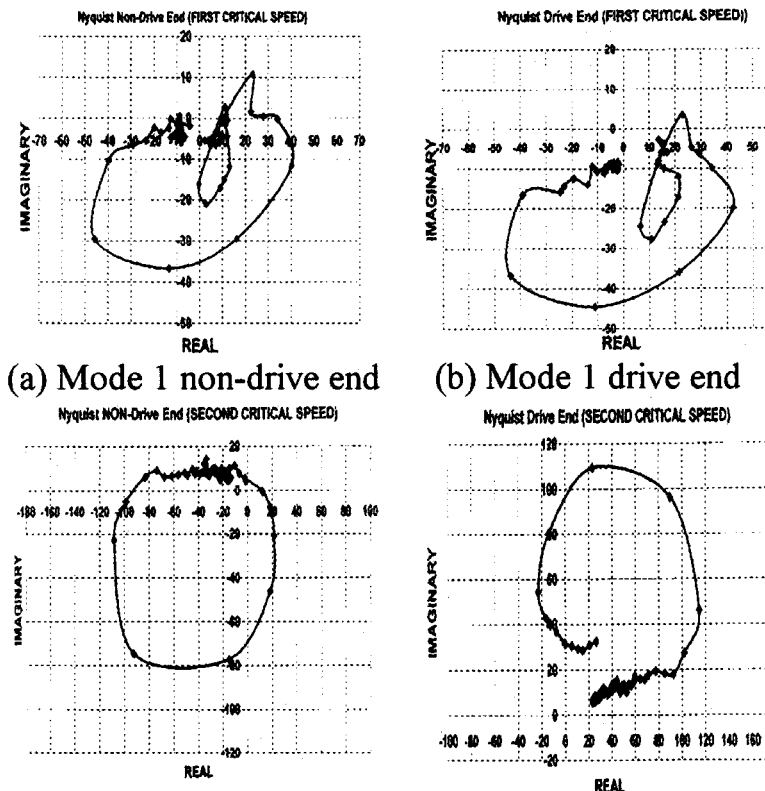


Fig. 5 Measured polar plots

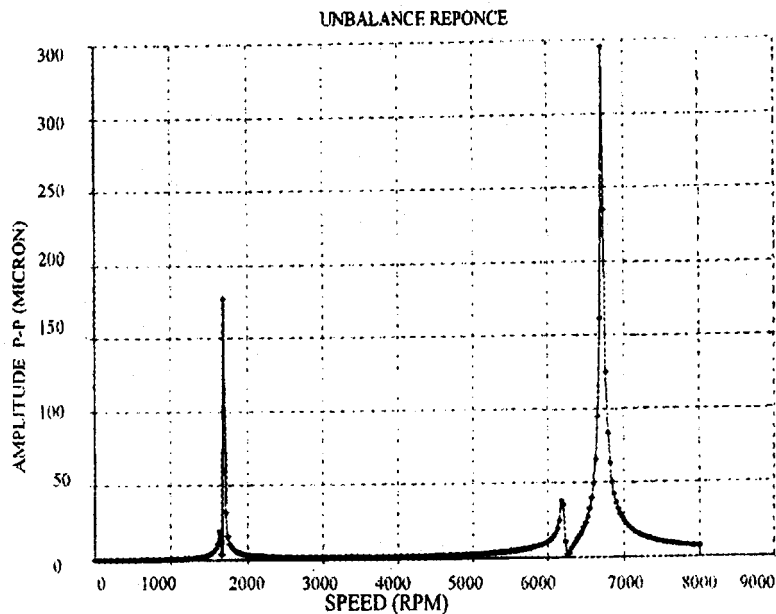


Fig. 6 Bode plot calculated from the calibrated model

ing the properties of the modal circle. This method of calculating the modal loss factor depends on plotting the Nyquist plot of the rotor response around the critical speeds, Fig. 5.

This plot was then fitted with a circle, [13]. In this case, the modal loss factor is a function of the rate of change of the sweeping angle around the circle center with respect to the change of running speed. The modal loss factors were determined to be (0.023) and (0.0174) for the first and second modes, respectively.

Results

The procedure to balance the rotor is now quite clear. A calibrated rotor model has to be developed. Actually most of the work is done in obtaining a calibrated rotor model. Next based on the calibrated rotor model, calculated mode shape data in the complex form of Eq. (3) are tabulated and normalized according to Eq. (6). Next the complex modal mass is calculated from the calibrated model according to Eq. (7). The loss factor is then determined from the polar plot of the coastdown test for each of the modes. Finally the measured vibration at the critical speeds, in orbital form Eq. (2), and the above calculated and measured modal data, are inserted into Eqs. (12) and used to calculate the balancing weights directly.

For the test rig shown in Fig. 1, the vibration was measured in the horizontal and vertical directions at the same axial position between the nondrive end bearing and the last disk. The amplitude

and phase of the vibration in each direction were measured, which means that the measurements in each direction were represented as complex numbers. The horizontal and vertical measurements were combined in one complex number as shown in Eq. (2). The same measurements were repeated at the drive end, and the results were consistent.

In order to balance these two modes, the two balancing planes were chosen at the second and fourth disks referenced from the drive-end side. By substituting in the system of Eqs. (12), two systems of equations (one for each mode) are solved to obtain the magnitude and phase of the correction masses required to balance the concerned modes. The correction masses results are complex numbers, which can be transformed into magnitude and angular location. The result in correction masses for the first mode were $2.3\text{e-}3$ kg (2.3 gm) at 92 deg and $2.2\text{e-}3$ kg (2.2 gm) at 92 deg at the second and fourth disks, respectively, and for the second mode $1.2\text{e-}3$ kg (1.2 gm) at 118 deg and $1.2\text{e-}3$ kg (1.2 gm) at -61 deg at the second and fourth disks, respectively. It was judged that these magnitudes of the correction masses were not quite suitable for this test rig. Recall that the amplitude response in the model calibration was not adequately matched. Taking a conservative approach, it was decided to reduce the magnitude of the balancing weights, but keeping the phase locations as calculated. This is common practice in the field, where conservative weight estimates

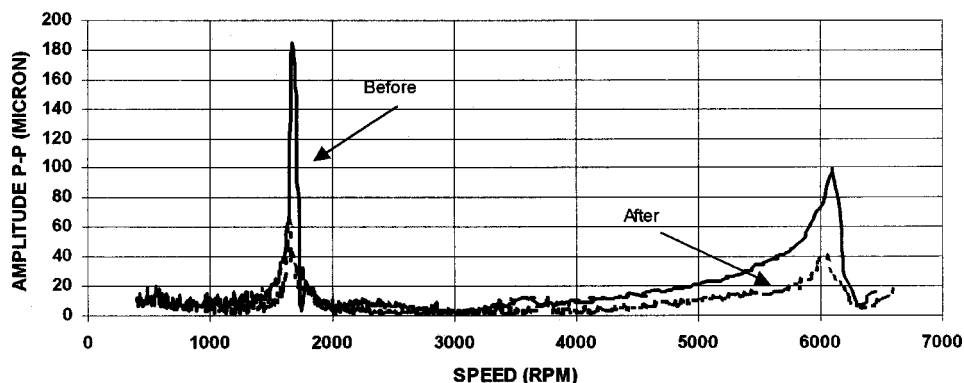


Fig. 7 Vibration measurements before and after balancing

are used before committing to the larger weights. If the conservative weights succeed in balancing the machine, the balancing effort is deemed successful.

The actual installed correction masses for the first mode were 0.6e-3 kg (0.6 gm) at 92 deg and 0.6e-3 kg (0.6 gm) at 92 deg at the second and fourth disks, respectively, and for the second mode were 1e-3 kg (1.0 gm) at 118 deg and 1e-3 kg (1.0 gm) at -61 deg at the second and fourth disks, respectively. The coast down test done after installing the correction masses is shown in Fig. 7. Comparing the coast down after balancing with that before balancing, it can be seen that the rotor is well balanced. The vibration levels away from resonance are within 20 micrometers (less than 1 mil), and at the first critical speed the level is around 40 micrometers. Considering that this rotor has very little damping, the attained vibration levels at the critical speeds are quite remarkable.

Discussion and Conclusion

In this paper, a conceptually different method for rotor balancing without trial runs is presented. The method is based on the use of complex rotor vibration measurement and complex mode shapes computed from a calibrated model. The method was quite successful in modally balancing a flexible rotor in a test rig. The method requires that a coast down test be measured on one bearing in both the horizontal and vertical direction and also in both magnitude and phase. These complex data are then assembled, with a calibrated rotor model, to obtain directly the balancing weights and locations.

The advantages of this technique are clear, particularly regarding the time and cost reduction in performing the balancing. In the test rig described in the paper, balancing of two modes was done by a single measurement, while in practice this machine would have required, at best, three different measurements. These numbers can be even more drastic for rotors requiring balancing in more planes. It can be argued that the time and effort saved by this balancing method, is actually spent in calibrating the rotor model. That is, the usual trial weights, which were used to calibrate the influence coefficients of the machine, require the same effort as calibrating a rotor model. Even though this argument is conceptually correct, yet it is incomplete, because a calibrated rotor requires more office work than field work, i.e., only the single coast-down test was necessary for us to calibrate the rotor model.

Moreover, a calibrated rotor model is a necessity rather than a luxury. We urge turbomachinery manufacturers to supply calibrated rotor models with their machines. Not only will this calibrated model be useful to balance the rotor, but also it can be used as a good indicator of the machine condition. A deviation in the response of the machine from that obtained from the calibrated model would indicate deterioration in machine condition and a fault initiation.

Several points need to be highlighted. First, this method assumes that the rotor can be run above the critical speeds within its speed range, whose corresponding modes need to be balanced. In some cases, this would not be possible, either due to the balance condition of the rotor or due to the lack of damping. It is then advised to balance the rotor using the usual influence coefficient or modal balancing methods, then trim balancing could be performed with the method outlined in this paper. Any future balancing of the rotor can then also be balanced according to the method outlined in this paper.

The second point is regarding calibrating the rotor model. We believe that the method outlined here for rotor modeling is fea-

sible in the field, while more advanced modal based methods, [14], would be difficult to apply. However, the proposed method requires a complex rotor dynamic model that would be somewhat difficult to calibrate. In general, the rotor presents no difficulties in modeling but the bearing and support stiffness and damping may need further testing to better estimate their properties. We have used simple well-known techniques to estimate the support stiffness and damping, but in some cases it may be necessary to apply local modal testing to estimate the bearing and support properties. Actually, calibrating models should be the subject of further research to develop a robust methodology for rotor model calibration.

Third the influence coefficient matrix obtained numerically or by testing can be used to calibrate a rotor model and to evaluate deviations in machine condition. However, this is also a subject for future research, i.e., calibrating rotor models using influence coefficients and their subsequent application in machine condition monitoring.

This leaves us with future work to be done. As outlined above, a robust methodology for rotor model calibration, including perhaps local modal testing and/or influence coefficient calibration would be required. As for extensions to the current paper, we believe that the method needs to be applied in other testing environments, particularly on rotors with fluid film bearings, as well as on actual field machinery.

Finally, to summarize the paper, the presented balancing technique, which requires the measurement of the orbital vibration, and the knowledge of the orbital natural modes, is tested on a test rig with two flexible modes with very little damping. The method was quite effective in locating the balancing weights to reduce the vibration levels of the test rig, and obtain quite satisfactory response at the critical speeds. This clearly illustrates the strength of the proposed method.

References

- [1] Thearle, E. L., 1934, "Dynamic Balancing of Rotating Machinery in the Field," *Trans. ASME*, **56**, pp. 745-753.
- [2] Goodman, T. P., 1964, "A Least-Squares Method for Computing Balance Corrections," *J. Eng. Ind.*, **86**(3), pp. 273-279.
- [3] Bishop, R. E. D., and Parkinson, A. G., 1963, "On the Isolation of Modes in the Balancing of Flexible Shafts," *Proc. Inst. Mech. Eng.*, **177**, pp. 407-423.
- [4] El-Shafei, A., 2001, "Static-Couple Balancing of Overhung Rotors," *Vibrations*, **17**(1), pp. 6-11.
- [5] Hundal, M. S., and Harker, R. J., 1966, "Balancing of Flexible Rotors Having Arbitrary Mass and Stiffness Distributions," *J. Eng. Ind.*, **88**, pp. 217-233.
- [6] Palazzolo, A. B., and Gunter, E. J., 1977, "Modal Balancing Without Trial Weights by a Modified Nyquist Plot Procedure," *Vibration Institute, Machinery Vibration Seminar*, Cherry Hill, NJ, Nov.
- [7] Gash, R., and Drechsler, J., 1978, "Modales Auswucher Elastischer Laufer Ohne Testgewichtssetzungen," *Colloid Sci.*, **(320)**, pp. 45-54.
- [8] Morton, P. G., 1985, "Modal Balancing of Flexible Shafts Without Trial Weights," *Proc. Inst. Mech. Eng.*, **199**(C1), pp. 71-78.
- [9] Meacham, W. L., Brawley, G. H., and Molis, S. J., 1986, "Balancing of Large Steam Turbine-Generator Using MoBal," *EPRI Rotating Machinery Dynamics, Bearings and Seals Symposium*, St. Louis, MO, Sept. 9-11.
- [10] Wiese, D., 1992, "Two New Procedures to Balance Flexible Rotors Without Test Runs," *Proceedings of the 5th International Conference on Vibrations in Rotating Machinery*, Bath, England, IMechE, London, pp. 557-568.
- [11] Preciado, E., and Bannister, R. H., 2000, "On the Balancing of Flexible Rotors Without Trial Runs," *Proceedings of the 7th International Conference on Vibrations in Rotating Machinery*, Nottingham, England, IMechE, London, pp. 121-130.
- [12] *RIMAP User Guide*, 2000, Version 1.2, RITEC, Egypt.
- [13] Ewins, D. J., 1986, *Modal Testing: Theory and Practice*, Research Studies Press Ltd.
- [14] Kreuzinger-Janik, T., and Irretier, H., 2000, "Unbalance Identification of Flexible Rotors Based on Experimental Modal Analysis," *Proceedings of the 7th International Conference on Vibrations in Rotating Machinery*, Nottingham, England, IMechE, London, pp. 335-346.

E. Capiez-Lernout

Graduate Student
e-mail: capiez@univ-mlv.fr

C. Soize

Professor
e-mail: soize@univ-mlv.fr

Laboratory of Engineering Mechanics,
University of Marne-La-Vallée,
5, Bd Descartes,
77454 Marne-La Vallée Cedex 02, France

Nonparametric Modeling of Random Uncertainties for Dynamic Response of Mistuned Bladed Disks

The random character of blade mistuning is a motivation to construct probability models of random uncertainties. Recently, a new approach known as a nonparametric model of random uncertainties, based on the entropy optimization principle, was introduced for modeling random uncertainties in linear and nonlinear elastodynamics. This paper presents an extension of this nonparametric model for vibration analysis of structures with cyclic geometry. In particular this probability model allows the blade eigenfrequencies uncertainties and the blade-modal-shape uncertainties to be modeled.

[DOI: 10.1115/1.1760527]

1 Introduction

Structural dynamics of mistuned bladed disks remain a difficult problem in turbomachinery. Usually, vibration analysis of cyclic structures is performed using their cyclic symmetry and is formulated for one sector from which the dynamics of the structure is reconstructed (see, for instance, [1,2]). Nevertheless, manufacturing tolerances and dispersion of materials create uncertainties on geometry, boundary conditions and material properties of each blade. In the present paper, it is assumed that the disk is not mistuned. It is known that the phenomenon related to mistuning can induce strong vibrations for the forced response of the bladed disk and can produce spatial localization in the dynamic response of the blades as proved in the early works, [3,4]. A good understanding of the mistuning effects still remains a subject of interest. Mistuning has been the object of various researches these last decades and many numerical studies have been developed on both free and forced responses. The influence of several parameters as interblade coupling, strength mistuning or damping has also been explored, [5–7]. Moreover, the random nature of blade mistuning has been a motivation to construct probability models of random uncertainties to solve the corresponding stochastic responses and to perform statistical analyses in order to predict the effects of mistuning. Several researches have also been carried out to accurately estimate probability responses related to mistuning by using perturbation techniques with lumped parameter models, [8,9]. Reduced-order models have been extensively studied to reduce time costs for mistuned industrial turbomachinery bladed disks, [10–12]. In the context of the blade mistuning, a probabilistic parametric approach requires a complete probabilistic parametric description of the uncertain parameters, such as the use of stochastic fields for modeling uncertainties of constitutive equations, boundary conditions, and geometry. Such a probabilistic parametric method is not really simple to develop because of the large number of parameters which induce mistuning and which has to be identified. A good compromise widely used in most studies dealing with this subject, is to consider the Young modulus of blades as uncertain variables. In this case, blade eigenfrequencies are mistuned through the stiffness parameters but the corresponding eigenmodes remain unchanged with respect to the tuned case. Recently, Mignolet et al. [13,14] introduced a probability model

which is experimentally identified using the maximum likelihood principle in order to include blade-modal-shape mistuning or not. This model exhibits differences with respect to the parametric probability model currently used. Since the random character of blade mistuning is a motivation to construct probability models of random uncertainties, the present goal of this paper is to present an alternative nonparametric probability model to the usual parametric one for modeling random uncertainties in mistuning analysis. This nonparametric probabilistic approach allows both data uncertainties and model uncertainties to be taken into account while probabilistic parametric approaches do not allow model uncertainties to be modeled. Such a nonparametric approach has been recently introduced in linear elastodynamics, in time and frequency domain, [15,16]. This nonparametric probability model is directly constructed using the mean reduced matrix model, and using a probability model for symmetric positive-definite real random matrices, which is constructed using the entropy maximization principle with the available information. In Section 2, the mean reduced matrix model required by the nonparametric approach is presented. Section 3 is devoted to the nonparametric model of random uncertainties. Section 4 describes the strategy for mistuning analysis. In Section 5, a numerical example is given. In particular, a numerical comparison is carried out in order to give an information when the dispersion levels of the random matrices issued from the nonparametric and from the usual parametric approach are similar.

2 Construction of the Mean Reduced Matrix Model

Let Ω be a fixed structure with an N -order cyclic symmetry, submitted to external forces (the problem under consideration is related to rotating structures; in order to simplify the presentation, a fixed structure is considered; the methodology and the results presented can be extended without difficulty to the case of rotating structures). The geometrical domain, the viscoelasticity tensor, the mass density and the Dirichlet conditions related to the generating sector are invariant under the $2\pi/N$ rotation around the rotational axis of symmetry. Consequently, we are then interested in the linearized vibrations around a static equilibrium configuration considered as a natural state without prestresses whose model will be called the mean model of the structure. The vibration analysis is performed in the frequency band B . Structure Ω is divided into one disk Ω_d and N identical blades Ω_j , for j belonging to $\{0, \dots, N-1\}$. Let Σ_j be the coupling interface between blade Ω_j and disk Ω_d . It is assumed that the blades are only coupled by the disk. The disk is fixed on a part of its boundary. The mistuning is

Contributed by the Structures and Dynamics Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the S&D Division June 2002; final revision received December 2003. Associate Editor: M. Mignolet.

assumed to be statistically independent from blade to blade. Consequently, the nonparametric probability model of random uncertainties for a blade is described independently of the other blades. Furthermore, the probabilistic nonparametric approach requires the construction of a mean reduced matrix model for each uncertain substructure. This justifies the use of a dynamic substructuring method. The Craig and Bampton method, [17], is used in this paper but any other substructuring methods could be used. Then, the mean finite element model of the disk is assembled with the mean reduced model of each blade. The resulting mean matrix model will be called the mean reduced matrix model for the bladed disk. It should be outlined that the purpose of this paper is not to construct the most efficient reduced-order model adapted to the mistuning problematic. This has been thoroughly studied in the literature. The focus of this paper deals with the model of mistuning by using a nonparametric probabilistic approach. Thus, the construction of this mean reduced model for each blade is only justified by the use of a nonparametric probability model for describing the blade random uncertainties. For ω fixed in frequency band B, let $[A^d(\omega)]$ be the mean dynamic stiffness matrix of disk Ω_d with free coupling interface. Let n_d be the number of degrees-of-freedom of the disk. For the disk, the block decomposition with respect to the internal degrees-of-freedom and with respect to the coupling interface degrees-of-freedom yields

$$[A^d(\omega)] = \begin{bmatrix} [A_{ii}^d(\omega)] & [A_{i\Sigma}^d(\omega)] \\ [A_{i\Sigma}^d(\omega)]^T & [A_{\Sigma\Sigma}^d(\omega)] \end{bmatrix}. \quad (1)$$

Each blade j is reduced by the Craig and Bampton decomposition method, [17], with N_g fixed coupling interface modes. For ω fixed in frequency band B, let $[A_{\text{red}}^j(\omega)]$ be the mean reduced dynamic stiffness matrix of blade Ω_j with free coupling interface. Matrix $[A_{\text{red}}^j(\omega)]$ is defined by

$$[A_{\text{red}}^j(\omega)] = -\omega^2[M_{\text{red}}^j] + i\omega[D_{\text{red}}^j] + [K_{\text{red}}^j], \quad (2)$$

and its block decomposition with respect to the N_g generalized coordinates $\mathbf{q}^j$ and to the n_Σ coupling interface degrees-of-freedom is written as

$$[A_{\text{red}}^j(\omega)] = \begin{bmatrix} [\mathcal{A}^j(\omega)] & [A_c^j(\omega)] \\ [A_c^j(\omega)]^T & [A_\Sigma^j(\omega)] \end{bmatrix}. \quad (3)$$

Let $N_a = N_g + n_\Sigma$. The matrices $[M_{\text{red}}^j] \in \mathbb{M}_{N_a}^+(\mathbb{R})$, $[D_{\text{red}}^j] \in \mathbb{M}_{N_a}^+(\mathbb{R})$ and $[K_{\text{red}}^j] \in \mathbb{M}_{N_a}^+(\mathbb{R})$ are the mean reduced mass, damping and stiffness matrices respectively. The mean vector of the physical degrees-of-freedom is then given by

$$\begin{bmatrix} \mathbf{u}_i^j(\omega) \\ \mathbf{u}_\Sigma^j(\omega) \end{bmatrix} = [H^j] \begin{bmatrix} \mathbf{q}^j(\omega) \\ \mathbf{u}_\Sigma^j(\omega) \end{bmatrix}, \quad (4)$$

in which, for blade j , $[H^j]$ is the transformation matrix involved by the Craig and Bampton decomposition, [17], $\mathbf{u}_i^j(\omega)$ is the mean vector of the internal degrees-of-freedom and $\mathbf{u}_\Sigma^j(\omega)$ is the mean vector of the coupling interface degrees-of-freedom. The mean reduced matrix model of the bladed disk is then obtained by assembly which yields

$$\begin{bmatrix} [\mathcal{A}(\omega)] & [A_c(\omega)] & [0] \\ [A_c(\omega)]^T & [A_\Sigma(\omega)] + [A_{\Sigma\Sigma}^d(\omega)] & [A_{i\Sigma}^d(\omega)]^T \\ [0] & [A_{i\Sigma}^d(\omega)] & [A_{ii}^d(\omega)] \end{bmatrix} \begin{bmatrix} \mathbf{q}(\omega) \\ \mathbf{u}_\Sigma(\omega) \\ \mathbf{v}_i(\omega) \end{bmatrix} = \begin{bmatrix} \mathcal{F}(\omega) \\ \mathbf{f}_\Sigma(\omega) \\ \mathbf{g}_i(\omega) \end{bmatrix}, \quad (5)$$

where $\mathbf{q}(\omega)$ is the mean vector in $\mathbb{C}^{NN_g}$ defined by $\mathbf{q}(\omega) = \{\mathbf{q}^0(\omega), \dots, \mathbf{q}^{N-1}(\omega)\}$, where $\mathbf{u}_\Sigma(\omega)$ is the mean vector in $\mathbb{C}^{Nn_\Sigma}$ defined by $\mathbf{u}_\Sigma(\omega) = \{\mathbf{u}_\Sigma^0(\omega), \dots, \mathbf{u}_\Sigma^{N-1}(\omega)\}$ and where

$\mathbf{v}_i(\omega)$ is the mean vector of the $\mathbb{C}^{n_d - Nn_\Sigma}$ describing the internal degrees-of-freedom of the disk. We have $[\mathcal{A}(\omega)]^{jk} = [\mathcal{A}^j(\omega)]\delta_{jk}$, $[A_c(\omega)]^{jk} = [A_c^j(\omega)]\delta_{jk}$ and $[A_\Sigma(\omega)]^{jk} = [A_\Sigma^j(\omega)]\delta_{jk}$, where the upper subscript jk denotes the matrix block defining the connection between blade j and blade k . The right side of Eq. (5) defines the terms related to the mean external forces.

3 Nonparametric Model of Random Uncertainties for Blade Mistuning

In this section, the nonparametric model of random uncertainties related to the blade mistuning is constructed. The main results concerning the nonparametric probability model of random uncertainties in structural dynamics, [15,16], are used. Such a probability model is implemented from the mean reduced matrix model of each blade.

3.1 Random Reduced Matrix Model for the Bladed-Disk

Using the methodology derived from [15,18], the nonparametric probabilistic approach consists in modeling the reduced dynamic stiffness matrix for blade j as the random matrix

$$[A_{\text{red}}^j(\omega)] = -\omega^2[\mathbf{M}_{\text{red}}^j] + i\omega[\mathbf{D}_{\text{red}}^j] + [\mathbf{K}_{\text{red}}^j], \quad (6)$$

in which $[\mathbf{M}_{\text{red}}^j]$, $[\mathbf{D}_{\text{red}}^j]$, and $[\mathbf{K}_{\text{red}}^j]$ are $N_a \times N_a$ independent random matrices corresponding to the random reduced mass, damping, and stiffness matrices of blade j . The results and notation of Sec. 2 are transposed for the random case. We then deduce that

$$[A_{\text{red}}^j(\omega)] = \begin{bmatrix} [\mathcal{A}^j(\omega)] & [A_c^j(\omega)] \\ [A_c^j(\omega)]^T & [A_\Sigma^j(\omega)] \end{bmatrix}, \quad (7)$$

and the random stochastic system corresponding to Eq. (5) is written as

$$\begin{bmatrix} [\mathcal{A}(\omega)] & [A_c(\omega)] & [0] \\ [A_c(\omega)]^T & [A_\Sigma(\omega)] + [A_{\Sigma\Sigma}^d(\omega)] & [A_{i\Sigma}^d(\omega)]^T \\ [0] & [A_{i\Sigma}^d(\omega)] & [A_{ii}^d(\omega)] \end{bmatrix} \begin{bmatrix} \mathbf{Q}(\omega) \\ \mathbf{U}_\Sigma(\omega) \\ \mathbf{V}_i(\omega) \end{bmatrix} = \begin{bmatrix} \mathcal{F}(\omega) \\ \mathbf{f}_\Sigma(\omega) \\ \mathbf{g}_i(\omega) \end{bmatrix}, \quad (8)$$

in which $\mathbf{Q}(\omega)$, $\mathbf{U}_\Sigma(\omega)$ and $\mathbf{V}_i(\omega)$ are the $\mathbb{C}^{NN_g}$ -valued random vector of the generalized coordinates, the $\mathbb{C}^{Nn_\Sigma}$ -valued random vector of the degrees-of-freedom of the coupling interface and the $\mathbb{C}^{n_d - Nn_\Sigma}$ random vector of the internal degrees-of-freedom of the disk respectively.

3.2 Probability Model of the Random Matrices for One Blade.

Since the random uncertainties are statistically independent from blade to blade, random matrices $[A_{\text{red}}^0(\omega)], \dots, [A_{\text{red}}^{N-1}(\omega)]$ are statistically independent. We can limit the construction of the probability model to one blade. Then, we have to construct the probability model of the random matrices $[\mathbf{M}_{\text{red}}^j]$, $[\mathbf{D}_{\text{red}}^j]$, and $[\mathbf{K}_{\text{red}}^j]$. Since each random matrix involves both generalized coordinates and physical degrees-of-freedom which are of different nature, it is necessary to construct the probability model on a matrix related to normalized quantities. A normalization of the random reduced matrix has to be done before constructing the probability model. In addition, the probability model has to be consistent with a mechanical problem and has to satisfy the following constraints which constitute the only available information: C1 the mean reduced matrix is equal to the mean value of the random reduced matrix; C2 the signature of the random reduced matrix is respected: it means that the random reduced matrix has to be positive definite (or semi-positive definite) if its corresponding mean reduced matrix is positive definite (or semi-positive definite); C3 the second-order moment of the

physical random response of the bladed disk has to exist. The probability model is then derived from these three constraints by using the entropy maximization principle. It should be outlined that constraints C2 and C3 do not allow the probability distribution to be independently constructed for each entry of the random matrices under consideration. Only a global approach can be used to construct the probability distribution of such random matrices. In particular constraint C2 need to construct the probability density distribution on the set of all the symmetric positive-definite matrices.

3.2.1 Normalization of Random Matrices $[\mathbf{M}_{\text{red}}^j]$, $[\mathbf{D}_{\text{red}}^j]$ and $[\mathbf{K}_{\text{red}}^j]$. Constraint C1 yields

$$\mathcal{E}\{[\mathbf{M}_{\text{red}}^j]\}=[\mathbf{M}_{\text{red}}^j], \quad \mathcal{E}\{[\mathbf{D}_{\text{red}}^j]\}=[\mathbf{D}_{\text{red}}^j], \quad \mathcal{E}\{[\mathbf{K}_{\text{red}}^j]\}=[\mathbf{K}_{\text{red}}^j], \quad (9)$$

The normalization of random matrices $[\mathbf{M}_{\text{red}}^j]$, $[\mathbf{D}_{\text{red}}^j]$, and $[\mathbf{K}_{\text{red}}^j]$ is constructed for that the mean value of each normalized random matrix is the unity matrix. Such a construction requires the factorization of the mean matrices. Since the blade with free coupling interface is considered, mean reduced mass matrix $[\mathbf{M}_{\text{red}}^j]$ belongs to $\mathbb{M}_{N_a}^+(\mathbb{R})$ while mean reduced stiffness and mean damping matrices $[\mathbf{K}_{\text{red}}^j]$ and $[\mathbf{D}_{\text{red}}^j]$ belong to $\mathbb{M}_{N_a}^0(\mathbb{R})$ due to rigid-body modes. The rank of matrices $[\mathbf{K}_{\text{red}}^j]$ and $[\mathbf{D}_{\text{red}}^j]$ is noted N_a^+ . The Cholesky factorization of matrix $[\mathbf{M}_{\text{red}}^j]$ is

$$[\mathbf{M}_{\text{red}}^j]=[\mathbf{L}_M^j]^T[\mathbf{L}_M^j], \quad (10)$$

in which $[\mathbf{L}_M^j]$ is a real $N_a \times N_a$ upper triangular matrix. Since matrix $[\mathbf{K}_{\text{red}}^j]$ is semi-positive definite and block diagonal, we have

$$[\mathbf{K}_{\text{red}}^j]=[\mathbf{L}_K^j]^T[\mathbf{L}_K^j], \quad (11)$$

in which $[\mathbf{L}_K^j]$ is a $N_a^+ \times N_a$ real diagonal matrix. Since matrix $[\mathbf{D}_{\text{red}}^j]$ is semi-positive definite, we have

$$[\mathbf{D}_{\text{red}}^j]=[\mathbf{L}_D^j]^T[\mathbf{L}_D^j], \quad (12)$$

in which $[\mathbf{L}_D^j]$ is a $N_a^+ \times N_a$ real matrix obtained by solving the spectral problem related to matrix $[\mathbf{D}_{\text{red}}^j]$. Each random matrix is then written as

$$[\mathbf{M}_{\text{red}}^j]=[\mathbf{L}_M^j]^T[\mathbf{G}_M^j][\mathbf{L}_M^j], \quad [\mathbf{D}_{\text{red}}^j]=[\mathbf{L}_D^j]^T[\mathbf{G}_D^j][\mathbf{L}_D^j], \\ [\mathbf{K}_{\text{red}}^j]=[\mathbf{L}_K^j]^T[\mathbf{G}_K^j][\mathbf{L}_K^j], \quad (13)$$

in which $[\mathbf{G}_M^j]$, $[\mathbf{G}_D^j]$ and $[\mathbf{G}_K^j]$ are normalized random matrices such that

$$\mathcal{E}\{[\mathbf{G}_M^j]\}=[\mathbf{G}_M^j]=[\mathbf{I}_{N_a}], \quad \mathcal{E}\{[\mathbf{G}_D^j]\}=[\mathbf{G}_D^j]=[\mathbf{I}_{N_a^+}], \\ \mathcal{E}\{[\mathbf{G}_K^j]\}=[\mathbf{G}_K^j]=[\mathbf{I}_{N_a^+}]. \quad (14)$$

3.2.2 Construction of the Probability Model of Normalized Random Matrices $[\mathbf{G}_M^j]$, $[\mathbf{G}_D^j]$, $[\mathbf{G}_K^j]$. Constraints C2 and C3 which constitute the second part of the available information is written as

$$[\mathbf{M}_{\text{red}}^j] \in \mathbb{M}_{N_a}^+(\mathbb{R}), \quad [\mathbf{D}_{\text{red}}^j] \in \mathbb{M}_{N_a}^0(\mathbb{R}), \quad [\mathbf{K}_{\text{red}}^j] \in \mathbb{M}_{N_a}^0(\mathbb{R}), \quad (15)$$

$$\{[\mathbf{Q}(\omega) \quad \mathbf{U}_{\Sigma}(\omega) \quad \mathbf{V}_i(\omega)]^T, \omega \in \mathbb{B}\} \\ \text{is a second-order stochastic process.} \quad (16)$$

It can be shown, [16], that the normalized random reduced matrices have to be such that Eq. (14) holds and such that

$$[\mathbf{G}_M^j] \in \mathbb{M}_{N_a}^+(\mathbb{R}), \quad [\mathbf{G}_D^j] \in \mathbb{M}_{N_a^+}^+(\mathbb{R}), \quad [\mathbf{G}_K^j] \in \mathbb{M}_{N_a^+}^+(\mathbb{R}), \quad (17)$$

$$\mathcal{E}\{\|[\mathbf{G}_M^j]^{-1}\|_F^2\} < +\infty, \quad \mathcal{E}\{\|[\mathbf{G}_D^j]^{-1}\|_F^2\} < +\infty, \\ \mathcal{E}\{\|[\mathbf{G}_K^j]^{-1}\|_F^2\} < +\infty. \quad (18)$$

The dispersion level of these three normalized random matrices can be controlled by the positive real parameters δ_M^j , δ_D^j , δ_K^j defined by

$$\delta_E^j = \left\{ \frac{\mathcal{E}\{\|[\mathbf{G}_E^j] - [\mathbf{G}_E^j]\|_F^2\}}{\|[\mathbf{G}_E^j]\|_F^2} \right\}^{1/2} \quad \text{with } E = \{M, D, K\}. \quad (19)$$

From Eq. (13), it can be deduced that these parameters allow the dispersion level of random matrices $[\mathbf{M}_{\text{red}}^j]$, $[\mathbf{D}_{\text{red}}^j]$, and $[\mathbf{K}_{\text{red}}^j]$ to be controlled. Below, $[\mathbf{G}]$ denotes $[\mathbf{G}_M^j]$, $[\mathbf{G}_D^j]$ and $[\mathbf{G}_K^j]$, n denotes N_a , N_a^+ and N_a^+ and δ denotes δ_M^j , δ_D^j and δ_K^j . Using the entropy maximization principle with available information defined by Eqs. (14), (17), and (18), it can be proved, [15,16], that the probability density function of random matrix $[\mathbf{G}]$ with respect to the volume element

$$\tilde{d}\mathbf{G} = 2^{n(n-1)/4} \prod_{1 \leq i \leq j \leq n} d[G]_{ij}, \quad (20)$$

is written as

$$p_{[\mathbf{G}]}([\mathbf{G}]) = \mathbf{1}_{\mathbb{M}_n^+(\mathbb{R})}([\mathbf{G}]) \times C_G \times (\det([\mathbf{G}]))^{(1-\delta^2)(2\delta^2)^{-1}(n+1)} \\ \times e^{-(n+1)(2\delta^2)^{-1}\text{tr}([\mathbf{G}])}, \quad (21)$$

where C_G is the positive constant such that

$$C_G = \frac{(2\pi)^{-n(n-1)/4} \left(\frac{n+1}{2\delta^2} \right)^{n(n+1)(2\delta^2)^{-1}}}{\prod_{j=1}^n \Gamma\left(\frac{n+1}{2\delta^2} + \frac{1-j}{2} \right)}, \quad (22)$$

in which $\Gamma(z)$ is the gamma function defined for all $z > 0$ by $\Gamma(z) = \int_0^\infty t^{z-1} e^{-t} dt$. Equation (21) shows that the entries $[\mathbf{G}]_{jk}$ of random matrix $[\mathbf{G}]$ are dependent random variables. The covariance tensor $C_{jk,j'k'}^G = \mathcal{E}\{([\mathbf{G}]_{jk} - [\mathbf{G}]_{jk})([\mathbf{G}]_{j'k'} - [\mathbf{G}]_{j'k'})\}$ is such that

$$C_{jk,j'k'}^G = \frac{\delta^2}{n+1} \{ \delta_{j'k} \delta_{jk'} + \delta_{jj'} \delta_{kk'} \}. \quad (23)$$

The variance of random variable $[\mathbf{G}]_{jk}$ is then given by

$$V_{jk}^G = \frac{\delta^2}{n+1} (1 + \delta_{jk}). \quad (24)$$

With the available information defined by Eqs. (14), (17), and (18), and using the entropy maximization principle, it can be proved that random matrices $\{[\mathbf{G}_M^j], [\mathbf{G}_D^j], [\mathbf{G}_K^j], j \in \{0, \dots, N-1\}\}$ are independent.

3.2.3 Algebraic Representation of Normalized Matrices $[\mathbf{G}_M^j]$, $[\mathbf{G}_D^j]$, $[\mathbf{G}_K^j]$. The following algebraic representation of random positive-definite symmetric real matrix $[\mathbf{G}]$ allows a procedure for the Monte Carlo numerical simulation of random matrix $[\mathbf{G}]$ to be defined. Random matrix $[\mathbf{G}]$ is written as

$$[\mathbf{G}] = [\mathbf{L}_G]^T [\mathbf{L}_G], \quad (25)$$

in which $[\mathbf{L}_G]$ is an $n \times n$ real upper triangular random matrix such that

- (1) random variables $\{[\mathbf{L}_G]_{jj'}, j \leq j'\}$ are independent;
- (2) for $j < j'$, real-valued random variable $[\mathbf{L}_G]_{jj'}$ can be written as $[\mathbf{L}_G]_{jj'} = \sigma_n U_{jj'}$, in which $\sigma_n = \delta(n+1)^{-1/2}$ and where $U_{jj'}$ is a real-valued Gaussian random variable with zero mean and variance equal to 1;
- (3) for $j = j'$, positive-valued random variable $[\mathbf{L}_G]_{jj}$ can be written as $[\mathbf{L}_G]_{jj} = \sigma_n \sqrt{2V_j}$ in which σ_n is defined above and

where V_j is a positive-valued gamma random variable whose probability density function $p_{V_j}(v)$ with respect to dv is written as

$$p_{V_j}(v) = \mathbb{1}_{\mathbb{R}^+}(v) \frac{1}{\Gamma\left(\frac{n+1}{2\delta^2} + \frac{1-j}{2}\right)} v^{(n+1)/2\delta^2 - (1+j)/2} e^{-v}. \quad (26)$$

4 Mistuning Analysis

The first reduction described in Sec. 2 is only justified by the use of the nonparametric approach and keeps all the physical degrees-of-freedom of the disk. A second global reduction is thus necessary in order to accelerate the numerical computations.

4.1 Second Modal Reduction. We introduce a vector $\mathbf{Q}(\omega)$ such that

$$\begin{bmatrix} \mathbf{Q}(\omega) \\ \mathbf{U}_\Sigma(\omega) \\ \mathbf{V}_i(\omega) \end{bmatrix} = [\Psi] \mathbf{Q}(\omega), \quad (27)$$

where the columns of the matrix $[\Psi]$ are the $\tilde{n}$ eigenvectors related to the $\tilde{n}$ first eigenvalues of the mean generalized eigenvalue problem related to Eq. (5). It should be noted that these calculations are carried out by using the cyclic symmetry properties. Premultiplying Eq. (8) by $[\Psi]^T$ and using Eq. (27) yields matrix equation

$$[A(\omega)] \mathbf{Q}(\omega) = \mathbf{F}(\omega). \quad (28)$$

The random vector of the physical degrees-of-freedom for blade j is then given by

$$\begin{bmatrix} \mathbf{U}_i^j(\omega) \\ \mathbf{U}_\Sigma^j(\omega) \end{bmatrix} = [H^j] \begin{bmatrix} \mathbf{Q}^j(\omega) \\ \mathbf{U}_\Sigma^j(\omega) \end{bmatrix}, \quad (29)$$

in which, for blade j , $\mathbf{U}_i^j(\omega)$ is the C^{n_i} -valued random vector of the internal degrees-of-freedom, $\mathbf{U}_\Sigma^j(\omega)$ is the C^{n_Σ} -valued random vector of the coupling interface degrees-of-freedom and $\mathbf{Q}^j(\omega)$ is the C^{N_g} -valued random vector of the generalized coordinates. The stochastic equation related to Eq. (28) is solved frequency-by-frequency using the Monte Carlo numerical simulation. The two bases are related to deterministic quantities concerning the mean model. It should be noted that these bases are consistent from a mathematical point of view for the stochastic system. Concerning the dimension of numerical applications, a stochastic convergence analysis with respect to the model reduction has to be performed in order to validate the numerical results.

4.2 Random Magnification Factor. The observation is the random dynamic magnification factor $\{\mathbf{B}(\omega), \omega \in \mathbb{B}\}$ which is the positive real-valued stochastic process indexed by $\mathbb{B}$ such that

$$\mathbf{B}(\omega) = \sup_{j \in \{0, \dots, N-1\}} \mathbf{B}^j(\omega), \quad \mathbf{B}^j(\omega) = \frac{\|\mathbf{U}^j(\omega)\|}{\|\mathbf{U}^j\|_\infty}, \quad (30)$$

$$\|\mathbf{U}^j\|_\infty = \sup_{\omega \in \mathbb{B}} \|\mathbf{U}^j(\omega)\|,$$

in which, for blade j , $\mathbf{U}^j(\omega)$ is the random vector of the physical degrees-of-freedom and $\mathbf{U}^j(\omega)$ is the mean vector of the physical degrees-of-freedom. The random dynamic magnification factor over narrow-frequency band $\mathbb{B}$ is defined by

$$B_\infty = \sup_{\omega \in \mathbb{B}} \mathbf{B}(\omega). \quad (31)$$

The realizations of random observation B_∞ are deduced from the Monte Carlo numerical simulations and mathematical statistics are used for estimating probability quantities related to random variable B_∞ .

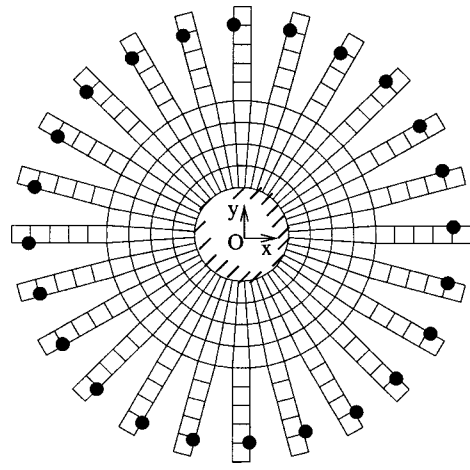


Fig. 1 Finite element mesh of the bladed disk. Input force localization (symbol ●).

5 Numerical Example for a Bladed Disk

A simple example is presented in order to illustrate the application of the proposed nonparametric probability model of random uncertainties.

5.1 Definition of the Mean Finite Element Model. The bladed disk considered (see Fig. 1) is constituted of a disk and 24 blades which are modeled by using linear thin plane theory in bending mode. The membrane and the bending motions are fully decoupled. We are only interested in the out-plane displacements. The bladed disk is located in the plane (Ox, Oy) of a cartesian coordinate system. It should be noted that, in the case of such a rotating structure, the outplane vibration response would not be affected by gyroscopic terms. The disk is made of a homogeneous and isotropic material with constant thickness 5×10^{-3} m, inner radius 3.5×10^{-2} m, outer radius 0.1 m, mass density 7860 kg/m^3 , Poisson ratio 0.25 and Young modulus $1.89 \times 10^{11} \text{ N/m}^2$. A Dirichlet condition is applied along the internal boundary defined by the inner radius. Each blade is made of a homogeneous and isotropic material with length 7×10^{-2} m, width 8.5×10^{-3} m, linear decreasing thickness from 5×10^{-3} m to 1×10^{-3} m, mass density 7860 kg/m^3 , Poisson ratio 0.25, and Young modulus $2 \times 10^{11} \text{ N/m}^2$. A damping model is added for the bladed disk, corresponding to a hysteretic model with a mean loss factor $\eta = 0.0004$. The mean finite element model shown in Fig. 1 is constituted of 312 (with 4 nodes) bending plate elements and has 1296 degrees-of-freedom such that the degree-of-freedom number of one blade is $n_a = 36$, the degree-of-freedom number of the disk is $n_d = 576$ and the degree-of-freedom number of a blade disk coupling interface is $n_\Sigma = 6$. Figure 2 displays the eigenfrequencies of the generalized eigenproblem associated with the tuned mechanical system in function of the circumferential wave number m . A frequency “veering” can clearly be seen and leads us to choose the frequency band of analysis $\mathbb{B} = [7040, 7200] \text{ Hz}$ corresponding to a circumferential wave number $m = 4$. A four-engine order excitation is then applied to excite the modes of the veering zone. Each blade j is submitted to a unit transverse load vector $\mathbf{f}^j(\omega)$ such that $\mathbf{f}^j(\omega)_k = \mathbb{1}_{\mathbb{B}}(\omega) \delta_{k, (2d+1)} e^{(2i\pi m j)/N}$ in which $m = 4$ and where d is the node number of the applied force. Concerning the nonparametric probability model, it is assumed that $\delta_M^j = \delta_D^j = 0$ for all the blades (no uncertainty in mass and damping). The mistuning is introduced by fixing the value $\delta_K^j = \delta_K$ for the blades (uncertainty in stiffness). For the parametric approach, standard deviation $\sigma_K^j = \sigma$ is deduced from Eq. (41). Let $\nu = \omega/(2\pi)$ be the circular frequency associated with angular frequency ω . As an example, for the nonparametric model with $\delta_K = 0.01$, Fig. 3

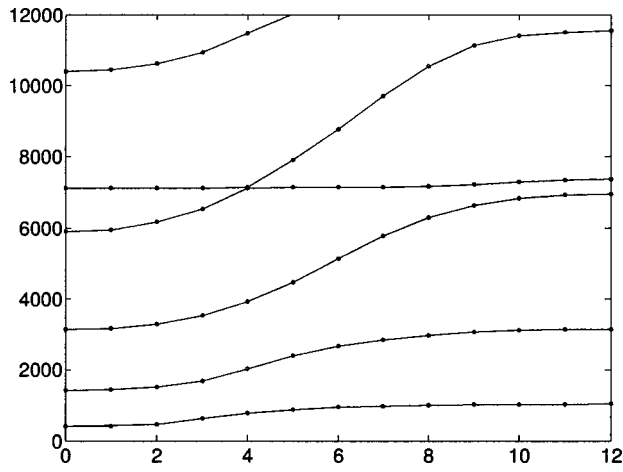


Fig. 2 Graph of the eigenfrequencies values (in Hz) of the tuned bladed disk versus the circumferential wave number m

shows the graph of the tuned response (dashed thick line), the graph of one realization of the random responses of the 24 blades (thin lines) and of its upper bound (thick line) with respect to frequency band B. Figure 4 shows a realization of $B^j(\omega)$ for the frequency $\nu=7108.9$ Hz which corresponds to the largest amplification that can occur in band B. The well known localization phenomenon is exhibited for blades 11 and 12 for which the largest amplifications is obtained.

5.2 Convergence Analysis. A convergence analysis has been carried out. Since the second-order mean convergence yields the convergence in law, the convergence analysis can be limited to the second-order convergence of the sequence $\{\mathbf{B}(\omega_0, N_g, \tilde{n})\}_{N_g, \tilde{n}}$ of random variables, that is to say to construct the function $(N_g, \tilde{n}) \mapsto \|\mathbf{B}(\omega_0, N_g, \tilde{n})\|$ in which $\omega_0 = 2\pi\nu_0$ is a fixed frequency and where

$$\|\mathbf{B}(\omega_0, N_g, \tilde{n})\|^2 = \mathcal{E}\{\mathbf{B}(\omega_0, N_g, \tilde{n})^2\}. \quad (32)$$

This norm is estimated by $\|\mathbf{B}(\omega_0, N_g, \tilde{n})\| \approx \text{Conv}(n_s, N_g, \tilde{n})$ in which

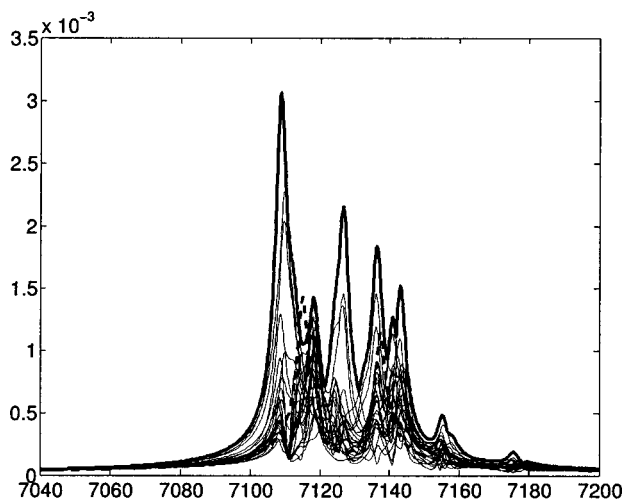


Fig. 3 (1) Displacement forced response (m) of the tuned system with respect to the excitation frequency (Hz): $\nu \mapsto \|U^j(\nu)\|$ (thick dashed line). (2) Graph of one realization θ_1 of the random displacement forced response (m) with respect to the excitation frequency (Hz): $\nu \mapsto \max_{j \in \{0, \dots, N-1\}} \|U^j(\nu, \theta_1)\|$ (thick line), $\nu \mapsto \|U^j(\nu, \theta_1)\|$, $j \in \{0, \dots, N-1\}$ (thin lines).

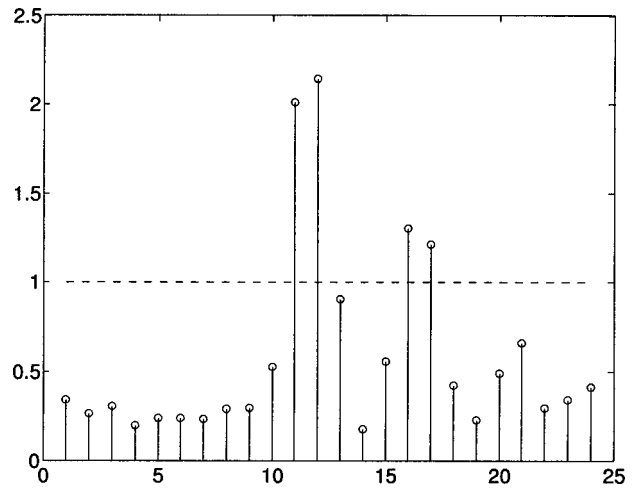


Fig. 4 Spatial localization: graph of one realization θ_1 of magnification factor $j \mapsto B^j(\nu, \theta_1)$ for all the blades

$$\text{Conv}(n_s, N_g, \tilde{n})^2 = \frac{1}{n_s} \sum_{i=1}^{n_s} \mathbf{B}(\omega_0, N_g, \tilde{n}, \theta_i)^2, \quad (33)$$

where θ_i is realization number i and where n_s is the number of realizations used in the Monte Carlo numerical simulation. Figure 5 shows the graph $n_s \mapsto \text{Conv}(n_s, N_g, \tilde{n})$ for several values of the couple $(N_g, \tilde{n})$ and for $\nu_0 = 7108$ Hz. A good convergence is obtained for $n_s = 1500$, $N_g = 8$ and $\tilde{n} = 120$. These numerical parameters are used for all the computations.

5.3 Comparison With the Usual Probability Parametric Approach. In order to give an information on the differences between the nonparametric probabilistic approach and a parametric probabilistic one, a numerical comparison is carried out. We choose as the usual parametric probability model, the Young modulus uncertainties of the blades which yield the blade eigenfrequencies mistuning. Such an approach leads the random stiffness matrix to be written as

$$[\mathbf{K}^j] = (1 + \Lambda_K^j) [\mathbf{K}^j], \quad (34)$$

where $1 + \Lambda_K^j$, which is proportional to the Young modulus, is a real-valued random variable whose probability distribution is uniform with a mean value equal to 1. Its standard deviation σ_K^j is

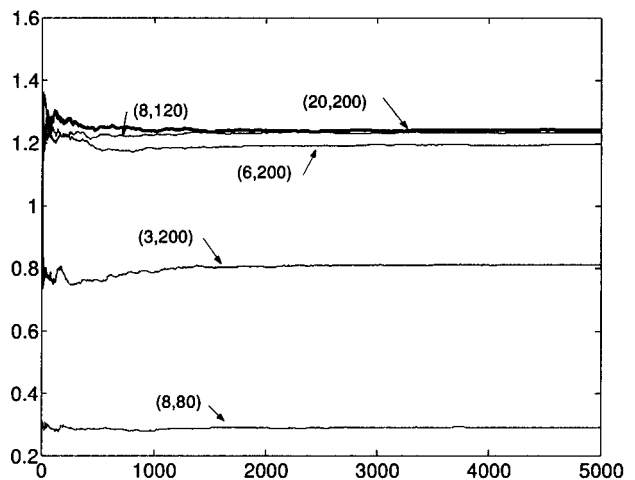


Fig. 5 Convergence analysis: graph of functions $n_s \mapsto \text{Conv}(n_s, N_g, \tilde{n})$ for several values of the couple $(N_g, \tilde{n})$

such that $1 + \Lambda_K^j$ remains positive almost surely. The random reduced stiffness matrix $[\mathbf{K}_{\text{red}}^{j,\text{para}}]$ related to the parametric probabilistic model of blade j is given by

$$[\mathbf{K}_{\text{red}}^{j,\text{para}}] = [\mathbf{L}_K^j]^T [\mathbf{G}_K^{j,\text{para}}] [\mathbf{L}_K^j], \quad [\mathbf{G}_K^{j,\text{para}}] = (1 + \Lambda_K^j) [\mathbf{I}_{N_a^+}]. \quad (35)$$

From Eqs. (34) and (35), we deduce that

$$\mathcal{E}\{[\mathbf{K}_{\text{red}}^{j,\text{para}}]\} = [\mathbf{K}_{\text{red}}^j]. \quad (36)$$

Parameter $W_K^{j,\text{para}}$ is then introduced such that

$$\begin{aligned} W_K^{j,\text{para}} &= \mathcal{E}\{\|[\mathbf{K}_{\text{red}}^{j,\text{para}}] - [\mathbf{K}_{\text{red}}^j]\|_F^2\}, \\ &= \mathcal{E}\{\|[\mathbf{L}_K^j]^T ([\mathbf{G}_K^{j,\text{para}}] - [\mathbf{I}]) [\mathbf{L}_K^j]\|_F^2\}. \end{aligned} \quad (37)$$

This scalar parameter measures the dispersion of random reduced stiffness matrix $[\mathbf{K}_{\text{red}}^{j,\text{para}}]$. It can easily be seen that $W_K^{j,\text{para}}$ is equal to the sum of the second-order moments of the eigenvalues of the matrix $[\mathbf{L}_K^j]^T ([\mathbf{G}_K^{j,\text{para}}] - [\mathbf{I}]) [\mathbf{L}_K^j]$. Since Λ_K^j is a centered random variable and since $\|[\mathbf{K}_{\text{red}}^j]\|_F^2 = \text{tr}([\mathbf{K}_{\text{red}}^j]^2)$, Eq. (37) yields

$$W_K^{j,\text{para}} = (\sigma_K^j)^2 \text{tr}([\mathbf{K}_{\text{red}}^j]^2). \quad (38)$$

In order to establish the comparison between the two probability approaches, the similar parameter W_K^j defined by $W_K^j = \mathcal{E}\{\|[\mathbf{L}_K^j]^T ([\mathbf{G}_K^j] - [\mathbf{I}]) [\mathbf{L}_K^j]\|_F^2\}$ is introduced. Using Eqs. (13), (23), and (24) yields

$$W_K^j = \frac{(\delta_K^j)^2}{N_a^+ + 1} (\text{tr}([\mathbf{K}_{\text{red}}^j])^2 + \text{tr}([\mathbf{K}_{\text{red}}^j]^2)), \quad (39)$$

in which δ_K^j is the parameter controlling the dispersion of random matrix $[\mathbf{G}_K^j]$. From Eqs. (9) and (36), we deduce that both random stiffness matrices $[\mathbf{K}_{\text{red}}^j]$ and $[\mathbf{K}_{\text{red}}^{j,\text{para}}]$ have the same mean value,

$$\mathcal{E}\{[\mathbf{K}_{\text{red}}^j]\} = \mathcal{E}\{[\mathbf{K}_{\text{red}}^{j,\text{para}}]\} = [\mathbf{K}_{\text{red}}^j]. \quad (40)$$

For given σ_K^j , the equation allowing the calculation of δ_K^j is obtained in writing that $W_K^j = W_K^{j,\text{para}}$ and yields

$$(\sigma_K^j)^2 = \frac{(\delta_K^j)^2}{N_a^+ + 1} \left(1 + \frac{\text{tr}([\mathbf{K}_{\text{red}}^j])^2}{\text{tr}([\mathbf{K}_{\text{red}}^j]^2)} \right). \quad (41)$$

5.4 Probability Density Function at a Given Frequency

The objective of this paper being to introduce a new nonparametric probabilistic approach of random uncertainties for blade mistuning analysis, the presentation is limited to a simple case (it is clear that this method is particularly well adapted to complex situations). In this condition, the probabilistic approaches are limited to homogeneous stiffness uncertainties from one blade to another one such that $\delta_K^j = \delta_K$ and $\sigma_K^j = \sigma_K$, for all j in $\{0, \dots, N-1\}$. The sensitivity of $\mathbf{B}(\omega_0)$ at frequency $\nu_0 = 7108$ Hz is analyzed with respect to parameter δ_K . Equation (41) allows δ_K to be calculated giving standard deviation σ_K , such that $\delta_K = 0.01$ for σ_K fixed and equal to 0.055. The mistuning analysis is then carried out for the random magnification factor $\mathbf{B}^{\text{para}}(\omega_0^{\text{para}})$ at frequency $\nu_0^{\text{para}} = 7108.2$ Hz defined hereinafter. Frequencies ν_0 and ν_0^{para} are the frequencies for which the extreme values statistics over frequency band $\mathbb{B}$ are maximum and have been calculated using the Monte Carlo numerical simulation with 4000 realizations. We are then interested in comparing the density probability functions $p_{\mathbf{B}(\omega_0)}(b, \omega_0)$ and $p_{\mathbf{B}^{\text{para}}(\omega_0^{\text{para}})}(b, \omega_0^{\text{para}})$. Figure 6 shows the probability density functions $p_{\mathbf{B}(\omega_0)}(b, \omega_0)$ and $p_{\mathbf{B}^{\text{para}}(\omega_0^{\text{para}})}(b, \omega_0^{\text{para}})$ with $n_s = 8,500,000$ realizations in order to observe the largest amplifications that can occur. The support of the probability density function of the random magnification factor seems to be upper bounded by a finite value which is close to 3. This value can be compared to the value $(1 + \sqrt{N})/2 \approx 2.95$ proposed by Whitehead [3]. But for other values of δ_K , there are frequencies

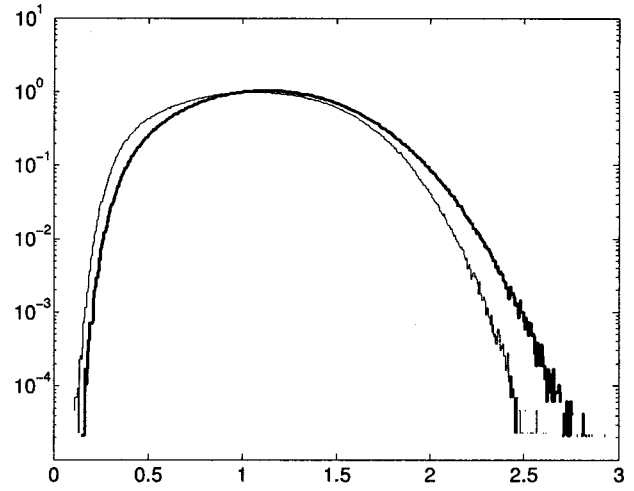


Fig. 6 Comparison of the nonparametric and the parametric models: probability density functions of random magnification factors $\mathbf{B}(\omega_0)$ and $\mathbf{B}^{\text{para}}(\omega_0^{\text{para}})$ in a logarithmic scale: $b \mapsto p_{\mathbf{B}(\omega_0)}(b, \omega_0)$ (thick line), $b \mapsto p_{\mathbf{B}^{\text{para}}(\omega_0^{\text{para}})}(b, \omega_0^{\text{para}})$ (thin line)

for which the upper bound is greater than 2.95. It is observed that the probability density function of random magnification factor $\mathbf{B}(\omega_0)$ for the nonparametric probability model differs from the probability density function of random magnification factor $\mathbf{B}^{\text{para}}(\omega_0^{\text{para}})$ for the parametric one.

5.5 Probability Density Function Over Frequency Band $\mathbb{B}$

A mistuning analysis is then carried out for the real positive random variable B_∞ . Figures 7 to 9 show the probability density functions $b \mapsto p_{B_\infty}(b)$ and $b \mapsto p_{B_\infty^{\text{para}}}(b)$ of random variables B_∞ and B_∞^{para} for $n_s = 1500$ realizations and for several mistuning rates of the nonparametric model: $\delta_K = 0.01$, $\delta_K = 0.02$ and $\delta_K = 0.04$, corresponding to $\sigma_K = 0.0055$, $\sigma_K = 0.011$ and $\sigma_K = 0.022$ for the parametric model. Below, $\mathcal{P}(A)$ denotes the probability of event A . Figures 10 to 12 show the corresponding mappings $b \mapsto \mathcal{P}(B_\infty > b)$ and $b \mapsto \mathcal{P}(B_\infty^{\text{para}} > b)$ in a logarithmic scale where b is the magnification factor level. The thick (or thin) lines are related to the nonparametric (or parametric) approach. These probability density functions are different. The maxima of probability density functions $p_{B_\infty}(b)$ and $p_{B_\infty^{\text{para}}}(b)$ do not occur for the same value of

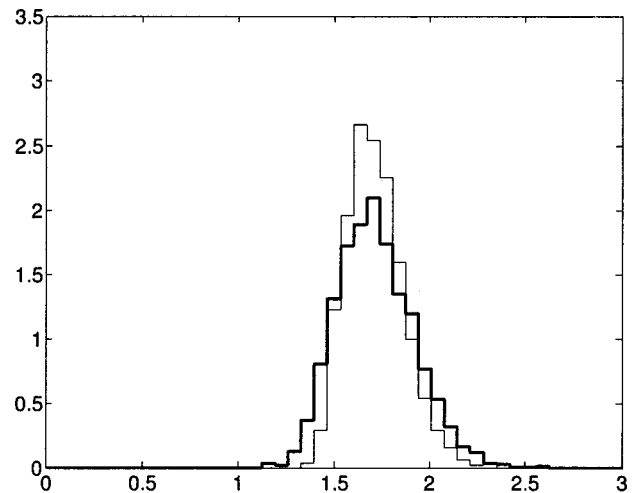


Fig. 7 Comparison of the nonparametric and the parametric models: graphs of the probability density functions $b \mapsto p_{B_\infty}(b)$ (thick line) and $b \mapsto p_{B_\infty^{\text{para}}}(b)$ (thin line) for $\delta_K = 0.01$

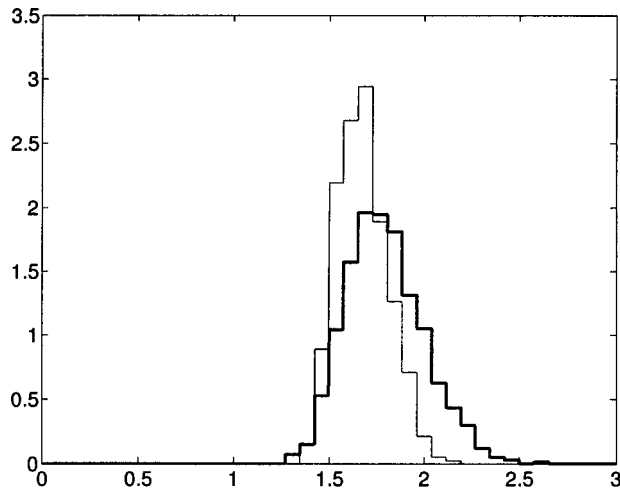


Fig. 8 Comparison of the nonparametric and the parametric models: graphs of the probability density functions $b \rightarrow p_{B_\infty}(b)$ (thick line) and $b \rightarrow p_{B_\infty^{\text{para}}}(b)$ (thin line) for $\delta_K=0.02$

b . In particular, for this simple example, it can be seen that the probability density function for the nonparametric approach is shifted to the right and predicts larger magnification factors than the parametric approach. Furthermore, although the random uncertainties have the same dispersion level for both probability models, probability density function $p_{B_\infty}(b)$ has a larger dispersion than probability density function $p_{B_\infty^{\text{para}}}(b)$. Consequently, the extreme values of the random magnification factor are larger for the nonparametric approach. A comparative parametric study with respect to the mistuning rate is done for both probability models. We are interested in studying $\mathcal{P}(B_\infty \leq b_p) = p$ and $\mathcal{P}(B_\infty^{\text{para}} \leq b_p) = p$ where b_p denotes the maximum magnification level reached with a probability p . In Fig. 13, the thick (or thin) lines represent $\delta_K \mapsto b_p(\delta_K)$ for the nonparametric (or parametric) model. The lower, middle, and upper curves are related to the probability level $p = 0.05$, $p = 0.50$, and $p = 0.95$, respectively. For both probability models, the amplification of the forced response presents a local maximum for a weak mistuning rate. Nevertheless, these local maxima are obtained for different mistuning rates according to the probability model used. Secondly, the sensitivity of the random magnification factor is analyzed with respect to the mean loss

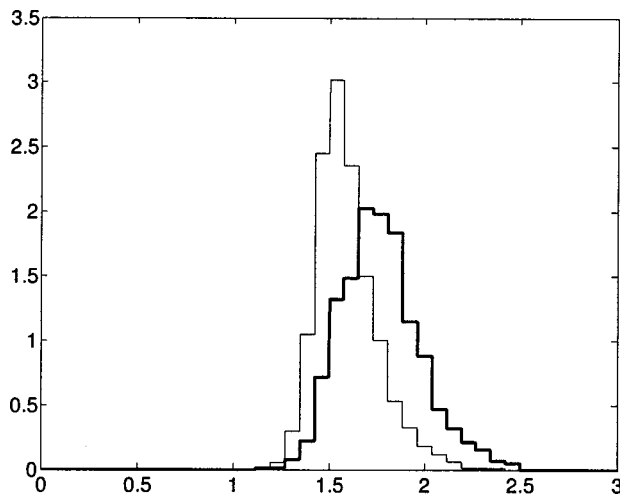


Fig. 9 Comparison of the nonparametric and the parametric models: graphs of the probability density functions $b \rightarrow p_{B_\infty}(b)$ (thick line) and $b \rightarrow p_{B_\infty^{\text{para}}}(b)$ (thin line) for $\delta_K=0.04$

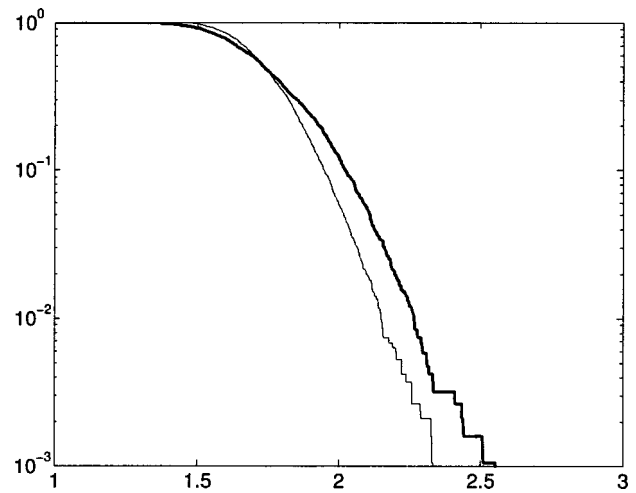


Fig. 10 Comparison of the nonparametric and the parametric models: graphs of $b \rightarrow \mathcal{P}(B_\infty > b)$ (thick line), $b \rightarrow \mathcal{P}(B_\infty^{\text{para}} > b)$ (thin line) in a logarithmic scale for $\delta_K=0.01$

factor η . For a mistuning rate $\delta_K=0.02$, Fig. 14 shows the graph $\eta \mapsto E\{B_\infty\}$ (thick line) and $\eta \mapsto E\{B_\infty^{\text{para}}\}$ (thin line). Figure 15 shows the graph $\eta \mapsto \sigma_{B_\infty}$ (thick line) and $\eta \mapsto \sigma_{B_\infty^{\text{para}}}$ (thin line).

6 Conclusion

In this paper, a nonparametric probability model of random uncertainties is proposed in order to analyze the mistuned bladed disks. By construction, the nonparametric probabilistic approach allows both data uncertainties and model uncertainties to be taken into account. Since the objective of such a nonparametric approach of random uncertainties is to take into account the model errors, it is natural that a parameter without direct physical meaning such as δ parameter, appears in the probability model. In return, such a new parameter has to be identified for such a dynamic system or can be used as a global parameter to analyze the sensitivity to random data and model uncertainties. With such a nonparametric probability model of random uncertainties, the random eigenvalues and the associated random eigenvectors of each uncertain blade are statistically dependent and result from a single random model. In order to validate the proposed probability

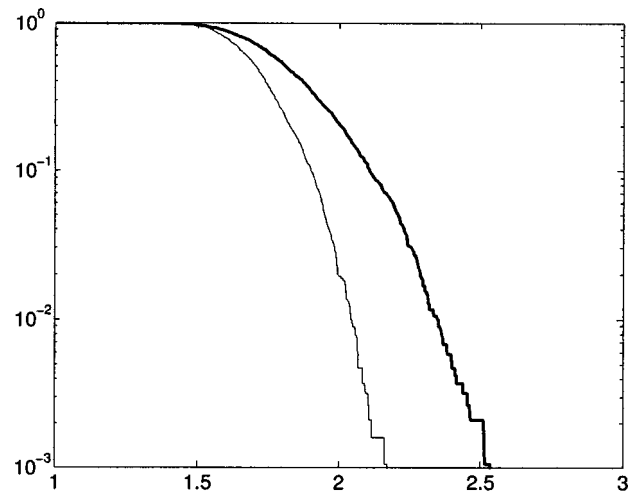


Fig. 11 Comparison of the nonparametric and the parametric models: graphs of $b \rightarrow \mathcal{P}(B_\infty > b)$ (thick line), $b \rightarrow \mathcal{P}(B_\infty^{\text{para}} > b)$ (thin line) in a logarithmic scale for $\delta_K=0.02$

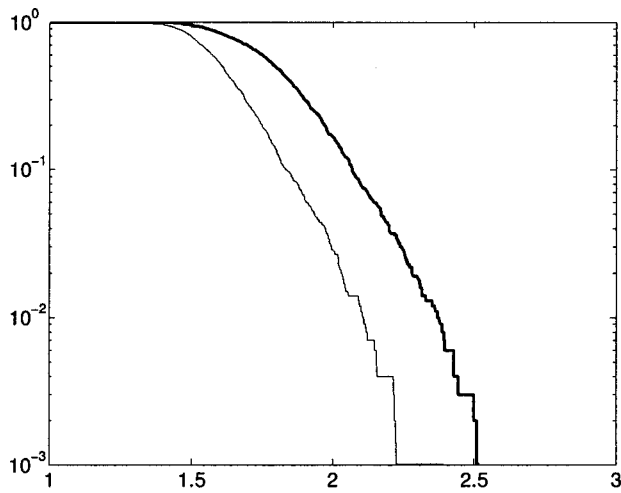


Fig. 12 Comparison of the nonparametric and the parametric models: graphs of $b \rightarrow \mathcal{P}(B_\infty > b)$ (thick line), $b \rightarrow \mathcal{P}(B_\infty^{\text{para}} > b)$ (thin line) in a logarithmic scale for $\delta_K = 0.04$

model, a simple example is presented, and this model is compared with a usual parametric probability model of random uncertainties. For a given frequency and for a given narrow frequency band, the numerical comparison carried out for this simple example exhibits some differences concerning the probability distributions of the random magnification factor.

Nomenclature

- n_a = number of degrees-of-freedom of a blade
- n_d = number of degrees-of-freedom of the disk
- n_i = number of internal degrees-of-freedom of a blade
- n_Σ = number of coupling interface degrees-of-freedom of a blade
- $\tilde{n}$ = dimension of the second global reduction

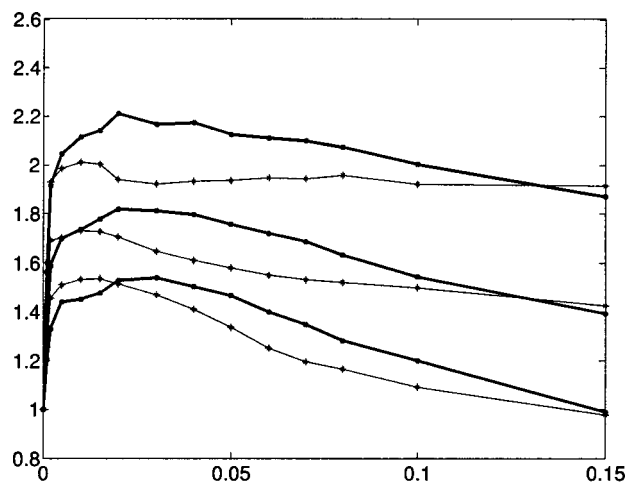


Fig. 13 Influence of the mistuning rate: graph of $\delta_K \rightarrow b_p(\delta_K)$ such that $\mathcal{P}(B_\infty \leq b_p) = p$. The thick (or thin) lines are related to the nonparametric (or parametric) model (the lower, middle and upper curves correspond respectively to $p=0.05$, $p=0.5$ and $p=0.95$).

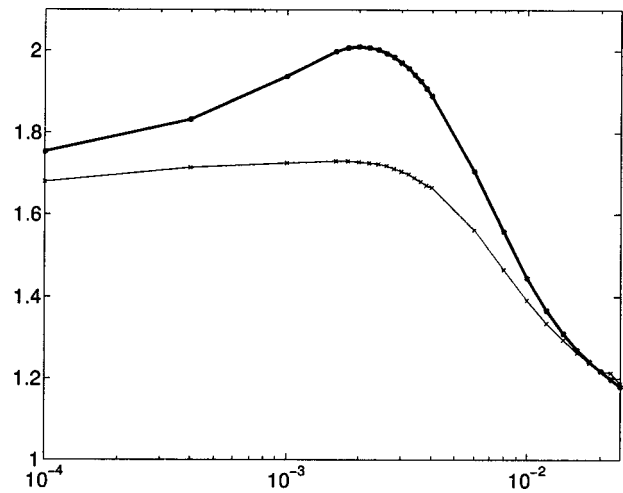


Fig. 14 Influence of the mean loss factor for a mistuning rate $\delta_K = 0.02$; graphs of $\eta \rightarrow E\{B_\infty\}$ (thick line), $\eta \rightarrow E\{B_\infty^{\text{para}}\}$ (thin line)

$\underline{q}(\omega)$ = mean vector of the generalized coordinates for all the blades

$\underline{q}^j(\omega)$ = mean vector of the generalized coordinates of blade j

$\underline{u}_i^j(\omega)$ = mean vector of the internal degrees-of-freedom of blade j

$\underline{u}_\Sigma(\omega)$ = mean vector of the coupling interface degrees-of-freedom for all the blades

$\underline{u}_\Sigma^j(\omega)$ = mean vector of the coupling interface degrees-of-freedom of blade j

$\underline{v}_i(\omega)$ = mean vector of the internal degrees-of-freedom of the disk

$[A^d(\omega)]$ = mean dynamic stiffness matrix of the disk

$[A_{\text{red}}^j(\omega)]$ = mean reduced dynamic stiffness matrix of blade j

$[A_{\text{red}}^j(\omega)]$ = random reduced dynamic stiffness matrix of blade j

B = frequency band of analysis

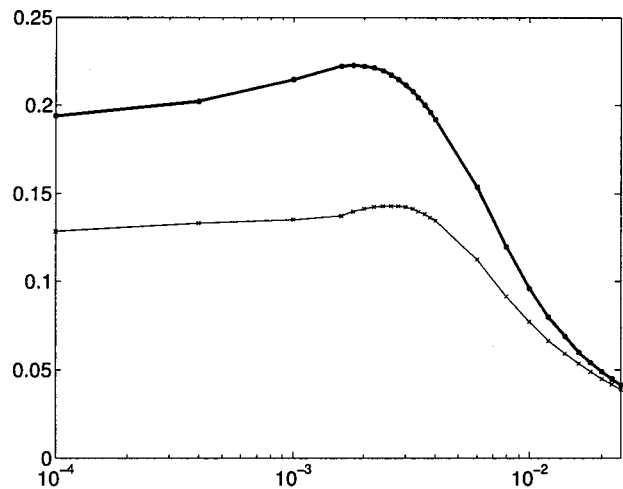


Fig. 15 Influence of the mean loss factor for a mistuning rate $\delta_K = 0.02$: graphs of $\eta \rightarrow \sigma_{B_\infty}$ (thick line), $\eta \rightarrow \sigma_{B_\infty^{\text{para}}}$ (thin line)

$\mathbf{B}^j(\omega)$ = random dynamic magnification factor of blade j
 $\mathbf{B}(\omega)$ = random dynamic magnification factor of the blades
 $\mathbf{B}_\infty$ = random dynamic magnification factor of the blades and over frequency band B
 $[\mathbf{G}_M^j], [\mathbf{G}_D^j], [\mathbf{G}_K^j]$ = random normalized mass, damping and stiffness matrices
 $[\mathbf{G}]$ = representing $[\mathbf{G}_M^j], [\mathbf{G}_D^j], [\mathbf{G}_K^j]$
 $[\mathbf{H}^j]$ = transformation matrix issued from the Craig and Bampton decomposition
 $[\mathbf{M}_{\text{red}}^j], [\mathbf{D}_{\text{red}}^j], [\mathbf{K}_{\text{red}}^j]$ = mean reduced mass, damping and stiffness matrices of blade j
 $[\mathbf{M}_{\text{red}}^j], [\mathbf{D}_{\text{red}}^j], [\mathbf{K}_{\text{red}}^j]$ = random reduced mass, damping and stiffness matrices of blade j
 N = number of blades
 N_g = number of fixed coupling interface modes of the disk
 N_a = rank of the reduced mass matrix of the blade
 N_a^+ = rank of the reduced damping and stiffness matrices of the blade
 $\mathbf{Q}(\omega)$ = random vector of the second global generalized coordinates
 $\mathbf{Q}(\omega)$ = random vector of the generalized coordinates for all the blades
 $\mathbf{Q}^j(\omega)$ = random vector of the generalized coordinates of blade j
 $\mathbf{U}_i^j(\omega)$ = random vector of the internal degrees-of-freedom of blade j
 $\mathbf{U}_\Sigma(\omega)$ = random vector of the coupling interface degrees-of-freedom for all the blades
 $\mathbf{U}_\Sigma^j(\omega)$ = random vector of the coupling interface degrees-of-freedom of blade j
 $\mathbf{V}_i(\omega)$ = random vector of the internal degrees-of-freedom of the disk
 $\delta_M^j, \delta_D^j, \delta_K^j$ = dispersion parameters related to the random mass, damping, stiffness matrices
 δ = representing $\delta_M^j, \delta_D^j, \delta_K^j$
 η = mean loss factor
 ν_0 = fixed frequency used in the probability nonparametric approach
 ω_0 = representing $2\pi\nu_0$
 Σ_j = blade disk coupling interface
 $[\Psi]$ = transformation matrix issued from the second global reduction
 Ω = bladed disk
 Ω_d = disk
 Ω_j = blade j
 $[\mathbf{A}]^T$ = transpose of matrix $[\mathbf{A}]$
 $\text{tr}([\mathbf{A}])$ = trace of matrix $[\mathbf{A}]$
 $\det([\mathbf{A}])$ = determinant of square matrix $[\mathbf{A}]$
 $\|\mathbf{U}(\omega)\|$ = Hermitian norm of vector $\mathbf{U}(\omega)$ such that $\|\mathbf{U}(\omega)\| = (\mathbf{U}(\omega)^* \mathbf{U}(\omega))^{1/2}$

$\|\mathbf{U}\|_\infty$ = infinity norm of vector $\mathbf{U}(\omega)$ such that $\|\mathbf{U}\|_\infty = \sup_\omega \|\mathbf{U}(\omega)\|$
 $\|[\mathbf{A}]\|_F$ = Frobenius norm of matrix $[\mathbf{A}]$ such that $\|[\mathbf{A}]\|_F = \text{tr}([\mathbf{A}][\mathbf{A}]^T)^{1/2}$
 $[\mathbf{I}_n]$ = $n \times n$ identity matrix
 $\mathcal{E}$ = mathematical expectation
 $\mathbf{M}_n^+(\mathbf{R})$ = set of all the positive-definite symmetric $n \times n$ real matrices
 $\mathbf{M}_n^{+0}(\mathbf{R})$ = set of all the positive (semi-positive definite) symmetric $n \times n$ real matrices
 $\mathcal{P}(X < x)$ = probability that $\{X < x\}$
 δ_{ij} = Kronecker symbol such that $\delta_{ij} = 1$ if $i = j$ and $\delta_{ij} = 0$ if $i \neq j$
 $\mathbf{1}_B(b)$ = indicatrix function on any set B such that $\mathbf{1}_B(b) = 1$ if $b \in B$ and $\mathbf{1}_B(b) = 0$ if $b \notin B$

References

- [1] Thomas, D. L., 1979, "Dynamics of Rotationally Periodic Structures," *Int. J. Numer. Methods Eng.*, **14**, pp. 81–102.
- [2] Ohayon, R., and Soize, C., 1998, *Structural Acoustics and Vibration*, Academic Press, San Diego, CA.
- [3] Whitehead, D. S., 1966, "Effects of Mistuning on the Vibration of Turbomachine Blades Induced by Wakes," *J. Mech. Eng. Sci.*, **8**(1), pp. 15–21.
- [4] Ewins, D. J., 1969, "The Effects of Detuning Upon the Forced Vibrations of Bladed Disks," *J. Sound Vib.*, **9**(1), pp. 65–69.
- [5] Wei, S. T., and Pierre, C., 1988, "Localization Phenomena in Mistuned Assemblies for Cyclic Symmetry—Part II: Forced Vibrations," *ASME J. Vib., Acoust., Stress, Reliab. Des.*, **110**(4), pp. 439–449.
- [6] Lin, C. C., and Mignolet, M. P., 1996, "Effects of Damping and Damping Mistuning on the Forced Vibration Response of Bladed Disks," *J. Sound Vib.*, **193**(2), pp. 525–543.
- [7] Lin, C. C., and Mignolet, M. P., 1997, "An Adaptive Perturbation Scheme for the Analysis of Mistuned Bladed Disks," *ASME J. Eng. Gas Turbines Power*, **119**, pp. 153–160.
- [8] Griffin, J. H., and Hoosac, T. M., 1984, "Model Development and Statistical Investigation of Turbine Blade Mistuning," *ASME J. Vib., Acoust., Stress, Reliab. Des.*, **106**, pp. 204–210.
- [9] Mignolet, M. P., and Lin, C. C., 1993, "The Combined Closed Form—Perturbation Approach to the Analysis of Mistuned Bladed Disks," *ASME J. Turbomach.*, **115**, pp. 771–780.
- [10] Yang, M.-T., and Griffin, J. H., 2001, "A Reduced-Order Model of Mistuning Using a Subset of Nominal System Modes," *ASME J. Eng. Gas Turbines Power*, **123**, pp. 893–900.
- [11] Bladh, R., Castanier, M. P., and Pierre, C., 2001, "Component-Mode-Based Reduced Order Modeling Techniques for Mistuned Bladed Disks—Part I: Theoretical Models," *ASME J. Eng. Gas Turbines Power*, **123**, pp. 89–99.
- [12] Seinturier, E., Dupont, C., Berthillier, M., and Dumas, M., 2002, "A New Aeroelastic Model for Mistuned Bladed Disks," *AIAA Paper No. 2002-1533*.
- [13] Mignolet, M. P., Rivas-Guerra, A. J., and Delor, J. P., 2001, "Identification of Mistuning Characteristics of Bladed Disks From Free Response Data (Part I)," *ASME J. Eng. Gas Turbines Power*, **123**, pp. 395–403.
- [14] Rivas-Guerra, A. J., Mignolet, M. P., and Delor, J. P., 2001, "Identification of Mistuning Characteristics of Bladed Disks From Free Response Data (Part II)," *ASME J. Eng. Gas Turbines Power*, **123**, pp. 404–411.
- [15] Soize, C., 2000, "A Nonparametric Model of Random Uncertainties for Reduced Matrix Models in Structural Dynamics," *Probab. Eng. Mech.*, **15**(3), pp. 277–294.
- [16] Soize, C., 2001, "Maximum Entropy Approach for Modeling Random Uncertainties in Transient Elastodynamics," *J. Acoust. Soc. Am.*, **109**(5), pp. 1979–1996.
- [17] Craig, Jr., R. R., and Bampton, M. C. C., 1968, "Coupling of Substructures for Dynamic Analyses," *AIAA J.*, **6**(7), pp. 1313–1319.
- [18] Soize, C., and Chebli, H., 2003, "Random Uncertainties Model in Dynamic Substructuring Using a Nonparametric Probabilistic Model," *J. Eng. Mech.*, **129**(4), pp. 449–457.

Anders Angantyr

e-mail: anders.angantyr@cad.luth.se

Jan Olov Aidanpää

e-mail: jan-olov.aidanpaa@cad.luth.se

Department of Applied Physics and Mechanical
Engineering,
Division of Computer Aided Design,
Luleå University of Technology,
Universitetsområdet, Porsön,
SE 97187 Luleå, Sweden

A Pareto-Based Genetic Algorithm Search Approach to Handle Damped Natural Frequency Constraints in Turbo Generator Rotor System Design

The detailed design of a turbo generator rotor system is highly constrained by feasible regions for the damped natural frequencies of the system. A major problem for the designer is to find a solution that fulfills the design criterion for the damped natural frequencies. The bearings and some geometrical variables of the rotor are used as the primary design variables in order to achieve a feasible design. This paper presents an alternative approach to search for feasible designs. The design problem is formulated as an optimization problem and a genetic algorithm (GA) is used to search for feasible designs. Then, the problem is extended to include another objective (i.e., multiobjective optimization) to show the potential of using the optimization formulation and a Pareto-based GA in this rotordynamic application. The results show that the presented approach is promising as an engineering design tool. [DOI: 10.1115/1.1760529]

Introduction

The detailed design stage of a turbo generator rotor system is an iterative process involving several conflicting objectives and constraints. One attractive approach is to look upon this design process as a multiobjective optimization problem, [1]. The generator rotor system design is highly constrained by feasible regions for the damped natural frequencies (complex eigenvalues). Therefore one of the most important tasks for the designer is to find designs that fulfill these constraints. Previous works in rotordynamic optimization applications with eigenvalue constraints are done by Chen and Wang [2] and Lee and Choi [3]. Multiple objectives with eigenvalue constraints have been studied by Shiau and Chang [4] and Shiau, Kuo, and Hwang [5]. In [2–5] other optimization methods than a genetic algorithm (GA) were used. In [6] Choi and Yang use a GA to lower the first natural frequency in a rotor-bearing system and in [7] they discuss the immune-genetic algorithm (IGA) for multiobjective rotor-bearing problems. In [8] Choi and Yang apply this to a problem with two bearings. In [2–7] constraints for maximum three eigenvalues were used and the damping of the modes were not taken into account.

The objective functions involved in the detailed rotor system design are expected to have complex shapes. Depending on the chosen design variables, the objective functions may even be discontinuous. During the past two decades GAs have successfully been used for optimization of difficult problems. The interest of application of GAs in the area of rotordynamics has recently started, [6–8]. GAs are, however, yet only applicable on problems with computationally cheap objective function evaluations. Foundations of GA are found in Goldberg [9] and a comprehensive survey with engineering applications is given by Gen and Cheng [10]. In [11] Andersson gives a survey of different methods for multiobjective problems. GAs are well suited for multiobjective problems since the search is done from a set of solutions. A common goal for multiobjective GAs is to evolve the Pareto optimal

tradeoff surface for the objectives, [1]. To achieve a robust algorithm a Pareto-based ranking scheme is preferable since it is insensitive to the shape of the Pareto optimal front, [12].

A drawback of GAs has been the lack of robust methods to incorporate constraints in the search algorithms. A frequently used approach is to add a penalty term to the objective value, [13,14]. Another interesting way to handle a constrained problem is to transform the constraints into objectives. The problem is then a multiobjective unconstrained problem. This approach is addressed as one method by Coello in [15], Michalewicz in [16], and Fonseca and Fleming in [17].

In this paper this approach is applied to the damped natural frequency constraints. The constraints are mapped on a scalar function and formulated as a design objective. Not only the frequencies, but also the damping ratio for each mode is taken into account. Furthermore, there is no restriction in the number of modes involved in the search. First a real-number encoded GA is applied to search for a feasible solution. Once a feasible solution is found, a second objective is included and a Pareto-based GA, [12], is applied to extend the search, i.e. multiobjective optimization. A Pareto-based GA has not been used earlier in optimization of rotor-bearing systems. The used rotordynamic system model is kept relatively simple in this study. The bearings are here described by linear, isotropic coefficients without cross coupling effects. In a real situation, a more complex model of the rotor system is needed.

In the first section a description of the design problem is given. Factorial experiments are then conducted in section two in order to get some knowledge of the behavior of the damped natural frequencies. In section three a single objective optimization is performed for the new function describing the degree of feasibility with respect to the damped natural frequency constraints. The optimization is also extended to include minimum rotor length as a design objective in the last section. The length of the rotor is related to costs and may, as well as the dynamical objectives, be an important objective at the system level in generator applications.

Contributed by the Structures and Dynamics Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the S&D Division October 2002; final revision received December 2003. Associate Editor: M. Mignolet.

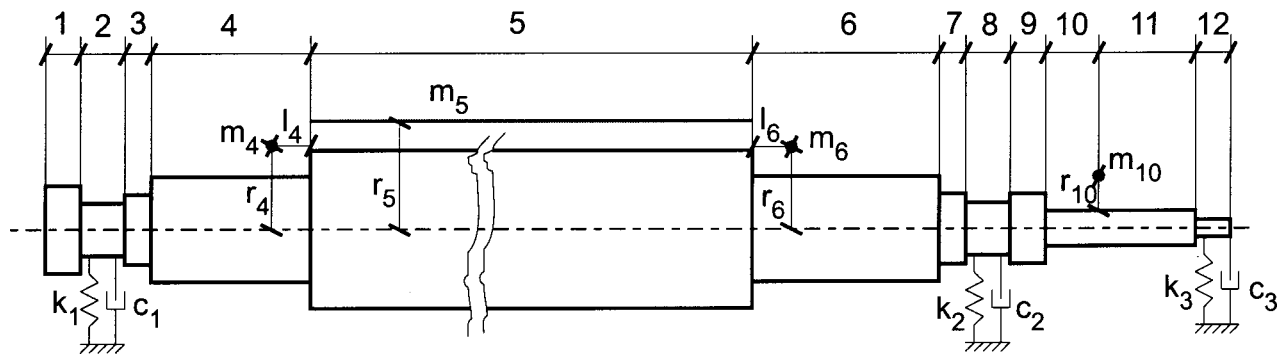


Fig. 1 Schematic sketch of rotor-bearing model

Generator Rotor System Design

The first stage in the generator design process is a basic design where the overall dimensions are specified with respect to electrical, thermal, and mechanical considerations. The next step is the detailed design of the rotor system and the stator. These activities are carried out in parallel. The detailed design of the rotor system also includes choice of bearing types, bearing designs and support structure (pedestal) designs. In [18] Rajan et al. state that “The design of a rotor-bearing system is an iterative process in which the parameters that influence the design are modified until the desired design objectives are achieved.” The detailed design process of a turbo generator rotor system is no exception from this statement. The cause of these iterations is that several blurred objectives and constraints exist for the design. One of the most important among these is the design criterion for the damped natural frequency constraints. A damped natural frequency analysis of the rotor-bearing system is today a computationally cheap analysis which gives much information about the system. Fundamentals of stability and damped critical speeds are given by Lund in [19].

The model of the rotor-bearing system is a discrete model implemented in ARDAS. ARDAS is an in-house code specially developed for rotordynamic analyses based on the transfer matrix method and a substructure method. The complexity of the model is chosen such that it should capture the most important effects found in a real system. A schematic sketch of the model with geometry and bearing parameters is found in Fig. 1. Nominal values of the design parameters are given in Table 1.

The bearings are for simplicity assumed to be isotropic and described by linearized coefficients for stiffness and damping. The retaining rings in Sections 4 and 6 are modeled as ring masses

(m_4 and m_6) rigidly attached to the rotor. An additional mass (m_5) is uniformly applied to Section 5 to model the weak copper conductors. The exciter at the end of Section 10 is also modeled as a ring mass (m_{10}) rigidly attached to the rotor.

In this case, the damped natural frequency constraints are composed by three infeasible regions.

1. A stability margin, $\bar{\zeta}_1$, is required for the lower modes which are passed during acceleration to the operational speed.
2. No modes are allowed within a range around the operational speed, $\bar{n}_{dr}$. If there exist modes in this range they should be highly damped, $\bar{\zeta}_2$.
3. The third region is similar to the second but the range is around the double rotational speed. The required damping for modes in this region is $\bar{\zeta}_3$.

The cause of the third infeasible region is that modes close to the double rotational speed will be excited by gravity since the real rotor have anisotropic bending stiffness. The infeasible regions for the damped natural frequencies are indicated by the colored rectangular areas in Fig. 2.

Generally there are two different possibilities to handle infeasible damped natural frequencies. The first is to make design changes which shifts the frequencies into feasible regions. The second method is to increase the damping for the infeasible damped natural frequencies. A serious problem is, however, to see the effect of design changes on all the damped natural frequencies simultaneously.

The chosen numerical values of the damped natural frequency

Table 1 Nominal values of design parameters. The length of the section i is L_i . D_i is outer diameter and d_i is inner diameter for section i .

Bearing Parameters					
k_1	c_1	k_2	c_2	k_3	c_3
$2.0 \cdot 10^9$ N/m	$8.0 \cdot 10^6$ Ns/m	$2.0 \cdot 10^9$ N/m	$8.0 \cdot 10^6$ Ns/m	$1.8 \cdot 10^8$ N/m	$9.0 \cdot 10^5$ Ns/m
Geometrical parameters					
Section i	L_i	D_i	d_i	Additional mass	
1	490 mm	425 mm	0	...	
2	540 mm	340 mm	0	...	
3	310 mm	425 mm	0	...	
4	1800 mm	590 mm	0	$m_4 = 1670$ kg, $r_4 = 445$ mm, $l_4 = 300$ mm	
5	4300 mm	890 mm	0	$m_5 = 1.11$ kg/mm, $r_5 = 500$ mm	
6	2100 mm	590 mm	110 mm	$m_6 = 1670$ kg, $r_6 = 445$ mm, $l_6 = 300$ mm	
7	310 mm	425 mm	110 mm	...	
8	540 mm	340 mm	110 mm	...	
9	350 mm	425 mm	110 mm	...	
10	490 mm	245 mm	110 mm	$m_{10} = 825$ kg, $r_{10} = 225$ mm	
11	1200 mm	245 mm	110 mm	...	
12	190 mm	180 mm	110 mm	...	

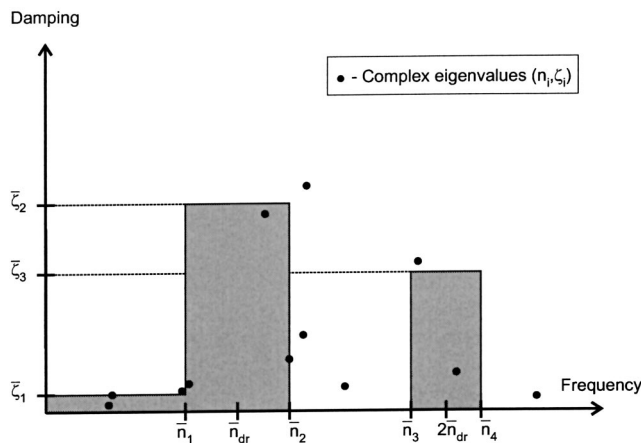


Fig. 2 Damped natural frequency constraints for a generator rotor system. The colored regions are infeasible regions for the damped natural frequencies.

constraints for a 50 Hz generator (operating speed 3000 rpm) used in this paper are based on experience and given in Table 2.

The resulting damped natural frequencies (marked “×”) with frequency less than 7500 rpm and damping less than 20% for the nominal design are plotted in Fig. 3. The corresponding eigenmodes are given in Fig. 4.

With this design, the task for the designer would be to increase the damping for the first two modes and change the frequency for mode 3 and 4.

Factorial Experiments

Factorial experiments are performed in order to study the parameters effect on the damped natural frequencies. Since no single response variable exist it is not possible to perform a factor effect estimation. The purpose with the factorial experiments rather is to attain some knowledge of the objective space for the damped natural frequencies. The parameters that possibly can be changed at this stage of the design process are chosen as factors. The factors with corresponding levels are given in Table 3.

The high and low levels of the bearing factors are set to the vertical and horizontal properties of two existing cylindrical journal bearings. The low level of the geometrical factors are set to their nominal values and the high levels are set with respect to practical limitations.

Since the computational cost is not of major concern full factorial experiments are performed. In the first experiment the bearing factors effect on the damped natural frequencies are studied. The geometrical factors are kept at their nominal levels (low). An intermediate level is also introduced only to get a more dense distribution of the damped natural frequencies in the frequency-damping plane. The results for the 3^6 full factorial experiment are found in Fig. 5. The effect of changes in all factors is tested in a second 2^{11} experiment and the result is found in Fig. 6.

From Fig. 5 it is clear that changes in the bearings mainly affects the damping ratio, when the geometry factors are constant. Figure 6 indicates that a single damped natural frequency can be placed in a feasible region. However, no design is found which simultaneously fulfills the design criterion for all the damped natural frequencies. The design indicated by rings in Fig. 5 and

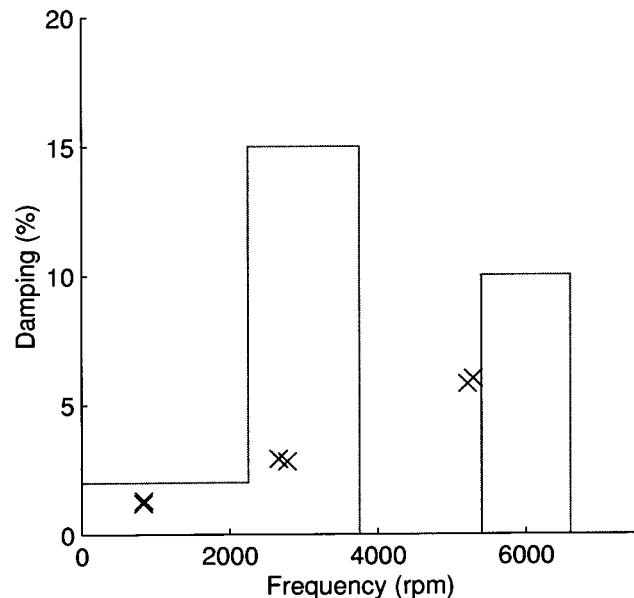


Fig. 3 Damped natural frequencies for nominal rotor-bearing system design. The design criterion is marked by the lines.

Fig. 6 is the best design found with respect to a function defined in the next section about optimization. Still remains the important question whether feasible designs exist if all the factors are chosen as design variables. This is the main issue for the next section.

Optimization

The damped natural frequency constraints may be expressed by a number of inequality constraints for the modal damping. The number of constraints as well as the damping level for each constraint is, however, dependent on the problem. A function ($D(\mathbf{X})$) is then formulated to describe the degree of feasibility of the damped natural frequency constraints. The basic idea behind this function is that it should decrease if the infeasible damped natural frequencies are moved in the directions indicated by the arrows in Fig. 7. With all damped natural frequencies outside the shaded region, which corresponds to a feasible design, the function is zero. Hence, $D(\mathbf{X}) \geq 0$. The function is explained in the Appendix. The function is basically a sum of terms where each term describes an infeasible damped natural frequency. No weights are used since no preference between the damped natural frequencies is done. Each term can take a value within the range [0,1] depending on the actual position within the infeasible area. The damped natural frequency constraints are fulfilled for a design that satisfies $D(\mathbf{X})=0$. The search for feasible designs with respect to the damped natural frequency constraints is now the problem of minimizing $D(\mathbf{X})$.

A first search with the design variables $\mathbf{X} = [k_1, c_1, k_2, c_2, k_3, c_3]$ is performed to find what possibly can be achieved if only changes in the bearings are allowed. The next search is performed with $\mathbf{X} = [k_1, c_1, k_2, c_2, k_3, c_3, L_3, L_7, L_9, L_{10}, L_{11}]$ as design variables. The additional geometry variables are chosen since these are the only geometry variables that may be changed at the current stage of the design process.

Table 2 Numerical values of damped natural frequency constraints

ζ_1	ζ_2	ζ_3	n_1	n_{dr}	n_2	n_3	n_4
2.0%	15.0%	10.0%	2250 rpm	3000 rpm	3750 rpm	5400 rpm	6600 rpm

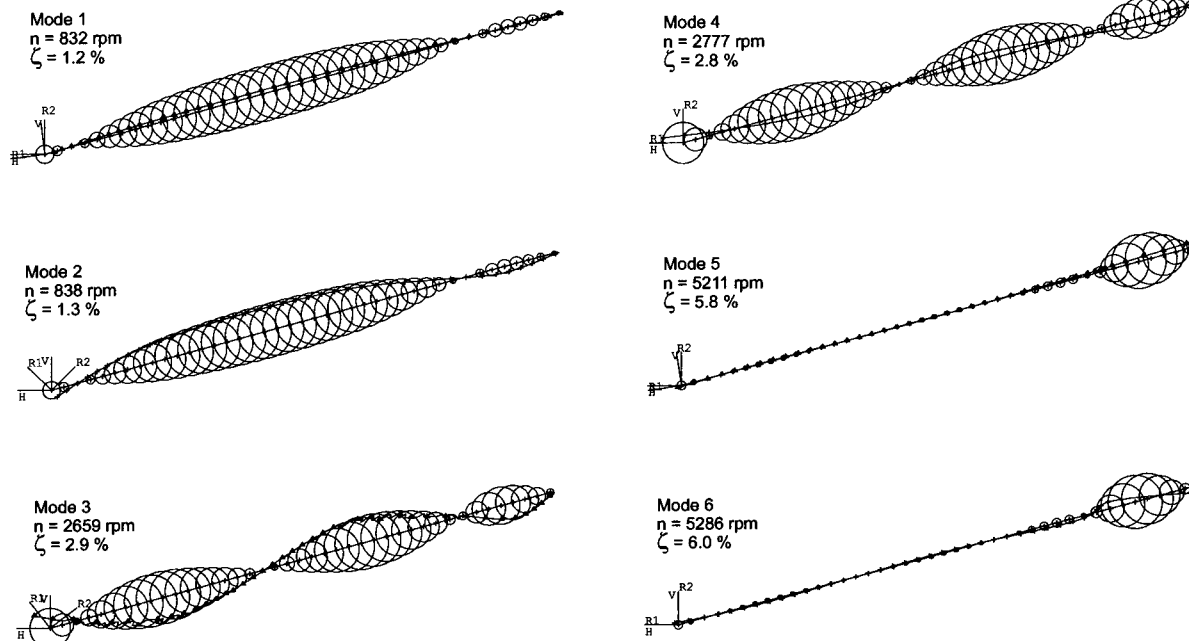


Fig. 4 Eigenmodes for nominal rotor-bearing system design

Due to the electrical requirements the other geometry variables are fixed. The side constraints for each design variable are set to the low and high factor levels found in Table 3.

The chosen search algorithm is a GA with real-number encoding. The GA parameters are set to values that should result in a stable algorithm for the global search. Fine-tuning of all parameters is necessary in order to achieve an efficient algorithm. For definition of GA parameters and methods see Gen and Cheng [10]. Roulette-wheel selection is used with linear ranking and the selection pressure 1.9. Extended intermediate crossover is used to produce new offsprings, [20]. The mutation probability is the same in every generation and selected such that on average one variable per individual is mutated. The used mutation operator is explained in [20]. This operator produces a small mutation step with high probability and large mutation step with low probability. The maximum mutation step size is 10% of the range given by the side constraints for the actual design variable. 90% of the population is replaced in each generation. Hence the reinsertion scheme is of elitist type. The code used is the GEATbx 3.30, which is a toolbox for MATLAB written by Hartmut Pohlheim [21].

Table 3 Factor levels

Factor	Bearing Factors	
	Low level	High level
k_1	$1.0 \cdot 10^9$ N/m	$3.0 \cdot 10^9$ N/m
c_1	$2.0 \cdot 10^6$ Ns/m	$14.0 \cdot 10^6$ Ns/m
k_2	$1.0 \cdot 10^9$ N/m	$3.0 \cdot 10^9$ N/m
c_2	$2.0 \cdot 10^6$ Ns/m	$14.0 \cdot 10^6$ Ns/m
k_3	$1.2 \cdot 10^8$ N/m	$2.4 \cdot 10^8$ N/m
c_3	$4.0 \cdot 10^5$ Ns/m	$14.0 \cdot 10^5$ Ns/m
Factor	Geometrical Factors	
	Low level	High level
L_3	310 mm	910 mm
L_7	310 mm	910 mm
L_9	350 mm	650 mm
L_{10}	490 mm	790 mm
L_{11}	1200 mm	1500 mm

The search result is found in Fig. 8 and Fig. 9. The population is run for a predefined number of generations hence no convergence criterion is used.

Figure 8 shows that the best design found results in $D(\mathbf{X}) = 0.593$. This is found in generation 169. Figure 9 shows that a feasible design ($D(\mathbf{X}) = 0$) is found in the 65th generation for the used design variables and side constraints.

It is now known that at least one feasible design exists for $\mathbf{X} = [k_1, c_1, k_2, c_2, k_3, c_3, L_3, L_7, L_9, L_{10}, L_{11}]$. An important question to raise at this point is therefore if other feasible designs exist. A natural continuation of the study would be to tune the search

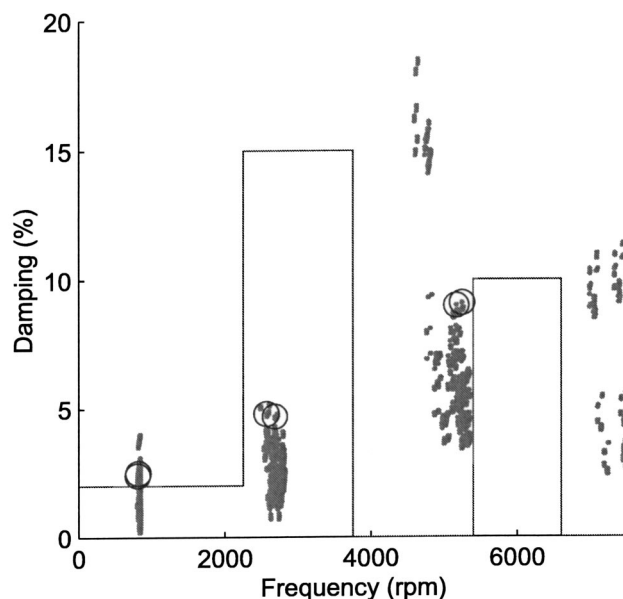


Fig. 5 Damped natural frequency results for 3^6 full factorial experiment for bearing factors. The geometrical factors are kept at their nominal values.

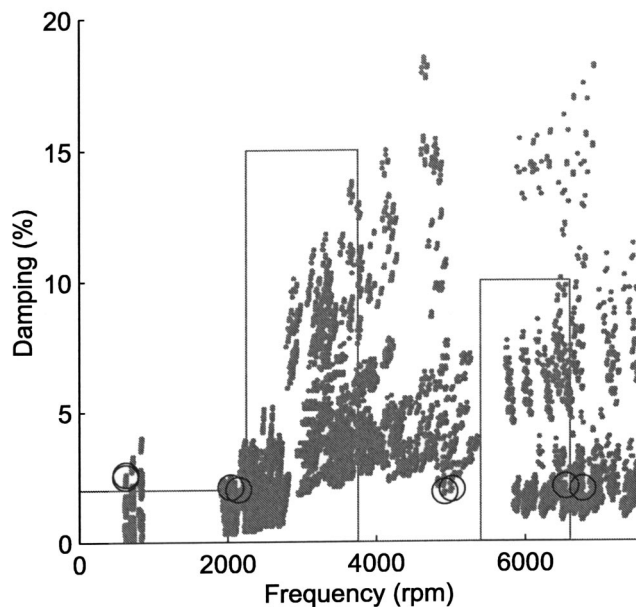


Fig. 6 Damped natural frequency results for 2^{11} full factorial experiment for all factors

algorithm so that the population evolves the whole set of feasible designs. All feasible designs may, however, not be of interests since other objectives also exist. One interesting objective is the increase of rotor length which is necessary in order to achieve a feasible design. The increase of rotor length from the nominal value is formulated as

$$L(X) = L_3 - L_{3,nom} + L_7 - L_{7,nom} + L_9 - L_{9,nom} + L_{10} - L_{10,nom} + L_{11} - L_{11,nom} \quad (1)$$

The nominal values for the lengths are indicated by “nom” as extension of the index in Eq. (1). These nominal values are the same as the low levels of the factors in Table 3. Since an increased rotor length is associated with additional costs, it is of interests to keep the increase at a minimum. The search for interesting designs is now the problem of minimizing $D(X)$ and $L(X)$.

The most interesting design in this case is clearly a design that satisfies $D(X)=0$ and minimizes $L(X)$. Infeasible designs may still contain important information and it is of interest to see the tradeoff (Pareto optimal) curve for these objectives. This would be

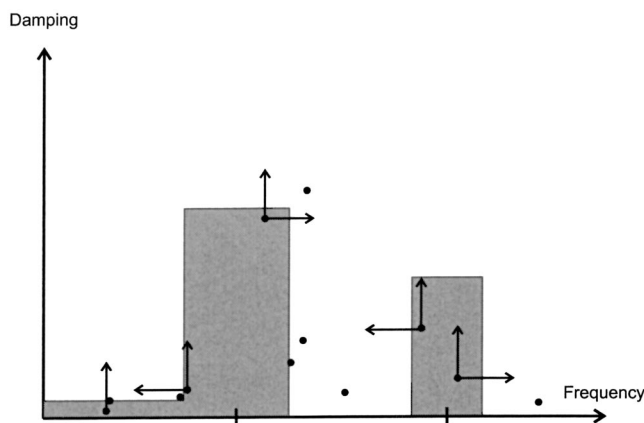


Fig. 7 Directions for movements of infeasible damped natural frequencies that results in a better design with respect to the damped natural frequency constraints

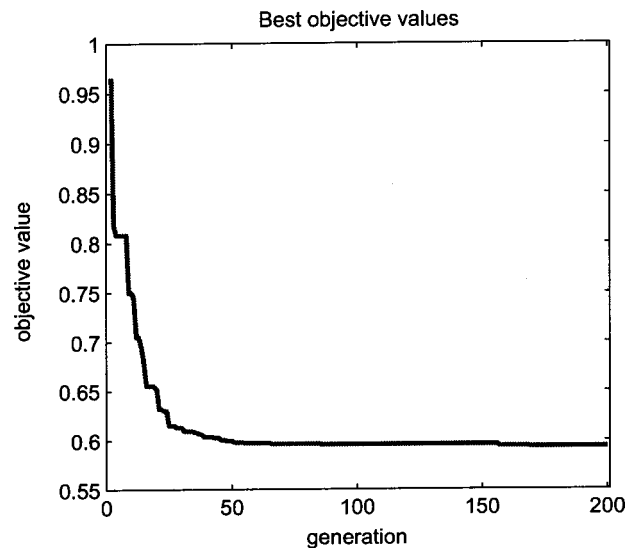


Fig. 8 Search result for $\min(D(X))$ as objective and $X = [k_1, c_1, k_2, c_2, k_3, c_3]$ as design variables. The population size is 30.

especially important if economic objectives are included and other types of design variables are used. This could for example be bearing types, i.e., categorical variables.

A search for the Pareto optimal front between the both objectives with $X = [k_1, c_1, k_2, c_2, k_3, c_3, L_3, L_7, L_9, L_{10}, L_{11}]$ as design variables is done. The used population size is 200 and 20% of the population is replaced in each generation. The algorithm is therefore more of steady-state type. For definition of a steady state GA see p. 108 in [22]. The high elitism is used since no valuable genetic information about the Pareto optimal front should get lost. The ranking scheme by Fleming and Fonseca [12] is used but solutions with exactly the same values in one objective are ranked equally if they are not dominated by other solutions. This makes it possible to evolve several feasible designs. The sharing method described by Gen and Cheng at p. 111 in [10] is used. The sharing

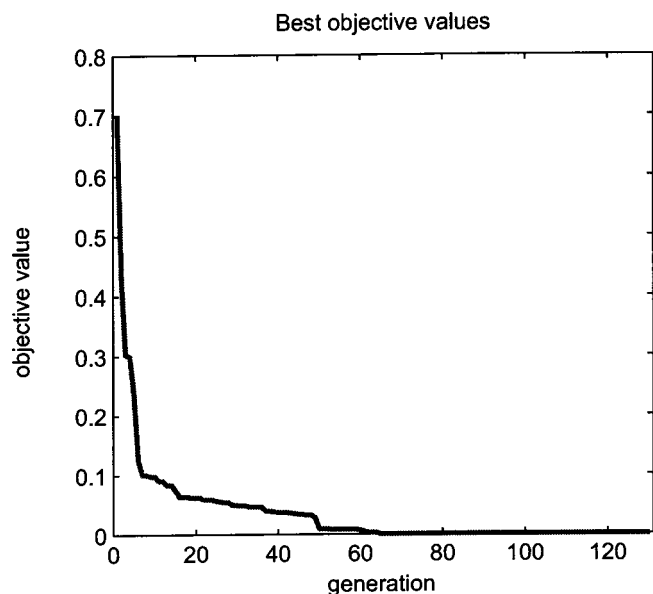


Fig. 9 Search result for $\min(D(X))$ as objective and $X = [k_1, c_1, k_2, c_2, k_3, c_3, L_3, L_7, L_9, L_{10}, L_{11}]$ as design variables. The population size is 60.

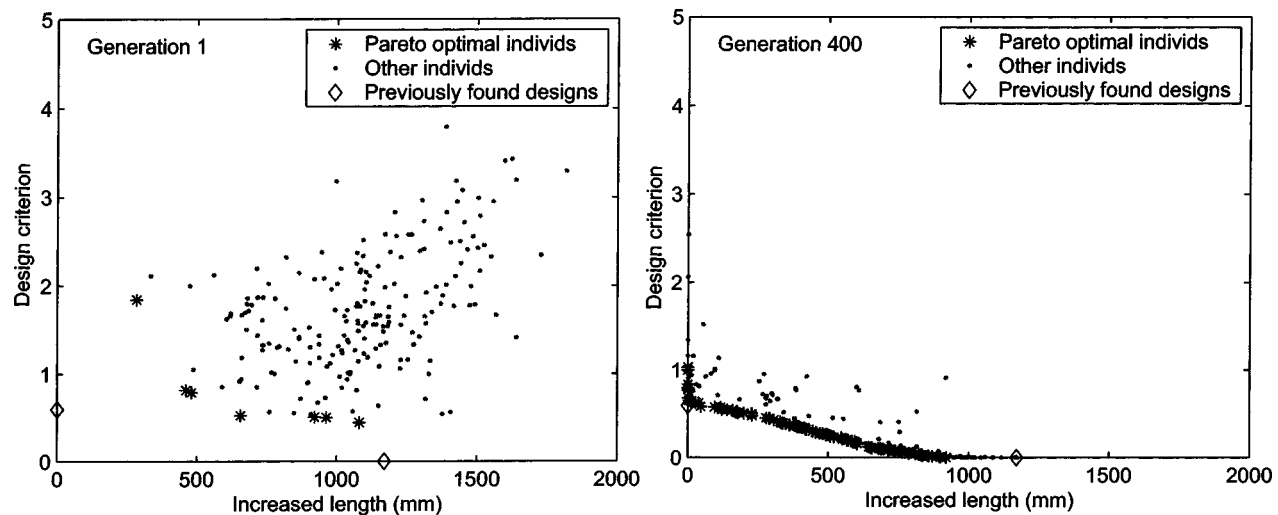


Fig. 10 The initial and 400th generation in the objective space

is based on the distance between designs in the phenotype space (design variable space). All the design variables are linearly scaled on the range $[0,1]$. The sharing parameters are set to the GEATbx default values. In this particular case they become $\sigma_{\text{share}}=1.02$ and $\alpha=1$. It is however shown that it can be better to base the sharing on the distance between pair wise designs in the objective space, [23]. Sharing on the objective space attempts to achieve a more uniform distribution of solutions at the global Pareto optimal front. For more information on Pareto based GAs, see [1,12].

The initial and 400th generation are given in Fig. 10. The previously best found design and feasible design for the single objective optimization problem $\min(D(X))$ are also found in Fig. 10. Twelve feasible designs ($D(X)=0$) are found in the 400th generation. The best found feasible design with respect to $\min(L(X))$ results in $L(X)=918$ mm which is better than the previously found feasible design that resulted in $L(X)=1171$ mm. The multiobjective approach results in a better design than the single objective approach.

Results and Conclusions

A new approach to search for feasible designs in a turbo generator rotor system has been shown. In this case a feasible design is when the damped natural frequencies satisfies a design criterion according to Fig. 2. In Table 4 the results from the two factorial experiments and three optimizations are summarized. Each optimization is evaluated against the objective function $D(X)$ where $D(X)=0$ corresponds to a feasible design. Two sets of factors or design variables are evaluated namely; bearing properties ($k_1, c_1, k_2, c_2, k_3, c_3$) and bearing properties together with some lengths of the rotor ($k_1, c_1, k_2, c_2, k_3, c_3, L_3, L_7, L_9, L_{10}, L_{11}$). In the table, $L(X)$ is the increased length of the rotor with respect to the original design. It is normally an advantage if changes are only made in the bearings since changes in the rotor geometry will imply higher cost.

Table 4 shows that no feasible design was found in the factorial experiment. However, if all design variables are included a fea-

Table 4 Summary of best designs with respect to $\min(D(X))$ found in factorial experiments and single objective search. For the multiobjective search the best design found is defined by $D(X)=0$ and $\min(L(X))$. Factors or design variables within parentheses are held constant in the actual experiment or optimization.

	Factorial Experiments		Optimization		
	Full 3^6 with bearing factors	Full 2^{11} with all factors	Single objective with bearing design variables	Single objective with all design variables	Multiobjective with all design variables
$k_1/10^9$ (N/m)	1.0	1.0	1.0	1.4	1.0
$c_1/10^6$ (Ns/m)	2.0	14.0	2.4	8.9	3.8
$k_2/10^9$ (N/m)	1.0	1.0	1.0	1.0	1.0
$c_2/10^6$ (Ns/m)	8.0	14.0	4.4	11.0	12.0
$k_3/10^8$ (N/m)	1.2	1.2	1.2	1.7	2.4
$c_3/10^5$ (Ns/m)	4.0	14.0	5.5	13.0	13.0
L_3 (mm)	(310)	910	(310)	727	770
L_7 (mm)	(310)	910	(310)	841	762
L_9 (mm)	(350)	350	(350)	371	351
L_{10} (mm)	(490)	490	(490)	500	492
L_{11} (mm)	(1200)	1200	(1200)	1392	1203
$D(X)$	0.686	0.0724	0.593	0	0
$L(X)$ (mm)	(0)	1200	(0)	1171	918
Objective function evaluations	729	2048	4566	3516	16,200

sible design could be achieved in the single objective optimization. By including the $\min(L(\mathbf{X}))$ objective another feasible design with shorter rotor length was found.

The results show that the method of using a Pareto-based GA and multiobjective optimization with the damped natural frequency constraints as one objective is interesting in this rotordynamic application. Normally the design of a turbo generator is based on simple linear dynamical analyses. A serious problem for the designer is, however, how to handle multiple conflicting objectives. With the knowledge of the Pareto optimal tradeoff surface, for the objectives, the designer is better equipped to make a rational choice for the final design.

The presented way to handle the damped natural frequency constraints may also be applicable in other rotordynamic systems. It is also shown that the search easily can be extended to include more objectives. The computational effort will however increase if the dimension of the objective space is increased. Mating restrictions may be needed in order to zoom in on the most important ranges of the objectives. In this paper a linear and isotropic bearing model is used without cross coupling effects. Therefore, a better bearing-pedestal model is required in order to achieve results of more practical interests. The direction for the author's further research is therefore to investigate the possibility to include a bearing code in the search algorithm.

Acknowledgments

The first author gratefully acknowledges Alstom Power for the feedback of the ARDAS software, engineering knowledge, and funding of the project.

Appendix

The function, $D(\mathbf{X})$, describing the degree of feasibility with respect to the damped natural frequency constraints is shown by these rows of coding.

```

D(X)=0
for i=1 to N
  if  $0 < n_i \leq \bar{n}_1$  and  $\zeta_i < \bar{\zeta}_1$ 
     $D(\mathbf{X}) = D(\mathbf{X}) + (\zeta_i - \bar{\zeta}_1) / \bar{\zeta}_1$ 
  elseif  $\bar{n}_1 < n_i \leq \bar{n}_{dr}$  and  $\zeta_i < \bar{\zeta}_2$ 
     $D(\mathbf{X}) = D(\mathbf{X}) + ((\zeta_i - \bar{\zeta}_1)(n_i - \bar{n}_1)) / (\bar{\zeta}_2(\bar{n}_{dr} - \bar{n}_1))$ 
  elseif  $\bar{n}_{dr} < n_i < \bar{n}_2$  and  $\zeta_i < \bar{\zeta}_2$ 
     $D(\mathbf{X}) = D(\mathbf{X}) + ((\zeta_i - \bar{\zeta}_1)(\bar{n}_2 - n_i)) / (\bar{\zeta}_2(\bar{n}_2 - \bar{n}_{dr}))$ 
  elseif  $\bar{n}_2 < n_i \leq 2\bar{n}_{dr}$  and  $\zeta_i < \bar{\zeta}_3$ 
     $D(\mathbf{X}) = D(\mathbf{X}) + ((\zeta_i - \bar{\zeta}_1)(n_i - \bar{n}_2)) / (\bar{\zeta}_3(2\bar{n}_{dr} - \bar{n}_2))$ 
  elseif  $2\bar{n}_{dr} < n_i < \bar{n}_4$  and  $\zeta_i < \bar{\zeta}_3$ 
     $D(\mathbf{X}) = D(\mathbf{X}) + ((\zeta_i - \bar{\zeta}_1)(\bar{n}_4 - n_i)) / (\bar{\zeta}_3(\bar{n}_4 - 2\bar{n}_{dr}))$ 
  end
end

```

N is the number of damped natural frequencies with frequency less than $\bar{n}_4$, see Fig. 2 for definition of the other variables. Only infeasible damped natural frequencies will give a contribution to $D(\mathbf{X})$. The magnitude of an infeasible eigenvalues contribution to $D(\mathbf{X})$ depends on its location within the infeasible region.

References

- [1] Andersson, J., 2001, "Multiobjective Optimization in Engineering Design," Ph.D. thesis No. 675, Linköping University, Linköping.
- [2] Chen, T. Y., and Wang, B. P., 1993, "Optimum Design of Rotor-Bearing Systems With Eigenvalue Constraints," *ASME J. Eng. Gas Turbines Power*, **115**, pp. 256–260.
- [3] Lee, D. S., and Choi, D. H., 2000, "Reduced Weight Design of a Flexible Rotor With Ball Bearing Stiffness Characteristics Varying With Rotational Speed and Load," *ASME J. Vib. Acoust.*, **122**, pp. 203–208.
- [4] Shiau, T. N., and Chang, J. R., 1993, "Multi-Objective Optimization of Rotor-Bearing System With Critical Speed Constraints," *ASME J. Eng. Gas Turbines Power*, **115**, pp. 246–255.
- [5] Shiau, T. N., Kuo, C. P., and Hwang, J. R., 1994, "Multiobjective Optimization of a Flexible Rotor in Magnetic Bearings With Critical Speeds and Control Current Constraints," *Proc. of the International Gas Turbine Aeroengine Congress and Exposition*, ASME, New York, pp. 1–13.
- [6] Choi, B. K., and Yang, B. S., 2000, "Optimum Shape Design of Rotor Shafts Using Genetic Algorithm," *ASME J. Vib., Acoust., Stress, Reliab. Des.*, **6**, pp. 207–222.
- [7] Choi, B. K., and Yang, B. S., 2001, "Multiobjective Optimum Design of Rotor-Bearing Systems With Dynamic Constraints Using Immune-Genetic Algorithm," *ASME J. Eng. Gas Turbines Power*, **123**, pp. 78–81.
- [8] Choi, B. K., and Yang, B. S., 2001, "Optimal Design of Rotor-Bearing Systems Using Immune-Genetic Algorithm," *ASME J. Vib. Acoust.*, **123**, pp. 398–401.
- [9] Goldberg, D. E., 1989, *Genetic Algorithms in Search, Optimization and Machine Learning*, Addison-Wesley, Reading, MA.
- [10] Gen, M., and Cheng, R., 2000, *Genetic Algorithms & Engineering Optimization*, John Wiley and Sons, New York.
- [11] Andersson, J., 2000, "A Survey of Multiobjective Optimization in Engineering Design," Technical Report No. LiTH-IKP-R-1097, Linköping University, Linköping.
- [12] Fonseca, C. M., and Fleming, P. J., 1995, "An Overview of Evolutionary Algorithms in Multiobjective Optimization," *Evol. Comput.*, **3**, pp. 1–18.
- [13] Gen, M., and Cheng, R., 1996, "A Survey of Penalty Techniques in Genetic Algorithms," *Proc. of the IEEE International Conference on Evolutionary Computation*, IEEE, Piscataway, NJ, pp. 804–809.
- [14] Richardson, J. T., Palmer, M. R., Liepins, G., and Hilliard, M., 1989, "Some Guidelines for Genetic Algorithms With Penalty Functions," *Proc. of the 3rd International Conference on Genetic Algorithms*, Morgan Kaufmann Publishers, San Francisco, pp. 191–197.
- [15] Coello Coello, Carlos A., 1999, "A Survey of Constraint Handling Techniques Used With Evolutionary Algorithms," Technical Report Lania-RI-99-04, Laboratorio Nacional de Informática Avanzada, Xalapa.
- [16] Michalewicz, Z., 1995, "A Survey of Constraint Handling Techniques in Evolutionary Computation Methods," *Proc. of the 4th Annual Conference on Evolutionary Programming*, MIT Press, Cambridge, MA, pp. 135–155.
- [17] Fonseca, C. M., and Fleming, P. J., 1995, "Multiobjective Optimization and Multiple Constraint Handling With Evolutionary Algorithms I: A Unified Formulation," Research Report 564, University of Sheffield, Sheffield, UK.
- [18] Rajan, M., Rajan, S. D., Nelson, H. D., and Chen, W. J., 1987, "Optimal Placement of Critical Speeds in Rotor-Bearing Systems," *ASME J. Vib., Acoust., Stress, Reliab. Des.*, **109**, pp. 152–157.
- [19] Lund, J. W., 1974, "Stability and Damped Critical Speeds of a Flexible Rotor in Fluid-Film Bearings," *J. Eng. Ind.*, **96**, pp. 509–517.
- [20] Mühlstein, H., and Schlierkamp-Voosen, D., 1993, "Predictive Models for the Breeder Genetic Algorithm: I. Continuous Parameter Optimization," *Evol. Comput.*, **1**, pp. 25–49.
- [21] Pohlheim, H., 1998, "Development and Engineering Application of Evolutionary Algorithms," Ph.D. Thesis, Technical University Ilmenau, Ilmenau.
- [22] Deb, K., 2002, *Multi-Objective Optimization using Evolutionary Algorithms*, John Wiley and Sons, London.
- [23] Fonseca, C. M., and Fleming, P. J., 1993, "Genetic Algorithms for Multiobjective Optimization: Formulation, Discussion and Generalization," *Proc. of the 5th International Conference on Genetic Algorithms*, Morgan Kaufmann Publishers, San Francisco, pp. 416–423.

Lateral Forces From Single Gland Rotor Labyrinth Seals in Turbines

Bum Ho Song

Seung Jin Song

School of Mechanical and Aerospace
Engineering,
Seoul National University,
Seoul 151-742, Korea

Even though interest in labyrinth seal flows has increased recently, an analytical model capable of predicting turbine flow response to labyrinth seals is still lacking. Therefore, this paper presents a new model to predict flow response in an axial turbine stage with a shrouded rotor. A concentric model is first developed, and this model is used to develop an eccentric model. Basic conservation laws are used in each model, and a nonaxisymmetric sealing gap is prescribed for the eccentric model. Thus, the two models can predict the evolution of a uniform upstream flow into a nonuniform downstream flow. In turbines with concentric shrouded rotors, the seal flow is retarded in the axial direction and tangentially underturned. In turbines with eccentric shrouded rotors, flow azimuthally migrates away from and pressure reaches its peak near the maximum sealing gap region. Finally, the rotordynamic implications of such flow nonuniformities are discussed and compared against eccentric unshrouded turbine predictions. [DOI: 10.1115/1.1690771]

Introduction

Turbines can be broadly classified into shrouded and unshrouded turbines. Shrouded turbines have seals at rotor tips and/or stator hubs for aerodynamic and structural reasons. The seals enhance aerodynamic efficiency by reducing leakage through the gap between rotating and stationary components. The seals also improve blades' vibrational characteristics. Furthermore, such seals can be classified into contact and noncontact seals. Often, noncontact seals are used in turbomachinery due to high relative speeds. Noncontact seals include labyrinth, honeycomb, and overlap seals. Among these, the labyrinth seals are widely used due to their simple geometry. A typical labyrinth seal is composed of two or more sharp sealing knives separated by kinetic energy dissipating chambers (Fig. 1).

Relatively little research had previously been done on labyrinth seal effects in turbines. Recently, however, manufacturers have begun to show significant interest in turbine shroud flows to improve efficiency. Labyrinth seal flow research had initially focused on understanding and optimizing labyrinth seal knife contraction coefficients (e.g., Varmes [1]). Subsequently, Denton and Johnson [2] developed a seal leakage flow model by assuming that the leakage flow is caused by an axial pressure difference across the seal. Through their experiment, Denton and Johnson [2] validated their prediction that the leakage mass flow would increase linearly with sealing gap. More recently, Pfau et al. [3] made flow measurements in the labyrinth seal cavity of an annular turbine cascade. The data showed that the leakage flow was axially retarded and tangentially underturned relative to the passage flow.

Research has also been conducted on aerodynamic losses due to labyrinth seals in rotating environments. In a single-stage turbine, Peters et al. [4] measured total pressure and flow angle to determine its performance. Then, Hunter and Manwaring [5] reconfirmed Denton and Johnson's [2] finding, that the seal leakage flow's circumferential momentum remains essentially unaltered, in a two stage turbine. Thus, the consequent mismatch (between the leakage flow and the passage flow) in circumferential momentum at the seal cavity exit decreased efficiency. To mitigate such detrimental effects, Wallis et al. [6] tested turning devices built inside the labyrinth seal of a multistage turbine to extract additional work from the leakage flow.

Less research has been conducted on the aerodynamic effects of nonaxisymmetric sealing gaps in shrouded turbines. Such effects are important for aerodynamic and structural reasons because non-axisymmetric seal gap can cause aerodynamic forces which can promote rotordynamic instability (Den Hartog [7], Thomas [8], Alford [9], Ehrich [10], and Ehrich and Childs [11]). A lumped parameter model for the flow inside a labyrinth seal cavity, with an inlet and an exit, was initially developed by Kostyuk [12]. Then, Iwatsubo [13] extended the model to account for the seal depth variation in an eccentric labyrinth seal by adopting a harmonic solution approach. Furthermore, Millsaps and Martinez-Sanchez [14] experimentally and analytically examined unsteady aerodynamic effects in a whirling labyrinth seal. The seal entry flow with nonaxisymmetric swirl and the change in swirl created by friction inside the seal gland were found to cause rotordynamic instability.

The current state of the art can be summarized as follows. Experimentally, limited labyrinth seal flow data have been obtained. Analytically, models have been developed to predict flows within the seal glands of concentric and eccentric labyrinth seals. However, such models focus solely on the seal leakage flow, when, in reality, the seal leakage flow interacts with the main passage flow. Thus, an analytical model capable of predicting global turbine flow field response (i.e., both *leakage and passage flows*) in shrouded turbines under concentric and eccentric conditions is still lacking. Therefore, the objective of this study is to develop such a model, and, thus, help understand the long length scale aerodynamic effects of labyrinth seals. The current research scope is limited to labyrinth seals in axial turbines, and an analytical approach has been adopted.

Model Description

This section introduces a new flow model for an axial turbine with a shrouded rotor and a recessed casing. The model is a combination of concentric and eccentric parts. The concentric part is an axisymmetric, two-dimensional, meridional plane analysis, examining radial flow redistribution due to axisymmetric rotor sealing gap. The eccentric part is a nonaxisymmetric, two-dimensional, radial plane analysis focusing on azimuthal flow redistribution caused by nonaxisymmetric sealing gap distribution. Incompressible flow is assumed, limiting the validity of current model to low speed turbines.

Concentric Model. In the concentric part, rotor inlet flow is assumed to be uniform. Upon going through the stage, the flow splits into seal leakage and main passage flows (Fig. 2). Viscous effects are assumed to be limited to within the seal cavity, and

Contributed by the International Gas Turbine Institute (IGTI) of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Paper presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Amsterdam, The Netherlands, June 3–6, 2002; Paper No. 2002-GT-30335. Manuscript received by IGTI, Dec. 2001, final revision, Mar. 2002. Associate Editor: E. Benvenuti.

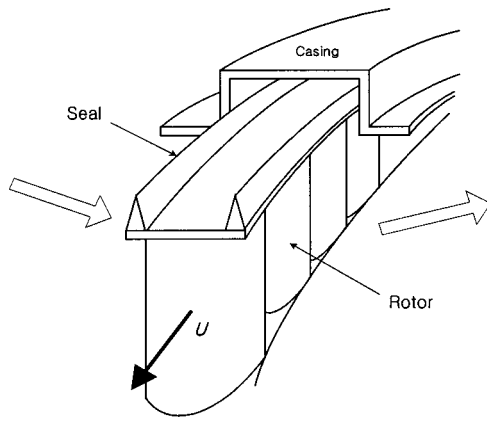


Fig. 1 A cutaway section of a single gland turbine labyrinth seal

inviscid flow is assumed elsewhere. Station 1 is the stator inlet, and Station 2 is the stator exit. Station 3r is the rotor trailing edge, and Station 3 is the rotor exit. Also, the axial, tangential, and radial directions are denoted by x , y , and z , respectively.

To analyze the seal flow, Millsaps and Martinez-Sanchez's model [14] has been adopted. A labyrinth seal is shown schematically in Fig. 3. The seal knife height is h , and the seal's length is l . The sealing gap is δ , and the cavity depth is $h + \delta$. The leakage flow rate is affected mainly by the sealing gap δ , and the radial velocity of the leakage flow re-entering the annulus at the rotor exit is determined by the axial gap a . From Millsap's model, the following equation for the seal leakage mass fraction λ can be obtained.

$$\lambda = \frac{1}{\phi} \frac{\delta_e \mu_e}{H} \sqrt{\frac{2(\delta_i \mu_i)^2}{(\delta_i \mu_i)^2 + (\delta_e \mu_e)^2}} \sqrt{\left(\frac{p_2 - p_3}{\rho U^2}\right) + \frac{\phi^2}{2}} \quad (1)$$

The leakage mass fraction is a function of the operating flow coefficient ($\phi = c_x/U$), contraction coefficients (μ_i and μ_e), sealing gaps (δ_i and δ_e), and the pressure drop across the rotor $(p_2 - p_3)/\rho U^2$.

For steady axisymmetric swirling flows in the seal cavity, the following equation for c_y^*/c_{x0} can be obtained from the tangential momentum equation.

$$\frac{1}{8} \left[f_s \left(\frac{l}{H} + \frac{2a}{H} + \frac{2(h+\delta)}{H} \right) - f_r \left(\frac{l}{H} + \frac{4h}{H} \right) \right] \left(\frac{c_y^*}{c_{x0}} \right)^2 + \left[\lambda + \frac{f_r}{4} \left(\frac{l}{H} + \frac{4h}{H} \right) \frac{U}{c_{x0}} \right] \frac{c_y^*}{c_{x0}} - \left[\lambda \frac{c_{y2}}{c_{x0}} + \frac{f_r}{8} \left(\frac{l}{h} + \frac{4h}{H} \right) \left(\frac{U}{c_{x0}} \right)^2 \right] = 0 \quad (2)$$

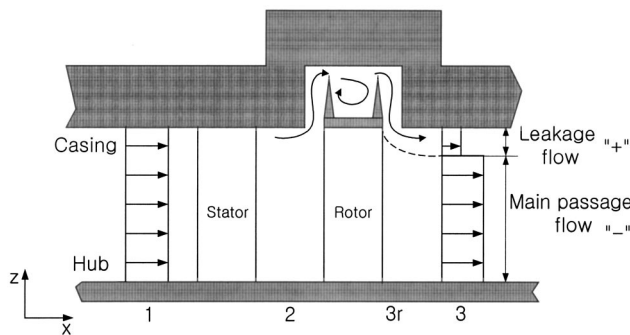


Fig. 2 Schematic view of a turbine stage with shrouded rotor

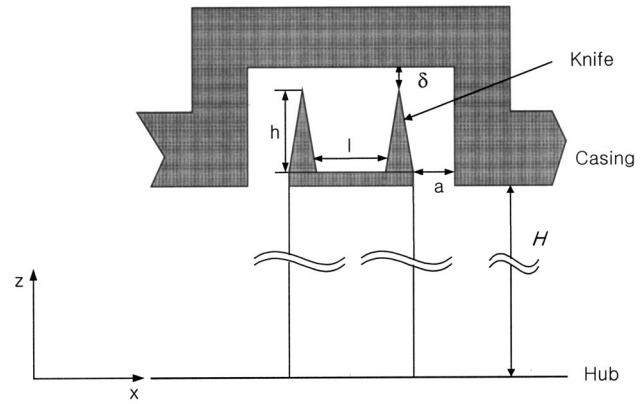


Fig. 3 Labyrinth seal parameters

Since the two friction factors f_r and f_s depend on c_y^*/c_{x0} , an iterative scheme is employed to solve for c_y^*/c_{x0} .

The next section presents a model for analyzing the blade scale effects of the seal leakage flow. The control volume for both leakage and passage flows is shown in Fig. 4. For the leakage flow at Station 3r, the axial velocity is assumed to be zero, and tangential velocity c_{y3r}^+ is assumed to be equal to c_y^* .

$$\frac{c_{x3r}^+}{c_{x0}} = 0 \quad (3)$$

$$\frac{c_{y3r}^+}{c_{x0}} = \frac{c_y^*}{c_{x0}} \quad (4)$$

Due to seal leakage λ , the axial velocity of the main passage flow is

$$\frac{c_{x3r}^-}{c_{x0}} = (1 - \lambda). \quad (5)$$

Also, the assumption of perfect guidance by the blades leads to

$$\frac{c_{y3r}^-}{c_{x0}} = \frac{U}{c_{x0}} - \frac{c_{x3r}^-}{c_{x0}} \tan \beta_3. \quad (6)$$

Next, Bernoulli equation for the main passage flow from location 3r to 3 yields

$$\frac{p_{3r} - p_3}{\frac{1}{2} \rho U^2} = \left(\frac{c_{x0}}{U} \right)^2 \left[\left(\frac{c_{x3}^-}{c_{x0}} \right)^2 - \left(\frac{c_{x3r}^-}{c_{x0}} \right)^2 \right] \quad (7)$$

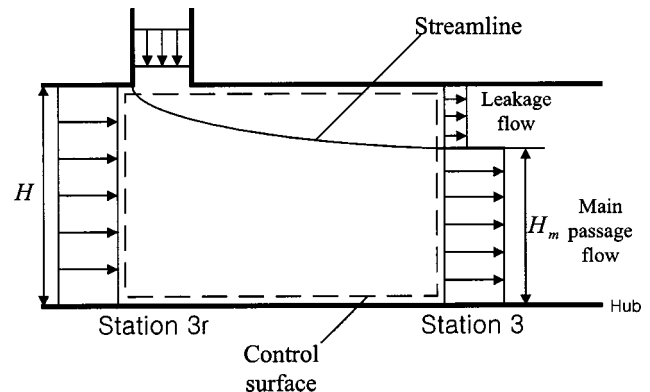


Fig. 4 Control volume for axial momentum analysis

and the axial momentum equation can be used to obtain ($p_{3r} - p_3$) as

$$\frac{p_{3r} - p_3}{\frac{1}{2}\rho U^2} = 2 \left(\frac{c_{x0}}{U} \right)^2 \left[\left(\frac{c_{x3}^-}{c_{x0}} \right)^2 + \frac{\lambda}{1-\lambda} \left(\frac{c_{x3}^+}{c_{x0}} \right)^2 - \left(\frac{c_{x3r}^-}{c_{x0}} \right)^2 \right]. \quad (8)$$

At this point, seal and passage flows can be assumed to either mix out or remain unmixed. In reality, some mixing does occur between the two flows, and for loss calculations, it is important to consider mixing as a source of entropy (The equations for the mixing model are given in Appendix A). However, the seal and passage flows are assumed to remain separate in the current model for the following reasons. First, the focus of the current model is not on loss but on blade loading and flow turning. Second, experimental data from Hunter and Manwaring [5] show that a distinct seal flow layer exists at downstream distances greater than one blade chord length. Third, in the case of an eccentric unshrouded turbine, lateral force predictions from Qiu [15] (mixing model) and Song and Martinez-Sanchez [16,17] (nonmixing model) show that only the predictions from the latter agree with experimental results.

Thus, from the continuity constraint for each stream

$$\frac{c_{x3}^+}{c_{x0}} = \frac{\lambda}{(1-\lambda)(1-A)} \frac{c_{x3r}^-}{c_{x0}} \quad (9)$$

$$\frac{c_{x3}^-}{c_{x0}} = \frac{1}{A} \frac{c_{x3r}^-}{c_{x0}}. \quad (10)$$

Here, $A = H_m/H$ is the ratio of the main passage flow area to the total flow area.

Substituting Eqs. (9) and (10) into Eqs. (7) and (8) yields the following equation for A .

$$\frac{1}{2}A^3 + \left(\left(\frac{\lambda}{1-\lambda} \right)^2 - \frac{3}{2} \right) A^2 + \frac{3}{2}A - \frac{1}{2} = 0 \quad (11)$$

Knowing A and λ , the axial velocity of the leakage flow can be obtained as

$$\frac{c_{x3}^+}{c_{x0}} = \frac{\lambda}{(1-A)} \quad (12)$$

and tangential velocity remains the same,

$$\frac{c_{y3}^+}{c_{x0}} = \frac{c_y^*}{c_{x0}}. \quad (13)$$

Next, the axial velocity of the main passage flow can be determined as

$$\frac{c_{x3}^-}{c_{x0}} = \frac{(1-\lambda)}{A} \quad (14)$$

and tangential velocity is assumed to remain constant,

$$\frac{c_{y3}^-}{c_{x0}} = \frac{c_{y3r}^-}{c_{x0}}. \quad (15)$$

For the main stream, $(p_2 - p_3)/1/2\rho U^2$ is equivalent to the stagnation enthalpy drop, given by Euler's turbine equation, minus the kinetic energy gain.

$$\left(\frac{p_2 - p_3}{\frac{1}{2}\rho U^2} \right)^- = \frac{c_{x0}}{U} \left[\frac{c_{y2}}{c_{x0}} - \frac{c_{y3}^-}{c_{x0}} \right] + \frac{1}{2} \left(\frac{c_{x0}}{U} \right)^2 \left[\left(\frac{c_{x3}^-}{c_{x0}} \right)^2 + \left(\frac{c_{y3}^-}{c_{x0}} \right)^2 \right] - \frac{1}{2} \left(\frac{c_{x0}}{U} \right)^2 \left[\left(\frac{c_{x2}}{c_{x0}} \right)^2 + \left(\frac{c_{y2}}{c_{x0}} \right)^2 \right]. \quad (16)$$

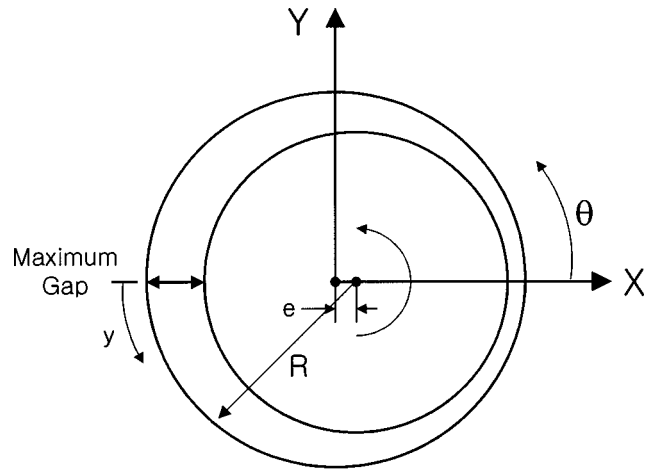


Fig. 5 Coordinate system for the eccentric analysis

Equation (16) is in turn needed for the seal flow analysis in Eq. (1). Therefore, the system of Eqs. (1), (2), (11), and (16) are solved simultaneously.

Eccentric Model. This section presents the eccentric shrouded turbine model. The approach of Song and Martinez-Sanchez [16,17] has been modified. The rotor offset, e , is assumed to be much smaller than the annulus height. Therefore, the sealing clearance is given by

$$\delta = \bar{\delta} + \text{Re}[\hat{e}e^{i(y/R)}] \quad (17)$$

where $\bar{\delta}$ is the mean rotor sealing gap, and y is the distance from the maximum sealing gap in the azimuthal direction (Fig. 5).

The concentric analysis relates the flow variables at $x=0^-$ to those at $x=0^+$, equivalent to far upstream (Station 0) and far downstream (Station 3), respectively, in the concentric model. The concentric analysis shows that the nondimensional flow variables depend on the local sealing gap δ/H and flow coefficient ϕ . Thus, the perturbation quantities in pressure and velocity for both upstream and downstream flows can be determined as shown in Appendix B.

From the perturbations in flow variables, lateral forces, or rordynamic excitation forces, can be predicted. The tangential force exerted on the turbine by the fluid per azimuthal length is defined as

$$f_y = \lambda q(c_{y2} - c_{y3}^+) + (1-\lambda)q(c_{y2} - c_{y3}^-) \quad (18)$$

and its perturbation is given as

$$f'_y = \bar{\lambda} \bar{q} (\bar{c}_{y2} - \bar{c}_{y3}^+) \left[\frac{\lambda'}{\lambda} + \frac{q'}{q} + \frac{c'_{y2} - c'_{y3}^+}{\bar{c}_{y2} - \bar{c}_{y3}^+} \right] + (1-\bar{\lambda}) \bar{q} (\bar{c}_{y2} - \bar{c}_{y3}^-) \times \left[\frac{-\lambda'}{1-\lambda} + \frac{q'}{q} + \frac{c'_{y2} - c'_{y3}^-}{\bar{c}_{y2} - \bar{c}_{y3}^-} \right]. \quad (19)$$

However, nonaxisymmetric rotor sealing gap induces azimuthal flow redistribution, and this flow redistribution results in a non-axisymmetric upstream pressure distribution (Eq. (B5)). The passage flow static pressure in the rotor region $\langle p_h \rangle$ is approximated as the average of pressures at the rotor inlet and exit.

$$\begin{aligned} \langle p_h \rangle &= \frac{p_2 + p_3}{2} = p_2 - \frac{p_2 - p_3}{2} \\ &= p_2 - \frac{1}{2} \frac{\rho}{2} [(c_{x3}^-)^2 \tan^2 \beta_3 - (c_{y2}^- - U)^2] \end{aligned} \quad (20)$$

and its perturbation is given as

Table 1 Pfau et al.'s turbine cascade parameters (Pfau et al. [3])

Parameter	Value
β_2	0 deg
β_3	68 deg
δ_i/H	0.01
δ_e/H	0.01
h/H	0.18
a/H	0.16
l/H	0.55
μ_i	0.65*
μ_e	0.76*

*From Varmes [1].

$$\langle p_h \rangle' = p_2' - \frac{\rho}{2} [\tan^2 \beta_3 \bar{c}_{x3} c_{x3}' - (c_{y2} - U) c_{y2}']. \quad (21)$$

Also, the pressure inside the seal cavity is defined to be

$$\langle p^* \rangle = p_2 - (p_2 - p^*) = p_2 - \rho U^2 \left[\frac{1}{2} \left(\frac{\lambda \phi}{\delta/H \mu_1} \right)^2 - \frac{1}{2} \phi^2 \right] \quad (22)$$

whose perturbation is given as

$$\langle p^* \rangle' = p_2' - \rho U^2 \left[\left(\frac{\lambda \phi}{\delta/H \mu_1} \right)^2 \left(\frac{\lambda'}{\lambda} + \frac{\phi'}{\phi} - \frac{\delta'}{\delta} \right) - \phi \phi' \right]. \quad (23)$$

Thus, the net pressure acting on the shrouded rotor is

$$\langle p \rangle = [1 - (R_t/R_h)] \langle p_h \rangle + (R_t/R_h) \langle p^* \rangle \quad (24)$$

and its perturbation is

$$\langle p \rangle' = [1 - (R_t/R_h)] \langle p_h \rangle' + (R_t/R_h) \langle p^* \rangle'. \quad (25)$$

Upon projection onto the X , Y -axes, the total rotordynamic excitation force coefficients can be obtained as

$$(\alpha_Y + i\alpha_X)_{(total)} = \frac{-\hat{f}_Y + iL\langle \hat{p} \rangle}{|f_Y(e/H)|} \quad (26)$$

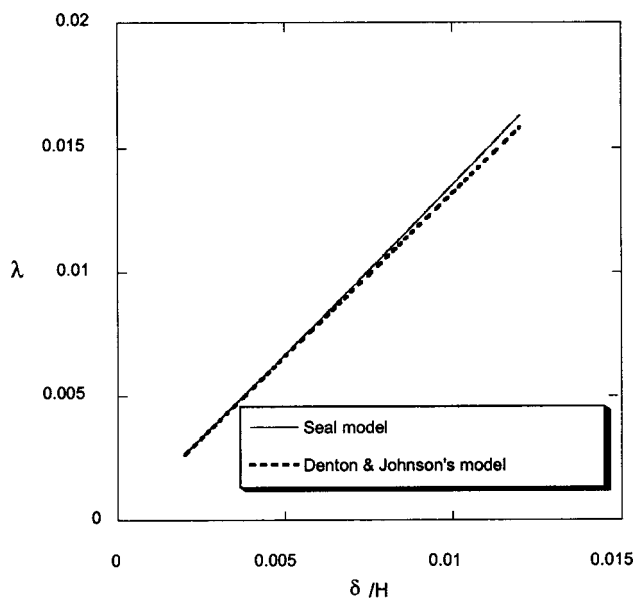


Fig. 6 Leakage mass flow amount versus sealing gap for Pfau et al. [3] annular cascade

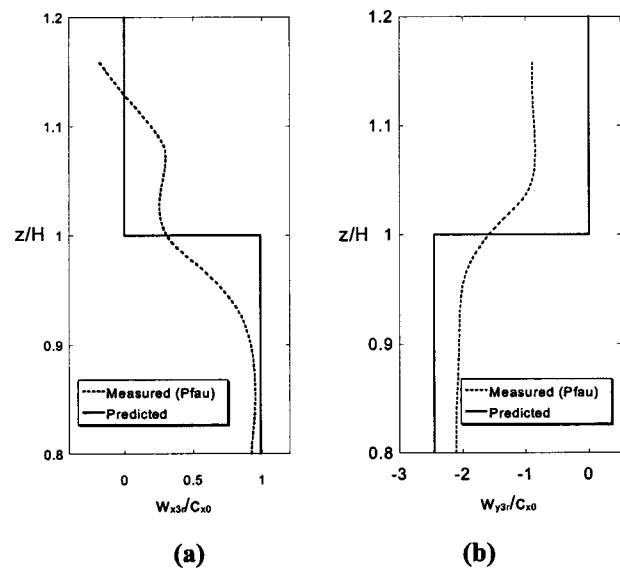


Fig. 7 Radial distributions of relative axial and tangential velocities at seal cavity exit (Station 3r)

The total coefficients $\alpha_{(total)}$ are composed of contributions from tangential force asymmetry f'_y in Eq. (19), and pressure asymmetry $\langle p \rangle'$ in Eq. (25). Forces along the rotor offset are called direct forces and are denoted with a subscript X . Forces perpendicular to the rotor offset are called cross forces and are denoted with subscript Y .

Model Predictions

Concentric Model Predictions. The concentric model has been applied to two turbines to verify its predictions. The first is a hypothetical turbine with blading identical to the annular turbine cascade tested by Pfau et al. [3]. Table 1 lists the blade angles, sealing gaps, seal gland depth, axial gap, and seal pitch used in the experiment. The values of contraction coefficients, μ_i and μ_e , have been calculated according to Varmes [1].

Figure 6 shows the seal leakage mass fractions vs. sealing gap predicted by the new model and Denton and Johnson's model. The two predictions' magnitude, linearity and sensitivity to the sealing gap agree with each other. For each nondimensional sealing gap of 1.0%, the model predicts a leakage mass fraction of about 1.4%. Unfortunately, experimental leakage mass fraction data from Pfau et al.'s [3] turbine do not exist for comparison. However, since Denton and Johnson's model, [2], has been verified by an earlier experiment, such agreement between the new model and Denton and Johnson's model gives credibility to the new model.

Figure 7 shows the radial distributions of the predicted and measured axial velocity (Fig. 7(a)) and relative tangential velocity

Table 2 Peters et al.'s turbine parameters (Peters et al. [4])

Parameter	Value
ϕ_D	0.35
ψ_D	1.2
R_D	0.46
δ_i/H	0.017
δ_e/H	0.017
h/H	0.08
a/H	0.04
l/H	0.6
μ_i	0.65*
μ_e	0.76*

*From Varmes [1].

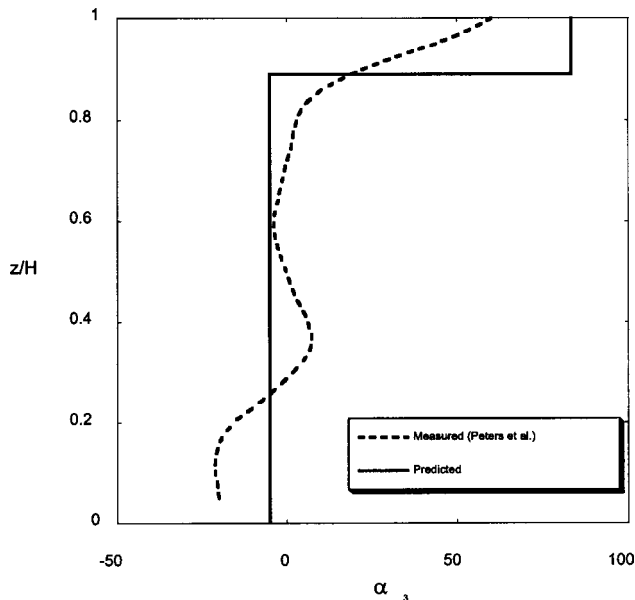


Fig. 8 Absolute flow angle at the rotor exit for Peters et al. [4] turbine

(Fig. 7(b)) at the seal cavity exit (Station 3r). At Station 3r, the model assumes a leakage flow with zero axial momentum. Therefore, the axial velocity of the leakage flow is zero. Also, the axial velocity of the main passage flow is decreased to less than c_{x0} to satisfy the overall continuity constraint. Thus, the seal leakage flow is axially stopped and unloads the entire turbine blade (Fig. 7(a)). The relative tangential velocities for the main passage flow and the leakage flow are predicted to be -2.44 and 0 , respectively (Fig. 7(b)). Since tangential velocities are defined to be positive in the direction of rotation, the seal leakage flow is underturned. The leakage flow remains axial because (i) the upstream velocity is purely axial; and (ii) due to the lack of rotation, the viscous effects do not alter the tangential momentum of the leakage flow. These predictions agree qualitatively with Pfau et al.'s experimental data which have been area averaged in the pitchwise direction. The measured values are different from the predicted values due to the mixing between the passage and leakage flows.

To compare against experimental data from a rotating turbine, Peters et al.'s [4] turbine has been selected because the turbine had a shrouded rotor and flow angle data were available. Table 2

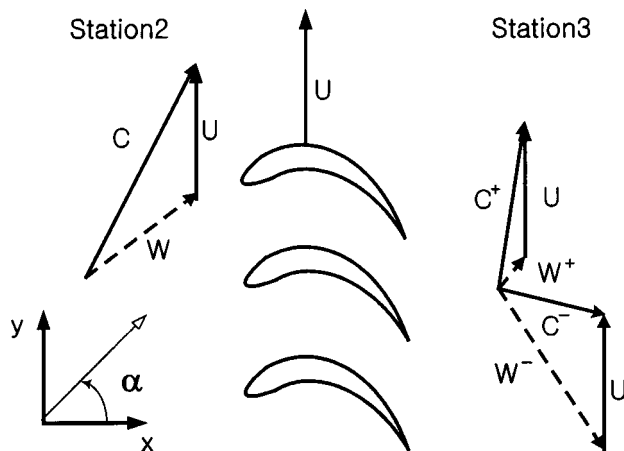


Fig. 9 Velocity vectors at Stations 2 and 3

Table 3 Eccentric shrouded turbine parameters (Song and Martinez-Sanchez [16,17])

Parameter	Value
ϕ_D	0.58
ψ_D	1.5
R_D	0.208
δ_i/H	0.02
δ_e/H	0.02
h/H	0.32
a/H	0.04
l/H	1
μ_i	0.65*
μ_e	0.76*

*From Varnes [1].

lists the turbine's specifications. Again, contraction coefficient values have been adopted from Varnes' empirical relations.

Figure 8 shows the radial distributions of predicted and measured absolute flow angle at the rotor exit (Station 3). The predicted flow angles for the main passage flow and the seal leakage flow are about -5 deg and 80 deg, respectively. The absolute flow angle is positive in the direction of rotation; therefore, the leakage stream is again predicted to be underturned. Though the magnitudes are slightly different, the measured data show similar trends. The measured flow angle for the passage flow is different from predicted value due to deviation and secondary flow. Also, the measured flow angle for the seal leakage flow is different from the predicted value because of entrainment by the passage flow. The predicted upstream and downstream velocity triangles for this turbines are shown schematically in Fig. 9. Although velocity data are unavailable, Fig. 9 clearly shows the underturning and the axial momentum defect of the leakage flow.

Eccentric Model Predictions. This section presents the radius scale predictions for a shrouded turbine. To facilitate comparison with an unshrouded turbine, blading identical to the unshrouded turbine in Song and Martinez-Sanchez [16,17] has been selected. Furthermore, the unshrouded turbine predictions, which have been validated, are also given as reference. The turbine's parameters are given in Table 3. The sealing gaps were assumed to be 2%.

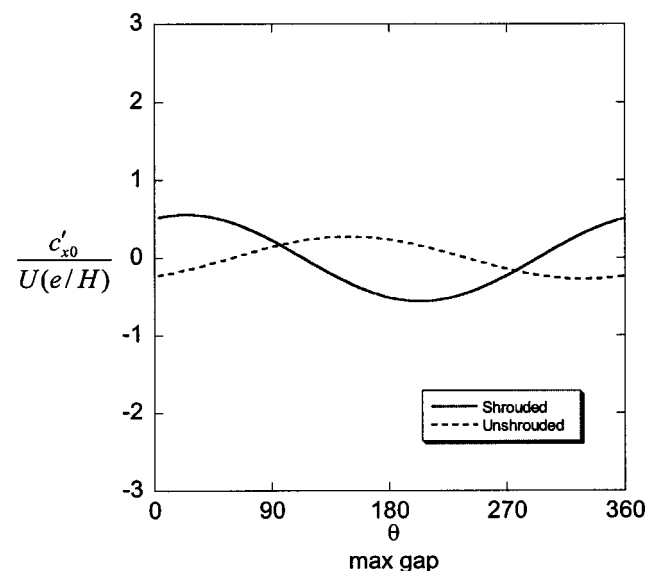


Fig. 10 Upstream axial velocity perturbation versus azimuthal angle; unshrouded prediction from Song and Martinez-Sanchez [16,17]

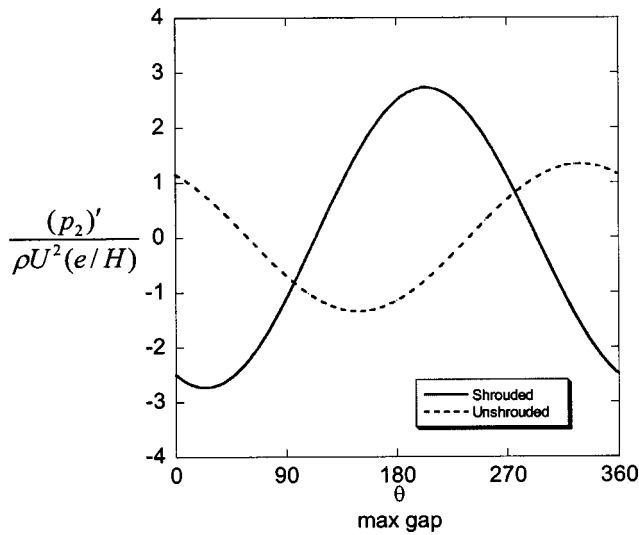


Fig. 11 Upstream pressure perturbation versus azimuthal angle; unshrouded prediction from Song and Martinez-Sanchez [16,17]

The plots of predicted upstream axial velocity perturbation vs. azimuthal location θ are shown in Fig. 10 for both shrouded and unshrouded cases. The minimum gap is at $\theta=0$ deg and the maximum gap is $\theta=180$ deg. The upstream flow migrates preferentially to the minimum gap region for the following reasons. As shown previously, the seal leakage flow is axially retarded relative to the main passage flow, and the amount, or the downstream thickness, of the leakage layer increases linearly with the sealing gap. Thus, the “blockage” effects are larger near the maximum gap and force the flow towards the minimum gap. Then from the Bernoulli relation, the pressure decreases as flow accelerates, and the resulting upstream pressure pattern can be seen in Fig. 11. In the unshrouded turbine, the tip clearance flow is axially accelerated relative to the passage flow, resulting in opposite trends.

The perturbation of the passage flow static pressure in the rotor region $\langle p_h \rangle'$ (i.e. the average of the inlet and the outlet) is plotted versus θ in Fig. 12 for both shrouded and unshrouded cases. In the shrouded case, the pressure has its maximum near the maximum

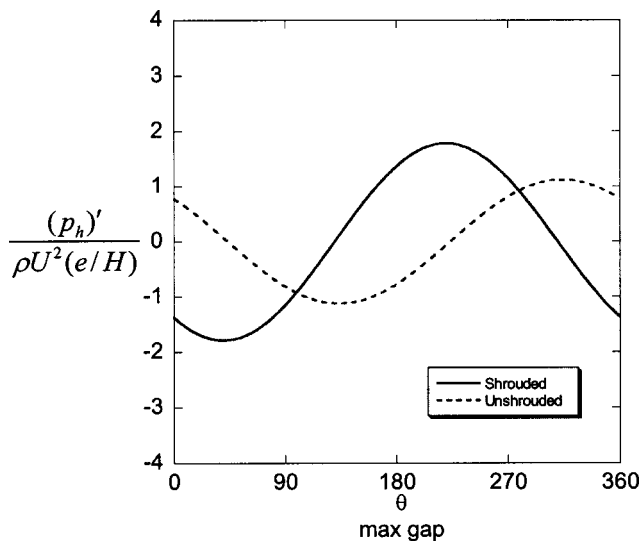


Fig. 12 Perturbation in the rotor region flow pressure versus azimuthal angle; unshrouded prediction from Song and Martinez-Sanchez [16,17]

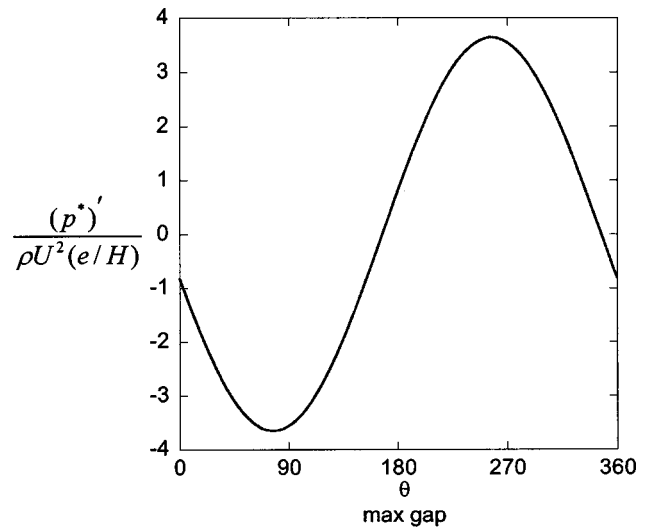


Fig. 13 Pressure perturbation in the seal gland versus azimuthal angle

gap ($\theta=180^\circ$) because $\langle p_h \rangle'$ is mainly affected by the pressure perturbation upstream of the turbine. For the unshrouded turbine, the competing effects between preferential flow migration towards the bigger gap and the consequent λ' places minimum p_h at about 150 deg (Song and Martinez-Sanchez [16,17]). Also, a graph of the pressure perturbation of the seal leakage flow in the seal gland $\langle p^* \rangle'$ versus θ is shown in Fig. 13. This effect exists only in shrouded turbines. With axisymmetric upstream flow, an eccentric seal would introduce nonaxisymmetric mass and tangential momentum flux into the seal cavity, resulting in a maximum seal gland pressure at 270 deg (Millsaps and Martinez-Sanchez [14]). However, Millsaps and Martinez-Sanchez's model does not consider either the interaction between the leakage and the passage flows or the nonaxisymmetric upstream and downstream flow conditions. The current model does consider both effects, and, mainly due to the upstream nonuniformity in pressure (Fig. 11), the maximum seal gland pressure is located ahead of 270 deg (Fig. 13). According to Eq. (25), $\langle p^* \rangle'$ is the dominant pressure

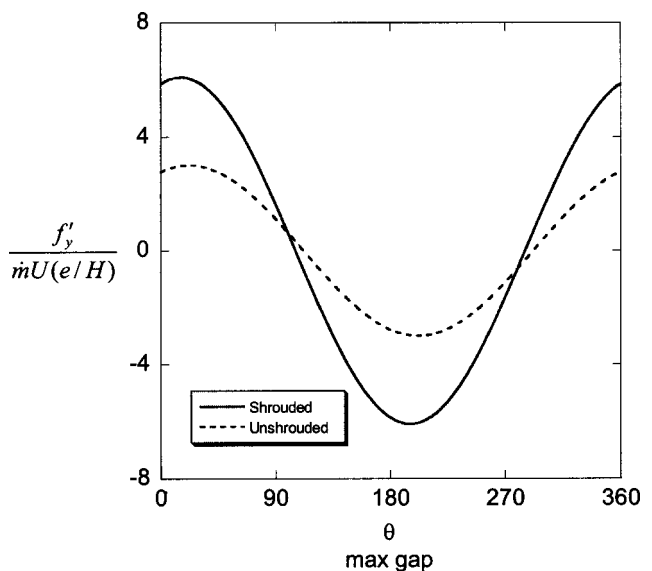


Fig. 14 Rotor blade loading perturbation versus azimuthal angle; unshrouded prediction from Song and Martinez-Sanchez [16,17]

Table 4 Excitation force coefficients for the shrouded turbine

Direction	$\alpha_{(wd)}$	$\alpha_{(p)}$	$\alpha_{(total)}$
<i>X</i>	−1	0.3	−0.7
<i>Y</i>	3.6	3.4	7

Table 5 Excitation force coefficients for the unshrouded turbine (Song and Martinez-Sanchez [16,17])

Direction	$\alpha_{(wd)}$	$\alpha_{(p)}$	$\alpha_{(total)}$
<i>X</i>	−0.9	−1.8	−2.7
<i>Y</i>	1.8	1.4	3.2

effect because the effects of $\langle p_h \rangle'$ are diminished by acting radially inward on the hub and radially outward on the shroud. Furthermore, the amplitude of $\langle p^* \rangle'$ is larger than that of $\langle p_h \rangle'$ because $\langle p^* \rangle'$ is more sensitive to the local leakage flow rate λ' than $\langle p_h \rangle'$.

Figure 14 shows plots of tangential force distribution (Eq. (19)) versus θ for both shrouded and unshrouded turbines. Between the two cases, the phases are similar while the magnitude is larger in the shrouded turbine. The tangential force perturbation is mainly affected by the leakage fraction perturbation, λ' . The phases are similar for both because λ' has its maximum near the maximum gap in both cases. However, the magnitude of λ' is larger in the shrouded turbine because λ' is determined by the perturbation in the axial pressure difference between the rotor inlet and exit. On the other hand, λ' in the unshrouded turbine is determined by the perturbation in blade loading which is less sensitive to eccentricity.

Rotordynamic Coefficient Predictions. The excitation rotordynamic coefficients predicted for shrouded and unshrouded turbines are given in Tables 4 and 5, respectively. The coefficients due to blade loading variation are $\alpha_{(wd)}$'s and those due to pressure variation are $\alpha_{(p)}$'s. Both models predict that the blade loading variation induces a positive cross force, which promotes a forward whirl, and a negative direct force. However, the shrouded turbine model predicts a bigger $\alpha_{Y(wd)}$ because it predicts a much bigger variation in the leakage flow rate λ' . The direct forces due to $\alpha_{X(wd)}$ remain similar. The pressure asymmetry effects are magnified (larger $\alpha_{(p)}$'s) in the shrouded turbine due to the seal effects. In both turbines, $\alpha_{Y(p)}$'s are positive, adding to the cross force due to f_y' . On the other hand, the shrouded turbine model predicts a positive $\alpha_{X(p)}$ while the unshrouded turbine model predicts a negative $\alpha_{X(p)}$. Overall, the current model predicts that the shrouded turbine has a bigger cross force and a smaller direct force than the unshrouded turbine.

Conclusions

This study's objective has been to gain physical understanding of aerodynamic effects caused by labyrinth seals in axial turbines with a shrouded rotor. The following conclusions can be made.

1. A turbine passage flow model incorporating labyrinth seal effects has been developed.
2. Relative to the passage flow, the seal leakage flow is under-turned and axially retarded.
3. The seal leakage flow rate increases linearly with the sealing gap.
4. With eccentricity, the upstream flow migrates azimuthally toward the smaller sealing gap.
5. In a shrouded turbine, the seal gland pressure has its maximum at $\theta \approx 270$ deg, and turbine torque has its maximum at $\theta \approx 0$ deg.

6. Both torque asymmetry and pressure asymmetry contribute to the positive cross force and thus promote a forward whirl, or rotordynamic instability.

Acknowledgments

Support from the EESRI (01-M-02) of the Korea Ministry of Commerce, Industry, and Energy and the Micro Thermal System Research Center of the Korea Science and Engineering Foundation is gratefully acknowledged.

Nomenclature

<i>a</i>	= axial gap
<i>A</i>	= area ratio
<i>c</i>	= absolute flow velocity
<i>D</i>	= turbine mean diameter
<i>D_h</i>	= hydraulic diameter
<i>e</i>	= magnitude of rotor offset
<i>E_i</i>	= eigenvector for downstream perturbations
<i>F_X</i>	= lateral force in the direction of the offset
<i>F_Y</i>	= lateral force perpendicular to the direction of the offset
<i>f</i>	= Darcy friction factor
<i>H</i>	= annulus height
<i>H_m</i>	= passage flow thickness
<i>h</i>	= seal gland depth
<i>l</i>	= seal pitch
<i>p</i>	= pressure
<i>q</i>	= mass flux per unit circumferential length
<i>R</i>	= turbine mean radius
<i>U</i>	= turbine rotational speed at the mean radius
<i>w</i>	= relative flow velocity
<i>X</i>	= direction along the rotor offset
<i>x</i>	= axial flow direction
<i>Y</i>	= direction perpendicular to the rotor offset
<i>y</i>	= azimuthal flow direction
<i>z</i>	= radial flow direction
α	= absolute flow angle
α_X	= direct excitation force coefficient
α_Y	= cross excitation force coefficient
β	= relative flow angle
Δ	= thickness of the seal leakage flow
δ	= sealing gap
ϕ	= flow coefficient; velocity potential
λ	= nondimensional seal leakage flow rate
μ	= contraction coefficient
ρ	= density
τ	= viscous shear stress exerted on the fluid
ψ	= work coefficient

Subscripts

0	= far upstream
1	= stator inlet
2	= stator exit
3 <i>r</i>	= rotor exit near the seal cavity
3	= rotor exit
<i>h</i>	= hub
<i>i</i>	= first seal knife
<i>e</i>	= second seal knife
<i>r</i>	= rotor
<i>s</i>	= stator
<i>t</i>	= tip
0−	= near upstream of stage on the radius scale
0+	= near downstream of stage on the radius scale

Superscripts

−	= main passage flow
+	= seal leakage flow
*	= flow inside the seal cavity
'	= nonaxisymmetric perturbation
—	= azimuthal mean, or axisymmetric value

$\hat{\Delta}$ = complex amplitude

Appendix A

This section presents blade scale equations when the seal and passage flows are assumed to mix. From the continuity equation, the axial velocity at Station 3 is the same as the upstream axial velocity.

$$\frac{c_{x3}}{c_{x0}} = 1 \quad (A1)$$

From the tangential momentum equation, the tangential velocity at Station 3 can be obtained.

$$\frac{c_{y3}}{c_{x0}} = \lambda \frac{c_{y3r}^+}{c_{x0}} + (1 - \lambda) \frac{c_{y3r}^-}{c_{x0}} \quad (A2)$$

Thus, Eq. (16) can be simplified as

$$\frac{p_2 - p_3}{\frac{1}{2} \rho U^2} = \frac{c_{x0}}{U} \left[\frac{c_{y2}}{c_{x0}} - \frac{c_{y3}}{c_{x0}} \right] + \frac{1}{2} \left(\frac{c_{x0}}{U} \right)^2 \left[\left(\frac{c_{y3}}{c_{x0}} \right)^2 - \left(\frac{c_{y2}}{c_{x0}} \right)^2 \right]. \quad (A3)$$

Appendix B

This section presents a summary of the eccentric model developed by Song and Martinez-Sanchez [16,17].

Irrational upstream flow is given as

$$\nabla^2 \phi = 0 \quad (B1)$$

where ϕ is the velocity potential. Near upstream of the stage at $x=0^-$, the axial velocity is

$$c_x(0^-, y, t) = \text{Re}[c_x(-\infty) + \hat{K}_0 e^{i(y/R)}] \quad (B2)$$

where $\hat{K}_0$ is the complex amplitude of axial velocity perturbation as the flow approaches the turbine. Far upstream, $c_x(-\infty) = \bar{c}_{x0}$.

With these conditions, the solution to $\nabla^2 \phi = 0$ yields

$$c_x(x, y, t) = \bar{c}_{x(-\infty)} + \text{Re}[\hat{K}_0 e^{x/R + i(y/R)}] \quad (B3)$$

$$c_y(x, y, t) = \text{Re}[i \hat{K}_0 e^{x/R + i(y/R)}] \quad (B4)$$

Consequently, the upstream pressure is given by

$$p(x, y, t) = p(-\infty) - \text{Re}[\rho(c_x(-\infty) - i \Omega R) \hat{K}_0 e^{x/R + i(y/R)}]. \quad (B5)$$

Downstream of the stage, the flow consists of two regions—the seal leakage flow and the main passage flow. The continuity equation for each stream can be written as

$$\frac{\partial \Delta}{\partial t} + \frac{\partial(c_x^+ \Delta)}{\partial x} + \frac{\partial(c_y^+ \Delta)}{\partial y} = 0 \quad (B6)$$

$$\frac{\partial(H - \Delta)}{\partial t} + \frac{\partial(c_x^-(H - \Delta))}{\partial x} + \frac{\partial(c_y^-(H - \Delta))}{\partial y} = 0 \quad (B7)$$

where H is annulus height and Δ is the thickness of the seal leakage flow. The momentum equations can be written as

$$\frac{\partial \mathbf{c}^\pm}{\partial t} + (\mathbf{c}^\pm \cdot \nabla) \mathbf{c}^\pm + \frac{1}{\rho} \nabla p = 0 \quad (B8)$$

where all vectors are two-dimensional (x, y) .

Assuming $e \ll H$, Δ , c_x^+ , c_y^+ , c_x^- , c_y^- , and p can each be expressed as a mean value plus a small perturbation. For example,

$$\Delta = \bar{\Delta} + \Delta'. \quad (B9)$$

The perturbation part is assumed to be the first harmonic of the angle y/R with an exponentially decaying amplitude, associated with potential effects. For example,

$$\Delta' = \text{Re}[\hat{\Delta} e^{\alpha x + i(y/R)}]. \quad (B10)$$

After substitution of equations such as Eq. (A10) into the Eqs. (A6)–(A8) and linearization, the following system of equations can be obtained:

$$\begin{bmatrix} \alpha \bar{\Delta} & i \bar{\Delta}/R & 0 & 0 & A_{15} & 0 \\ 0 & 0 & \alpha(H - \bar{\Delta}) & i(H - \bar{\Delta})/R & A_{25} & 0 \\ A_{31} & 0 & 0 & 0 & 0 & \alpha \\ 0 & A_{42} & 0 & 0 & 0 & i/R \\ 0 & 0 & A_{53} & 0 & 0 & \alpha \\ 0 & 0 & 0 & A_{64} & 0 & i/R \end{bmatrix} \begin{bmatrix} \hat{c}_x^+ \\ \hat{c}_y^+ \\ \hat{c}_x^- \\ \hat{c}_y^- \\ \hat{\Delta} \\ \hat{p}/\rho \end{bmatrix} = 0 \quad (B11)$$

where

$$A_{15} = A_{31} = A_{42} = \alpha \bar{c}_x^+ + i \frac{\bar{c}_y^+}{R} \quad (B12)$$

$$A_{53} = A_{64} = \alpha \bar{c}_x^- + i \frac{\bar{c}_y^-}{R} \quad (B13)$$

$$A_{25} = -\alpha \bar{c}_x^- - i \frac{\bar{c}_y^-}{R}. \quad (B14)$$

Upon determining eigenvalues α_i and eigenvectors E_i , the non-trivial homogeneous solution to the system of equations above can be written as

$$\begin{pmatrix} \hat{c}_x^+, \hat{c}_x^-, \hat{c}_y^+, \hat{c}_y^-, \hat{\Delta}, \hat{p}/\rho \end{pmatrix} = \sum_{i=1}^5 \hat{K}_i E_i \quad (B15)$$

where the complex constants $\hat{K}_i$'s have to be determined.

In the eccentric model, downstream location $x=0^+$ is Station 3 in the concentric model. Thus, the eccentric perturbation quantities at $x=0^+$ are given as

$$\begin{bmatrix} \hat{c}_x^+ \\ \hat{c}_y^+ \\ \hat{c}_x^- \\ \hat{c}_y^- \\ \hat{\Delta} \\ \hat{p}/\rho \end{bmatrix}_{x=0^+} = \left[\frac{\hat{c}_x(0^-)}{U} \frac{\partial}{\partial \phi} + \left(\frac{e}{H} \right) \frac{\partial}{\partial \bar{H}} \right] \begin{bmatrix} c_{x3}^+ \\ c_{y3}^+ \\ c_{x3}^- \\ c_{y3}^- \\ \Delta \\ p/\rho \end{bmatrix}_{\text{Station 3}} \quad (B16)$$

where the partial derivatives are to be calculated from the concentric model.

Now, Eqs. (B15) and (B16) can be set equal to each other. Then, the terms due to (e/H) act as forcing terms which determine the particular complex amplitudes $\hat{K}_i$'s. Thus, upstream and downstream perturbations in velocity and pressure can be determined.

References

- [1] Varmes, G., 1960, "A Fluid Mechanics Approach to the Labyrinth Seal Leakage Problem," ASME J. Basic Eng., **82**(2), pp. 265–275.
- [2] Denton, J. D., and Johnson, C. G., 1976, "Tip Leakage Loss of Shrouded Turbine Blades," CEGB Report R/M/N848.
- [3] Pfau, A., Treiber, M., Sell, M., and Gyarmathy, G., 2000, "Flow Interaction From the Cavity of an Axial Turbine Blade Row Labyrinth Seal," ASME Paper No. 2000-GT-481.
- [4] Peters, P., Breisig, V., Giboni, A., Lerner, C., and Pfost, H., 2000, "The Influence of the Clearance of Shrouded Rotor Blades on the Development of the Flowfield and Losses in the Subsequent Stator," ASME Paper No. 2000-GT-478.
- [5] Hunter, Scott D., and Manwaring, Steven R., 2000, "Endwall Cavity Flow Effects on Gaspath Aerodynamics in an Axial Flow Turbine—Part 1: Experimental and Numerical Investigation," ASME Paper No. 2000-GT-651.
- [6] Wallis, A., Denton, J., and Demargne, A., 2000, "The Control of Shroud Leakage Flows to Reduce Aerodynamic Losses in a Low Aspect Ratio, Shrouded Axial Flow Turbine," ASME Paper No. 2000-GT-475.

- [7] Den Hartog, J. P., 1956, *Mechanical Vibrations*, 4th Ed., McGraw-Hill, New York, pp. 319–321.
- [8] Thomas, H. J., 1958, “Unstable Natural Vibration of Turbine Rotors Induced by the Clearance Flow in Glands and Blading,” *Bull. de l’A.I.M.*, **71**(11/12), pp. 1039–1063.
- [9] Alford, J., 1965, “Protecting Turbomachinery From Self-Excited Rotor Whirl,” *ASME J. Eng. Gas Turbines Power*, **87**, pp. 333–334.
- [10] Ehrich, F. F., 1976, “Self-Excited Vibration,” *Shock and Vibration Handbook*, 2nd Ed., McGraw-Hill, New York, pp. 1–5.
- [11] Ehrich, F. F., and Childs, D. W., 1984, “Self Excited Vibration in High Performance Turbomachinery,” *Mech. Eng. (Am. Soc. Mech. Eng.)*, pp. 66–79.
- [12] Kostyuk, A. G., 1972, “A Theoretical Analysis of the Aerodynamic Forces in the Labyrinth Glands of Turbomachines,” *Teploenergetica*, **19**(11), pp. 29–33.
- [13] Iwatsubo, T., 1980, “Evaluation of Instability Forces in Labyrinth Seals in Turbines or Compressors,” *NASA CP-2133*, pp. 139–169.
- [14] Millsaps, K., and Martinez-Sanchez, M., 1994, “Dynamic Forces From Single Gland Labyrinth Seals—Part I: Ideal and Viscous Decomposition,” *ASME J. Turbomach.*, **116**, pp. 686–693.
- [15] Qiu, Y. J., 1985, “An Investigation of Destabilizing Blade Tip Forces for Shrouded and Unshrouded Turbines,” S.M. thesis, Department of Aeronautics and Astronautics, M.I.T., Cambridge, MA.
- [16] Song, S. J., and Martinez-Sanchez, M., 1997, “Rotordynamic Forces Due to Turbine Tip Leakage—Part I: Blade Scale Effects,” *ASME J. Turbomach.*, **119**, pp. 695–703.
- [17] Song, S. J., and Martinez-Sanchez, M., 1997, “Rotordynamic Forces Due to Turbine Tip Leakage—Part II: Radius Scale Effects and Experimental Verification,” *ASME J. Turbomach.*, **119**, pp. 704–713.
- [18] Denton, J. D., 1993, “Loss Mechanism in Turbomachines,” *ASME J. Turbomach.*, **115**, pp. 621–656.

Yangbing Zeng

C. F. Lee¹

e-mail: cflee@uiuc.edu

Department of Mechanical and Industrial
Engineering,
University of Illinois at Urbana-Champaign,
140 Mechanical Engineering Building,
1206 West Green Street,
Urbana, IL 61801

Computations of Air/Fuel Preparation Process in a Port-Injected Spark-Ignition Engine During Cold-Starting Phase

A numerical study has been performed of the air/fuel preparation process in a cold-starting port-injected spark-ignition engine. The latest models were implemented for spray impingement and multicomponent vaporization of the droplet and wall film accounting for finite diffusion in the liquid. The infinite diffusion model was found insufficient for predicting vaporization in this engine, and the single-component fuel representation yields results significantly different from those from the multicomponent one. The operating parameters studied included injection timing, swirl, speed, target path, enrichment, and fuel accumulation. In-cylinder measurements were compared and good agreement was achieved. Detailed quantitative analysis of the air/fuel preparation of the engine was reported. [DOI: 10.1115/1.1760524]

Introduction

Emission regulation has become more stringent due to environmental concerns. For port fuel-injected (PFI) spark-ignition engines, a disproportional amount of unburned hydrocarbon (HC) emission occurs during the cold-starting phase. About 65% to 80% of the total tailpipe HC emission was measured during the first 5 minutes in a 40-minute test following the federal test procedure, [1]. Extensive studies have been performed to identify the cause and search for a possible remedy both numerically and experimentally. Three factors are identified as the most significant contributors: (1) ineffective catalyst, (2) fuel enrichment, and (3) incomplete combustion due to the nonhomogeneity of the air/fuel mixture, [2]. The primary cause for the last two factors is the relatively cold surface temperature at the intake port and the cylinder wall, [3]. A popular design for an injection path is to target the spray at the valve or the intake port wall in order to take advantage of high surface temperature when the engine is fully warmed up. During the cold-starting phase, however, the surface temperature of the valve or wall is fairly low. As a consequence, most of the deposited fuel converts into a thin fuel film, and only a small fraction of this fuel film is brought into the cylinder by vaporization, stripping and back-flow atomization. It is estimated that only 30% to 40% of injected fuel enters the cylinder in the same cycle, [4,5]. To compensate for this fuel loss, extra fuel is injected in order to formulate a combustible air/fuel vapor mixture (i.e., fuel enrichment).

It is well established that promoting of fuel vaporization and controlling the amount of the fuel induced into the cylinder are the keys to overcoming cold-starting problems through optimizing engine operation and design parameters. The most related parameters include swirl intensity, turbulence, injection timing and target position. Higher swirl intensity increases the fuel vaporization rate and improves mixture uniformity, [6]. Stronger turbulence produces more uniform mixture due to better mixing, [7,8]. Re-

tarding injection time until the intake valve opens significantly reduces intake port wall wetting, however, HC emission increases rather than decreases because of more severe in-cylinder wall wetting, [3]. Fuel volatility also plays an important role in determining mixture quality, and fuels with lower boiling points have better cold-starting ability, [9].

With much improvement in computer resource and numerical model capacity, CFD becomes an important tool in engine design and research. Extensive numerical studies have been carried out for individual sub-processes (e.g., vaporization, atomization, spray/wall impingement) and for the complete air/fuel mixture preparation process. Among those sub-processes, one of the most important processes is fuel vaporization. Traditional computations were made with the assumption of a single component fuel for simplicity although actual fuels are mixtures of hydrocarbons (i.e., multicomponent). Under this assumption, the faster vaporization of light components and the slower vaporization of heavy components are not captured. Consequently, the mixing and combustion process cannot be adequately represented. Different from a single-component fuel, vaporization of multicomponent fuel is characterized by a more transient process in that the fuel composition always varies spatially and transiently during the whole lifetime, [10–12]. In the literature, there are extensive studies on vaporization of multicomponent droplets and films (two forms of fuel that exist in engines), and several kinds of models have been introduced, [12–20]. However, most of these models are either too costly in computational resource or insufficiently accurate, [10–12]. Zeng and Lee [10–12] recently developed two multicomponent vaporization models for droplets and films, respectively. The models include effects of important mechanisms involved, such as preferential vaporization, finite diffusion, internal circulation, surface regression, and nonideal behavior in high-pressure environments. In particular, these two models account for the nonuniform distributions inside the droplet/film through tracing the surface and average values for temperature and mass fraction, and thus their computational cost is only slightly higher than those of infinite diffusion type models but is much lower than those of other detailed and complicated models in the literature with comparable accuracy.

Another important process is spray/wall impingement. For port-injected engines, a significant amount of droplets hit the wall of

¹To whom correspondence should be addressed.

Contributed by the Internal Combustion Engine Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the ICE Division, May 2001; final revision received Nov. 2003. Associate Editor: D. Assanis.

the intake port and the cylinder after they leave the injector, especially with closed-valve injection. Much progress has been made on modeling the outcome of impingement and subsequent film dynamics, [15,16]. Consistent with parcel representation for sprays, a model with a particle method was developed by O'Rourke and Amsden [16] for impingement, film dynamics and the effect of film on mass, momentum, and energy transports. Trujillo et al. [21] further improved this model by implementing into it a formula for a deposition rate with the effect of roughness and a more detailed treatment for secondary droplets from splashing breakup.

A numerical study is performed in this study to quantify effects of engine parameters on air/fuel preparation quality at the cold-starting phase. Distinct from most of prior studies, the fuel used is a multicomponent fuel with similar distillation characteristics to those of gasoline. The studied parameters include injection timing, swirl intensity, engine speed, target path, fuel enrichment, and fuel accumulation. The preparation quality is analyzed in terms of fuel vaporization, wall wetting and fuel vapor in-cylinder distribution. The computed engine is a Ford production engine. In the rest of this paper, a brief description of the models used will be given first, followed by computational details. In the Results and Discussions section, effects of various parameters on air/fuel preparation processes will be discussed in detail, and comparisons with in-cylinder fuel distribution measurements will also be presented.

Numerical Model

KIVA-3V2, [22,23], is used as the code in this study, which has been widely used in engine computations due to its capacity in modeling turbulence, spray and chemical reaction and its flexibility in handling the engine-like geometry. However, this code is limited to a single component fuel. This limitation is removed by the implementation of two newly developed vaporization models, [10–12]. In addition, the impingement model is replaced by the model developed by Trujillo et al. [21], which accounts for splashing. Since the details of the models in the original KIVA-3V2 code can be found in the reports of Amsden [22,23], only the newly implemented models are presented here. These new models and the resulting code have been verified extensively against experimental data on droplet lifetime, surface temperature history, and film height over a wide range of ambient temperature, pressure and relative velocity between fuel and gas. The details can be found in Refs. [10–12].

Droplet Vaporization Model. Due to the diffusion resistance inside the droplet and preferential vaporization, the property distributions are nonuniform in general. Two extreme modes of vaporization could take place depending on the rate of droplet surface regression (vaporization rate), namely batch distillation process and diffusion-limited process. The batch distillation process occurs when the vaporization is very slow, and it can be well described by the infinite diffusion model. The diffusion-limited process takes place when the vaporization rate is very large, in which the core composition of the droplet is essentially frozen. Unfortunately, the vaporization occurring in engines is neither batch distillation nor diffusion limited modes. As a matter of fact, both the combined effect of preferential vaporization and finite diffusion (resulting in nonuniform temperature and mass fraction distribution) and the effect of surface regression (altering the temperature and mass distribution) are important. As such, the temperature and mass fraction are not uniform within the droplet. To account for this nonuniformity, conventional model needs to solve a spatially dependent partial equation, which requires tremendous computational time. In the present study, an ordinary differential equation with respect to time t is used to describe the evolution of this non-uniformity represented by the difference between the surface and average values, [10],

$$\frac{d\Phi_d}{dt} = \frac{0.2E(\text{Pe}_V)\phi R/D_\ell - \Phi_d}{E(\text{Pe}_V)R^2/s_1^2 D_\ell} \quad (1)$$

where Φ is either the temperature T or mass fraction Y_i , ϕ is either the heat or species mass flux at the droplet surface. E is the correction factor for the effect of surface regression, which was quantified by a nondimensional variable Pe_V defined as

$$\text{Pe}_V = R \sum \omega_i / D_\ell \quad (2)$$

which represents the ratio of either the thermal or mass diffusion time scale to the surface regression time scale. Mathematically, $E(\text{Pe}_V)$ is defined as the ratio of the difference of surface and average values at the specific Pe_V to the difference at $\text{Pe}_V=0$ subject to the same boundary conditions, [10]. Zeng and Lee [10] developed a mathematical formula for $E(\text{Pe}_V)$ based on a semi-analytical solution. In addition, to account for the enhancement of mass and thermal diffusion from internal circulation, an enlarged diffusivity is used to replace the physical diffusivity as suggested by Jin and Borman [19].

Film Vaporization Model. Based on the boundary layer equations and the Prandtl mixing-length theory, O'Rourke and Amsden [16] developed a modified wall function to determine the transfer rate between the film and the adjacent gas. Following the same strategy, Zeng and Lee [11] extended it to include multicomponent fuel effects. The species vaporization rate ω_i is given by

$$\omega_i = H Y_i s_i B_i, \quad (3)$$

where H is the overall diffusivity above the film having a formula similar to the wall function, B is the transfer number, and s is used to account for the effect of high vaporization on diffusion.

In general, the mass fraction and temperature distributions within the film are nonuniform because of finite thermal and mass diffusion speeds. The full set of governing equations, at least in the direction normal to film for mass and energy transport, are required to solve in order to provide accurate values for determining the vaporization rate. Zeng and Lee [11] reduced these governing equations to a set of ordinary differential equations through the usage of a third-order polynomial to simulate the temperature and mass fraction profiles. With this method, the surface temperature is given by

$$\frac{dT_s}{dt} = 6 \frac{\alpha_\ell}{\delta^2} \left(\frac{q}{k_\ell} \delta - 3T_s - T_w + 4T_m \right). \quad (4)$$

The difference between the surface and average mass fractions for component i is given by

$$\frac{d(Y_{i\ell s} - Y_{i\ell m})}{dt} = \frac{4\Gamma_{\ell i}}{\rho \delta^2} [\eta - 3(Y_{i\ell s} - Y_{i\ell m})]. \quad (5)$$

The above equations combined with overall mass and energy conservation equations will give instantaneous mass fraction and surface temperature for determination of the film vaporization rate.

Phase Equilibrium and Property Evaluation. In a high-pressure environment, the assumption of idea gas and liquid behavior is invalid. Consequently, Raoult's law is not appropriate to give the thermo-dynamical equilibrium relation between gas and liquid phases at the droplet/film surface. To account for the effect of nonideal behavior at high pressure, the Peng-Robinson equation of state was used to calculate the compressibility factor and fugacity for gas and liquid phases, [24]. By equating species fugacities of liquid and gas phases, the phase equilibrium relation can be determined. Accurate property evaluation is crucial for computations of multicomponent fuel vaporization. In this study, all properties are temperature and composition-dependent with the correction of high pressure if applicable, [24].

Table 1 Engine specification and injection parameters

Stroke	7.9 cm	Injection duration	11 msec
Bore diameter	8.2 cm	Cone angle	10 deg
Compression ratio	9.7	Cone angle thickness	10 deg
Saunter mean diameter	210 μ m	Injection velocity	23 m/sec

Impingement Model. The impingement model developed by Trujillo et al. [21] was used in this study. Based on the injection conditions, the droplets splash, spread, rebound, or stick. The outcome of impingement depends on a nondimensional variable, K . The value of K for the onset of splash is determined by film height and roughness. When splash occurs, part of the mass is converted into film while part of the mass becomes secondary droplets. Different from the method in the KIVA-3V2, addition parcels are added for secondary droplets.

Computational Detail

All the above models have been implemented into the KIVA-3V2 code with extensive validations, [10–12,21]. Validations include comparing computed droplet life time, drop size, and surface temperature temporal history, and film height against available experimental data, details can be found in [10–12]. In this study, the computed engine is a Ford 2.5L, V-6 production engine that has been investigated intensively in the Engine Laboratory in the University of Illinois at Urbana-Champaign, [3,25]. The major parameters for this engine are listed in Table 1. The multicomponent fuel is composed of 15% isopentane, 20% hexane, 45% isooctane, and 20% decane by mass producing a distillation curve similar to that of a California Phase II gasoline, [3], and isooctane is used for the single-component fuel computations.

The paper presents results for 12 cases with different operating conditions and computational parameters. The varied conditions include injection timing, swirl intensity, engine speed, target path and the amount of the injected fuel. High swirl is accomplished by deactivating the secondary intake valve. Note that the spray was injected only in the primary intake port. Closed-valve injection starts at 0 deg ATDC (end of the compression stroke, all crank angles hereafter are defined relative to this reference time), while open-valve injection starts at 360 deg ATDC when the intake valve is fully open. The primary target path, Target 1 is located near the center (valve stem) of the primary valve top surface when the valve fully opens and in the same plane as the middle plane of the primary intake valve. Target 2 is obtained by rotating the injection angle upward by 4.4 deg in the middle plane, therefore the impingement location is in the front half of the valve surface closer to the spark plug. The injected spray has a solid cone shape, a spray angle of 10 deg and an SMD of 210 μ m, [21]. The computational details are summarized in Table 2. The mesh is generated using the KIVA software, and each individual cell size is around $2 \times 2 \times 2$ mm³.

Results and Discussion

Fuel Representation and Vaporization Model. The importance of the fuel vaporization model and fuel representation was first examined. Three computations (Cases 1, 2, and 3) were conducted for closed-valve injection with different fuel representations and vaporization models. Figure 1 shows the variation of the total and in-cylinder fuel vapor mass (normalized by the injection mass) versus crank angle. The vaporization rate decreases gradually with increasing crank angle before the intake valve opens. This decrease is primarily due to most of the spray being converted into film particles. Right before the intake valve opens, little vaporization takes place because the film on the intake port wall and the valve stem approaches equilibrium with the surrounding gas. Once the intake valve opens, the intake charge motion significantly increases the vaporization rate by transporting the accumulated vapor mass and part of the film mass into the

Table 2 Computational parameters for each case

	Injection Timing	RPM	Injected Fuel	Swirl	Target	Model	Fuel
Case 1	Closed	1200	2.34 mg	Low	1	Present	Multi
Case 2	Closed	1200	2.34 mg	Low	1	Infinite	Multi
Case 3	Closed	1200	2.34 mg	Low	1	Present	Single
Case 4	Open	1200	2.34 mg	Low	1	Present	Multi
Case 5	Closed	1200	2.34 mg	High	1	Present	Multi
Case 6	Open	1200	2.34 mg	High	1	Present	Multi
Case 7	Closed	200	2.03 mg	Low	1	Present	Multi
Case 8	Closed	1200	2.34 mg	Low	2	Present	Multi
Case 9	Open	1200	2.34 mg	Low	2	Present	Multi
Case 10	Closed	1200	9.36 mg	Low	1	Present	Multi
Case 11	Closed	1200	9.36 mg	Low	1	Present	Single
Case 12 (Multi-Cycle)	Closed	1200	2.34 mg per cycle	Low	1	Present	Multi

cylinder. However, this promotion of vaporization diminishes gradually after the intake valve fully opens. Since little liquid fuel enters the cylinder, the promotion of vaporization due to the raised temperature during compression is not observed. The decrease in in-cylinder fuel vapor mass is because some vapor is pushed back into the intake port during the initial stage of the compression stroke around 540 deg CA when the intake valve is still open.

Compared to the single component fuel computation (Case 3), the multicomponent fuel computation (Case 1) produces a much faster vaporization rate initially due to the faster vaporization of light components. However, vaporization slows down at later times because the amount of light components diminishes. At the end of the compression stroke, the two fuel representations yield similar total vaporized fuel mass and the induced in-cylinder fuel vapor mass. Contrasting the multicomponent models, the infinite diffusion model yields more vapor than the present model. This is because the infinite diffusion model (with uniform temperature and concentration) overestimates the surface temperature of liquid film under cold-starting conditions resulting in a faster vaporization rate.

The vapor mass of a light component (pentane) and a heavy component (octane) using a multicomponent fuel representation is shown in Fig. 2. It is seen that the infinite diffusion model overestimates the vaporization rate for light components, underesti-

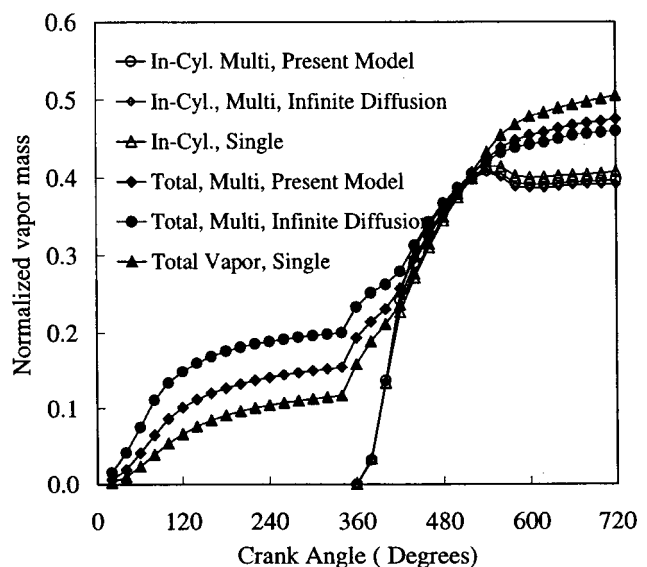


Fig. 1 Variation of fuel vapor mass versus crank angle (effect of fuel representation and vaporization model Cases 1, 2, and 3)

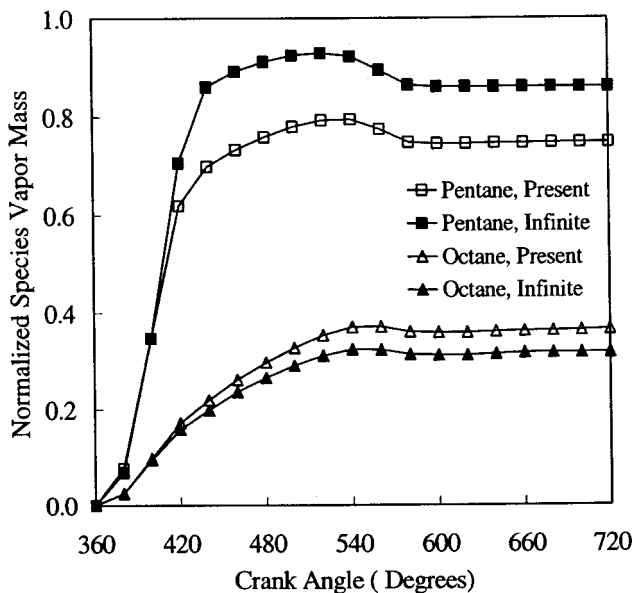


Fig. 2 Variation of species fuel vapor mass versus crank angle (Cases 1 and 2)

mates for heavy components. Due to finite diffusion, the actual surface mass fraction is lower than the average mass fraction for light components, and the reverse is true for heavy components. It is also shown that fuel volatility plays an important role in the vaporizing behavior of the PFI engine. For Case 2, nearly 95% of the pentane vaporized, and in contrast, only around 30% of the octane vaporized. The amount of in-cylinder fuel vapor and fuel film left at the end of the compression stroke is shown in Table 3. For the fuel film, the mass fraction of the heaviest component, decane, is almost double that of the initial composition, while that of the lightest component pentane is only one-seventh. It is worth to point it out that this trend is reverse for the vapor phase.

The reason for smaller difference at later times between the single component and multicomponent fuel representation is the reduction in the mass of light components. Under a fuel enrichment condition (typical of cold-starting engine operations), there is a greater mass of light components, and thus it would be expected that the difference between the two fuel representations would become larger. This expectation is supported by the comparison between Cases 10 and 11 shown in Fig. 3, where four times stoichiometric fuel was injected. Nearly 30% difference is observed for the in-cylinder fuel vapor mass using the two fuel representations. These results indicate that it is very inaccurate to use a single component fuel to represent fuel vaporization for cold starting computations. Hereafter, all computations were made with a multicomponent fuel representation and the present vaporization models.

Table 3 Composition of in-cylinder fuel at the end of compression stroke

	Case 1 Vapor	Case 2 Vapor	Case 3 Vapor	Case 1 Liquid	Case 2 Liquid	Case 3 Liquid
C5 (%)	28	33		2	2	
C6 (%)	29	29		8	10	
C8 (%)	41	36	100	41	44	100
C10 (%)	1	1		49	44	
Total (mg)	9.29	9.16	9.54	0.48	0.55	0.425

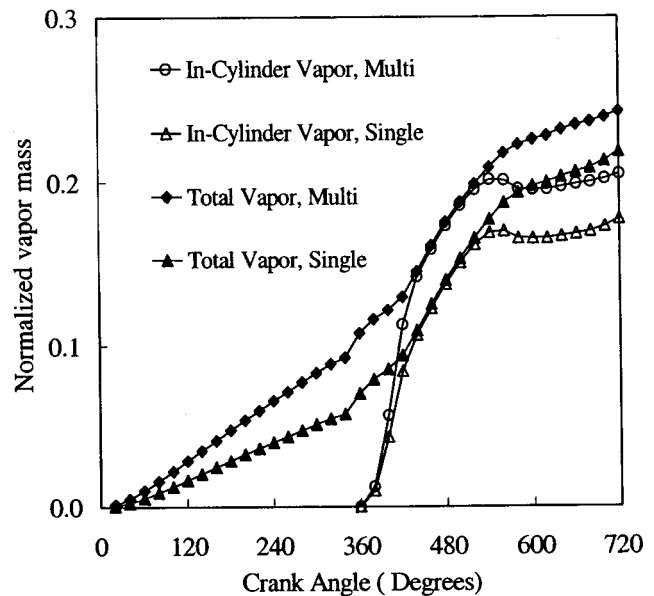


Fig. 3 Variation of fuel vapor mass versus crank angle (effect of fuel representation and vaporization model under fuel enrichment, Cases 10 and 11)

Injection Timing. Now, we proceed to present the effects of various engines operating conditions on air/fuel mixture preparation starting from injection timing. Whenever it is possible, computational results for fuel distribution were compared with the published exciplex fluorescence images, [3,25], for in-cylinder vapor distribution to further verify the models.

The typical injection event occurs as the intake valves close, while there is theoretical interest on open-valve injection (i.e., the injection starts after the intake valves open). The liquid entry is completely different for these two different injection timings as shown in Figs. 4 and 5. For closed-valve injection, most liquid fuel stays on the intake port walls and there are only very few air-borne droplets. For the open-valve injection, part of the injected spray hits the intake port wall, the valve stem and the front corner of the valve top surface, and some droplets enter directly the cylinder. Some droplets hitting the wall splash and produce smaller droplets; which are brought into the cylinder by the intake charge. Some droplets entering the cylinder hit the exhaust valve or the cylinder wall on the exhaust side.

The in-cylinder total liquid and vapor mass for different injection timing are shown in Fig. 6. The increase in liquid mass oc-

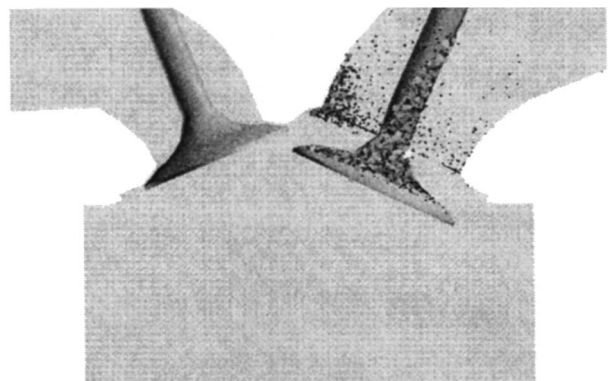


Fig. 4 Liquid entry for closed-valve injection at 430 deg CA (Case 1)

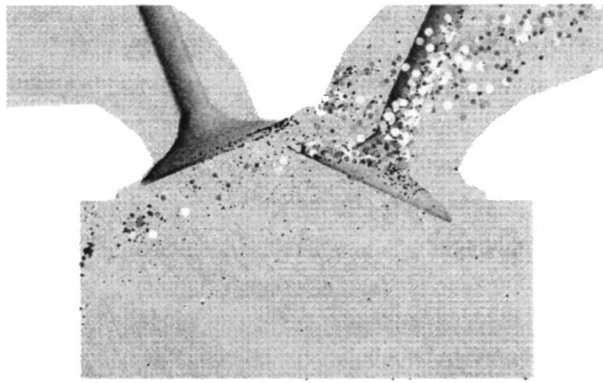


Fig. 5 Liquid entry for open-valve injection at 430 deg CA (Case 4)

curing between 620 deg CA and 640 deg CA is due to the closure of the intake valve. Some film on the valve seat is squeezed into the cylinder or the intake port depending on its position. The general trend shown here is that more total fuel is induced into the cylinder using open-valve injection than closed-valve injection because of the spray directly entering the cylinder. Since more liquid enters the cylinder, the air/fuel mixture quality is degraded. This liquid is a significant source of inhomogeneities in the vapor fuel distribution, and could result in higher hydrocarbon emissions.

Swirl Intensity. It has been reported that swirl can significantly promote air/fuel mixing. Its effect for closed-valve injection is analyzed here by examining the results for Cases 1 and 5. Figure 7 shows the in-cylinder total liquid and vapor mass for these two cases. It is apparent that high swirl brings more vapor fuel into the cylinder. This is due to the nearly doubled intake charge velocity through the primary intake port in which the injection takes place. Most of the in-cylinder vapor comes from the vaporization of film along the valve and intake port wall surfaces. The larger intake charge velocity of the high swirl case enhances the convective transport above the film resulting in higher vaporization rates. The effects of swirl for open-valve injection are

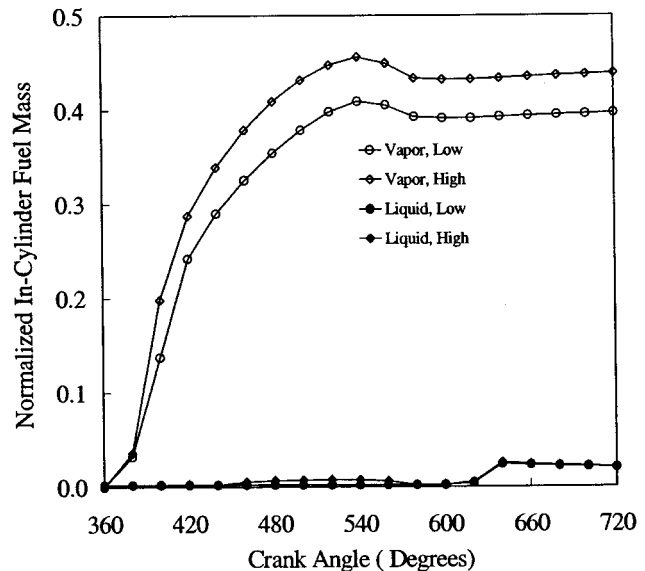


Fig. 7 Variation of in-cylinder fuel mass versus crank angle (effect of swirl intensity for closed-valve injection, Cases 1 and 5)

shown in Fig. 8 (Cases 4 and 6). Similar effects on the in-cylinder vapor were observed in these open-valve cases.

The fuel vapor distribution evolutions for the closed-valve injection with low and high swirl cases are shown in Fig. 9 (Case 1) and Fig. 10 (Case 5). For comparison, the results from single-shot exciplex fluorescence measurements, [25], are also plotted. All images are for the vapor signal intensity corresponding to hexane on the axial plane 5 mm below the bottom of the cylinder head. For both cases, the vapor first appears under the exhaust valve and then moves to other regions by means of bulk charge motion and mass diffusion. In the high swirl case, a clockwise swirl flow pattern brings fuel vapor everywhere and produces a better mixing. For the low swirl case, the fuel vapor tends to be confined at the top half of the plane. The computational images agree reasonably well with those from experiments, except during the initial

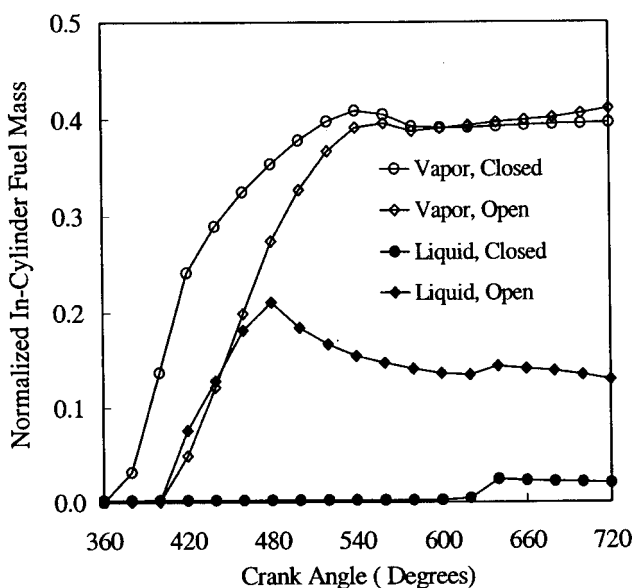


Fig. 6 Variation of in-cylinder fuel mass versus crank angle (effect of injection timing, Cases 1 and 4)

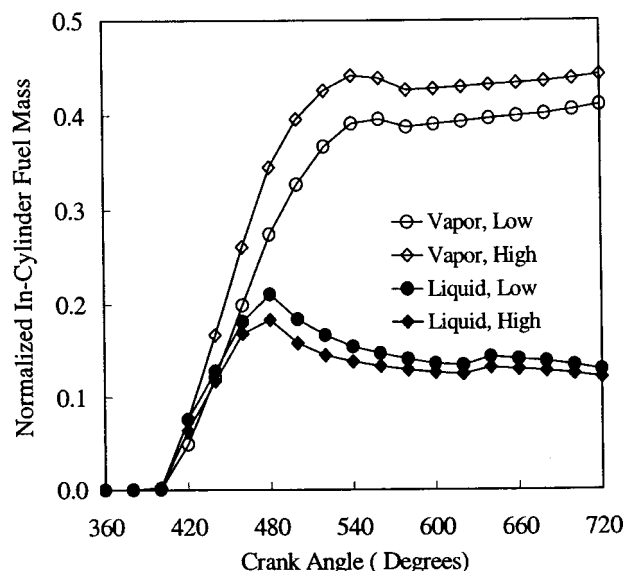
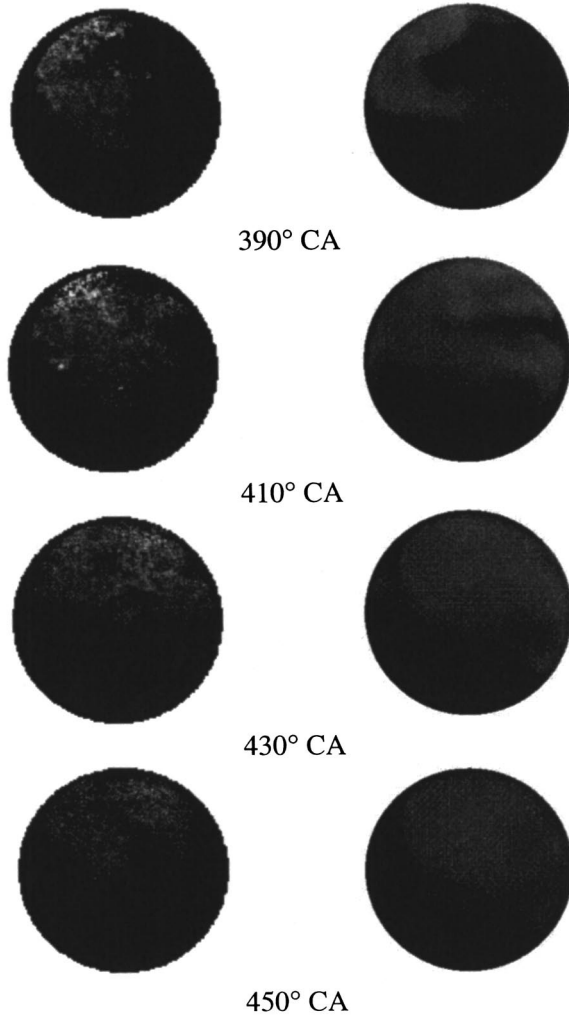
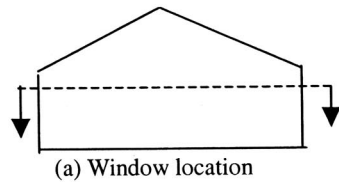
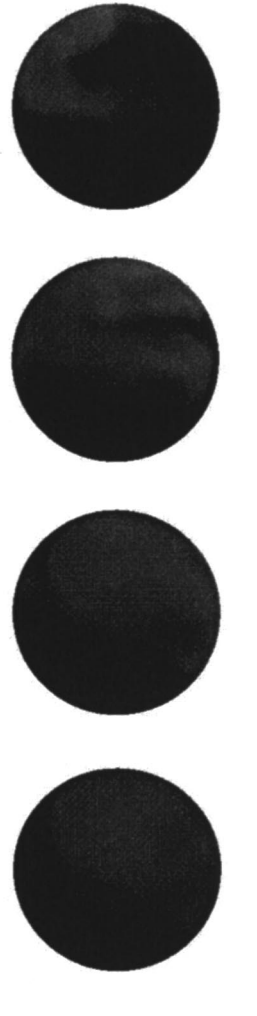


Fig. 8 Variation of in-cylinder fuel mass versus crank angle (effect of swirl intensity for open-valve injection, Cases 4 and 6)



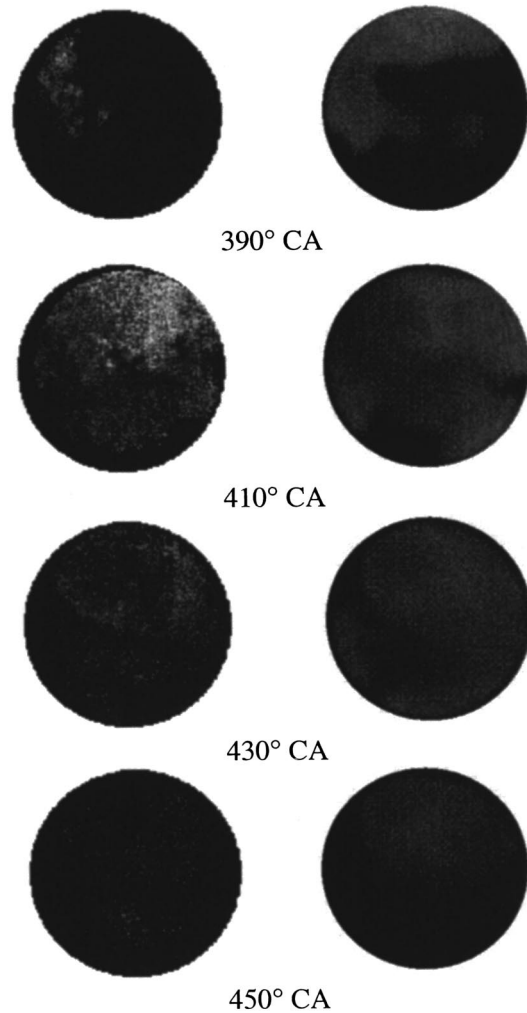
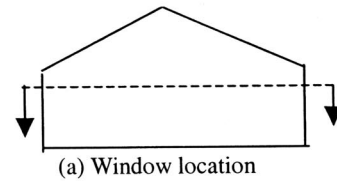
(b) Experiment



(c) Computation

Fig. 9 Vapor phase distribution of hexane for Case 1

stages of the high swirl case. Computations predict that more fuel enters the cylinder initially, especially for the high swirl case. A possible reason for this discrepancy is that back-flow process is not well captured. As the intake valves open, a strong back-flow is generated due to the pressure differential across the intake valves, and this back-flow pushes backward the fuel vapor accumulated in the intake port wall. The back-flow is stronger in the high swirl case because of the smaller exit area since the secondary intake valve is deactivated. The fuel distributions on the middle plane of the cylinder for these two cases at two crank angles are shown in Figs. 11 through 14. Figure 11 shows both the computed vapor density and averaged vapor fluorescence image of the light component C_6H_{14} , [3], in the center plane of the cylinder between two intake valves at the early stage of the intake stroke (410 deg CA) for Case 1. The box in the computational image indicates the



(b) Experiment

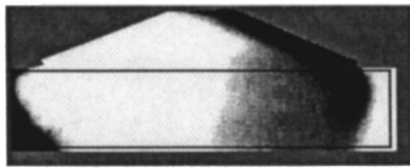


(c) Computation

Fig. 10 Vapor phase distribution of hexane for Case 5

imaging region of the corresponding experiment. It can be seen that the computed result agrees reasonably well with the measured one. The vapor images for the late stage of the compression stroke (670 deg CA) are shown in Fig. 12. The distribution becomes more disperse and uniform due to the mixing. Similar images for Case 5 were shown in Figs. 13 and 14. Good agreement between the computational and the experimental images was also obtained. Contrary to Case 1, more fuel is accumulated in the left side of the cylinder at the early stage of the intake stroke for the high swirl case.

Engine Speed. This study considers two engine speeds, 1200 rpm and 200 rpm, which are representative for high load and low load conditions, respectively. The effects of the engine speed can be seen in Fig. 15 which compare the in-cylinder fuel mass at different engine speeds. More in-cylinder fuel vapor is induced for the low-speed case (39% versus 46%). However, these results are

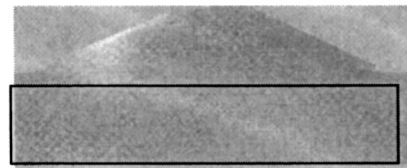


(a) computation



(b) measurement

Fig. 11 Vapor phase distribution of hexane at 410 deg CA for Case 1

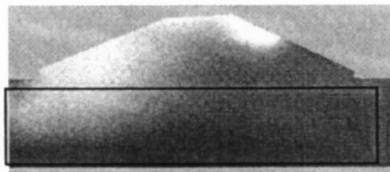


(a) computation



(b) measurement

Fig. 14 Vapor phase distribution of hexane at 670 deg CA for Case 5

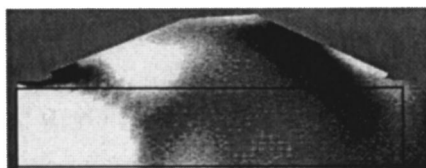


(a) computation

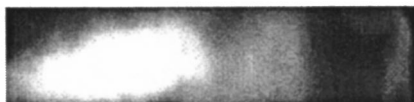


(b) measurement

Fig. 12 Vapor phase distribution of hexane at 670 deg CA for Case 1



(a) computation



(b) measurement

Fig. 13 Vapor phase distribution of hexane at 410 deg CA for Case 5

relatively close considering the large difference in engine speeds (1200 rpm versus 200 rpm). For the low speed case, the gas velocities in the intake port and cylinder are lower, and thus the vaporization rate is slower. However, this effect is almost negated by the fact that more time is available for vaporization at low engine speeds. Comparison of fuel distributions at the end of the compression stroke indicates that the mixtures have similar degrees of uniformity. Again in spite of more time available for mixing in the low-speed case, the turbulence intensity is also lower. These results agree with the conclusion drawn by Kelly-Zion et al. [3] that engine speed has a negligible effect on air/fuel mixture preparation.

Target Path. Target path is an important design parameter for PFI engines. Two target paths are compared here for both the closed-valve injection strategy and the open-valve strategy. Target 1 is located near the center (valve stem) of the primary valve top surface when the valve fully opens and is along the plane that passes through the centerline of the primary intake valve. In con-

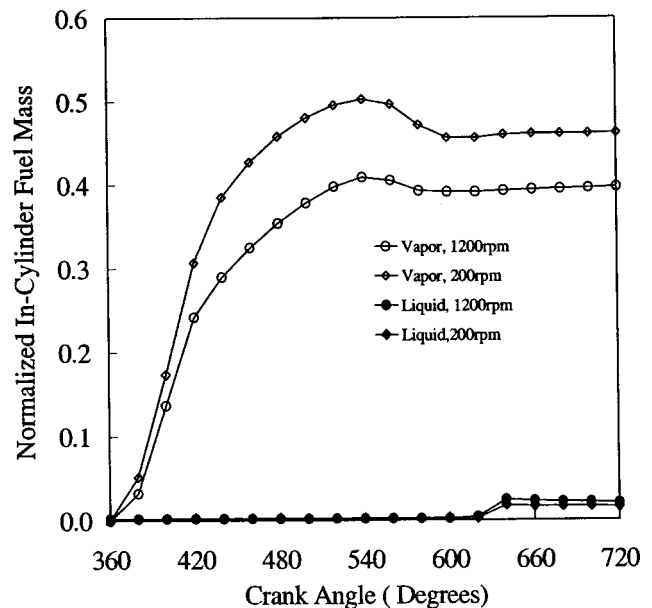


Fig. 15 Variation of in-cylinder fuel mass versus crank angle (effect of engine speed, Cases 1 and 7)

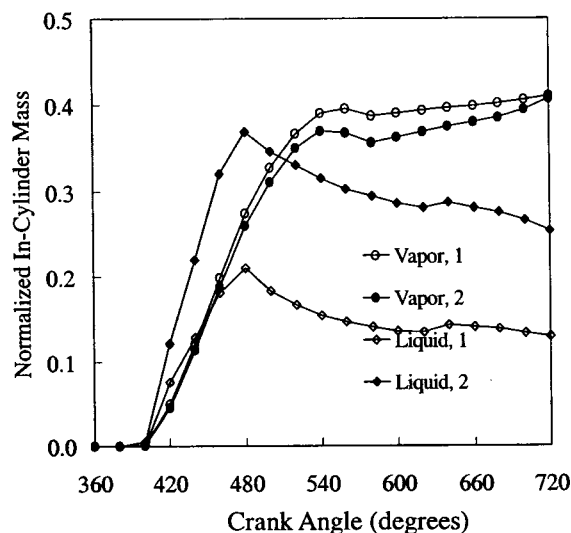


Fig. 16 Variation of in-cylinder fuel liquid mass versus crank angle (effect of target path for open-valve injection, Cases 4 and 9)

trast, the injection angle is rotated upward by 4.4 deg relative to Target 1 in the centerline plane for Target 2. Figures 16 and 17 show the results for in-cylinder liquid and vapor mass. For open-valve injection, more liquid fuel enters the cylinder using Target 2 because more spray droplets bypass the primary intake valve surface. This results in a corresponding increase in the in-cylinder fuel vapor mass at late times. The trend for the closed-valve injection is somewhat different from that of the open-valve injection. Less fuel is vaporized for Target 2, and roughly the same amount of in-cylinder liquid fuel remains at the end of the compression stroke. The differences between the two target paths are due to the different wall wetting areas as shown in Fig. 18. For closed-valve injection, the vaporization of air-borne droplets only accounts for a small part of the total vaporization, and thus the film area significantly influences the total vaporization rate. Target 2 produces a smaller film area and thus a lower total vaporization

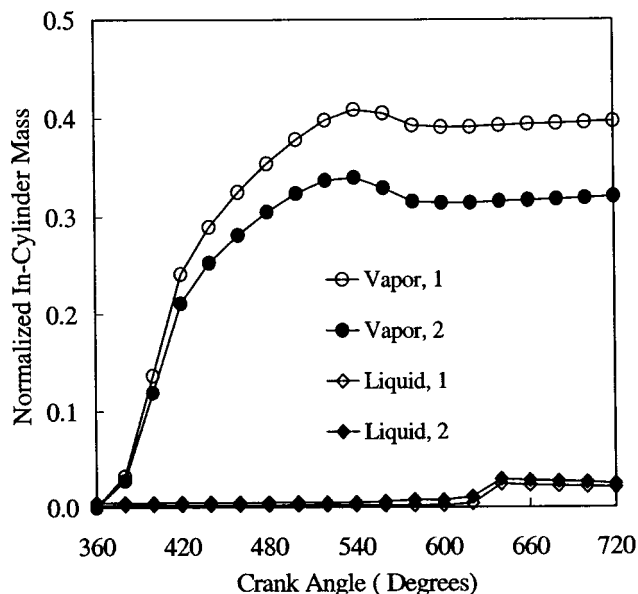


Fig. 17 Variation of in-cylinder fuel vapor mass versus crank angle (effect of target path for closed-valve injection, Cases 1 and 8)

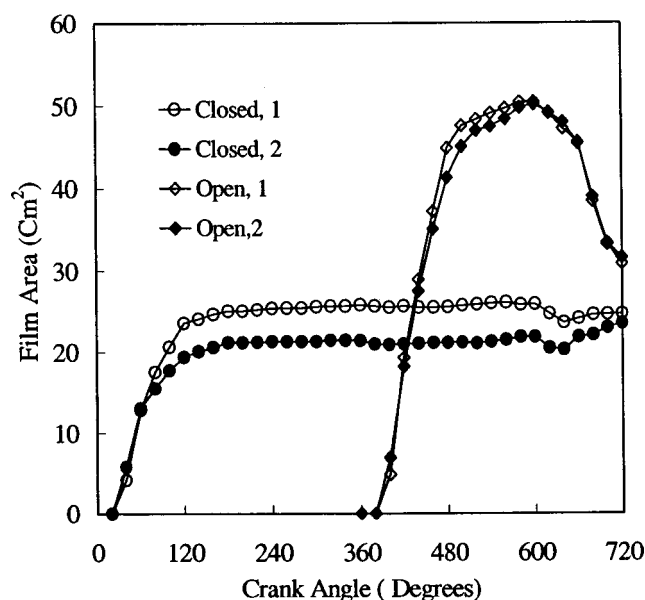


Fig. 18 Variation of film area versus crank angle (effect of target path, Cases 1, 4, 8, and 9)

rate. Open-valve injection results in a larger film area than the closed-valve injection, which contributes to the faster vaporization for open-valve injection. The rapid decrease of the film area for the open-valve injection at the later part of the compression stroke is due to the vaporization of in-cylinder film. These results suggest that the ideal target position should maximize the film area. Two benefits are gained from doing this. One is the faster vaporization rate, and the other is that less liquid enters the cylinder because the film height decreases with an increasing film area.

Fuel Enrichment. To produce a combustible air/fuel vapor mixture, substantial fuel enrichment is necessary during the cold starting phase that is believed to be one of primary causes for high HC emissions. A computation with four times stoichiometric fuel (Case 10) was performed and compared with results for stoichiometric case (Case 1, note that the results are normalized by the

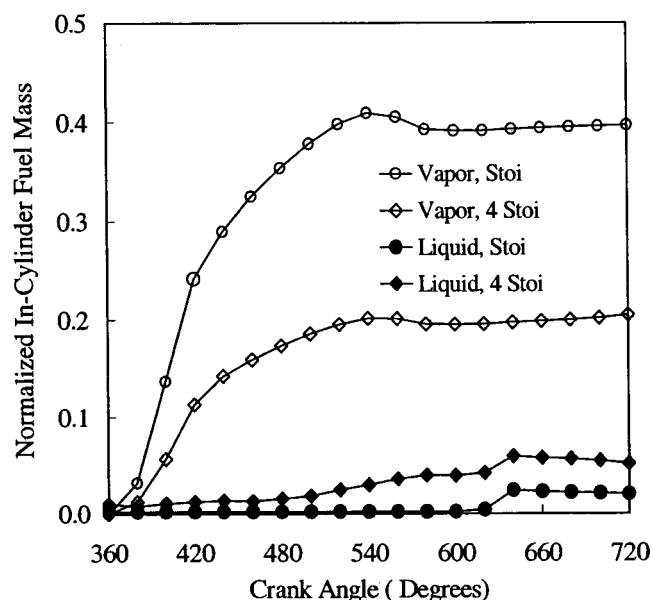


Fig. 19 Variation of in-cylinder fuel mass versus crank angle (effect of fuel enrichment, Cases 1 and 10)

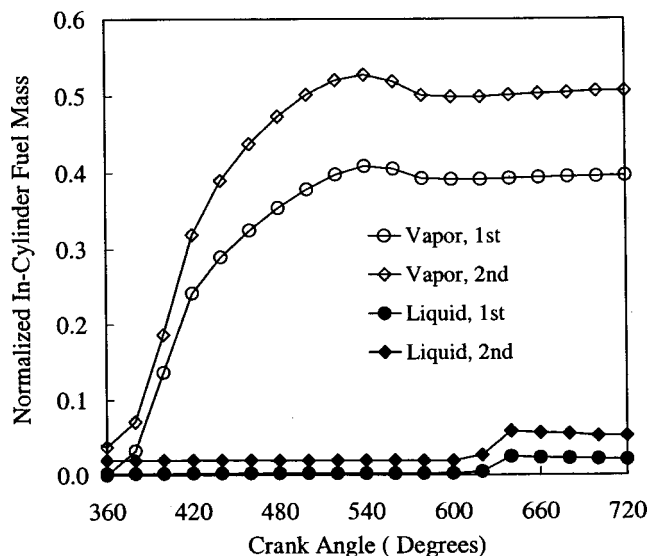


Fig. 20 Variation of in-cylinder fuel mass versus crank angle (effect of fuel accumulation, Cases 1 and 12)

injection mass) shown in Fig. 19. The important findings are that the percentage of fuel vapor entering the cylinder is decreased more than one-half, whereas the percentage for liquid fuel is increased almost four times. When more fuel is injected, the resulting film area does not increase proportionally. Consequently, the total vaporization rate increase is limited resulting in a decrease in the percentage of vaporized fuel. However, the film thickness increases significantly with more fuel injected, which results in more liquid induced into the cylinder.

Fuel Accumulation. To examine the effects of fuel accumulation, a computation over two engine cycles was performed. The results are plotted in Fig. 20 for in-cylinder fuel and vapor mass through the first and second cycles. Due to fuel accumulation in the intake port in forms of both vapor and liquid, the induced fuel mass increase during the second cycle. Although the total fuel mass entering the cylinder is still less than the injected fuel mass, it can be expected that the former will nearly equal to the latter with continuous fuel accumulation, and thus a steady state will be reached. The negative effect of fuel accumulation is that more liquid fuel enters the cylinder and remains in the form of wall films at the end of the compression stroke.

Summary

A numerical study has been performed of the air/fuel preparation process in a port-injected spark-ignition engine during the cold-starting phase. The importance of the vaporization model and fuel representation was first demonstrated. Computations for different engine operating parameters were then conducted to study the effects of these parameters on mixture quality. The results can be summarized as follows:

1. The infinite diffusion model widely used in multidimensional computation does not sufficiently represent the complicated vaporization characteristic of multi-component fuels in a cold-starting engine.
2. Single-component fuel computations result in less vapor-induced into the cylinder than multicomponent fuel computations, especially under fuel enrichment cases.
3. The effects of various engine operation parameters were examined and discussed in detail. Compared to closed-valve injection, open-valve injection leads to more fuel entering the cylinder and a significant portion remains liquid. High swirl increases the in-cylinder fuel vapor mass for both

closed-valve and open-valve injections and leads to a more uniform fuel vapor in-cylinder distribution. Engine speed has a relatively weak influence on mixture quality. The injection path (i.e., spray targeting) is important since it determines the liquid distribution and the resulting film area. With more fuel injected, the percentage of the vaporized fuel decreases, while the liquid fuel remaining at the end of compression stroke increases significantly. Fuel accumulation in the intake port results in more liquid fuel entering the cylinder.

Acknowledgments

This work was supported in part by the National Science Foundation under grant No. CTS-9734402, with Farley Fisher as technical monitor, and by Ford Motor Company. We also thank Dar-Lon Chang and Yu-Yu Lee for their assistance in preparing the manuscript, and Mario Trujillo for the assistance in implementing the impingement model in this work.

Nomenclature

- B = transfer number
- D = diffusion coefficient
- E = ratio between the difference of surface and average values at the specific Pe_V and that at $Pe_V = 0$
- H_Y = overall mass transfer coefficient of the film
- K = splash coefficient
- R = droplet radius
- T = temperature
- Y = mass fraction
- q = heat flux to the film
- t = time
- Φ = diffusion variable
- Γ = mass diffusion coefficient
- ρ = density
- α = thermal diffusivity
- δ = film height of the wall
- ω = film vaporization rate
- η = species mass diffusion flux at the film surface
- s = film vaporization coefficient

Subscripts

- d = difference between the surface and mean values
- g = gas phase
- i = species index
- ℓ = liquid phase
- m = average value
- s = surface value
- w = value at the wall

References

- [1] Gallopoulos, N., 1992, "Bridging the Present to the Future in Personal Transportation—The Role of Internal Combustion Engines," SAE Paper 920721.
- [2] Sampson, M. J., and Heywood, J. B., 1995, "Analysis of Fuel Behavior in the Spark-Ignition Engine Start-Up Process," SAE Paper 950678.
- [3] Kelly-Zion, P. L., Styron, J. P., Lee, C. F., Lucht, R. P., Peters, J. E., and White, R. A., 1998, "Multicomponent Liquid and Vapor Fuel Measurements in the Cylinder of a Port Injected, Spark Ignition Engine," *27th Symposium (International) on Combustion*, Combustion Institute, Pittsburgh, PA, pp. 2111–2117.
- [4] Horie, K., Takahashi, H., and Akazaki, S., 1995, "Emissions Reduction During Warm-Up Period by Incorporating a Wall-Wetting Fuel Model on the Fuel Injection Strategy During Engine Starting," SAE Paper 952478.
- [5] Naitoh, K., Takagi, Y., Kokita, H., and Kuwahara, K., 1994, "Numerical Prediction of Fuel Secondary Atomization Behavior in SI Engine Based on Oval-Parabola Trajectories (OPT) Model," SAE Paper 940526.
- [6] Shimizu, R., Matsumoto, S., Furuno, S., Murayama, M., and Kojima, S., 1992, "Measurement of Air-Fuel Mixture Distribution in a Gasoline Engine Using LIEF Techniques," SAE Paper 922356.
- [7] Meyer, J., Haug, M., Schreiber, M., and Unverzagt, S., 1995, "Controlling Combustion in an Spark Ignition Engine by Quantitative Fuel Distribution," SAE Paper 950107.

- [8] Zhao, F.-Q., Taketomi, M., Nishida, K., and Hiroyasu, H., 1993, "Quantitative Imaging of the Fuel Concentration in a SI Engine With Laser Rayleigh Scattering," SAE Paper 932641.
- [9] Chen, K. C., Dewitte, K., and Cheng, W. K., 1995, "Fuel Effects and Enrichment Effects on Engine Starting and Warm-Up Behavior," SAE Paper 950065.
- [10] Zeng, Y., and Lee, C. F., 2002, "A Model for Multicomponent Spray Vaporization in a High Pressure and High Temperature Environment," ASME J. Eng. Gas Turbines Power, **124**, pp. 717–724.
- [11] Zeng, Y., and Lee, C. F., 2000, "A Multicomponent-Fuel Film-Vaporization Model for Multidimensional Computations," J. Propul. Power, **16**, pp. 964–973.
- [12] Zeng, Y., and Lee, C. F., 2002, "A Preferential Vaporization Model for Multicomponent Droplets and Sprays," Atomization Sprays, **12**(1), pp. 163–186.
- [13] Abraham, J., and Magi, V., 1998, "A Model for Multicomponent Droplet Vaporization in Sprays," SAE Paper 980511.
- [14] Ayoub, N. S., and Reitz, R. D., 1997, "Multidimensional Modeling of Fuel Effects and Split Injections on Diesel Engine Cold-Starting," J. Propul. Power, **13**, pp. 123–130.
- [15] Bai, C., and Gosman, A. D., 1996, "Mathematical Modeling of Wall Films Formed by Impinging Sprays," SAE Paper 960626.
- [16] O'Rourke, P. J., and Amsden, A. A., 1996, "A Particle Numerical Model for Wall Film Dynamics in Port-Injected Engines," SAE Paper 961961.
- [17] Megaridis, C. M., and Sirignano, W. A., 1990, "Numerical Modeling of a Vaporizing Multicomponent Droplet," *23rd Symposium (International) on Combustion*, The Combustion Institute, Pittsburgh, PA, pp. 1413–1421.
- [18] Abramzon, B., and Sirignano, W. A., 1989, "Droplet Vaporization Model for Spray Combustion Calculations," Int. J. Heat Mass Transfer, **32**, pp. 1605–1618.
- [19] Jin, J. D., and Borman, G. L., 1985, "A Model for Multicomponent Droplet Vaporization at High Ambient Pressures," SAE Paper 850264.
- [20] Law, C. K., 1976, "Multicomponent Droplet Combustion With Rapid Internal Mixing," Combust. Flame, **26**, pp. 219–233.
- [21] Trujillo, M. F., Mathews, W. S., Lee, C. F., and Peters, J. E., 2000, "Modeling and Experiment of Impingement and Atomization of a Liquid Spray on a Wall," Int. J. Eng. Res., **1**(1), pp. 87–105.
- [22] Amsden, A., 1999, "KIVA-3V, Release 2, Improvements to KIVA-3V," Los Alamos National Laboratory Report LA-13608-MS.
- [23] Amsden, A., 1997, "KIVA-3V: A Block-Structured KIVA Program for Engines With Vertical or Canted Valves," Los Alamos National Laboratory Report LA-13313-MS.
- [24] Reid, R. C., Prausnitz, J. M., and Poling, B. E., 1987, *The Properties of Gases and Liquids*, 4th Ed., McGraw-Hill, New York.
- [25] Styron, J. P., Kelly-Zion, P. L., Lee, C. F., Peters, J. E., White, R. A., and Lucht, R. P., 2000, "Multicomponent Liquid and Vapor Fuel Distribution Measurements in the Cylinder of a Port Injected, Spark Ignition Engine," SAE Paper 2000-01-0243.

A Characteristic Parameter to Estimate the Optimum Counterweight Mass of Symmetric In-Line Engines

R. Stanley

Mechanical Engineering Department,
Kettering University,
1700 W. Third Street,
Flint, MI 48504
e-mail: rstanley@kettering.edu

D. Taraza

Wayne State University,
5050 Anthony Wayne Drive,
Detroit, MI 48202
e-mail: taraza@me1.eng.wayne.edu

Dimensionless relationships that estimate the maximum bearing load of two symmetric in-line engines have been developed. These relationships were empirically determined by averaging the maximum bearing loads of several engine configurations with various geometric and operating parameters. It has been found that the maximum bearing load initially decreases with engine speed, reaches a minimum, and then increases quickly with engine speed. This minimum reflects a transition speed at which the contribution of the inertia force overcomes the contribution of the maximum pressure force to the maximum bearing load. The transition speed increases with an increase of counterweight mass and is a function of the geometric parameters and operating conditions of the engine. Rapid estimation of the main bearing loads is made possible by the use of the dimensionless relationships developed in the paper. [DOI: 10.1115/1.1559902]

Introduction

Automotive engineers are interested in minimizing the weight of the internal combustion engine. Engine weight should be lowered only if adequate component strength can be achieved and if the engine can perform within acceptable vibration limits. The slider/crank mechanism contributes significantly to the overall engine weight, so optimizing the design is desirable.

This paper focuses on the counterweights, which contribute to engine weight, vibration, and bearing load.

Vibration is an important consideration when designing the crankshaft. Several past studies have concluded that a lower counterweight mass will increase the engine noise level, [1,2] but other recent studies suggest the opposite, [3,4]. Although crankshaft vibrations are important to consider, a detailed analysis of crankshaft vibrations is beyond the scope of this paper, but will be considered in future investigations.

A combination of the maximum bearing load and the average bearing load will determine the fatigue of the main bearing, [5]. Calculating the bearing load as a function of counterweight mass is therefore an important part of the design process.

Recent modeling of the crankshaft is performed with complicated and proprietary models. Several finite element method have been developed to optimize crankshaft weight, [6–8]. These methods are complex and are not available to the general engineering community.

It is relatively straightforward to calculate the bearing load for a given engine configuration. However, there is currently no generic and fundamental relationship that can predict the maximum main bearing load as a function of counterweight mass, engine speed, cylinder pressure, and the geometric parameters of the engine. This study provides the design engineer with an uncomplicated method to estimate the maximum bearing load, based on engine parameters and steady-state conditions.

This model will provide the design engineer with reasonable results, which are accurate within 8%. It is intended to offer a realistic start in the design process before continuing to a more complicated computer model or physical prototype.

The “ideal” counterweight mass is defined such that the rotating inertia force of each cylinder is completely balanced by the corresponding set of counterweights. The counterweight mass that produces the lowest maximum bearing load is not normally the “ideal” counterweight mass because the load depends not only on the rotating inertia force, but also on the reciprocating inertia force and the pressure force.

Dimensionless relationships have been developed to estimate the percent of “ideal” counterweight mass that will result in the acceptable maximum bearing load, based on given operating conditions and geometric parameters. The percent of the “ideal” counterweight mass is given by the variable σ_{CW} and ranges from 0% to 100%.

These relationships have been developed for two common in-line four-stroke engines with symmetric crankshafts:¹ a four-cylinder with firing order 1-3-4-2 and a six-cylinder with firing order 1-5-3-6-2-4. In the following paper,

1. the forces and moments that contribute to bearing loading and any corresponding balancing conditions will be explained,
2. the engine forces, based on 1, above, will be transferred to the main bearing,
3. the dependence of bearing load on counterweight mass and engine speed will be graphically shown,
4. dimensionless parameters will be formulated to estimate the maximum bearing load, and
5. example problems will be solved.

Forces and Moments in the IC Engine

Forces and moments in the internal combustion (IC) engine result from two main sources: cylinder pressure and the inertia of the moving parts. For a given engine configuration, inertia forces and moments depend only on the engine speed and pressure forces and moments depend mainly on the load applied to the engine.

THE CRANK STAR—The crank star (or the first order vector star) is commonly used to evaluate crank configuration and resulting engine balance. Figure 1 is a schematic of a four-cylinder four-stroke in-line engine with firing order 1-3-4-2. The sketch at

¹Contributed by the Internal Combustion Engine Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the ICE Division, May 2001; final revision received by the ASME Headquarters, March 2002. Editor: D. N. Assanis.

¹A symmetric crankshaft is defined as follows: one-half of a symmetric crankshaft is a mirror image of the other half at its center. Only four-stroke engines with an even number of cylinders may have symmetric crankshafts.

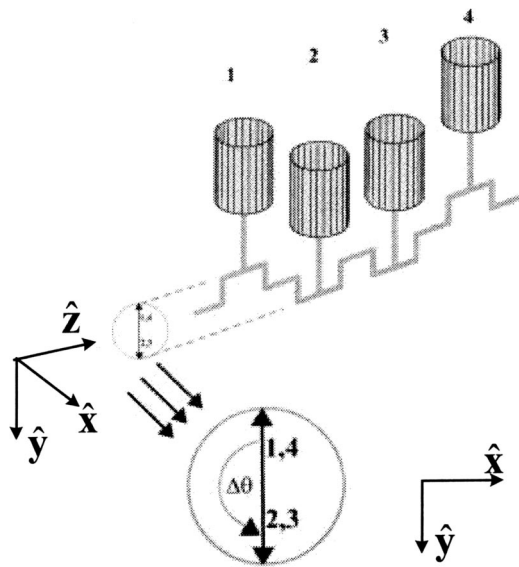


Fig. 1 Four-stroke four-cylinder in-line engine and crank star

the bottom of the figure is the crank star. When piston number 1 is at top dead center (TDC) position, cylinder $j=4$ is also at $\Delta\theta_j = 0$ deg (TDC) and cylinders $j=2$ and $j=3$ are $\Delta\theta_j = 180$ deg (bottom dead center—BDC). As the crankshaft rotates clockwise at an angle θ , the cranks of the remaining cylinders rotate $\theta_j = \theta + \Delta\theta_j$.

Figure 2 illustrates the forces and moments that contribute to IC engine unbalance. In the following section the forces and moments that cause engine unbalance will be explained and corresponding balancing conditions, if any, will be clarified. For more details on these unbalances and justifications of the assumptions made, see [9].

INERTIA FORCES—The inertia forces are separated into two categories: the rotating inertia force and the reciprocating inertia force.

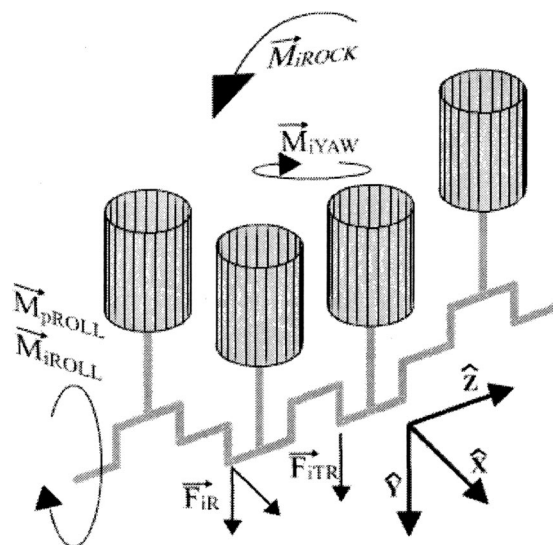


Fig. 2 Unbalancing forces and moments of a four-stroke four-cylinder engine

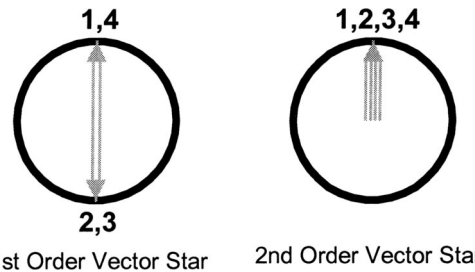


Fig. 3 First and second-order vector stars: 4-Cyl

The Reciprocating Inertia Force—The reciprocating inertia force F_{iTR} of each cylinder is given by the simplified two harmonic formula:²

$$\vec{F}_{iTR,j} = -m_{TR} \cdot R \cdot \omega^2 (\cos \theta_j + \Lambda \cos 2\theta_j) \hat{y} \quad (1.1)$$

where m_{TR} is the reciprocating mass, consisting of the mass of the piston, piston rings, piston pin, and about 1/3 of the mass of the connecting rod.

Four-Cylinder Engine—The reciprocating inertia force is not “naturally balanced” in this engine because it will cause a shaking force unless an external balancing mechanism is added. This shaking force will eventually be transmitted to the engine mounts.

This unbalance can be explained with the assistance of the first and second-order vector stars (Fig. 3). Here, the reciprocating inertia forces of the individual cylinders cancel each other for the first-order harmonic order and add in-phase for the second harmonic order. Hence, the net shaking unbalance of the four stroke/ four cylinder in-line engine is due to the second harmonic order component of the reciprocating inertia force.

The unbalance can be expressed analytically by expanding Eq. (1.1) in order to include all four cylinders, as follows:

$$\vec{F}_{iTR} = \sum_{j=1}^N m_{TR} \cdot R \cdot \omega^2 (\cos \theta_j + \Lambda \cos 2\theta_j) \hat{y} \quad (1.2)$$

Expanding Eq. (1.2) for this engine, the net shaking force is

$$\vec{F}_{iTR} = 4\Lambda \cdot m_{TR} \cdot R \cdot \omega^2 (\cos 2\theta) \hat{y} \quad (1.3)$$

Six-Cylinder Engine—The reciprocating inertia force of the six-cylinder engine is naturally balanced. This can be explained by the first and second order vector stars (Fig. 4), where the reciprocating inertia forces of the individual cylinders cancel each other for both the first and second harmonic order.

The counterweight mass adds only to the first harmonic component of the inertia force. Therefore, altering the counterweight mass will not influence the shaking force due to the reciprocating inertia force in any symmetric crankshaft with at least four cylinders.

²The third and higher-order harmonic terms contribute very little to the reciprocating inertia force, so they are ignored, [9].

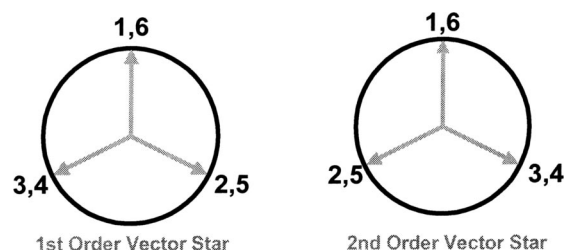


Fig. 4 First and second-order vector stars: 6-Cyl

The Rotating Inertia Force—the rotating inertia force F_{IR} each cylinder is given by

$$\vec{F}_{IR,j} = m_R \cdot R_R \cdot \omega^2 (\sin \theta_j \hat{x} - \cos \theta_j \hat{y}) \quad (1.4)$$

where m_R is the rotating mass and consists of the mass of the crank pin, the crank webs, and about 2/3 of the mass of the connecting rod. The parameter R_R represents the distance from the crankshaft center of rotation to the center of gravity of the rotating mass.

The vector star of the rotating inertia forces is identical to the first-order vector star of the reciprocating inertia force. The rotating inertia force of each cylinder is usually offset (at least partially) by counterweights placed on the opposite side of each crank. There are two counterweights per crank. Each counterweight force is given by

$$\vec{F}_{CWi,j} = -m_{CWi,j} \cdot R_{CWi,j} \cdot \omega^2 \begin{Bmatrix} \sin(\theta_j + \gamma_{i,j}) \hat{x} \\ -\cos(\theta_j + \gamma_{i,j}) \hat{y} \end{Bmatrix}, \quad i=1,2 \quad (1.5)$$

where $\gamma_{i,j}$ is the angle that counterweight “ i , j ” is offset with respect to the crank axis.

For complete balancing of each crank ($\sigma_{CW} = 100\%$), the following equation holds true:

$$(m_{CW1,j} \cdot R_{CW1,j} + m_{CW2,j} \cdot R_{CW2,j}) = m_R \cdot R_R, \quad j = 1, 2, \dots, N. \quad (1.6)$$

According to the first-order vector stars (Figs. 3 and 4), the rotating inertia force of both the four-cylinder and six-cylinder engines is naturally balanced with or without any counterweights. Hence, adding or subtracting counterweight mass has no effect on the shaking force due to the rotating inertia force.

INERTIA MOMENTS—The inertia moments are separated into three categories: the rolling moment (z -direction), the yawing moment (y -direction), and the rocking moment (x -direction).

The Inertia Rolling Moment—The inertia rolling moment M_{iROLL} of each cylinder is produced by the reciprocating inertia force and is given by

$$\vec{M}_{iROLL} = \sum_{j=1}^N m_{TR} \cdot (R \cdot \omega)^2 (\cos \theta_j + \Lambda \cos 2\theta_j) \left(\frac{\sin(\theta_j + \beta_j)}{\cos \beta_j} \right) \hat{z} \quad (1.7)$$

where β_j is the angle between the axis of the connecting rod and the cylinder axis. For a standard reciprocating mechanism, where there is no piston pin offset, β_j is given by

$$\beta_j = \sin^{-1}(\Lambda \cdot \sin \theta_j). \quad (1.8)$$

The rolling moment vector stars correspond to the vector stars of the reciprocating inertia force (Figs. 3 and 4). Accordingly, the four-cylinder engine will be unbalanced only by the second-order inertia rolling moment and the six cylinder engine will have no unbalance. Hence, the addition or subtraction of counterweight mass has no effect on this moment.

The Inertia Yawing and Rocking Moments—Both of these moments are naturally balanced because of the configuration of symmetric crankshafts, [10]. Therefore, adding or subtracting counterweight mass has no effect on either of these moments.

THE PRESSURE FORCE—The pressure force F_p of each cylinder is given by

$$\vec{F}_{p,j} = (P(\theta)_{CYL,j} - P(\theta)_{CC,j}) \cdot \frac{\pi}{4} B^2 \hat{y}. \quad (1.9)$$

The unbalance produced by the pressure force produces only the pressure rolling moment (see [9]).

THE PRESSURE ROLLING MOMENT—The pressure rolling moment M_{pROLL} is given by

$$\vec{M}_{pROLL,j} = (P(\theta)_{CYL,j} - P(\theta)_{CC,j}) \cdot \frac{\pi}{4} B^2 \cdot \frac{\sin(\theta_j + \beta_j)}{\cos \beta_j} \hat{z}. \quad (1.10)$$

It is impossible to balance the pressure rolling moment because it is a function of engine load, not engine speed. Correctly designed engine mounts isolate the unbalance caused by this moment.

It is important to note that any multicylinder in-line crank arrangement that naturally balances the inertia rocking moment can operate with no counterweights while maintaining the same balance. This is because the first harmonic of the reciprocating inertia force and the rotating inertia force add in-phase. So if the reciprocating inertia force causes no inertia rocking moment, the rotating inertia force must also cause no inertia rocking moment (with or without counterweights).

Loading on the Main Bearings

The reciprocating inertia force, the pressure force, and the rotating inertia force are transferred directly to the main bearings.

The transfer of the thrust force between the piston and the liner to the main bearings causes the pressure and inertia rolling moments. Although these rolling moments are not considered directly in counterweight design, the thrust force N_j that causes these moments must be applied as a lateral load on the bearing. The force N_j is given by

$$\vec{N}_j = (\vec{F}_{p,j} + \vec{F}_{iTR,j}) \frac{\sin \beta}{\cos \beta}. \quad (1.11)$$

For illustrative purposes, Fig. 5 is a sketch of the forces acting on the main bearings of two adjacent cylinders. The loading on the bearings is statically and dynamically indeterminate, but a reasonable approximation can be used, [11]. The loading on each bearing is assumed to be influenced only by the adjacent cranks. In Fig. 5, bearing B_2 shares the load of crank “1” with bearing B_1 and the load of crank “2” with bearing B_3 .

Details of the transmission of the forces to the bearings can be found in [10]. Significant points that are listed in [10] are

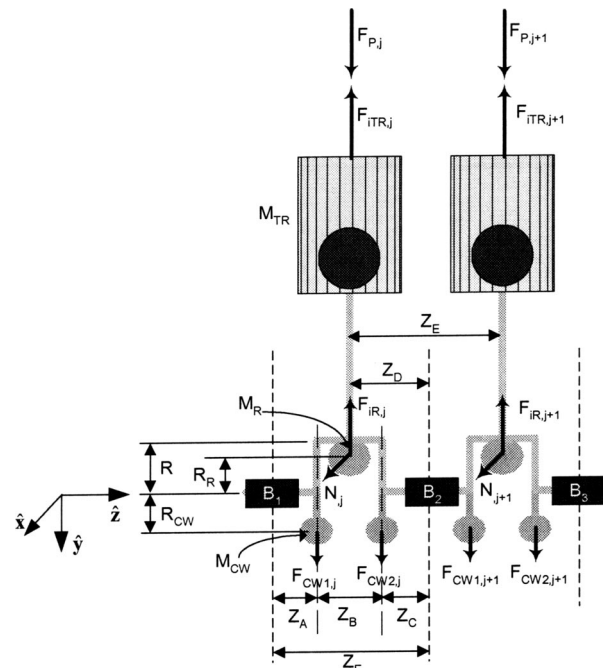


Fig. 5 Forces acting on the main bearing

Table 1 Engine properties

Bore B (mm)	114
Stroke S (mm)	135
Connecting Rod Length L (mm)	216
Reciprocating mass M_{TR} (kg)	3.455
Rotating mass M_R (kg)	5.870
Rotating mass radius R_R (mm)	51.3
Firing order: 4-cylinder	1-3-4-2
Firing order: 6-cylinder	1-5-3-6-2-4
Counterweight angle γ	0°
Max. cylinder pressure (bar)	~ 64
Engine load	$\sim \frac{1}{2}$ load

1. the rotating inertia force and the first harmonic component of the reciprocating inertia force add together in-phase in the y -direction.
2. the pressure force is near maximum at firing TDC. The reciprocating inertia force and the y -component of the rotating inertia force are at maximum here. The pressure forces and inertia forces are opposite in sign and subtract from each other.
3. from the items above, there are advantages and disadvantages realized when increasing the counterweight mass. Increasing the counterweight mass will
 - (a) reduce the average bearing load, due to a lower average inertia force: an advantage.
 - (b) increase the maximum bearing load at relatively low engine speeds. This is because the lower inertia force will not be large enough to overcome the high pressure force: a disadvantage.
 - (c) will add cost and weight to the crankshaft: a disadvantage.
 - (d) will increase the mass moment of inertia of each crank and therefore reduce the first natural frequency of the crankshaft (in torsion): a disadvantage.

A tradeoff must be made between the advantages and disadvantages listed above. To illustrate the details of this tradeoff, an analysis of a four-cylinder and a six-cylinder engine with properties shown in Table 1 follows.

Four-Cylinder Engine—Due to symmetry, the loads on bearing numbers 1 and 2 are identical to those of bearing numbers 5 and 4, respectively. Therefore, only bearing numbers 1, 2, and 3 are evaluated in the four-cylinder engine simulation.

Six-Cylinder Engine—Due to symmetry, the loads on bearing numbers 1, 2, and 3 are identical to those of bearing numbers 7, 6, and 5, respectively. Additionally, bearing numbers 2 and 3 (and likewise bearing numbers 5 and 6) are identical in magnitude, but shifted by 120 deg. Therefore, only bearing numbers 1, 2, and 4 are evaluated in the six-cylinder engine simulation.

The forces acting on the above bearings are illustrated in Figs. 6–10 at two different speeds: 100 rad/sec (955 rpm) and 300 rad/sec (2864 rpm).

Bearing #1 (Fig. 6)

1. The forces in bearing number one are identical for both the four and six-cylinder engine. This is because bearing number one is only affected by the forces in cylinder number one.
2. At 100 rad/sec, the maximum bearing force is only slightly lower with no counterweights than with 100% counterweights. This is because the inertia force is relatively small, so the pressure force reduction is also small. (This is true of all bearings at this low speed.)
3. At 300 rad/sec, the maximum bearing force is higher with no counterweights than with 100% counterweights. This is be-

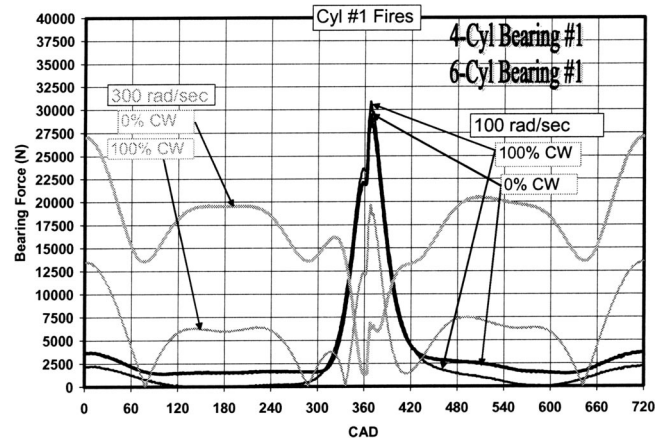


Fig. 6 Bearing #1, four-cylinder and six-cylinder

cause the high inertia force resulting from the zero counterweight mass overcomes the pressure force, thereby increasing the maximum bearing load.

Bearing #2, four-Cylinder (Fig. 7)

1. The maximum bearing force is mainly due to the pressure force and is independent of the counterweight percentage. Bearing number two has no net bearing force resulting from

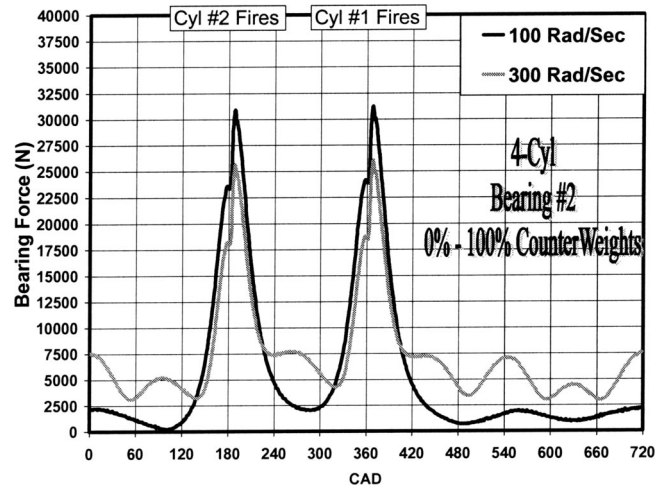


Fig. 7 Bearing #2, four-cylinder

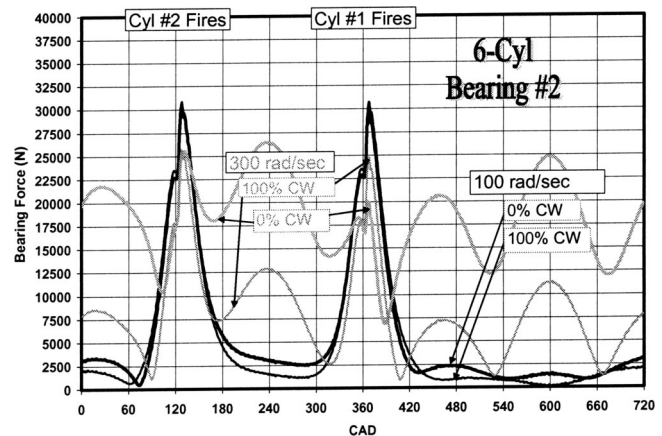


Fig. 8 Bearing #2, six-cylinder

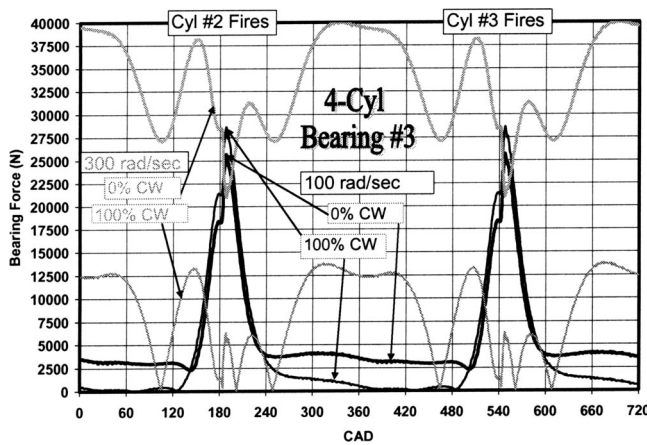


Fig. 9 Bearing #3 (middle bearing), four-cylinder

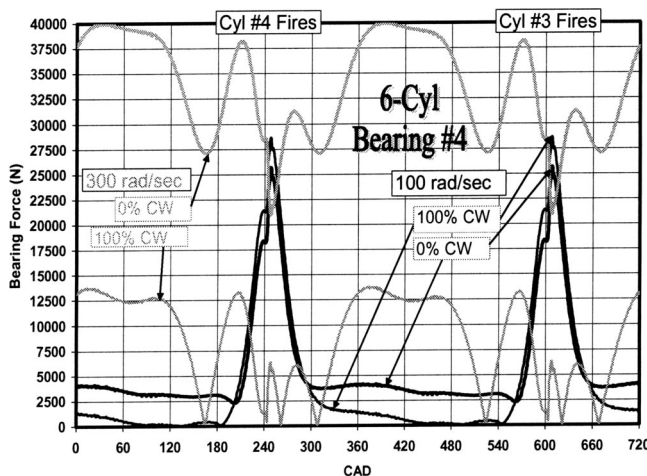


Fig. 10 Bearing #4 (middle bearing), six-cylinder

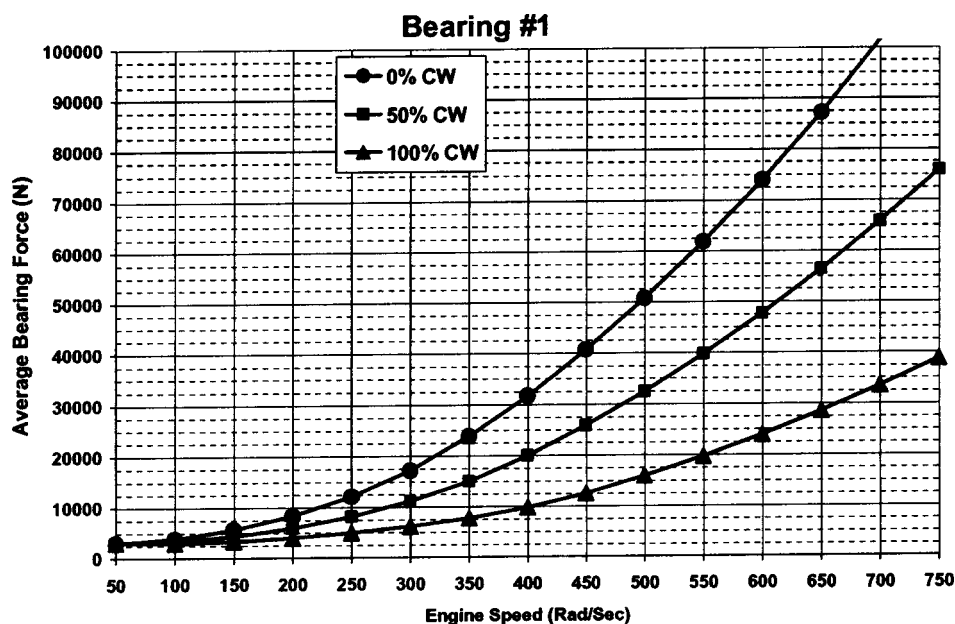


Fig. 11 Bearing #1 average bearing force 4-Cyl and 6-Cyl

the first harmonic order component of the reciprocating and the rotating inertia force in this crankshaft configuration. Adding counterweights therefore has no effect on the force in the bearing.

2. The only inertia force that affects the bearing force here is the second harmonic order component of the reciprocating inertia force. This is significantly lower than the pressure force at low and moderate engine speeds. Because of this, the maximum bearing force only varies slightly with engine speed.

Bearing #2, six-Cylinder (Fig. 8)

1. At 100 rad/sec, the inertia forces are small, so there is little difference between the maximum bearing load with 0%CW and 100%CW.
2. At 300 rad/sec, with 0%CW, the maximum force is due to inertia forces at ~240 deg, while with 100%CW, the maximum force is determined by the pressure force at ~360 deg. There is little difference between these two maximum values.
3. At 300 rad/sec, 0%CW, the lowest maximum bearing force occurs when cylinder number one fires at 360 deg and not when cylinder number two fires at 120 deg. This is because the vector combination of the inertia forces of cylinders one and two are larger at 360 deg than those of cylinders two and three at 120 deg; the second harmonic component of the reciprocating inertia force adds to the inertia force magnitude at 360 deg and subtracts from it at 120 deg. In both cases, these large inertia forces overcome the pressure forces and hence lower the bearing load.

Bearing #3, four-Cylinder (Fig. 9) and

Bearing #4, six-Cylinder (Fig. 10)

1. Both bearings experience the same bearing load magnitude, with a phase shift. This is because the adjacent cranks of these middle bearings are identically oriented, which is a characteristic of any symmetric crankshaft.
2. The average bearing force of this bearing is much larger than that of the other bearings. This is because the inertia forces of each adjacent crank add together in-phase.
3. At 100 rad/sec (relatively low speed), the maximum bearing load is due mainly to the pressure forces.

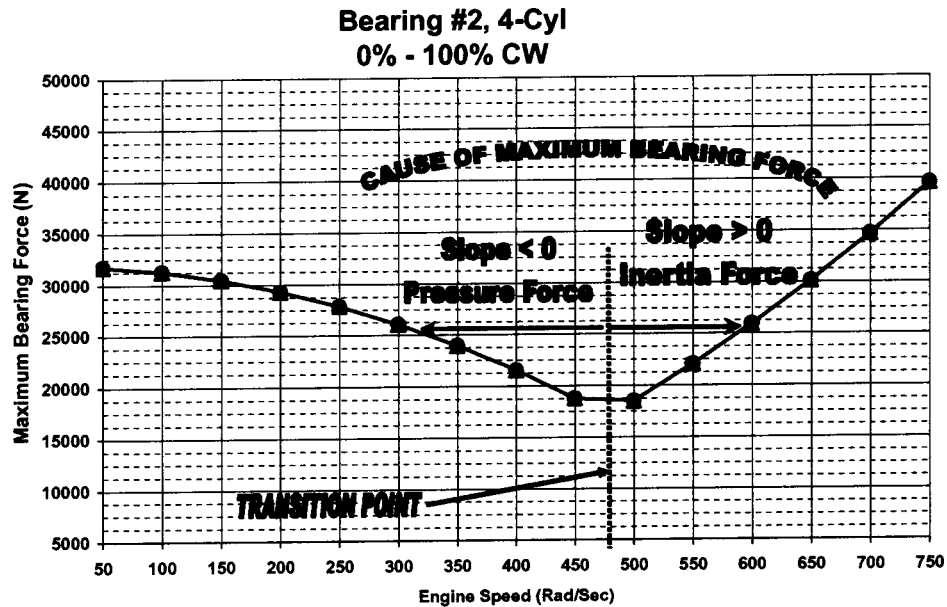


Fig. 12 Bearing #2, 4-Cyl maximum bearing force

4. At 300 rad/sec (relatively high speed), the maximum bearing load of is due mainly to the inertia forces.
5. The maximum bearing force at 300 rad/sec is much higher with no counterweights than with 100% counterweights. This is because 0% counterweight condition results in a higher inertia force and hence a higher maximum bearing force.

It is desirable to find a general relationship between the maximum bearing load and the average bearing load as a function of engine speed and counterweight mass. In order to do so, average and maximum values are shown graphically below. These figures summarize average and maximum bearing force values of selected bearings at speeds ranging from 50 rad/sec to 750 rad/sec. Comments and findings follow the graphs.

The following comments can be made about Figs. 11 through 16. (The gray arrows in Figs. 13 through 16 represent the path of the maximum bearing load in the considered engine configurations.)

1. According to Fig. 11, the average bearing load increases with engine speed and increases with a decrease in the counterweight mass. This is true for all bearings at all speeds and crankshaft configurations.
2. Referring to Fig. 12, as engine speed increases, the bearing load decreases, reaches a minimum, and then rapidly increases. This minimum is referred to as the transition maximum point. It is the point where the pressure forces no longer determine the maximum bearing force (at ~TDC)

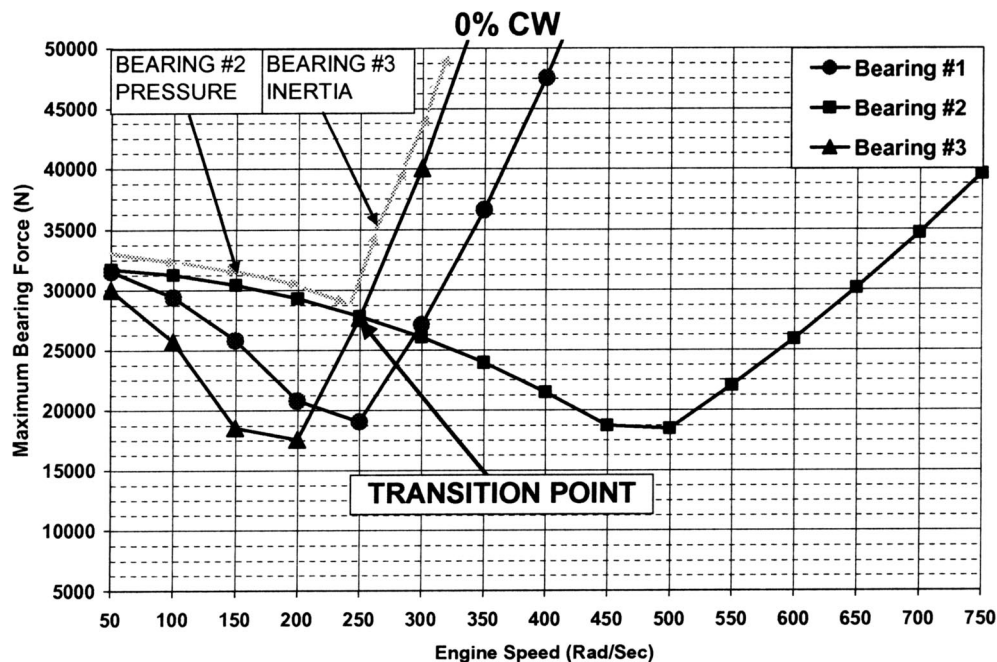


Fig. 13 Maximum bearing force: 4-Cyl, 0% CW

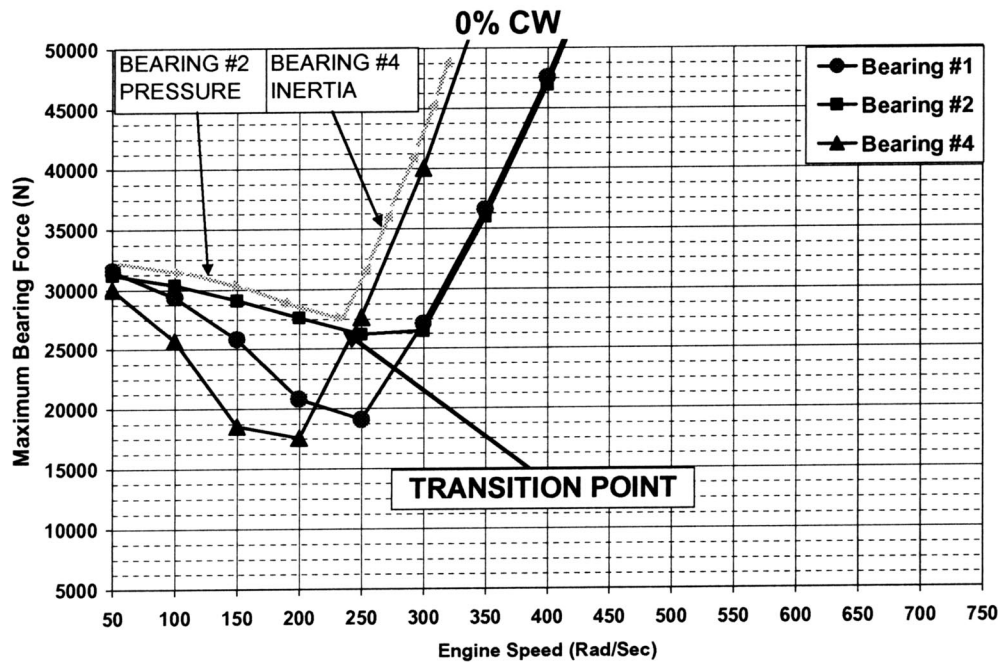


Fig. 14 Maximum bearing force: 6-Cyl, 0% CW

and the inertia forces take precedence as the bearing load mechanism (at BDC). The speed at which this occurs is referred to as the transition speed.

- At relatively low speeds, before the transition speed, the maximum bearing load occurs at bearing number two and is primarily due to the pressure forces. At higher speeds, the maximum bearing load occurs at the center bearing and is due to high inertia forces.
- The transition speed increases as the counterweight percentage increases. This is because the inertia forces “take over” at a lower engine speed as the counterweight percentage is decreased.

Bearing Characteristic Parameters

It is evident that there is a strong relationship between the counterweight mass, the maximum cylinder pressure, the maximum bearing load, and a combination of the rotating and the reciprocating inertia forces. With this in mind, a wide range of geometric parameters, cylinder pressure traces, and operating speeds were evaluated in a computer program (see Table 2). An averaging method was applied to the data and the bearing characteristic parameters were developed.

Following are the restrictions of these parameters:

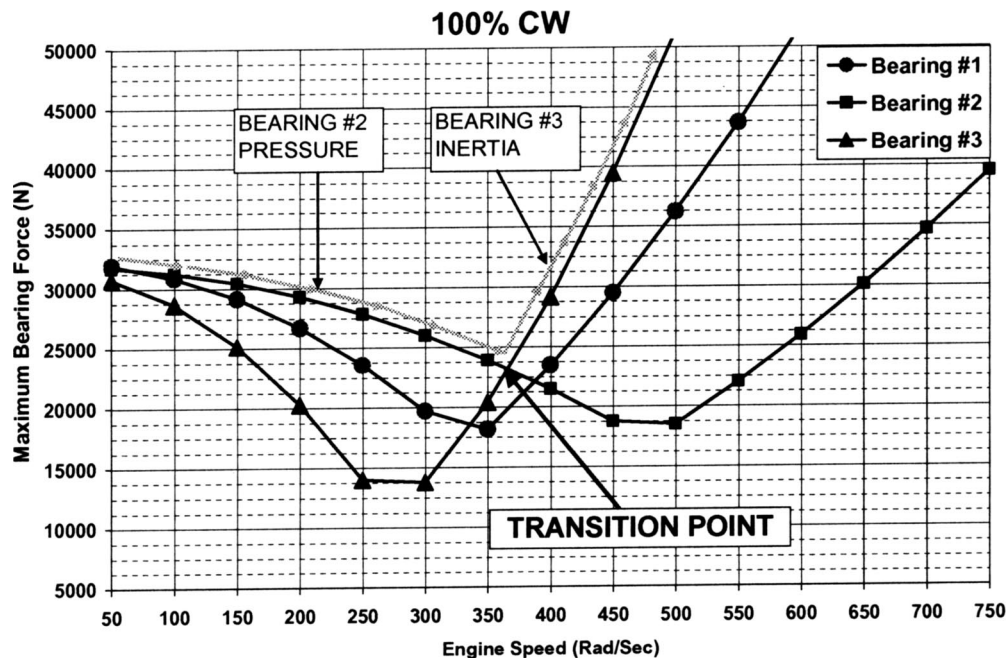


Fig. 15 Maximum bearing force: 4-Cyl, 100% CW

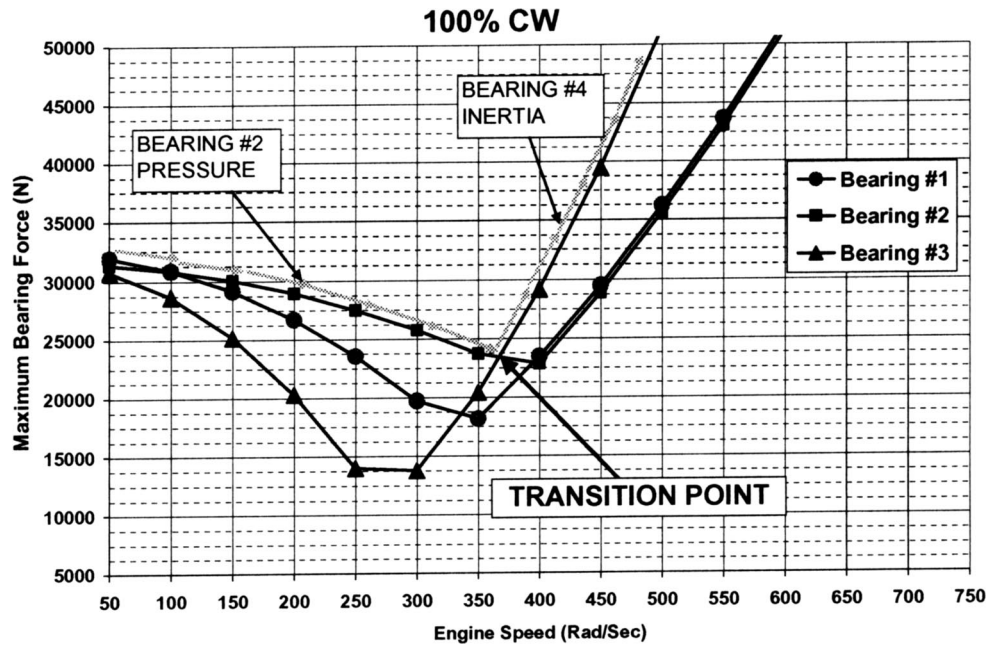


Fig. 16 Maximum bearing force: 6-Cyl, 100% CW

Table 2 Simulation range values

Parameter	Range
$\Lambda = R/L$	0.25–0.40
Stroke S (mm)	80–160
Reciprocating mass m_{TR} (kg)	0.8–5
Rotating mass m_R (kg)	1–7
Rotating mass radius R_R (mm)	40–80
Max. cylinder pressure (bar)	50–125
Engine speed (rad/sec)	50–500

1. The parameters are valid for a four-cylinder four-stroke inline engine with firing order 1-3-4-2 and a six-cylinder four-stroke engine with firing order 1-5-3-6-2-4.
2. All counterweights are assumed to be located 180 deg opposite to the respective crank pin ($\gamma_j = 0$).
3. The maximum bearing force represents the maximum bearing force of all bearings.

A combination of two dimensionless parameters may be used to estimate the maximum bearing load as a function of engine speed, peak cylinder pressure, and engine parameters. Figure 17 shows a relationship between the maximum bearing characteristic parameter B_{CP} and the maximum bearing force parameter F_{CP} for both engines configurations.

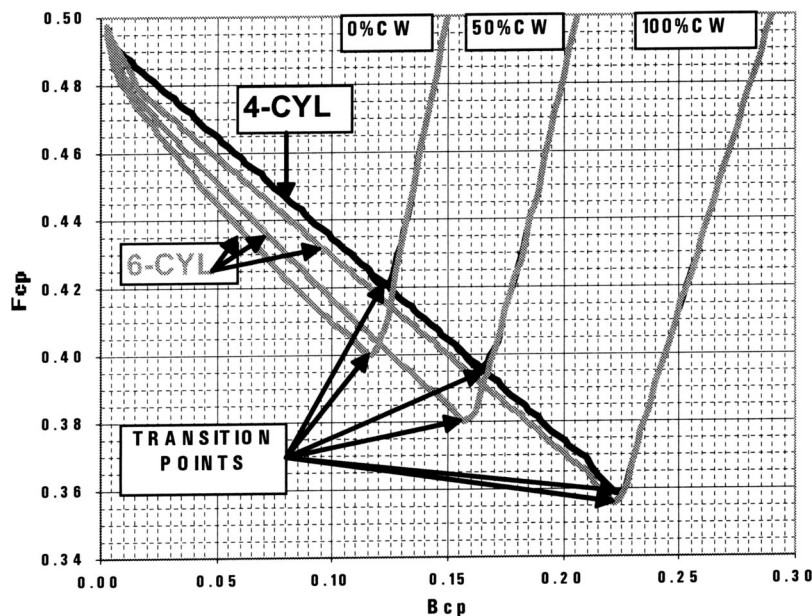


Fig. 17 Maximum bearing force parameter

Table 3 Four-cylinder model error analysis

%CW		Transition Point Value	Transition Point Range	% Error From Mean
0	B_{CP}	0.125	0.115–0.130	8%
50	B_{CP}	0.165	0.160–0.170	3%
100	B_{CP}	0.225	0.224–0.226	<1%
0	F_{CP}	0.420	0.405–0.425	4%
50	F_{CP}	0.395	0.380–0.410	4%
100	F_{CP}	0.360	0.350–0.360	<2%

The bearing characteristic parameter B_{CP} is defined as a weighted ratio of the maximum inertia force to the maximum pressure force and is given by

$$B_{CP} = \frac{\frac{1}{2}(\eta_R \cdot m_R \cdot R_R + \eta_{TR} \cdot m_{TR} \cdot R) \omega^2}{F_{P,MAX}} \quad (1.12)$$

The parameters η_R and η_{TR} are the contributions of the rotating inertia force and the reciprocating inertia force to the maximum bearing load, respectively. The parameter η_R lies between zero and 0.52 and η_{TR} lies between 0.48 and unity. These are given by

$$\eta_R = -0.28 \cdot \sigma_{CW}^2 - 0.24 \cdot \sigma_{CW} + 0.52 \quad (1.13)$$

$$\eta_{TR} = (1 - \eta_R) \quad (1.14)$$

The maximum bearing force parameters F_{CP} is defined as the ratio of the maximum bearing load to the maximum pressure force and is given by

$$F_{CP} = \frac{F_{B,MAX}}{F_{P,MAX}} \quad (1.15)$$

The parameters B_{CP} and F_{CP} shown in Fig. 17 represent an average of many simulations. The ranges of the transition points and corresponding errors are shown in Tables 3 and 4.

EXAMPLE PROBLEMS—Below are three solved example problems that demonstrate the usefulness of these parameters.

Problem #1 Statement. Estimate the maximum bearing load of the four cylinder engine ($\sigma_{CW}=50\%$) with properties listed in Table 1. The engine operates at $\omega = 175$ rad/sec.

Problem #1 Solution:

(1) The parameter $\sigma_{CW}=50\%$. From Eqs. (1.13) and (1.14),

$$\eta_R = -0.28 \cdot (0.50)^2 - 0.24 \cdot (0.50) + 0.52 = 0.33$$

$$\eta_{TR} = (1 - 0.33) = 0.67$$

(2) Use Eq. (1.12) to calculate B_{CP} ,

$$B_{CP} = \frac{\frac{1}{2}(\eta_R \cdot m_R \cdot R_R + \eta_{TR} \cdot m_{TR} \cdot R) \omega^2}{F_{P,MAX}} = \frac{\frac{1}{2}(0.33 \cdot 5.87 \cdot 0.0513 + 0.67 \cdot 3.45 \cdot .0675)(175)^2}{6.4E6 \frac{\pi}{4} \cdot 0.114^2}$$

Table 4 Six-cylinder model error analysis

%CW		Transition Point Value	Transition Point Range	% Error From Mean
0	B_{CP}	0.117	0.110–0.125	7%
50	B_{CP}	0.157	0.150–0.165	5%
100	B_{CP}	0.224	0.222–0.226	<1%
0	F_{CP}	0.400	0.380–0.420	5%
50	F_{CP}	0.380	0.360–0.400	5%
100	F_{CP}	0.355	0.350–0.360	<2%

$$B_{CP} = 0.060$$

Corresponding to this value on Fig. 17, $F_{CP}=0.046$.

(3) The maximum bearing load $F_{B,MAX}$ can be found by using Eq. (1.15):

$$F_{CP} = \frac{F_{B,MAX}}{F_{P,MAX}} \quad 0.46 = \frac{F_{B,MAX}}{6.4E6 \frac{\pi}{4} \cdot 0.114^2}$$

Solving yields

$$F_{B,MAX} \approx 30 \text{ kN} \quad \leftarrow \text{ANS}$$

Problem #2 Statement. Verify the transition point of the six cylinder engine ($\sigma_{CW}=0\%$) with the properties listed in Table 1.

Problem #2 Solution:

(1) The parameter $\sigma_{CW}=0$. From Eqs. (1.13) and (1.14),

$$\eta_R = -0.28 \cdot 0^2 - 0.24 \cdot 0 + 0.52 = 0.52$$

$$\eta_{TR} = (1 - 0.52) = 0.48$$

(2) Use Eq. (1.12) to calculate B_{CP} as a function of ω ,

$$B_{CP} = \frac{\frac{1}{2}(\eta_R \cdot m_R \cdot R_R + \eta_{TR} \cdot m_{TR} \cdot R) \omega^2}{F_{P,MAX}} = \frac{\frac{1}{2}(0.52 \cdot 5.87 \cdot 0.0513 + 0.48 \cdot 3.45 \cdot .0675) \omega^2}{6.4E6 \frac{\pi}{4} \cdot 0.114^2} \quad B_{CP} = 2.06E - 6 \cdot \omega^2$$

(3) The transition point in Fig. 17 corresponds to $B_{CP} \sim 0.117$. With this value, calculate $\omega_{\text{TRANSITION}}$:

$$0.117 = 2.06E - 6 \cdot \omega_{\text{TRANSITION}}^2$$

$$\omega_{\text{TRANSITION}} = 238 \text{ rad/sec} \quad \leftarrow \text{ANS}$$

This speed is reasonable. Figure 14 illustrates that the transition speed is a little lower than $\omega=250$ rad/sec.

Problem #3 Statement. An engineer wishes to estimate the percent of ideal counterweight mass to use for the six-cylinder engine shown in Table 1. He would like to design the counterweight mass such that the maximum bearing force is minimized at ~ 350 rad/sec. The maximum cylinder pressure is expected to be 84 bar.

Problem #3 Solution:

(1) Assume first that $\sigma_{CW}=0$. From Eqs. (1.13) and (1.14),

$$\eta_R = -0.28 \cdot 0^2 - 0.24 \cdot 0 + 0.52 = 0.52$$

$$\eta_{TR} = (1 - 0.52) = 0.48$$

(2) Using Eq. (1.15), calculate B_{CP} :

$$B_{CP} = \frac{\frac{1}{2}(\eta_R \cdot m_R \cdot R_R + \eta_{TR} \cdot m_{TR} \cdot R) \omega^2}{F_{P,MAX}} = \frac{\frac{1}{2}(0.52 \cdot 5.87 \cdot 0.0513 + 0.48 \cdot 3.45 \cdot .0675) \omega^2}{8.4E6 \cdot \frac{\pi}{4} \cdot 0.114^2} \quad B_{CP} = 1.57E - 6 \cdot \omega^2$$

(3) The transition point in Fig. 17 corresponds to $B_{CP} \sim 0.117$. With this value, calculate $\omega_{\text{TRANSITION}}$:

$$0.117 = 1.57E - 6 \cdot \omega_{\text{TRANSITION}}^2$$

$$\omega_{\text{TRANSITION}} = 273 \text{ rad/sec}$$

TOO LOW

(4) Try $\sigma_{CW}=50\%$. From above,

$$\eta_R = -0.28 \cdot (0.5)^2 - 0.24 \cdot (0.5) + 0.52 = \underline{0.33}$$

$$\eta_{TR} = (1 - 0.33) = \underline{0.67}$$

$$B_{CP} = \frac{\frac{1}{2}(0.33 \cdot 5.87 \cdot 0.0513 + 0.67 \cdot 3.45 \cdot .0675) \omega^2}{8.4E6 \cdot \frac{\pi}{4} \cdot 0.114^2}$$

$$B_{CP} = 1.49E - 6 \cdot \omega^2$$

The critical point in Fig. 17 corresponds to $B_{CP} \sim 0.157$:

$$0.157 = 1.49E - 6 \cdot \omega_{\text{TRANSITION}}^2$$

$$\omega_{\text{TRANSITION}} = 325 \text{ rad/sec}$$

STILL TOO LOW

(5) Try $\sigma_{CW}=100\%$. From above,

$$\eta_R = -0.28 \cdot (1)^2 - 0.24 \cdot (1) + 0.52 = 0$$

$$\eta_{TR} = (1 - 0) = 1$$

$$B_{CP} = \frac{\frac{1}{2}(0 \cdot 5.87 \cdot 0.0513 + 1 \cdot 3.45 \cdot .0675) \omega^2}{8.4E6 \cdot \frac{\pi}{4} \cdot 0.114^2}$$

$$B_{CP} = 1.36E - 6 \cdot \omega^2$$

The transition point in Fig. 17 corresponds to $B_{CP} \sim 0.225$

$$0.225 = 1.36E - 6 \cdot \omega_{\text{TRANSITION}}^2$$

$$\omega_{\text{TRANSITION}} = 407 \text{ rad/sec}$$

TOO HIGH

Therefore the counterweight mass should be between 50% and 100% of its ideal mass. Linear interpolation may be used to arrive at a reasonable σ_{CW} .

$$\sigma_{CW} = 50\% + \frac{350 - 325}{407 - 325} (100\% - 50\%) = \underline{65\%} \quad \leftarrow \text{ANS}$$

Conclusions

1. Dimensionless parameters have been developed that can be used to estimate the maximum bearing load in four-cylinder and six-cylinder symmetric in-line engines with given geometric properties and operating conditions. The parameters are accurate within an 8% margin of error for engines corresponding to the range of parameters listed in Table 2.
2. The maximum bearing load decreases with an increase in engine speed, reaches a minimum (the transition point), and then increases. This minimum represents the transition speed. At speeds lower than transition speed, the maximum bearing load is primarily determined by pressure forces. At higher speeds, it is mainly determined by inertia forces.
3. The transition speed increases with an increase in σ_{CW} .
4. The maximum bearing force is applied to bearing number two until the transition speed is reached. Beyond the transition speed, the maximum bearing force is applied to the middle bearing.
5. The average bearing load increases with engine speed and increases with a decrease in counterweight mass.
6. The middle bearing is consistently loaded by the largest average bearing force.

Future Considerations

1. The effect of counterweight mass on crankshaft vibration should be investigated.
2. Bearing characteristic parameters should be developed for common V-type engines and odd-numbered in-line cylinder arrangements.

Nomenclature

Λ	= ratio of the crank radius to the connecting rod length (R/L)
θ	= angular position of crank number 1 (rad)
θ_j	= angular position of crank "j" (rad)
$\Delta\theta_j$	= angular position of crank "j" with respect to crank number 1 (rad)
σ_{CW}	= percent of ideal counterweight mass
$\gamma_{i,j}$	= offset angle of counterweight mass "i,j" from 180 deg opposite of crank pin "j" (rad) ($i = 1, 2$)
β_j	= connecting rod angle of crank "j" (rad)
η_R	= weighing factor of rotating inertia forces
η_{TR}	= weighing factor of reciprocating inertia forces
ω	= crankshaft angular velocity (rad/sec)
$\omega_{\text{TRANSITION}}$	= transition speed (rad/sec)
B	= bore (mm)
B_1, B_2, B_3	= bearing numbers
B_{CP}	= maximum bearing characteristic parameter
B_{FP}	= maximum bearing force parameter
$F_{B,j}$	= force acting on bearing "j" (N)
$F_{CW,j}$	= Counterweight force of crank "j" (N)
$F_{iR,j}$	= rotating inertia force of crank "j" (N)
$F_{iTR,j}$	= reciprocating inertia force of crank "j" (N)
F_P	= pressure force (N)
j	= crank number
L	= connecting rod length (mm)
$m_{CW,j}$	= counterweight mass of crank "j" (Kg)
M_{iROCK}	= inertia rocking moment (Nm)
M_{iROLL}	= inertia rolling moment (Nm)
M_{iYAW}	= inertia yawing moment (Nm)
M_{pROLL}	= pressure rolling moment (Nm)
m_R	= rotating mass (Kg)
m_{TR}	= reciprocating mass (Kg)
N	= number of cranks
N_j	= normal force between the piston "j" and the cylinder liner (N)
$P(\theta)_{CC}$	= crankcase pressure (Pa)
$P(\theta)_{CYL}$	= cylinder pressure (Pa)
R	= crank radius (mm)
$R_{CW,j}$	= distance from the crank center to the center of gravity of counterweight mass "j" (Kg)
R_R	= distance from the crankshaft axis to the rotating mass (mm)
S	= stroke (mm)
$Z_A - Z_E$	= distances along the crankshaft (see Fig. 5) (mm)
$Z_{CW,j/o}$	= distance of counterweight "j" from the center of the crankshaft (mm)
$Z_{j/o}$	= distance of crank "j" from the center of the crankshaft (mm)

References

- [1] Eichhorn, U., Gassmann, S., and Schoenefeld, U, 1989, "Influence of Crankshaft Balancing Rate for a Gasoline Engine," *MTZ*, **50**, 10.
- [2] Schwarzel, W., and Bartsch, G., 1981, "The New 1.6L Opel 4-Cylinder OHC Engines," *MTZ*, **42**, 9.
- [3] Riad, M., and Ghazy, A, 1991, "Assessment of the Removal of the Crankshaft Balance Weights on Bearing Caps Vibration and Engine Noise," *SAE Paper No. 911058*.
- [4] Queregaesser, J., Meyer, J., Schaefer, E., and Wolschendorf, J., 1997, "The Effect of Cranktrain Design on Powertrain NVH," *SAE Paper No. 971994*.
- [5] Shigley, J., and Mischke, C., 1989, *Mechanical Engineering Design*, 5th Ed., McGraw-Hill, New York.

- [6] Prakash, V., Aprameyan, K., and Shrinivasa, U., 1998, "An FEM Based Approach to Crankshaft Dynamics and Life Estimation," SAE Paper No. 980565.
- [7] Mikulec, A., Reams, L., Chottiner, J., Page, R., and Lee, So-duk, 1998, "Cranktrain Component Conceptualization and Weight Optimization," SAE Paper No. 980566.
- [8] Raub, J., Jones, J., Kley, P., and Rebbert, M., 1999, "Analytical Investigation of Crankshaft Dynamics as a Virtual Engine Model," SAE Paper No. 1999-01-1750.
- [9] Bauer, Horst, 2000, *Bosch Automotive Handbook*, 5th Ed., Robert Bosch GmbH, Stuttgart, Germany, pp. 366–423.
- [10] Stanley, R., and Taraza, D., 2001, "Bearing Characteristic Parameters to Estimate the Optimum Counterweight Mass of a Six-Cylinder In-Line Engine," *ASME ICE Spring 2001 Conference Proceedings*, ASME, New York, **36**, p. 3.
- [11] Taraza, D., 1985, *Dinamica Motoarelor Cu Ardere Interna*, Editura Didactica Si Pedagogica, Bucharest, Romania.

Computational Modeling of Natural Gas Injection in a Large Bore Engine

Gi-Heon Kim

Allan Kirkpatrick

Charles Mitchell

Department of Mechanical Engineering,
Colorado State University,
Fort Collins, CO 80523

The topic of this paper is the computational modeling of gas injection through various poppet valve geometries in a large bore engine. The objective of the paper is to contribute to a better understanding of the significance of the poppet valve and the piston top in controlling the mixing of the injected fuel with the air in the cylinder. In this paper, the flow past the poppet valve into the engine cylinder is computed for both a low (4 bar) and a high pressure (35 bar) injection process using unshrouded and shrouded valves. Experiments using PLIF (planar laser induced fluorescence) are used to visualize the actual fluid flow for the valve geometries considered. The results indicate that for low injection pressures the gas flow around a typical poppet valve collapses to the axis of symmetry of the valve downstream of the poppet. At high pressure, the gas flow from this simple poppet valve does not collapse, but rather expands outward and flows along the cylinder wall. At high pressures, addition of a shroud around the poppet valve was effective in directing the supersonic flow toward the center of the cylinder. Additional computations with a moving piston show that at top dead center, the flammable volume fraction and turbulence intensity with high pressure shrouded injection are larger than for low pressure injection. [DOI: 10.1115/1.1762906]

1 Introduction

The topic of this paper is the computational modeling of gas injection and in-cylinder mixing in a large bore natural gas fueled engine. Natural gas fueled engines have been used for many years in stationary applications such as gas compression and electric power generation. In the United States alone, there are over 8000 large bore (bore >30 cm) slow speed (speed <500 rpm) natural gas engines in use. The most common configuration is a two-stroke cycle with direct injection of natural gas into the cylinder.

The injection of natural gas into the engine cylinder is usually done through the use of some type of mechanically or electrically actuated poppet valve. The effect of the poppet valve on the mixing of the gas is not fully understood. In contrast to a simple orifice, the poppet valve acts as a bluff body, over which the injected gas flows, producing an annular type flow pattern into the cylinder. One technology that has promise for improved engine performance and low emissions is the use of electronically controlled gas injectors operating at high injection pressures. Gas injection at high pressure can increase the momentum injected into the cylinder, increasing the mixture turbulence level and the overall fuel-air mixing.

In this paper we examine the injection of natural gas from two types of poppet valves, an unshrouded valve for low pressure injection (Type A) and a shrouded valve for high pressure injection (Type B). Both types of valves are currently used in practice in large bore natural gas fueled engines. In addition, the effect of the injection pressure on the subsequent mixing in the cylinder has not been extensively studied. Natural gas is typically injected at low pressures, 1–3 bar above manifold pressure. Since natural gas pipelines operate at pressures of the order of 35 bar, it is of interest to consider the use of this high pressure gas for direct injection into the engine cylinder.

There have been many studies of the injection and mixing of sonic gas flow from a simple orifice. As the gas injection pressure/cylinder pressure ratio increases, above a critical pressure ratio of

about 1.9, compressible flow phenomena such as rarefaction waves and shock formation need to be taken into consideration. Birch et al. [1] used gas chromatography, and Ewan and Moody [2] used shadowgraphy to study the behavior of underexpanded jets with injection/cylinder pressure ratios varying from 2 to over 50. Li et al. [3] used the method of characteristics and CFD modeling to compute the effect of the injection pressure on the gas jet width and the size and location of the Mach disk.

Many researchers investigating engine performance have used a virtual nozzle as an alternative to modeling the exact geometry of the gas injector. Downstream of the Mach disk, the jet flow field is subsonic, so the compressible flow features do not need to be computed. Recent examples of this approach are given in Ouellette and Hill [4] and Mather and Reitz [5]. This assumption has an advantage of decreasing the grid density requirements in the near field of the injection nozzle for an engine performance simulation, but it prevents the assessment of the effect of injector geometry and pressure. Numerical models of engine performance incorporating subsonic natural gas injection from a simple orifice into an engine cylinder are presented by Gaillard [6], Han et al. [7], and Gundappa et al. [8].

The studies of sonic and supersonic gas injection with a poppet valve are much more limited. Boyer et al. [9] used schlieren photography to study the injection and scavenging processes in a Cooper GMV engine. The results indicated that the injected gas impinged as a jet onto the piston top, and did not fully mix with the cylinder gases before impinging onto the piston. Two related areas are those of annular propulsive nozzles, which have been studied extensively for aerospace propulsion applications, Conley et al. [10]; and underexpanded coanda jets, in which a supersonic jet attaches to a adjacent surface, Sawada and Asami [11].

There are a number of methods used to quantify the mixing performance of a given injector. In this paper, two parameters, mixture mass fraction and fuel mass fraction are used. The mass fractions are divided into lean, flammable and rich categories. Papageorgakis and Assanis [12] used the flammable mixture mass fraction as a basis of comparison, while Abraham and Magi [13] used the flammable fuel mass fraction. The flammable mixture mass fraction, Eq. (1), is defined as the ratio of the flammable mass of mixture to the total mixture mass in the cylinder at

Contributed by the Internal Combustion Engine Division of ASME for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the ICE Division October 2002; final revision received by the ASME Headquarters December 2003. Associate Editor: D. Assanis.

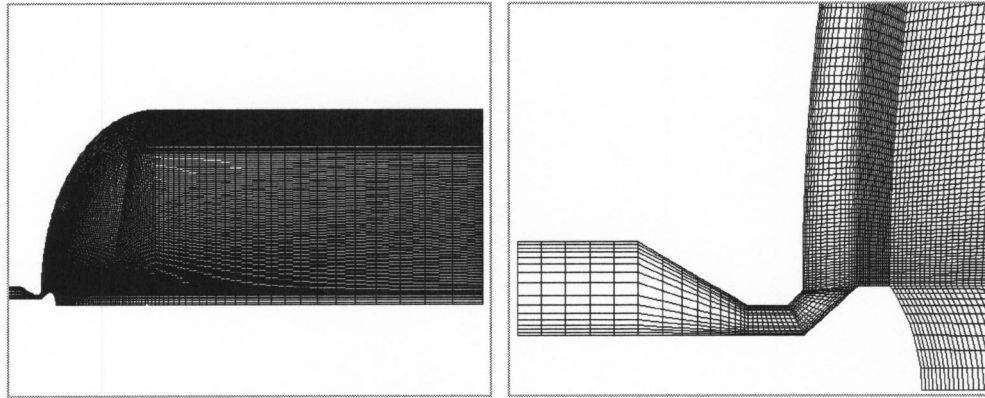


Fig. 1 Axisymmetric cylinder mesh for Type A poppet valve

any instant. The flammable fuel fraction, Eq. (2), is defined as the ratio of the mass of flammable fuel to the total fuel mass in the cylinder.

$$f_m = \frac{\text{flammable mass of mixture}}{\text{total mass of mixture}} \quad (1)$$

$$f_f = \frac{\text{mass of fuel included in flammable mixture}}{\text{total mass of fuel}} \quad (2)$$

The nonflammable (rich or lean) mixture mass fraction is related to the emissions performance, as it indicates the relative volume of the combustion chamber that may be susceptible to pollutant formation resulting from incomplete combustion. The flammable fuel mass fraction is related to the combustion efficiency, as it indicates the relative mass of fuel that is combustible.

2 Computational Experiments

The specific cylinder modeled in this paper is a Cooper GMV engine cylinder with a 35.5 cm cylinder bore and stroke, compression ratio of 9, and rated speed of 300 rpm. The intake ports for this two-stroke engine close at 134 deg before TDC, and the exhaust ports close at 103 deg before TDC. Start of injection is typically 120 deg BTDC for low pressure fuel injection and 115 deg BTDC for high pressure fuel injection. The low pressure poppet valve (Type A) is 2.39 cm in diameter, with a 0.635 mm maximum lift, and the shrouded high pressure valve (Type B) is 1.83 cm dia., with a 0.0635 cm lift. A smaller diameter and lift is required at higher pressures in order to keep an appropriate fuel mass flow rate due to increase in the density of the injected fuel gas. Both types of valves, as they are electronically controlled, have a top hat profile, in which the opening and closing times are much less than the fully open duration. The initial cylinder gases were assumed to be stationary, so that the effect of the two stroke scavenging flow is not included in this paper.

There were three different numerical models developed in sequence. The geometry of the CFD models included the complex details of the fuel poppet valve, piston crown, and cylinder head. The first model included the Type A valve configuration, the domed head and the cylinder wall but not the piston crown. This allowed investigation of the effect of the injection pressure on the fuel jet behavior in the cylinder in the absence of effects due to jet impingement on the piston. The poppet valve was fixed open at its maximum lift, and a constant injection (stagnation) pressure (4 bar and 35 bar) boundary condition was applied during the methane injection period. The second model represented an off-cylinder configuration with no cylinder head or sidewalls. This was developed in order to study the detailed flow structures in and immediately downstream of the valve. In addition, an off-engine experimental rig for this geometry was available for PLIF (planar laser

induced fluorescence) flow visualization and for subsequent comparison with and validation of the CFD predictions. The third model included a moving piston and simulated the compression stroke. This allowed investigation of the interaction of the fuel jet with both the moving piston top and the sidewalls.

The computational domain was discretized by using a variable size grid axisymmetric mesh. Grid adaptation was used to refine the mesh and locate the rapidly varying expansion and compression structures (including shock waves). Grid sensitivity computations indicated that at least ten grid points across the nozzle orifice were required in order to reduce the sensitivity of computational predictions to grid size in the nozzle region to an acceptable level. Two CFD solvers were used, Fluent, and Star CD. The turbulence model used was the standard $k-\epsilon$ model. The computations were carried out on a SUN Ultra Sparc workstation.

2.1 Injection Pressure Effects on Type A Valve Flow

Simulations were performed under two different pressure inlet conditions: a low-pressure supply (4 bar, pressure ratio 4:1) and a high-pressure supply (35 bar, pressure ratio 35:1). Computations for both supply pressures were performed for the typical low pressure poppet design designated as Type A. Figure 1 shows the mesh of the axis-symmetric domain used in these calculations, and represents the geometric configuration of the Cooper GMV large bore engine without a piston. Pure methane is injected through the valve into the cylinder, which is initially filled with quiescent air at 1 bar.

Flow patterns through the unshrouded valve operating at the low supply pressure and at the high supply pressure show clearly different tendencies. As shown in Figs. 2(a–c), in the case of low pressure injection, fuel jets collapse to the axis. On the other hand, the high pressure fuel jets spread outward and attach to the wall of cylinder head, as shown in Figs. 2(d–f). The left side of each figure in Fig. 2 is the methane mole fraction contour showing the extent of mixing and the right side is the velocity magnitude contour. The high velocity jet that forms in the high pressure case is of such strength that it never collapses back to the axis and instead impinges on and remains attached to the cylinder wall. Because of this, the high pressure jet would not impinge on the moving piston if it were present. Since this impingement and interaction with the piston is believed to be critical in enhancing mixing, the unshrouded design at high pressure is predicted to have poor mixing performance.

In general, the downstream behavior of the jet is probably of greatest interest as far as cylinder mixing is concerned. Still, the flow structures in and near the valve must be examined carefully since they impact directly the nature of the downstream jet. The initial stages of the injection process are shown in more detail in Figs. 3 and 4. The near field velocity magnitude contours (on the left-hand side) and the pressure contours (on the right-hand side)

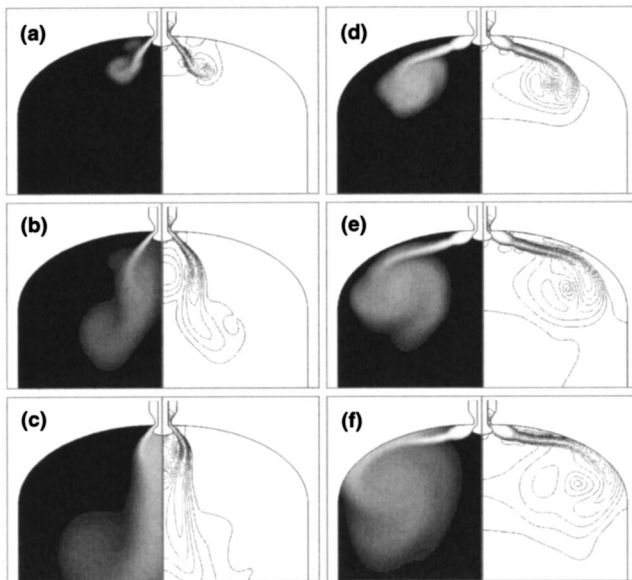


Fig. 2 Methane mole fraction and velocity magnitude for Type A poppet valve, (a–c) 4 bar injection, (d–f) 35 bar injection. (a) 1 ms after SOI, (b) 4 ms, (c) 7 ms, (d) 1 ms after SOI, (e) 2 ms, (f) 3 ms.

are shown in Fig. 3 for the low pressure injection, and in Fig. 4 for the high pressure injection. As shown in Fig. 3(a), at 2 ms after the start of injection, two local pressure maxima are indicated by points B and D, and two local minima and indicated by points A and C. The A and C minima correspond to the centers of vortices induced by the injected jet. Point D represents the jet tip. At point B, which is between the wall and the jet, mass accumulation oc-

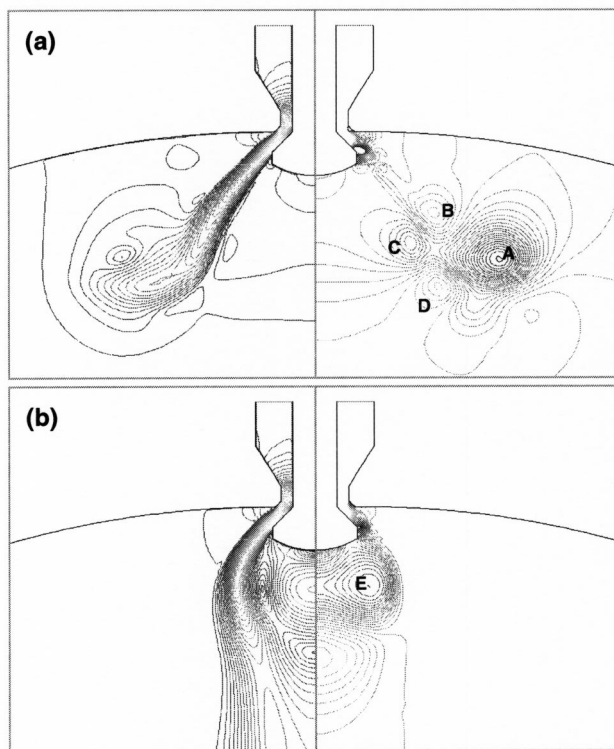


Fig. 3 Flow structures downstream of the Type A valve with 4 bar fuel injection. (a) 2 ms after SOI, (b) 7 ms after SOI.

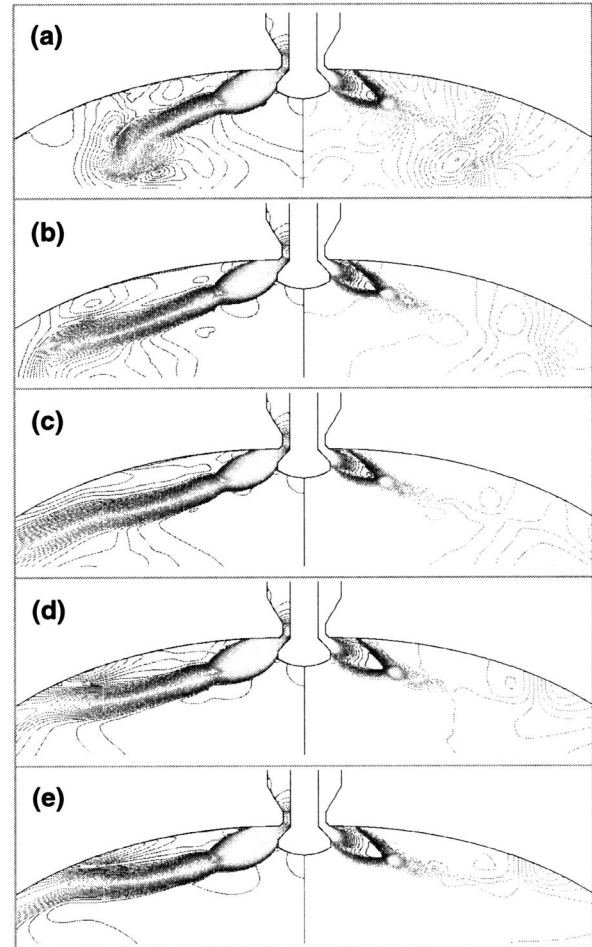


Fig. 4 Development of flow structures downstream of the Type A valve with 35 bar fuel injection; (a) 0.6 ms after SOI, (b) 1.2 ms, (c) 1.8 ms, (d) 2.4 ms, (e) 3.0 ms

curs due to the presence of the counterclockwise rotating vortex at A. Since the jet boundary blocks mass exchange through the jet, a pressure difference between the outer local maximum B and the inner local minimum C results. This positive pressure gradient across the jet forces the jet to collapse toward the centerline axis. Later in the injection process at 7 ms, shown in Fig. 3(b), a stationary vortex ring formed with a pressure minima at location E downstream of the valve pulls the jet to the centerline axis.

The sequence of images in Fig. 4 begin at 0.6 after the start of high pressure injection, and are 0.4 ms apart. In Fig. 4(a), injected fuel is expanded through the valve opening and a shock structure has already formed downstream and to the side of the valve exit. In Fig. 4(b), the fuel jet has not yet hit the cylinder wall, and there is no pressure gradient across the jet. In Fig. 4(c), the jet approaches the wall and an isolated circulation region is formed between jet and the wall of cylinder head. When the jet isolates the recirculation region completely, as indicated in Fig. 4(d), the pressure difference across the jet increases and stagnation occurs at the point of impingement on the wall. In Fig. 4(e), recirculation in the isolated region causes a further decrease in pressure and a significant pressure gradient across the jet is induced, pushing the fuel jet to the wall, creating a Coanda-type flow attached to the cylinder wall.

The steady-state flow through the type A valve is shown in Fig. 5. Figures 5(a) and 5(e) show the adapted grid for steady state computation of low and high pressure injection, respectively. Corresponding Mach number contours are presented in Figs. 5(b) and 5(f). The minimum Mach number contour scale value has a value

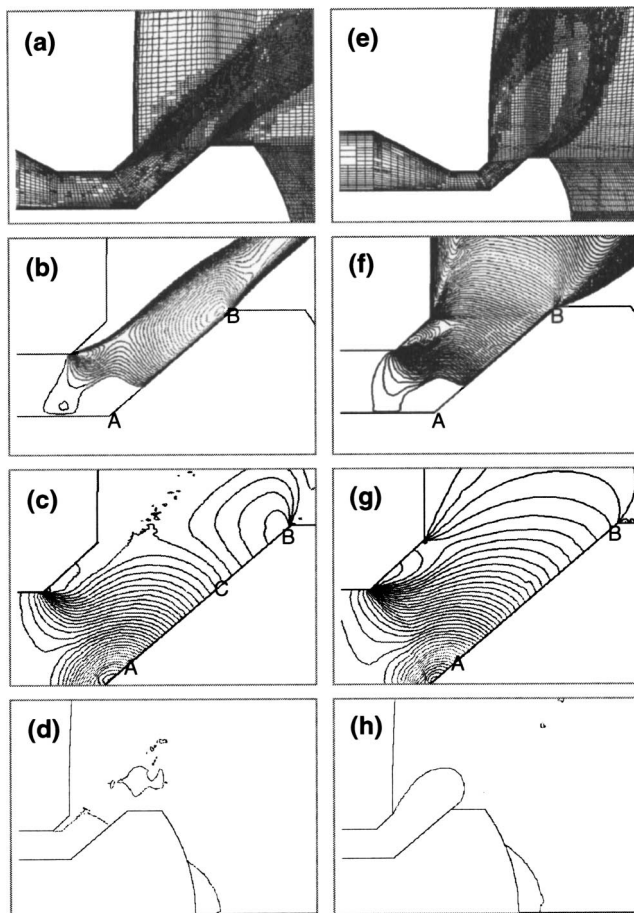


Fig. 5 Steady-state flow past Type A poppet valve, (a–d) 4 bar injection, (e–h) 35 bar injection. (a) Computational mesh, (b) Mach number contours, (c) static pressure contours, (d) cylinder pressure contour, (e) computational mesh, (f) Mach number contours, (g) static pressure contours, (h) cylinder pressure contour.

of one so that the area filled with contour lines represents the supersonic flow region. The fuel flow changes its direction and experiences compression when the bluff body of the valve head is encountered at point A, causing the complex transonic flow that is seen at this part of the valve. During the supersonic flow along the slope AB, expansion occurs continuously in both low and high pressure injection cases because the flow area increases in direct proportion to the distance from the cylinder axis. However, the flow patterns are quite different at point B depending on the injection pressure. In the low injection pressure case, the supersonic flow is compressed at point B while for the high pressure injection case the flow continues to expand. Static pressure contours in Fig. 5(c) show that the low pressure injection flow is overexpanded after passing point C, as the pressure falls below that of the cylinder. The flow is compressed by a shock structure at point B, since the pressure at the free jet boundary at point B must match the cylinder pressure. In contrast, with the high pressure injection case in Fig. 5(g), the flow continues to expand at and downstream of B through a centered expansion wave structure in order to meet the same free-jet boundary pressure condition. Pressure contour lines equal to the cylinder pressure, shown in Figs. 5(d) and 5(h), indicate where the flows have expanded to cylinder pressure.

2.2 Simulation of Off-Engine PLIF Test Cases. The meshes for unshrouded Type A and shrouded Type B valves used to simulate off-engine PLIF tests are presented in Fig. 6. The high pressure injection design included a shroud to direct the valve

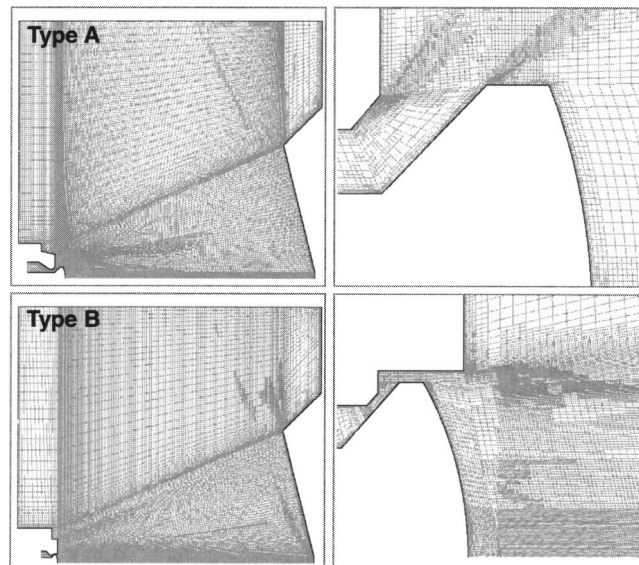


Fig. 6 Mesh for simulation of off-engine PLIF tests

outflow in the axial direction and reduce the radial expansion seen earlier in Fig. 4. The calculation domain shown includes the gas injector assembly and the piston top. In this set of computations, the jet is injected into an open cavity with no side walls. Since acetone seeded nitrogen was injected into ambient air at atmospheric pressure in the experimental tests, nitrogen was chosen as the injection gas in these computations. The details of the PLIF imaging experiment are described further in Olsen et al. [14].

The CFD computations are compared with the PLIF experimental results in Figs. 7 and 8. The left side of each frame is the PLIF image, and the computations appear on the right side. The supply/cylinder pressure ratio is 4 for the low pressure case, and 35 for the high pressure case. For the PLIF images the bright areas indicate regions of high intensity fluorescence from the acetone and thus also represent high concentrations of injected gas. For the CFD images the bright areas represent the injected gas concentration.

Figure 7 compares the transient injection flow for the low pressure injection case. The images are approximately 0.5 ms apart. The flow features contained in the CFD and the PLIF images are very similar. As the gas is injected into the cylinder, it flows over the poppet valve initially forming an S-shaped conical profile with a leading vortex, as shown in the first three images. The conical profile collapses to the axis downstream of the poppet valve, and the body of the jet behind the leading vortex has a relatively constant diameter. The vortex increases in size as it penetrates into the cylinder, and mixes with the cylinder gas.

The PLIF and CFD results for the high pressure injection case are shown in Fig. 8. The contrast between the high pressure jet with and without a shroud in place (see Fig. 4) is striking. With the shroud, the jet has a general similarity to an axially directed jet emanating from a hole of circular or annular cross section. A general qualitative agreement between experiment and computational prediction is apparent. A repeating pattern of low concentration zones followed by high concentration zones along the jet axis is evident in both sets of images. The location and extent of these zones correspond quite well at least for the first few occurrences. The flow downstream of the valve exit is supersonic and the high and low concentration zones result from the alternate reflection of compression and expansion waves from the axis and from the free boundary of the jet. The strength of successive compressions and rarefactions decreases due to viscous and turbulent

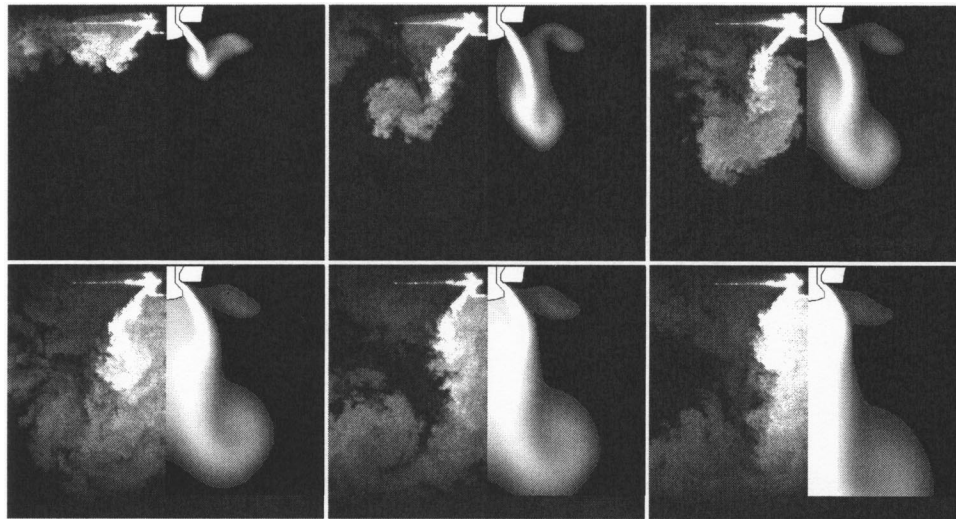


Fig. 7 Comparison of PLIF results and CFD with 4 bar injection

mixing losses. The jet diameter and spread rate with distance along the axis are approximately the same for both the PLIF and the computational images.

Contours of downstream static pressure and Mach number for the low-pressure injection case with the Type A (2.4 cm dia.) poppet valve are presented in Fig. 9 for a steady flow computation over a downstream distance of about eight valve diameters. In the same figure the corresponding centerline profiles of the static pressure (gage) and Mach number are also shown. Figure 10 shows the downstream contours and profiles for the high pressure injection case with the Type B (1.8 cm dia.) poppet valve for a downstream distance of about ten valve diameters. One of the important differences between the structures shown in these figures is that the low pressure injection case produce a subsonic downstream jet with a Mach number of about 0.5, while the high pressure injection produces a supersonic jet with a Mach number of 1.7. As mentioned in the previous section, the annular jet which occurs for low-pressure injection collapses and begins to merge at about 5 cm (2 in.) downstream from inlet. The static pressure along the axis reaches a local maximum at this stagnation point and decreases monotonically to the cylinder pressure. In contrast, the supersonic jet occurring for the high pressure injection case

has a shroud to direct the jet to the center and is repeatedly accelerated and decelerated due to the complex rarefaction and compression waves which result.

Axial velocity profiles in the radial direction are plotted at seven equally spaced downstream locations for low pressure injection with the Type A poppet valve and for high-pressure injection with the Type B poppet valve, respectively, in Figs. 11(a) and 11(b). Axial momentum flux distributions at the same locations are also presented in Fig. 11(c) and in Fig. 11(d). These figures illustrate the spatial development of the jets. The initially annular jets are eventually transformed into typical bell-shape profiles due to turbulent and molecular mixing in both cases. In the case of high pressure injection, the higher momentum inside the high pressure jet retards this transformation and, as a result of this, the bell shaped profile occurs much further downstream. The high-pressure jet has a much larger velocity gradient than the low-pressure jet because the jet radius is smaller and the core velocity is larger. This fact is important in that a large velocity gradient implies a good source for turbulence. Since high-level turbulence

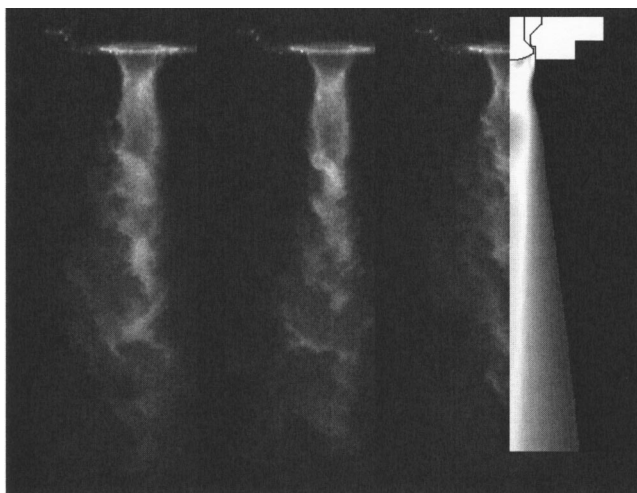


Fig. 8 Comparison of PLIF results and CFD with 35 bar injection

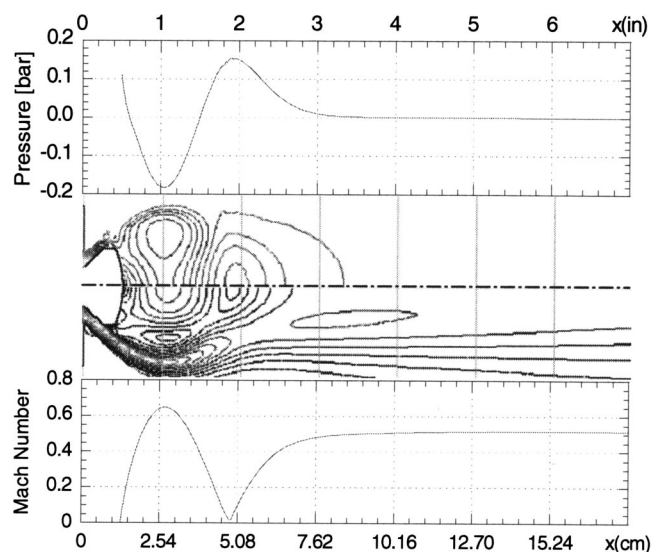


Fig. 9 Contours of static pressure and Mach number with corresponding axial profiles with 4 bar injection, steady flow computation

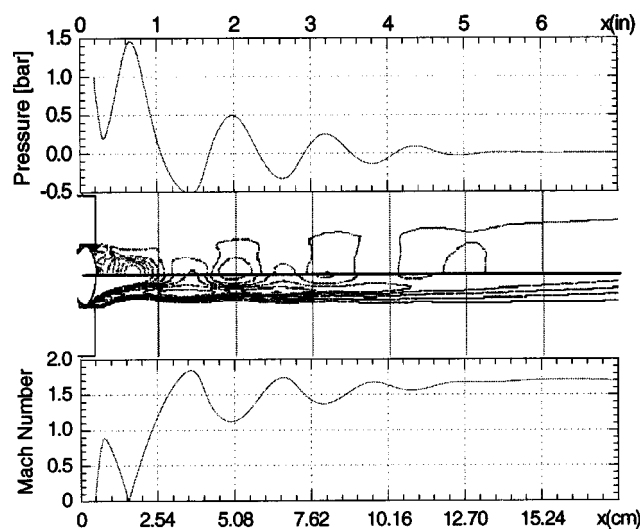


Fig. 10 Contours of static pressure and Mach number with corresponding axial profiles for 35 bar injection, steady flow computation

is believed to enhance fuel-air mixing and flame propagation, high pressure injection may have an advantage over low pressure injection in engine performance even in the absence of piston impingement effects. The axial momentum transfer rate is about 45 kg m s^{-2} for the low pressure injection case and about 100 kg m s^{-2} for the high pressure injection case, respectively.

Mass flow rates and energy flow rates passing through the same seven axial locations are plotted in Fig. 12. In spite of dissimilarity in the valve geometries and the operating pressure, the injected fuel mass rates are almost same in both gas admission systems. This is reflected by the total mass flow rates shown near the injection valves. The mass entrainment rate of the high pressure jet is higher, which implies that more air is being mixed into the fuel

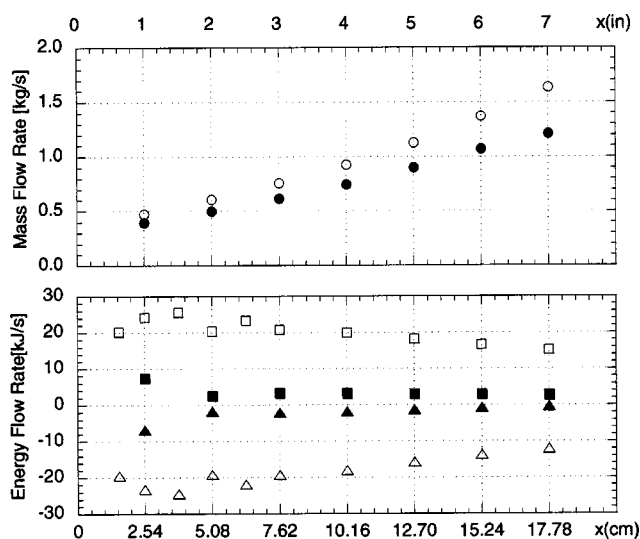


Fig. 12 Mass and energy flow rates along axial locations. Mass flow rate—(●) 4 bar injection, (○) 35 bar injection; kinetic energy flow rate—(■) 4 bar injection, (□) 35 bar injection; enthalpy flow rate—(▲) 4 bar injection, (△) 35 bar injection.

stream. Changes in the kinetic energy flow rate and enthalpy flow rate show opposite phase in both valve flows. This is to be expected and reflects the reversible and irreversible exchange between kinetic and thermal energy by waves and dissipation. In connection with this, the stagnation enthalpy flow rate, the sum of flow rates of kinetic energy and enthalpy, shows a small, monotonic increase in the downstream direction due to enthalpy inflow by entrainment. At the location where the high pressure valve flow begins to assume a bell-shaped profile there is approximately six or seven times as much kinetic energy in the jet as there is in the low pressure jet at the same axial location.

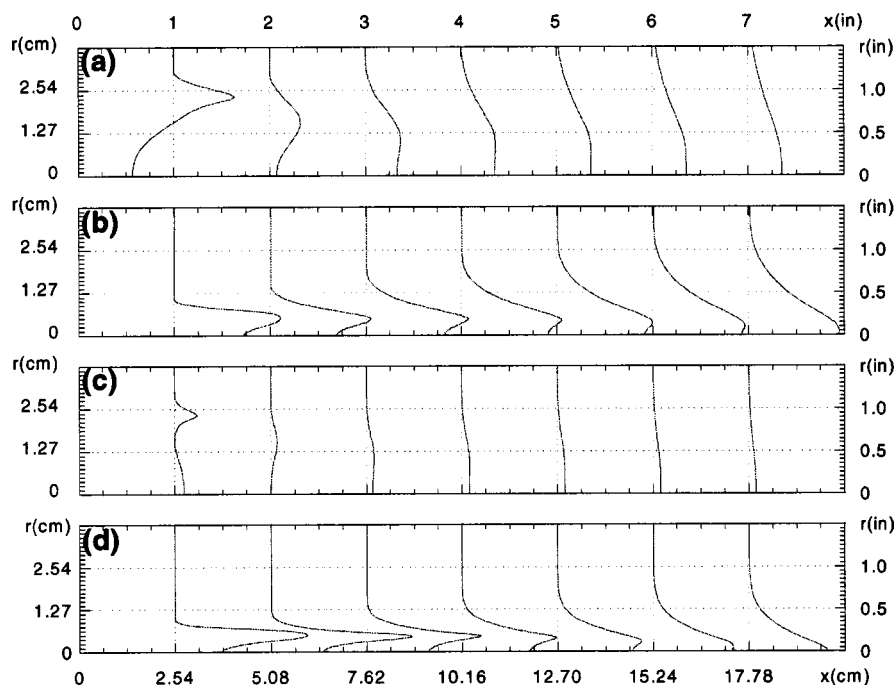


Fig. 11 Axial velocity (a–b) and momentum (c–d) profiles in radial direction at seven equally spaced downstream locations (a) 4 bar injection, (b) 35 bar injection, (c) 4 bar injection, (d) 35 bar injection

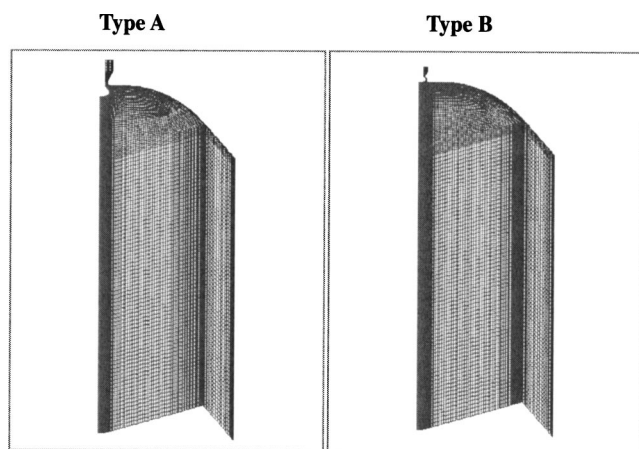


Fig. 13 Axisymmetric mesh for moving piston simulations

2.3 In-Cylinder Mixing Simulation With a Moving Piston.

Figure 13 shows the numerical grid systems used to compute gas injection in a cylinder with an upwardly moving piston. In this final set of computations the piston moves from bottom dead center (BDC) to top dead center (TDC). The unshrouded valve, Type A, was used for low pressure injection and the shrouded Type B for high pressure injection with a 35.5 cm cylinder bore and stroke. The initial cylinder pressure was assumed to be 1 bar, and

Table 1 In-cylinder mixing at top dead center

	P_i (bar)	SOI (deg btdc)	Duration (deg)	m_f (g)	ϕ	f_m	f_f
Short	35	115	20	1.81	0.66	0.84	0.95
	4	120	22	1.42	0.52	0.62	0.87
Long	35	122	32	2.89	1.06	0.98	0.99
	4	122	32	2.06	0.76	0.80	0.95

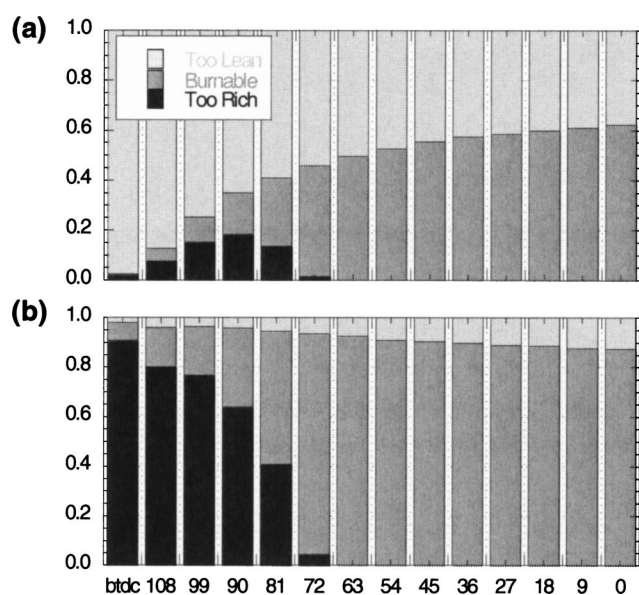


Fig. 14 Mixing fraction profiles with 4 bar injection. (a) Flammable mixture fraction, (b) flammable fuel fraction.

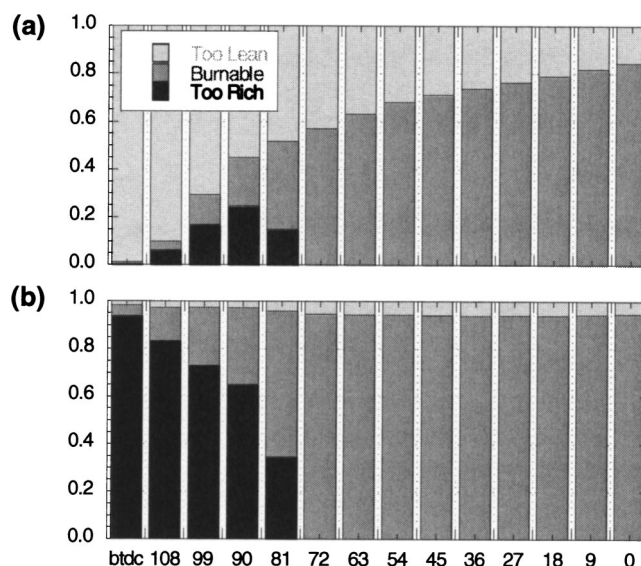


Fig. 15 Mixing fraction profiles with 35 bar injection. (a) Flammable mixture fraction, (b) flammable fuel fraction.

quiescent conditions were assumed at BDC. These axisymmetric computations were used to investigate the interaction between the injected fuel jet and the moving piston.

The mixture and fuel flammable fractions were computed by integrating over all of the computational cells in the cylinder volume. Calculating the flammable fuel fractions and flammable mixture fractions during the compression process for both cases facilitated comparison of the effectiveness of the fuel-air mixing for each injection system. A lean mixture in a cell was defined as one with an equivalence ratio less than 0.5, a flammable mixture has an equivalence ratio between 0.5 and 2.0, and a rich mixture has an equivalence ratio greater than 2.0. Our definitions of lean, rich and flammable mixtures are based on laminar flame propagation limits at atmospheric pressure and nominal atmospheric temperature. At top dead center, both pressure and temperature are sub-

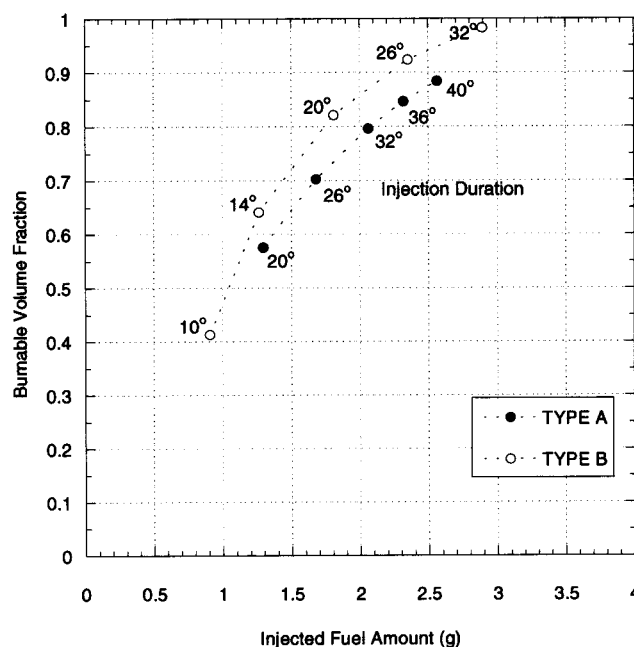


Fig. 16 Flammable mixture fractions at TDC as a function of injected fuel mass

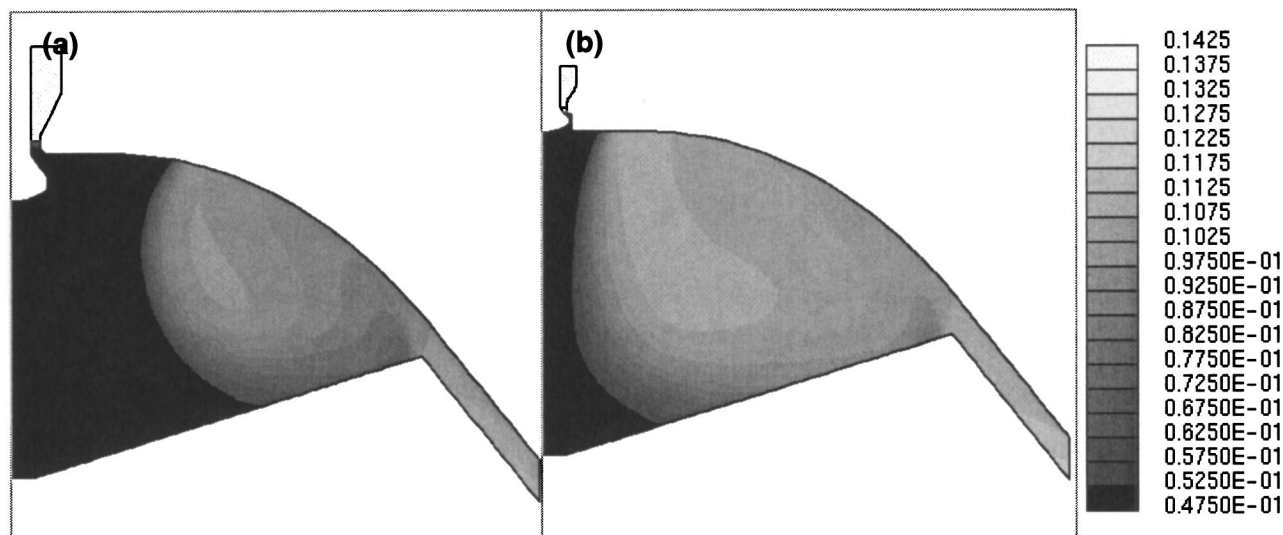


Fig. 17 Contours of methane mole fraction at TDC. (a) 4 bar injection, (b) 35 bar injection.

stantially higher. It is generally observed that the lean limit is not strongly affected by increasing pressure and temperature. Since our basic mixing problems exist on the lean limit our conclusions should still be valid in the main.

At an engine speed of 300 rpm, the time for the compression stroke is 100 ms. Two different injection cases were computed, a "short" duration of 20 deg crank angle (22 deg for the low pressure Type A case), which corresponds to 11 ms, and a 32 deg "long" duration of 18 ms, with the start of injection at about 120 deg before TDC. The specific start of injection (SOI) and injection duration are given in Table 1. The resulting injected fuel mass and overall equivalence ratios, which correspond to lean engine conditions, are given in Table 1. The total mass of fuel injected, $m_f(g)$ is less for the low pressure injection cases when the valve open times are approximately the same.

The computed flammable mixture fraction and the flammable fuel fraction profiles during the compression stroke for the short duration low and high-pressure cases are shown in Figs. 14 and 15. High flammable mixture fraction increasing rates appear about 5 deg crank angle after start of injection, for a duration of about 20 deg. This corresponds to the jet impingement period, indicating that jet impingement on the piston top enhances mixing. The val-

ues of the flammable mixture fraction (f_m) and the flammable fuel fraction (f_f) at TDC are listed in the last two columns of Table 1. In interpreting these results, recall from Figs. 11 and 12 that the high pressure jet has about twice the momentum flow rate and six times more kinetic energy flow rate than the low pressure jet by the time it impinges on the piston top. Examination of the computational results indicates that this additional momentum and energy produces more vigorous vortex structures in the cylinder as well as more substantial gas flow along the piston surface and cylinder wall.

In order to compare mixing results for the same injected fuel mass for both high and low-pressure injection, several additional computations were completed over a range of injection durations. Results of these calculations are presented in Fig. 16. In the figure, the flammable mixture fractions at TDC for the low and high pressure injection cases are plotted versus the amount of injected fuel. The start of injection (SOI) was fixed as 120 deg before TDC. The computational results indicate that for the same fuel mass injected, high pressure operation produces a 5–7% increase in flammable mixture fraction relative to low pressure operation.

The spatial distribution of the local fuel mole fraction at TDC for the "long" duration injection is shown in Fig. 17 for the low

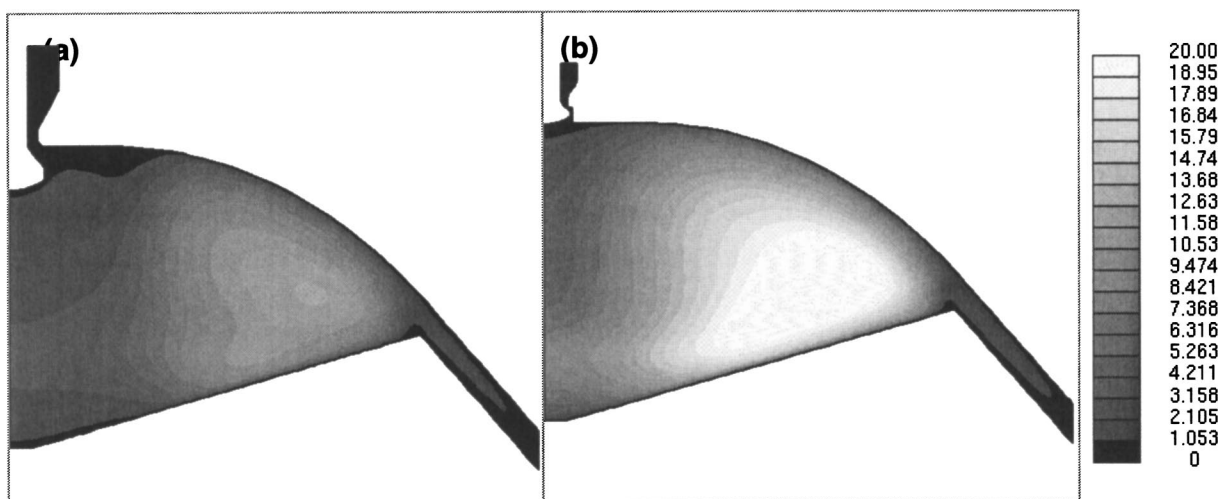


Fig. 18 Contours of turbulent kinetic energy (m^2/s^2) at TDC. (a) 4 bar injection, (b) 35 bar injection.

and high pressure injection cases. The lean areas for both pressure conditions occur in the interior of the cylinder. This is due to the redirection of the injected fuel by the top of the piston toward the sides of the cylinder. The relatively greater velocity of the high-pressure fuel jet allows for larger recirculation of the jet into the center of the cylinder and a corresponding larger fuel fraction in the cylinder center.

The spatial distribution of the turbulent kinetic energy (m^2/s^2) at TDC for the long duration injection computations is shown in Fig. 18. The turbulent kinetic energy is of interest as minimization of the combustion duration in an engine requires a high turbulent intensity. The high pressure jet creates increased turbulent kinetic energy relative to the low pressure jet, with the average value of $7.8 \text{ m}^2/\text{s}^2$ as compared to the average low-pressure value of $3.8 \text{ m}^2/\text{s}^2$. Clearly, at least some of the additional kinetic energy present in the high pressure injection becomes increased turbulent kinetic energy at TDC.

3 Conclusions

In this paper, the effects of injection pressure and valve geometry on fuel jet injection were investigated numerically. The results indicated that with an unshrouded valve, the low-pressure injection supersonic jet became over-expanded while flowing through the poppet valve, was rapidly compressed and slowed to subsonic speed after the valve exit, and eventually collapsed to the axis of symmetry of the valve downstream to form a "bell-shaped" subsonic jet. If the same valve was used with a high pressure supply, the supersonic flow was under-expanded in the valve and further expansion and spreading of the jet occurred after the valve exit, without jet collapse to the center of the cylinder.

At high pressures, addition of a shroud around the poppet valve was effective in directing the supersonic flow toward the center of the cylinder. With a shroud in place, the jet from a high pressure supply eventually formed a supersonic "bell-shaped" jet on the axis of symmetry some distance downstream of the valve exit. The downstream axial momentum flow from the high pressure shrouded valve was about twice that of the valve without a shroud operated at a low pressure, while the flow of kinetic energy was about six times as large. Computations performed for a cylinder model with a moving piston indicated that the shrouded valve operated at high pressure provided better mixing of fuel and air. The flammable mixture fraction at TDC was predicted to be 5% to 7% higher at TDC for high pressure injection.

Nomenclature

- f_m = flammable mixture fraction
- f_f = flammable fuel fraction
- m_f = mass of fuel (g)
- SOI = start of injection
- P_i = injection pressure (bar)
- ϕ = equivalence ratio

References

- [1] Birch, A. D., Brown, D., Dodson, M., and Swaffield, F., Dodson, M., 1984, "The Structure and Concentration Decay of High Pressure Jets of Natural Gas," *Combust. Sci. Technol.*, **36**, pp. 249–261.
- [2] Ewan, B. C. R., and Moodie, K., 1986, "Structure and Velocity Measurements in Underexpanded Jets," *Combust. Sci. Technol.*, **45**, pp. 275–288.
- [3] Li, Y., Kirkpatrick, A., Mitchell, C., and Willson, B., 2004, "Characteristic and Computational Fluid Dynamics Modeling of High Pressure Gas Jet Injection," *ASME J. Eng. Gas Turbines Power*, **126**, pp. 192–197.
- [4] Ouellette, P., and Hill, P., 2000, "Turbulent Transient Gas Injections," *ASME J. Eng. Gas Turbines Power*, **122**, pp. 743–753.
- [5] Mather, D. K., and Reitz, R., 2000, "Modeling the Effects of Auxiliary Gas Injection on Diesel Engine Combustion and Emissions," *SAE Paper 2000-01-0657*.
- [6] Gaillard, P., 1984, "Multidimensional Numerical Study of the Mixing of an Unsteady Gaseous Fuel Jet With Air in Free and in Confined Situations," *SAE Paper 840225*.
- [7] Han, Z., Tsao, K., and Abdalla, M., 1994, "Computation of the In-Cylinder Processes of a Natural Gas Engine," *SAE Paper 940213*.
- [8] Gundappa, M., Denlinger, M., Dulaney, K., Campbell, L., and McCarthy, J., 2000, "Computational Fluid Dynamics Modeling of Internal Combustion Engine Performance and Emissions," *Proc. Gas Machinery Conference*, Austin, TX.
- [9] Boyer, R., Craig, D., and Miller, C., 1954, "A Photographic Study of Events in a 14-In. Two-Cycle Gas Engine Cylinder," *Trans. ASME*, **76**, pp. 97–108.
- [10] Conley, R., Hoffman, J., and Thompson, H., 1985, "An Analytical and Experimental Investigation of Annular Propulsive Nozzles," *J. Aircr.*, **22**, pp. 270–276.
- [11] Sawada, K., and Asami, K., 1997, "Numerical Study on the Underexpanded Coanda Jet," *J. Aircr.*, **34**, pp. 641–647.
- [12] Abraham, J., and Magi, V., 1997, "Computations of Transient Jets: RNG k - ϵ Model Versus Standard k - ϵ Mode," *SAE Paper 970885*.
- [13] Papageorgakis, G., and Assanis, D., 1998, "Optimizing Gaseous Fuel-Air Mixing in Direct Injection Engines Using an RNG Based k - ϵ Model," *SAE Paper 980135*.
- [14] Olsen, D., Mastbergen, D., and Willson, B., 2001, "Planar Laser Induced Fluorescence Imaging of Gas Injection From Fuel Valves for Large Bore Natural Gas Engines," *Proc. ASME-ICE Fall Technical Conf.*, Vol. 37-2, Paper 2001-ICE-409, New Orleans, LA.

S. R. Krishnan
K. K. Srinivasan
S. Singh
S. R. Bell¹
K. C. Midkiff²
e-mail: cmidkiff@coe.eng.ua.edu

Department of Mechanical Engineering,
The University of Alabama,
290 Hardaway Hall,
Tuscaloosa, AL 35487-0276

W. Gong
S. B. Fiveland
M. Willi

Technical Center, Building F,
Caterpillar, Inc.,
P.O. Box 1875
Peoria, IL 61656-1875

Strategies for Reduced NO_x Emissions in Pilot-Ignited Natural Gas Engines

The performance and emissions of a single-cylinder natural gas fueled engine using a pilot ignition strategy have been investigated. Small diesel pilots (2–3% on an energy basis), when used to ignite homogeneous natural gas-air mixtures, are shown to possess the potential for reduced NO_x emissions while maintaining good engine performance. The effects of pilot injection timing, intake charge pressure, and charge temperature on engine performance and emissions with natural gas fueling were studied. With appropriate control of the above variables, it was shown that full-load engine-out brake specific NO_x emissions could be reduced to the range of 0.07–0.10 g/kWh from the baseline diesel (with mechanical fuel injection) value of 10.5 g/kWh. For this NO_x reduction, the decrease in fuel conversion efficiency from the baseline diesel value was approximately one to two percentage points. Total unburned hydrocarbon (HC) emissions and carbon monoxide (CO) emissions were higher with natural gas operation. The nature of combustion under these conditions was analyzed using heat release schedules predicted from measured cylinder pressure data. The importance of pilot injection timing and inlet conditions on the stability of engine operation and knock are also discussed.

[DOI: 10.1115/1.1760530]

Introduction

Achieving simultaneous reduction of NO_x and particulate emissions without compromising engine performance is the current focus of diesel engine research. Since this remains a difficult task, other alternatives to diesel combustion engines may be considered. Some of these include homogeneous charge compression ignition (HCCI) engines, spark-ignited natural gas combustion engines, and pilot-ignited natural gas combustion engines. Although HCCI promises diesel-like or higher efficiencies and much lower emissions, control of combustion in HCCI mode as well as achieving high brake mean effective pressure operation continue to pose serious challenges, [1]. Pilot-ignited natural gas combustion has been shown to match diesel efficiencies and produce significantly lower NO_x, [2], and particulate emissions, [3].

Pilot-ignited natural gas engines typically operate with natural gas as the primary fuel, which is either admitted into the intake manifold or injected directly into the cylinder early in the intake stroke. The natural gas-air mixture is ignited during the compression stroke by means of a small quantity of pilot diesel fuel. Therefore, the combustion process depends on pilot injection timing, relative amounts of pilot fuel and natural gas, in-cylinder charge temperature, and equivalence ratio.

Pilot-ignited natural gas combustion (also called dual fuel combustion) has been under investigation for many years, [4–6]. Conventional dual fuel engines use the stock diesel injection system to inject the pilot fuel, so the minimum pilot quantity possible is that for which the injection system is designed, i.e., the quantity used for idling the engine with diesel alone. To ensure acceptable operation at low loads, a relatively larger pilot quantity is often

required, [7]. The importance of pilot quantity vis-à-vis NO_x emissions in dual fuel engines is also well known, [8,9]. Previous research, [10], has shown that small pilot size can lead to lower NO_x emissions. Hence, there is a well-defined need to optimize pilot quantity relative to the gaseous fuel.

At very high loads combined with very high intake temperatures or pilot quantities, dual fuel engines are susceptible to knock, [11]. On the other hand, at low loads and high gaseous fuel substitutions, they are prone to misfire. These observations hold even for low-pilot operation, as will be discussed later, along with the results of the present work.

Experimental Setup

In this study, a single-cylinder direct-injection compression-ignition engine was used to investigate the performance and emissions characteristics of pilot-ignited natural gas combustion. The engine details are included in Table 1 and Fig. 1 shows a schematic of the experimental setup. The engine was coupled to a DC dynamometer through a torque meter. Intake, exhaust, coolant, and oil temperatures were measured using Type-K thermocouples. Crank angle resolved in-cylinder pressures were recorded using a piezoelectric transducer in combination with a shaft encoder. Injection timing of the pilot fuel was determined as the instant the injection pressure, measured as hoop-stress by a strain gage fitted on the injector, started decreasing. The cylinder pressure curves for 150 consecutive engine cycles were averaged and smoothed and a two-zone combustion model, [12], was used to derive heat release parameters from the average pressure curve. Compressed air was heated in an intake air tank to the required inlet air temperature and fed into the intake manifold of the engine along with natural gas. City natural gas was used and gas composition analysis showed that it contained 98.3% methane, 1.3% nitrogen, and less than 1% of ethane and carbon dioxide. A valve downstream of the exhaust tank was used to regulate exhaust backpressure.

Airflow rate was calculated by measuring pressure drop across a laminar flow element (LFE) and natural gas flow rate was measured using a thermal mass flow meter. The stock mechanical

¹Currently Dean, School of Engineering, University of Kansas, Lawrence, KS 66045.

²To whom correspondence should be addressed.

Contributed by the Internal Combustion Engine Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the ICE Division October 2002; final revision received September 2003. Associate Editor: D. Assanis.

Table 1 Engine details

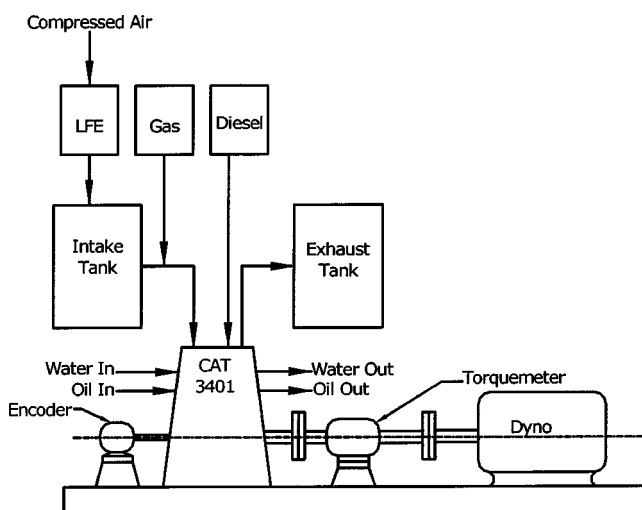
Engine make	Caterpillar 3401
Engine type	Single-cylinder, 4-stroke
Bore, mm	137
Stroke, mm	165
Compression ratio	14.5:1
Combustion system	Direct injection, Mexican hat
Fuel injection system	Electronic, common rail
Fuel injector	Pencil-type, four-holes
Injection timing	Variable
Engine rating	52 kW @ 2100 rev/min

diesel injection system, which was used for the baseline diesel tests, was replaced with an accumulator-type electronically controlled common-rail injection system. The original six-hole pencil-type diesel injector was replaced with a four-hole solenoid-operated injector. The injection system was capable of consistently injecting a minimum diesel quantity of 4.6 mm³/stroke and a maximum diesel quantity of 20 mm³/stroke. Additional details of the pilot injection system, its design, and operating principles are provided elsewhere, [13,14].

The emissions were sampled near the exhaust port and sent through a heated sample line to an integrated emissions bench comprised of Rosemount Analytical NGA 2000 series analyzers. Total hydrocarbon (HC) and NO_x measurements were performed on the hot, undried exhaust sample using a heated flame ionization detector (Model HHC) and two chemiluminescent detectors (Model HNOX), respectively. The two NO_x instruments provided several full-scale ranges from 0–10 ppm up to 0–10,000 ppm. Carbon monoxide and carbon dioxide were measured using the nondispersive infrared method and oxygen using the paramagnetic method in a separate analyzer (Model MLT).

Results and Discussion

The performance and emissions results obtained with pilot-ignited natural gas operation of the single-cylinder engine indicate the potential benefits of this mode of combustion. The overall set of experiments involved engine testing over a range of pilot injection timings, pilot quantities, and intake manifold conditions at a constant speed of 1700 rev/min and different loads. The following four sections of this paper focus on the results for full load (42 kW, 1220 kPa bmeq) engine operation at fixed pilot quantity and inlet conditions, and at different pilot injection timings; however, results for other loads are also provided in a later section. The full load tests were repeated on three separate days of testing and the

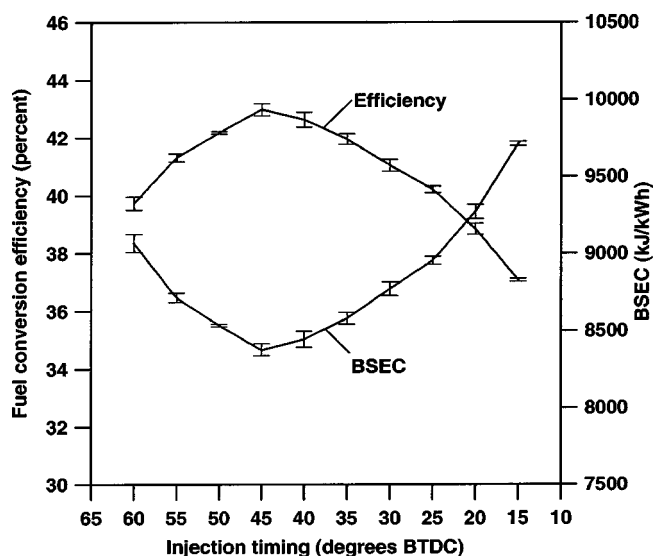
**Fig. 1 Schematic of experimental setup**

data repeatability is shown as error bars of one standard deviation about the mean. The intake manifold pressure (P_{in}), exhaust backpressure (P_{ex}), and inlet air temperature (T_{in}) were chosen to simulate a turbocharged engine and maintained at 202 kPa, 191 kPa, and 35°C, respectively. The pilot diesel quantity (m_{pilot}) was fixed at 3.3 g/min, which amounted to about 2–3% (depending on the fuel conversion efficiency) of the total injected energy associated with both fuels. These inlet and exhaust conditions and pilot quantity were chosen (by trial and error) to ensure knock-free operation at all injection timings. Also, as the pilot fuel injection system was of the accumulator-type, the magnitude of the injection pressure determined the amount of pilot fuel delivered, and one could not be varied independently of the other. The results reported in Figs. 1–8 were obtained at an injection pressure of 354 bar, which corresponded to the minimum possible pilot quantity of 4.6 mm³/stroke (approximately 3.3 g/min at 1700 rev/min).

Engine Performance

Figure 2 shows fuel conversion efficiency and brake specific energy consumption at different pilot injection timings. Fuel conversion efficiency is defined as the ratio of the brake power to the total rate of energy input into the engine calculated from the measured total diesel and natural gas flow rates multiplied by their respective lower heating values; brake specific energy consumption (BSEC) is the reciprocal of the fuel conversion efficiency in units of kJ/kWh. Some natural gas could survive the combustion process inside the cylinder or directly escape into the exhaust manifold during the valve overlap period. This is accounted for in the measured total natural gas flow rate used in calculating the total energy input rate. It must be noted that these efficiencies correspond to operation with a separate motor-driven fuel injection system and artificially boosted intake and exhaust manifold pressures that result in a positive pumping mean effective pressure. As a result, measured efficiencies are likely higher than would be obtained if the engine was used to drive the fuel injection system and an actual turbocharger was used to boost the intake pressure. Consequently, comparisons of efficiencies between various operating conditions of this engine are more meaningful than comparison to efficiencies of other engines.

As injection timing was advanced from 15 deg to 45 deg before top dead center (BTDC), fuel conversion efficiency increased from 38 to about 43%. Upon further advance of the injection timing, the efficiency decreased to about 40% at 60 deg BTDC.

**Fig. 2 Fuel conversion efficiency and brake specific energy consumption versus injection timing**

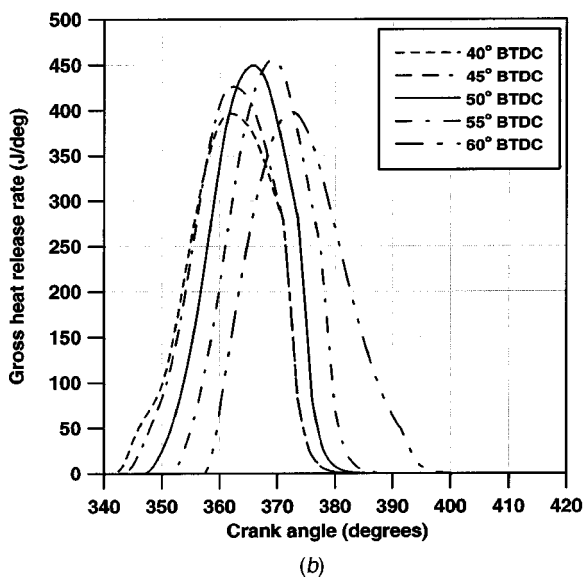
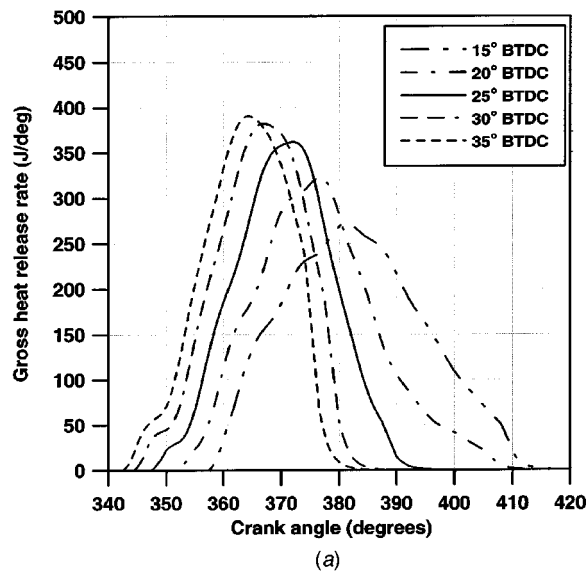


Fig. 3 (a) Gross heat release rate versus crank angle for injection timings between 15 deg and 35 deg BTDC. (b) Gross heat release rate versus crank angle for injection timings between 40 deg and 60 deg BTDC.

For reference, baseline diesel operation at the same speed (1700 rev/min), power (42 kW), and same intake and exhaust manifold conditions, yielded a fuel conversion efficiency of 41.3%. The stock mechanical diesel injection system was used for the baseline diesel operation and the injection timing was 22 deg BTDC.

The behavior of fuel conversion efficiency at different injection timings can be explained as follows. At a fixed speed and power, the fuel conversion efficiency for any injection timing depends on the start of heat release, the magnitude of peak heat release rate, the phasing of heat release about top dead center (TDC), and the duration of heat release. Thus, several heat release patterns can possibly result in the same power output; however, different heat release patterns will lead to different fuel conversion efficiencies. For instance, for a given amount of fuel, more work can be extracted during expansion if a high heat release occurs for a short time and occurs very close to TDC. When brake power and engine speed are held constant for all injection timings, the particular heat release pattern that provides for greater work per unit mass of fuel translates into higher fuel conversion efficiency.

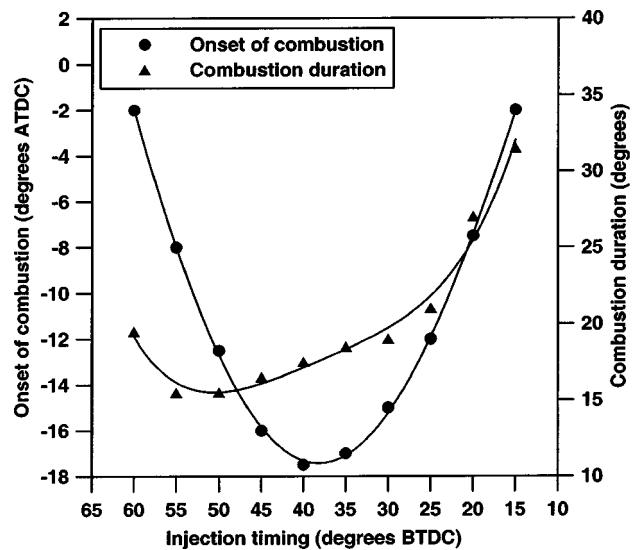


Fig. 4 Onset of combustion and combustion duration (10%–90%) versus injection timing

Figure 3(a) shows the gross apparent heat release curves for injection timings between 15 deg and 35 deg BTDC. As injection timing was retarded from 35 deg to 15 deg BTDC, the start of heat release was increasingly delayed, the overall heat release process was shifted away from TDC and the duration of heat release was increased. Further, the peak heat release rates were also decreased as injection timing was retarded from 35 deg to 15 deg BTDC. These effects contributed to the loss in fuel conversion efficiency between 35 deg and 15 deg BTDC. Figure 3(b) shows the gross heat release rate curves for injection timings between 40 deg and 60 deg BTDC. As injection timing was advanced from 40 deg to 60 deg BTDC, although the start of heat release was delayed, the duration of heat release did not change significantly. Nevertheless, the delayed start of heat release led to a decrease in fuel conversion efficiency for timings advanced beyond 45 deg BTDC.

Figure 4 shows the onset of combustion in degrees after top dead center (ATDC) and duration of combustion for the results shown in Figs. 3(a) and (b). The duration of combustion shown here is the period in crank angles from 10 to 90% mass burn. The onset of combustion was earliest for injection timings between 30 deg and 45 deg BTDC. For timings advanced beyond 45 deg

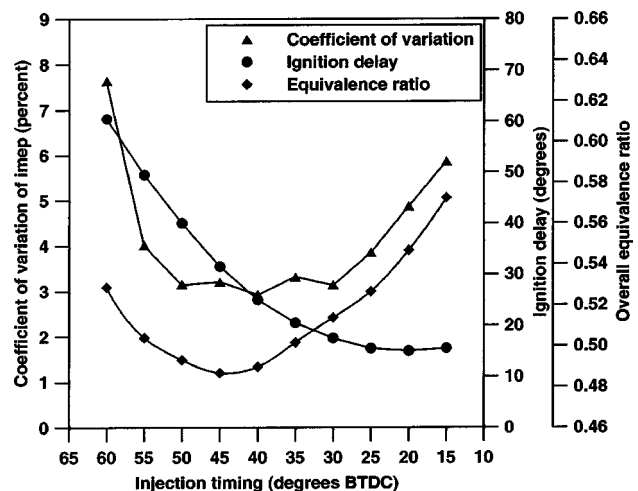


Fig. 5 Coefficient of variation of imep, ignition delay, and overall equivalence ratio versus injection timing

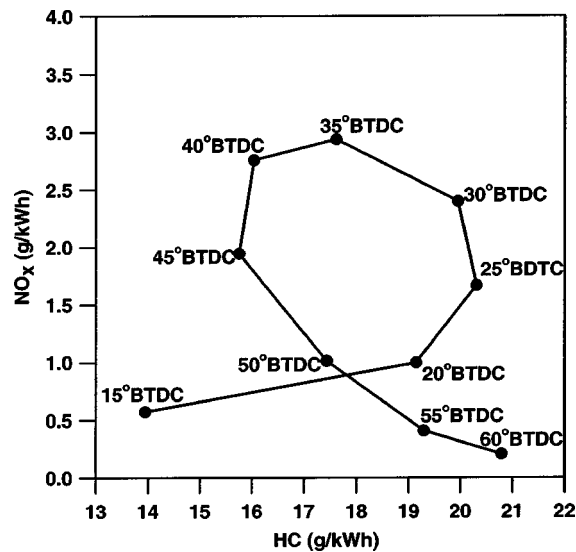


Fig. 6 Mean NO_x -HC tradeoff curve at different injection timings

BTDC or retarded beyond 30 deg BTDC, combustion began progressively closer to TDC. However, combustion duration did not follow the same trend. For instance, although combustion started at about the same time for both 15 deg and 60 deg BTDC, the combustion duration for the former was much longer than that of the latter. So, combustion occurred for a longer time and well into the expansion stroke for 15 deg BTDC, thereby reducing the fuel conversion efficiency compared to 60 deg BTDC. From a fundamental standpoint, there is a change in the nature of the combustion process as injection timing is advanced from 15 deg to 60 deg BTDC. In conventional dual fuel engines, pilot injection typically occurs close to TDC (around 15 deg BTDC) and the pilot ignites after an ignition delay period affected by local temperature and mixture composition. Under such conditions, the flame is initiated at the periphery of the pilot spray where the local equivalence ratio is near-stoichiometric, so most of the NO_x is formed in these regions where the local flame temperatures are high. Thus, NO_x formation in conventional dual fuel engines scales with pilot size. On the other hand, at the low pilot quantities and advanced injection

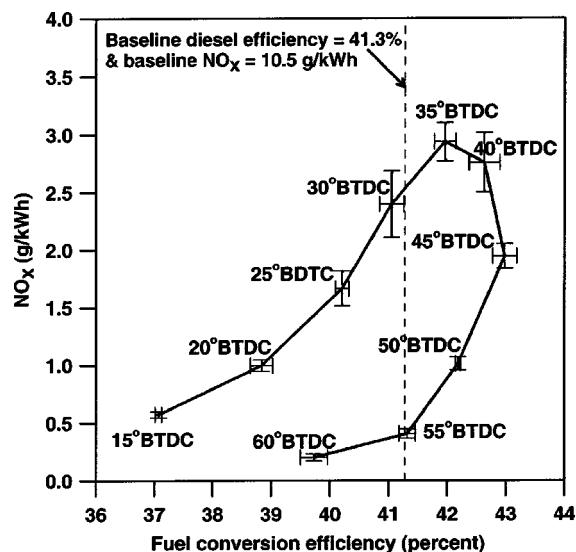


Fig. 7 NO_x -fuel conversion efficiency tradeoff curve at different injection timings

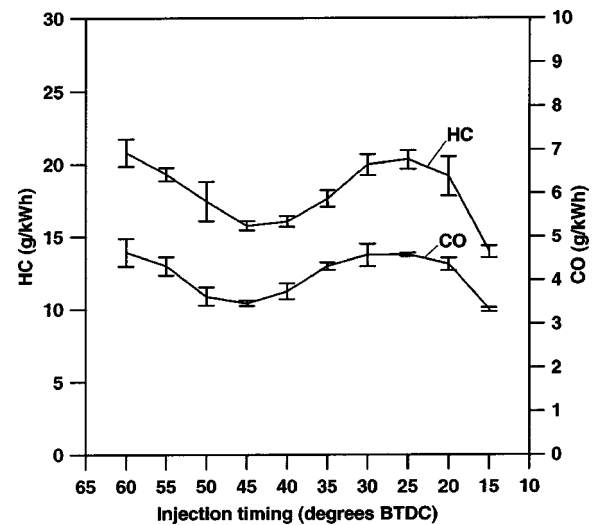


Fig. 8 HC and CO emissions versus injection timing

timings (such as 60 deg BTDC) that have been considered in this study, the combustion process is substantially different. Since a very small amount of pilot fuel is injected very early in the compression stroke, it disperses into the lean premixed natural gas-air mixture. As in-cylinder temperatures are not high enough to ensure fast diesel evaporation or ignition, the diesel-air mixture does not ignite immediately. In fact, conditions conducive for ignition are not obtained until the piston is close to TDC. The prolonged ignition delay period (observed in Fig. 5) allows more time for diesel evaporation, better dispersion of the diesel fuel in the cylinder, and greater diesel-air mixing prior to ignition. This likely results in a multitude of ignition centers that are well dispersed throughout the engine cylinder. These distributed ignition kernels lead to rapid flame propagation through the surrounding natural gas-air mixture at advanced injection timings such as 60 deg BTDC. The combustion duration shows little variation for injection timings between 30 deg and 60 deg BTDC when compared to timings retarded beyond 30 deg BTDC. For timings between 30 deg and 60 deg BTDC, since sufficient time is available for diesel-air mixing prior to ignition, the overall combustion process is driven more by the flame propagation speed, which does not vary much for the similar equivalence ratios and temperatures of the unburned natural gas-air mixture at those timings. On the other hand, as timing is retarded beyond 30 deg BTDC, the shorter time available for diesel-air mixing results in poor dispersion of the ignition centers in the cylinder, which in turn reduces flame propagation rates, thereby increasing the combustion duration.

Engine Stability and Knock

Pilot-ignited natural gas engines tend to exhibit cycle-to-cycle variations at high natural gas substitutions. These variations are manifested as torque fluctuations. To quantify these variations, the coefficient of variation (COV) of indicated mean effective pressure (imep) was calculated as the ratio of the standard deviation of imep of 150 successive engine cycles to the average imep, expressed as a percentage.

Figure 5 shows the variation of COV, ignition delay and fuel equivalence ratio with injection timing. Between 30 deg and 50 deg BTDC timings, there was little variation in COV. For timings more advanced than 50 deg BTDC or more retarded than 30 deg BTDC, COV increased. It is believed that COV is related to the cycle-to-cycle inconsistencies in ignition or combustion phasing with respect to TDC. Between 30 deg and 50 deg BTDC, the ignition delay was long enough for adequate diesel-air mixing before ignition and, hence, ignition was apparently more repeatable from one cycle to another. However, outside this range, the

variations in ignition or combustion phasing apparently increased depending on whether injection timing was advanced or retarded, respectively. The maximum value of COV was obtained at the most advanced timing of 60 deg BTDC. The high cycle-to-cycle variation for 60 deg BTDC injection timing is attributed to possible over-leaning of the diesel fuel in the cylinder in some engine cycles. The greater dispersion of diesel in the surrounding lean natural gas-air mixture may have led to over-leaning of some of the diesel-air mixture. This over-leaning may hinder the creation of local ignition centers in some regions or cause extinguishment of ignition centers after they are created. In either case, the overall effectiveness of the ignition process would be compromised, thus leading to poor combustion in those cycles which experience over-leaning. Depending on the extent to which local ignition is prevented and ignition centers are extinguished after creation, some engine cycles can experience totally or partially misfiring cycles at highly advanced injection timings. Thus, for 60 deg BTDC injection timing, cycle-to-cycle variations in the overall effectiveness of ignition appear to be the cause of high COV. On the other hand, for timings retarded beyond 30 deg BTDC, inconsistencies in ignition would not have been so important due to the shorter ignition delay periods and mixing times. For instance, with 15 deg BTDC injection timing, poor dispersion of diesel in the cylinder would have prevented over-leaning of the diesel, and ensured consistent ignition. However, as bulk of the combustion process occurred under progressively cooler cylinder temperatures in the expansion stroke, there would have been considerable cycle-to-cycle variations in burn rates, which result in cycle-to-cycle variations in imep. Therefore, variations in the phasing of the combustion process and burn rates appear to be the cause of high COV for 15 deg BTDC injection timing.

It is well known, [15], that knock in pilot-ignited natural gas (dual fuel) engines arises due to end-gas autoignition. Hence, the incidence of knock depends on the rate of combustion as well as its phasing relative to TDC. The occurrence of knock can be avoided by several measures including (but not restricted to) decreasing the equivalence ratio of the fresh charge (by increasing boost pressure), reducing the intake temperature, and employing exhaust gas recirculation (EGR). All of these measures lead to a reduction in preignition reaction rates in the unburned gas ahead of the flame, thereby suppressing the onset of autoignition. High engine loads typically involve higher in-cylinder temperatures, higher equivalence ratios, and faster combustion rates. Pilot-ignited natural gas engines are most susceptible to knock at full load conditions, especially when the combustion event is fast and occurs close to TDC. These conditions are similar to the requirements for high fuel conversion efficiency. Therefore, for the test results reported herein, 40 deg–50 deg BTDC timings, which had very high combustion rates and efficiencies, exhibited the greatest *inclination* to knock. However, for the results reported in this section, the inlet conditions (202 kPa and 35 degC) were fixed for all injection timings, and as mentioned before, were chosen so as to ensure knock-free operation at all timings.

NO_x Emissions

NO_x formation in engines is governed by the local conditions in the cylinder. It depends on the concentrations of nitrogen and oxygen, and is extremely sensitive to local temperatures. Therefore, the most favorable conditions for NO_x formation are obtained during the combustion process when high local temperatures exist along with adequate availability of nitrogen and oxygen. Since pilot-ignited natural gas engines operate under fuel-lean conditions, there is always excess air, which ensures availability of nitrogen and oxygen. Thus, NO_x formation in pilot-ignited natural gas engines is strongly governed by local temperatures.

The variation of full-load NO_x and HC emissions for a fixed pilot quantity and at different injection timings is shown in the form of a tradeoff curve in Fig. 6. NO_x emissions peaked at 35

deg–40 deg BTDC and decreased sharply as timing was advanced to 60 deg BTDC. Very low NO_x emissions were possible because of the very small pilot spray surrounded by lean, nearly homogeneous natural gas-air mixture. The overall equivalence ratio (based on both diesel and natural gas) for all injection timings was in the range of 0.49–0.57, as shown in Fig. 5, with the less efficient injection timings requiring higher natural gas fueling rates (pilot quantity being constant) and thus, higher equivalence ratios. For instance, the highest equivalence ratio of 0.57 was obtained for 15 deg BTDC timing while the least value of 0.49 was obtained for 45 deg BTDC timing. In addition to the NO_x reductions with advanced injection timings, there was also significant NO_x reduction for timings retarded beyond 35 deg BTDC. Yet, NO_x emissions for 15 deg BTDC timing were more than twice those for 60 deg BTDC timing. The reasons for the reduction in NO_x emissions for 60 deg BTDC and 15 deg BTDC injection timings are quite different. For 60 deg BTDC timing, since diesel was injected very early in the compression stroke, the ignition delay was high and, as previously mentioned, adequate time available for diesel-air mixing would have resulted in a lean, locally more homogeneous diesel-air-mixture at the time of ignition. Therefore, the local temperatures encountered during combustion would have been significantly lower, resulting in reduced NO_x emissions for 60 deg BTDC timing. It is also possible that at advanced timings such as 60 deg BTDC, the fraction of the total engine-out NO_x emissions formed due to combustion of natural gas alone (during flame propagation) would be more significant than for 35 deg BTDC injection timing; for the latter, the pilot's contribution to NO_x would be much greater than natural gas. For injection timings between 35 deg and 15 deg BTDC, the same quantity of pilot fuel was injected progressively later in the compression stroke when in-cylinder temperatures were higher and much closer to the self-ignition temperature range of diesel. As a result, the ignition delay period for 15 deg BTDC timing was much shorter. During this ignition delay period (about 15 deg CA), less time was available for diesel-air mixing. In addition, ignition occurred closer to TDC and bulk of the combustion process occurred in the expansion stroke. Therefore, the initial heat release rates, which are controlled by the amount of diesel-air mixture prepared before ignition, were relatively low. Moreover, this low heat release occurred later in the expansion stroke and over a longer duration, resulting in lower local temperatures, and thus, lower NO_x emissions. This reduction in NO_x levels obtained by retarding the pilot injection timing is similar to typical NO_x reduction strategies employed in conventional diesel and dual fuel engines.

To understand the overall relative merit of employing advanced or retarded injection timings to reduce NO_x emissions, it is important to consider the fuel conversion efficiencies and HC emissions associated with those timings. The behavior of NO_x and fuel conversion efficiencies at different injection timings is shown as a tradeoff curve in Fig. 7. NO_x emissions for the maximum efficiency timing (45 deg BTDC) were much higher than those for very retarded (15 deg BTDC) or very advanced (60 deg BTDC) timings. Along with NO_x reduction, efficiency also decreased when injection was advanced or retarded from 45 deg BTDC. However, it is interesting to note that the efficiency for 60 deg BTDC was about 3% greater than the efficiency for 15 deg BTDC and NO_x emissions were lower as well. Thus, it is clearly more beneficial to advance the pilot injection timing to reduce NO_x emissions and maintain minimal loss in fuel conversion efficiency.

HC and CO Emissions

Most of the unburned hydrocarbon emissions in pilot-ignited natural gas engines are unburned methane, [16], and in view of the fact that the natural gas used in the present work contained about 98% methane, the fraction of methane in the total exhaust hydrocarbons is expected to be high; however, this fraction was not measured in the experiments reported here. Now, although methane is a well-known greenhouse gas, methane emis-

Table 2 Best NO_x—efficiency results for low pilot-ignited natural gas operation at full load, which occurred at $T_{in}=45\text{degC}$, $m_{pilot}=4\text{ g/min}$

Efficiency (%)	Inj. Timing (degBTDC)	P_{in} (kPa)	NO _x (g/kWh)	HC (g/kWh)	CO (g/kWh)	COV (%)
39.3	65	240	0.07	30.8	6.3	7.8
40.2	65	229	0.10	24.3	5.4	7.3
41.3*	60	229	0.18	22.5	4.9	4.8
42.3	55	229	0.43	20.0	4.6	4.4
43.3	50	229	0.92	17.8	4.5	4.5

*Same as the baseline diesel efficiency value with stock mechanical injection system, similar intake and exhaust conditions, and 22 deg BTDC injection timing.

sions from engines are currently not regulated by the U.S. EPA. Nevertheless, it is important to reduce unburned methane emissions from pilot-ignited natural gas engines, at least from a fuel conservation standpoint.

The possible sources of HC emissions in engines include in-cylinder crevices, quenching of the flame front close to the cylinder walls, and bulk quenching of the fuel-air mixture in partially misfiring engine cycles. Moreover, in engines with continuous fuel aspiration into the intake manifold (like the present case), some unburned fuel could escape through the exhaust during the scavenging process when both intake and exhaust valves are open. The relative importance of these sources of HC emissions from pilot-ignited natural gas engines is difficult to determine. For instance, HC from in-cylinder crevices has been shown to be more important than wall quenching of the flame front in SI engines, [17]. However, crevice HC may not be as important in pilot-ignited natural gas engines since they operate at lean air-fuel ratios and, therefore, the amount of fuel trapped in the crevices may be relatively low. When partially misfiring engine cycles are encountered in pilot-ignited natural gas combustion, a significant portion of the fuel-air mixture may fail to undergo combustion. This is referred to as “bulk quenching,” which occurs in the bulk of the cylinder gases in partially misfiring engine cycles as opposed to “wall quenching” of the flame, which occurs close to the combustion chamber walls in every engine cycle. Further, since the single-wall quench distance has been shown to have an inverse dependence on charge equivalence ratio, [18], wall quenching becomes more pronounced at lean equivalence ratios. In view of the above observations, it can be said that bulk quenching and wall quenching are the dominant sources of HC in pilot-ignited natural gas engines.

The factors that affect HC oxidation are charge temperature during expansion, oxygen availability, and time available for oxidation. Since pilot-ignited natural gas combustion occurs under fuel-lean conditions, there is generally enough oxygen available for HC oxidation. Therefore, charge temperature and time available for oxidation are the critical parameters affecting HC oxidation in the expansion stroke. Ideally, for oxidation of HC, sufficiently high temperatures should be maintained for a long time in the expansion stroke.

CO emissions from engines are controlled primarily by the fuel-air equivalence ratio, [19]. For fuel-rich mixtures, CO emissions increase steadily with equivalence ratio and for fuel-lean mixtures, they vary little and are generally low.

Figure 8 shows the variation of HC and CO emissions with injection timing. For reference, baseline diesel operation at full load, intake and exhaust conditions similar to pilot-ignited natural gas operation, and 22 deg BTDC injection timing yielded HC and CO emissions of 0.87 g/kWh and 0.54 g/kWh, respectively. For natural gas operation, as injection timing was advanced from 45 deg to 60 deg BTDC, both HC and CO emissions increased. As previously mentioned, for timings advanced beyond 50 deg BTDC, the COV was considerably higher. Hence, these timings were more susceptible to partially misfiring cycles, resulting in greater bulk quenching of the flame, thereby leading to higher HC formation. Since charge temperatures would have been considerably lower due to partial misfire as well as delayed onset of combustion, oxidation of the “bulk-quenched-HC” would not be complete. Therefore, both CO and HC emissions increased from 45 deg to 60 deg BTDC.

As injection timing was retarded from 45 deg to 25 deg BTDC, HC, and CO emissions increased again. Since, the COV between these timings was relatively low and did not vary significantly, bulk quenching may not have occurred. However, it was already observed that between these timings, the peak heat release rates were lower and the onset of combustion was delayed. This delay could have resulted in the continuation of flame propagation later in the expansion stroke, when relatively lower in-cylinder temperatures could have increased the wall-quenched HC formation. Since this wall-quenched HC was formed after combustion, the lower charge temperatures could have slowed down its oxidation thus increasing the net HC and CO emissions.

Finally, Fig. 8 also shows that for timings retarded beyond 25 deg BTDC, both HC and CO emissions decreased, especially between 20 deg and 15 deg BTDC timings. From Figs. 3(a) and 4, it is seen that for both 20 deg and 15 deg BTDC timings, the heat release peaks were lower and the combustion durations were longer compared to 25 deg BTDC. Although slower combustion could have resulted in more wall-quenched HC for 20 deg and 15 deg BTDC, the time available for oxidation also increased since combustion duration increased. For 15 deg BTDC timing, which had the longest combustion duration, the time for which high temperatures persist in the cylinder would have increased and hence more complete oxidation of HC and CO could have occurred.

Summary of Overall Test Results

The results discussed to this point were not necessarily optimized for high fuel conversion efficiency or minimal NO_x. Preliminary tests to optimize engine-out NO_x emissions and efficiency were performed by operating the engine at full load and a range of intake pressures (200–240 kPa), intake temperatures (30°–50°C), and pilot fuel quantities (3.3–4.0 g/min). The best NO_x emissions obtained for different diesel-equivalent efficiencies were categorized from several different operating conditions. Table 2 is the summary of a possible “efficiency-NO_x” tradeoff for knock-free full-load operation at the aforementioned range of operating conditions. It may be seen that the lowest NO_x emissions recorded while maintaining baseline diesel-like efficiency (41.3%) were 0.18 g/kWh. The lowest NO_x emissions associated with the best efficiency (43.3%) increased to 0.92 g/kWh. On the other hand, if efficiency could be compromised by 2% compared

Table 3 Summary of efficiency and emissions results for different loads

Load (kW)	Bmep (kPa)	Efficiency (%)	Inj. Timing (degBTDC)	P_{in} (kPa)	T_{in} (degC)	m_{pilot} (g/min)	NO _x (g/kWh)	HC (g/kWh)	CO (g/kWh)	COV (%)
10.5	305	29.8	55	135	115	3.3	0.20	59.7	16.6	6.6
21.0	610	38.3	55	184	95	3.3	0.18	36.5	14.4	6.2
31.5	915	40.0	60	184	55	3.3	0.08	30.0	6.3	7.6
42.0	1220	41.3	60	229	45	4	0.18	22.5	4.9	4.8

to baseline diesel, NO_x emissions could be reduced to about 0.07 g/kWh. It is interesting to note that the best NO_x emissions for all fuel conversion efficiencies were obtained at advanced injection timings. Accompanying these low NO_x emissions, however, were significantly higher HC and CO emissions, and higher COV. Thus, there exists a clear tradeoff between NO_x emissions on the one hand and fuel conversion efficiency, HC emissions, and engine stability on the other. Table 3 summarizes the best NO_x emissions obtained at diesel-equivalent efficiencies for different loads. For all loads, NO_x emissions of 0.2 g/kWh or lower were achieved at advanced injection timings. However, HC and CO emissions increased significantly at lower loads. It may also be observed that low-load operation requires much higher intake temperatures and lower boost pressures to achieve good fuel conversion efficiencies and engine stability.

Conclusions

A single-cylinder compression-ignition engine was used to investigate different strategies for achieving low NO_x and good efficiency performance from pilot-ignited natural gas combustion. The results obtained lead to the following important conclusions:

1. Satisfactory pilot-ignited natural gas engine operation was achieved with low pilot quantities (2–3% by energy).
2. Significant reduction in NO_x emissions levels to approximately 0.2 g/kWh while maintaining full load baseline diesel fuel conversion efficiency (41.3%) were possible by employing relatively advanced (around 55 deg–60 deg BTDC) pilot injection timings.
3. For slightly lower efficiencies (baseline diesel efficiency less 1 or 2%), the NO_x emissions could be reduced to the range of 0.07–0.10 g/kWh; however, these NO_x benefits were accompanied by some penalty in HC emissions and engine stability.
4. Fuel conversion efficiencies 1–2% greater than baseline diesel value were also possible and the best NO_x under these conditions was about 0.43–0.92 g/kWh.
5. Carbon monoxide and unburned hydrocarbon emissions were higher than corresponding diesel values for all conditions, particularly at low loads.
6. Cycle-to-cycle variations in engine operation increased with very early (60 deg BTDC) or very late (15 deg BTDC) pilot injection.

Acknowledgments

Financial support from Caterpillar, Inc., the Alabama Department of Economic and Community Affairs, and The Center for Advanced Vehicle Technologies at The University of Alabama,

which receives partial funding from the U.S. Dept. of Transportation Federal Highway Administration under Grant DTFH61-99-X-00007, is gratefully acknowledged.

References

- [1] Flowers, D. L., Aceves, S. M., Smith, R., Torres, J., Girard, J., and Dibble, R., 2000, "HCCI in a CFR Engine: Experiments and Detailed Kinetic Modeling," SAE Paper No. 2000-01-0328.
- [2] Workman, J., and Beshouri, G. M., 1990, "Single Cylinder Testing of a High Pressure Electronic Fuel Injector for Low NO_x Emission Dual Fuel Engines Part II: Optimization, Startability and Inflammability Testing," ASME J. Eng. Gas Turbines Power, **112**, pp. 422–430.
- [3] Karim, G. A., 2000, "Combustion in Gas-Fuelled Compression Ignition Engines," *In-Cylinder Flows and Combustion Processes*, The Soichiro Honda Lecture—2000, Paper No. 2000-ICE-299, **35-1**, ASME, New York.
- [4] Elliot, M. A., and Davis, R. F., 1951, "Dual-Fuel Combustion in Diesel Engines," Ind. Eng. Chem., **43**(12), pp. 2854–2864.
- [5] Moore, N. P. W., and Mitchell, R. W. S., 1955, "Combustion in Dual-Fuel Engines," *Proceedings of the Joint Conference on Combustion*, Oct. 25–27, Institution of Mechanical Engineers, London, pp. 300–309.
- [6] Karim, G. A., 1968, "Combustion in Dual Fuel Engines—A Status Report," *Eighth International Conference on Combustion Engines*, Congress Palace, Brussels, May 6–10, CIMAC (International Council on Combustion Engines).
- [7] Karim, G. A., 1991, "An Examination of Some Measures for Improving the Performance of Gas Fuelled Diesel Engines at Light Load," SAE Paper No. 912366.
- [8] Abd Alla, G. H., Soliman, H. A., Badr, O. A., and Abd Rabbo, M. F., 2000, "Effect of Pilot Fuel Quantity on the Performance of a Dual Fuel Engine," Energy Convers. Manage., **41**, pp. 559–572.
- [9] Ouelette, P., Douville, B., Hill, P. G., and Ursu, B., 1998, " NO_x Reduction in a Directly Injected Natural Gas Engine," *Gas Engines and Alternative Fuels*, Vol. **31-3**, ASME, New York, Paper No. 98-ICE-151.
- [10] Krishnan, S. R., Biruduganti, M., Mo, Y., Bell, S. R., and Midkiff, K. C., 2002, "Performance and Heat Release Analysis of a Pilot-Ignited Natural Gas Engine," *International Journal of Engine Research*, **3**(3), pp. 171–184.
- [11] Karim, G. A., 1983, "The Dual Fuel Engine of the Compression Ignition Type—Prospects, Problems and Solutions—A Review," SAE Paper No. 831073.
- [12] Krishnan, S. R., 2001, "Heat Release Analysis of Dual Fuel Combustion in a Direct Injection Compression Ignition Engine," M.S. thesis, The University of Alabama.
- [13] Gebert, K., Beck, N. J., Barkhimer, R. L., Wong, H. C., and Wells, A. D., 1996, "Development of Pilot Fuel Injection System for CNG Engine," SAE Paper No. 961100.
- [14] Gebert, K., Beck, N. J., Barkhimer, R. L., and Wong, H. C., 1997, "Strategies to Improve Combustion and Emission Characteristics of Dual-Fuel Pilot Ignited Natural Gas Engines," SAE Paper No. 971712.
- [15] Felt, A. E., and Steele, W. A., 1962, "Combustion Control in Dual-Fuel Engines," SAE Trans., **70**, pp. 644–653.
- [16] Karim, G. A., Liu, Z., and Jones, W., 1993, "Exhaust Emissions From Dual Fuel Engines at Light Load," SAE Paper No. 932822.
- [17] Alkidas, A. C., 1999, "Combustion-Chamber Crevices: The Major Source of Engine-Out Hydrocarbon Emissions Under Fully Warmed Conditions," Prog. Energy Combust. Sci., **25**(3), pp. 253–273.
- [18] Adamczyk, A. A., and Lavoie, G. A., 1978, "Laminar Head-On Flame Quenching—A Theoretical Study," SAE Paper No. 780969.
- [19] Heywood, J. B., 1988, *Internal Combustion Engine Fundamentals*, McGraw-Hill, New York.

Large Eddy Simulation of the Anti-Erosion Characteristics of the Ribbed-Bend in Gas-Solid Flows

J. R. Fan

Professor
e-mail: fanjr@mail.hz.zj.cn

K. Luo

Assistant Professor

X. Y. Zhang

Assistant Professor

K. C. Cen

Professor

Institute for Thermal Power Engineering and
CE&EE,
Zhejiang University,
Hangzhou 310027, P.R. China

In order to find out the ribs with the best anti-erosion efficiency, the erosion effects in three 90° duct bend gas-solid flows with different ribs are investigated and compared with that in the bare bend. Three different kinds of ribs are studied, which have square cross section, isosceles right-angled triangle shape cross section and isosceles triangle shape cross section, respectively. The arrangement and the geometry dimensions of the ribs are the same. The gas phase flow field is solved by the large eddy simulation technique considering the interaction between the gas and the particles, whereas the particles are tracked by Lagrangian method. The results exhibit that the erosion rates of all the ribbed bends are greatly reduced due to the adding of the ribs compared with that of the bare bend. Among the three different kinds of ribs, the isosceles right-angled triangle ribs have the highest anti-erosion ability. The reason is that the particle impact velocity on the walls in the isosceles right-angled triangle ribbed bends is decreased to the utmost and the corresponding particle impact angle is altered adequately due to the special shape of the ribs. In addition, the rib erosion rate, the sidewall erosion rate and the bend wall erosion rate are calculated separately. It is found that the rib erosion rate is half of the wall erosion rate, and the sidewall erosion rate is so low that it can be omitted compared with the bend wall erosion rate. [DOI: 10.1115/1.1760523]

Introduction

The gas-particle flows in the duct bend are frequently encountered in engineering applications such as injecting coal into entrained-flow gasifiers, pneumatic conveying of powders in transport lines, sand blasting etc. The material erosion problems stand out because the solid particles transporting through the duct bend impact on the bend wall with the higher velocity and the corrosive angle, which can threaten the machine reliability and shorten the service life of equipments. In addition, the erosion rate of a bend could be 50 times higher than that of a straight duct under the same test conditions (Tilly [1]). So, it is necessary to find out some higher efficiency methods for protecting bends from erosion to conserve the engineering flow system equipments.

Erosion of metal surface by solid particles has been widely studied, [2,3]. The results show that three factors greatly affect the metal erosion by particle impact. The first is the properties of the impinging particles, such as the particle type, the mechanical properties, the size distribution and the geometrical shape. The second is the nature of target materials, such as the material properties, the duct orientation and curvature, and the surface conditions. The third is the system operation conditions including the particle impact velocity, the impact angle, the particle number density, and the properties of the carrier fluid. As far as the same particles and the same target material nature are concerned, the third factor is the most important to evaluate the potential erosion. To minify the particle impact velocity on the metal surface or to change the particle impact angle to an apt range can reduce the duct erosion largely.

So far, several methods have been developed to protect bends from erosion in the gas-solid flows, such as reducing particle loading from the flow, locating the position of major erosion zone and coating high-hardness alloy or applying protection lining in that particular area and optimizing the equipment design by keeping the particles velocity and the impinging angle away from the se-

rious erosion range. Fan et al. [4] introduced a finned tube erosion protection method. Song et al. [5] have demonstrated that adding ribs to the straight pipe would result in erosion reduce.

In the present paper, the method in which a number of ribs with certain cross section shape are fixed on the outer wall of the inside duct bend is deeply used to study the erosion between different geometry shape ribs. The difference of the rib geometry shape leads to the different degree of changing the pattern of gas-solid flow, altering the impact angle and reducing the impact velocity, which eventually results in the difference of the bend erosion. From the results, we can get the best choice of the ribs.

Numerical Method and Mathematic Models

The following assumptions are made first in the simulation:

- The gas-particle two-phase flow in the bend is in steady state, and fully developed.
- The gas phase is Newtonian, incompressible, and isothermal.
- The solid phase is composed of 20–80 μm spheres with a Gaussian size distribution.
- The gas-particle two-phase flow is considered as a dilute phase flow, and the inter-particle collisions are omitted. Particle rotation and electrostatic forces are negligible and be omitted also. However, particle-wall impacts are considered significantly.

Computational Domain and the Grid System

In this paper, three different shape ribs fixed on the duct bends are numerically simulated, respectively. The bare bend is marked with A, whereas the other three bends with ribs of the square cross section, the isosceles right-angled triangle cross section and the isosceles triangle cross section are marked with B, C, and D, respectively. Considering the relationship between the rib shape and the erosion, the present paper compares the erosion of ribs with different cross section under the prerequisite conditions that their dimensions and arrangement are equal. The initial gas-particle flow parameters for different cases are just the same as

Contributed by the Power Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the F&C Division August 12, 2001; final revision received February 8, 2004. Associate Editor: D. Y. S. Lou.

Table 1 The initial gas-particle flow parameters

Parameter	Value or Range
Particle arithmetic mean diameter $\bar{d}_p$ (μm)	50
Particle diameter range d_p (μm)	20–80
Gas volume flow Q_v (m^3/h)	30
Volume flow Q_s with standard state (m^3/h)	59.2769
Crack degree Ω	0.99673
Reynolds number Re_g	8.826×10^4
Cross section of square duct A_s (mm^2)	20×20
Fluid average velocity V_m (m/s)	41.2
Volume density of solids C_v	0.327%
Weight density of solids C_w	84.483%
Two-phase mixture density ρ_m (kg/m^3)	7.74
Bend curvature R/D	3

that of the experiment and numerical simulation (Yao et al. [6]), detailed listing in Table 1, and the detailed parameters for different ribs are listed in Table 2.

Figure 1(a) shows the computational domain and the rib arrangement of case C, which is similar to the other cases. Figure 1(b) shows the basic grid system in $y=0.01$ plane of case A. The $20 \times 20 \times 100$ grids are used for all cases. There are 20 intervals in x -direction, 20 in y -direction, and 100 in z -direction with 40 intervals in the straight section and 60 in the curved section.

Governing Equations of the Gas-Phase Flow

In large eddy simulation, we adopt the well-known subgrid scale model (Smagorinsky [7]) to simulate the gas flow field. The previous simulation (Yao et al. [6]) is based on the one-way coupling, i.e., ignoring the particles' counteractive forces and considering only the fluid affects the particle dispersion, which may be valid when the volume fraction of particles is under 10^{-6} . However, the local volume fraction of particles may reach 25 times of the mean volume fraction of the whole flow region (Squires et al. [8]). In these conditions, the particles can modify the flow field and this modification will affect the particle motion and the erosion in turn. So, the two-way coupling method is necessary for the accurate simulation. If a particle's momentum coupling effect can be approximated to a point force, the volume-averaged three-dimensional N-S equations for fluid can be expressed as

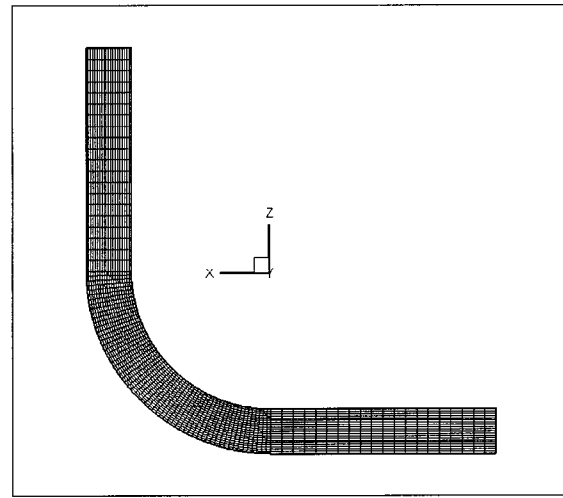
$$\text{Continuum equation: } \frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\text{momentum equations: } \frac{\partial \bar{u}_i}{\partial t} = - \frac{\partial}{\partial x_j} [\bar{u}_i \bar{u}_j] - \frac{\partial}{\partial x_i} \left[\frac{\bar{p}}{\rho} \right] + \frac{\partial}{\partial x_j} \left[(\nu + \nu_t) \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] + \bar{F}_{pi} \quad (2)$$

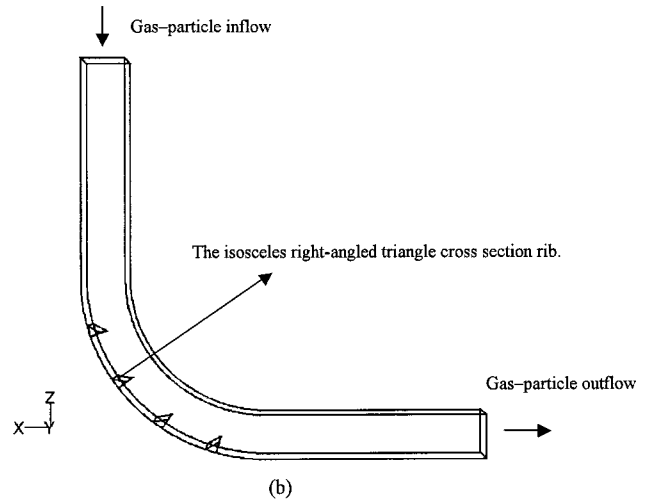
where $\nu_t = (C_s \Delta)^2 |\bar{S}|$, $|\bar{S}| = (2\bar{S}_{ij}\bar{S}_{ij})^{1/2}$, $\bar{S}_{ij} = 1/2(\partial \bar{u}_i / \partial x_j + \partial \bar{u}_j / \partial x_i)$, $\Delta = (\Delta x \cdot \Delta y \cdot \Delta z)^{1/3}$ and $C_s = 0.15$ is used in this work. $\bar{F}_{pi}$ is the interaction term between the gas and the particles, $\bar{F}_{pi} = \sum_j^N F_{ip} = \sum_j^N \rho_p' (v_i - \bar{u}_i) / \tau$, where ρ_p' is the local bulk density of a particle, N is the total number of particles in the control volume, v_i is the velocity of the j th particle, and u_i is the corresponding flow velocity at the position of that particle which can

Table 2 The parameters of different ribs

Ribs	Height (mm)	Width (mm)	Length (mm)	Rib Number
Case A	5	5	5	0
Case B	5	5	5	4
Case C	5	5	5	4
Case D	5	5	5	4



(a)



(b)

Fig. 1 (a) The computational domain and rib arrangement of case C. (b) The basic grid system in $y=0.01$ m plane of case A.

be obtained by interpolation. τ is defined in Sec. 2.3. Here, we use the summation convention with respect to repeated indices.

Simulation of the Particle Phase

Lagrangian Formulation for the Particles. The solid phase is treated by the Lagrangian approach and the particles are followed along their trajectories through the flow field.

The equations of particle motion in a three-dimensional Cartesian coordinate system are given below:

$$\frac{dx_p}{dt} = u_p \quad \frac{dy_p}{dt} = v_p \quad \frac{dz_p}{dt} = w_p \quad (3)$$

$$\frac{du_p}{dt} = \frac{1}{\tau} (u_g - u_p) - g \quad \frac{dv_p}{dt} = \frac{1}{\tau} (v_g - v_p) \quad \frac{dw_p}{dt} = \frac{1}{\tau} (w_g - w_p) \quad (4)$$

where x_p , y_p , and z_p are the location of a particle in the x , y , z -directions, respectively; u_g , v_g , w_g and u_p , v_p , w_p are the instantaneous velocities of the gas phase and the particle, respectively. $\tau = \rho_p d_p^2 / 18 \mu f$ is the particle relaxation time, with $f = C_D / (24 / Re)$. The drag coefficient, C_D , is defined by Sommerfeld and Zivkovic ([9]):

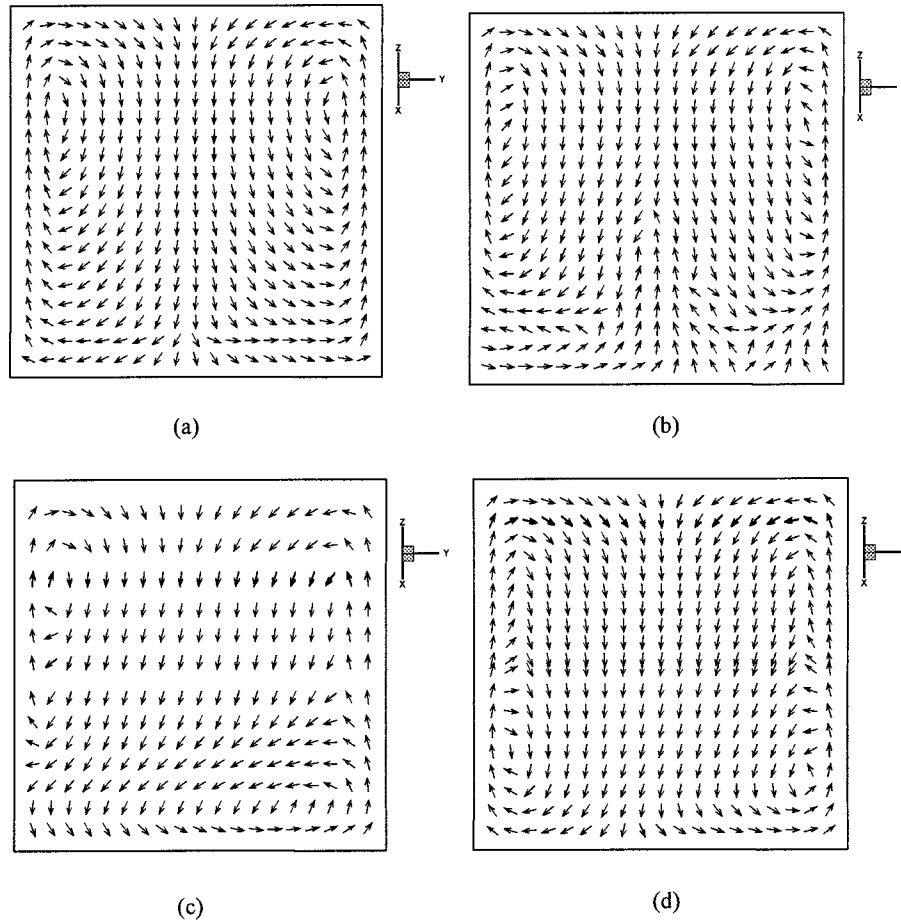


Fig. 2 The velocity vector of the gas flow at 45° cross section in cases A, B, C, and D. (a) Case A. (b) Case B. (c) Case C. (d) Case D.

$$C_D = \frac{24}{\text{Re}_p} f_p \quad \text{where} \quad \text{Re}_p = \frac{\rho d_p |v_g - v_p|}{\mu} \quad (5)$$

$$f_p = 1 + 0.15 \text{Re}_p^{0.687}, \quad \text{for } \text{Re}_p \leq 1000;$$

$$f_p = 0.0183 \text{Re}_p, \quad \text{for } \text{Re}_p > 1000.$$

A stochastic particle dispersion model, [10], is used to model the effect of turbulence on the particle motion in the pipe turbulent flows. The particle is assumed to interact with an eddy over a certain period of time, which is the smaller one of the eddy lifetime and the transition time. Equations (3) and (4) are solved by a fourth-order Runge-Kutta scheme with adaptive step length.

Model of Particle Impact and Rebound. Grant and Tabakoff [11] studied the particle rebound from a wall experimentally, and found that the change in particle momentum due to impact is mainly a function of the particle impact velocity v and its incidence angle β . The following empirical correlations are used for the impact restitution ratio:

$$\frac{v_{n2}}{v_{n1}} = 1.0 - 0.4159\beta_1 - 0.4994\beta_1^2 + 0.292\beta_1^3 \quad (6)$$

$$\frac{v_{t2}}{v_{t1}} = 1.0 - 2.12\beta_1 + 3.0775\beta_1^2 - 1.1\beta_1^3 \quad (7)$$

in which v_n and v_t represent the particle velocity impact components normal and tangential to the wall, respectively. Subscripts 1 and 2 refer to the conditions before and after impact, respectively. β_1 is the angle between the incident velocity and the tangent to the surface.

Calculation of the Wall Mass Erosion. Experimental results reported by Tabakoff et al. [12] indicate that the erosion of a target material depends on the particle impact velocity and the impingement angle. Measurements were made of the erosion rate for coal ash particles impacting on a steel surface. An empirical correlation is established for the erosion mass parameter, E , which is defined as the ratio of the mass loss of the target to the mass of the impinging particles,

$$E = K_1 \left\{ 1 + C_K \left[K_2 \sin \left(\frac{90}{\beta_0} \beta_1 \right) \right] \right\}^2 W_1^2 \cos^2(1 - R_1^2) + K_3 (W_1 \sin \beta_1)^4 \quad (8)$$

where W_1 and β_1 are the impact velocity and impingement angle, respectively. $R_1 = 1 - 0.0016W_1 \sin \beta_1$, $\beta_0 = 20$ deg is the angle of maximum erosion. While $\beta_1 \leq 3\beta_0$, $C_K = 1$; while $\beta_1 > 3\beta_0$, $C_K = 0$. $K_1 = 1.505101 \times 10^{-6}$, $K_2 = 0.296007$, $K_3 = 5.0 \times 10^{-12}$.

In order to describe the efficiency of protecting erosion for different kind of ribs, two important parameters are defined, the erosion rate and the relative erosion rate. The erosion rate is defined as $E = \Delta M_s / M_a$, i.e., the ratio of the eroded mass of wall material ΔM_s to the mass of impacting abrasive particles M_a . E is a function of particle impacting velocity and impacting angle.

The relative erosion rate, E/E_0 , is defined as the ratio of the erosion rate E of a ribbed bend to that of the bare bend E_0 . This parameter can show the effectiveness of different ribs to reduce erosion.

Besides, in the present paper, the erosion rate refers to the sum of the rib erosion rate and the wall erosion rates. E_r refers to the

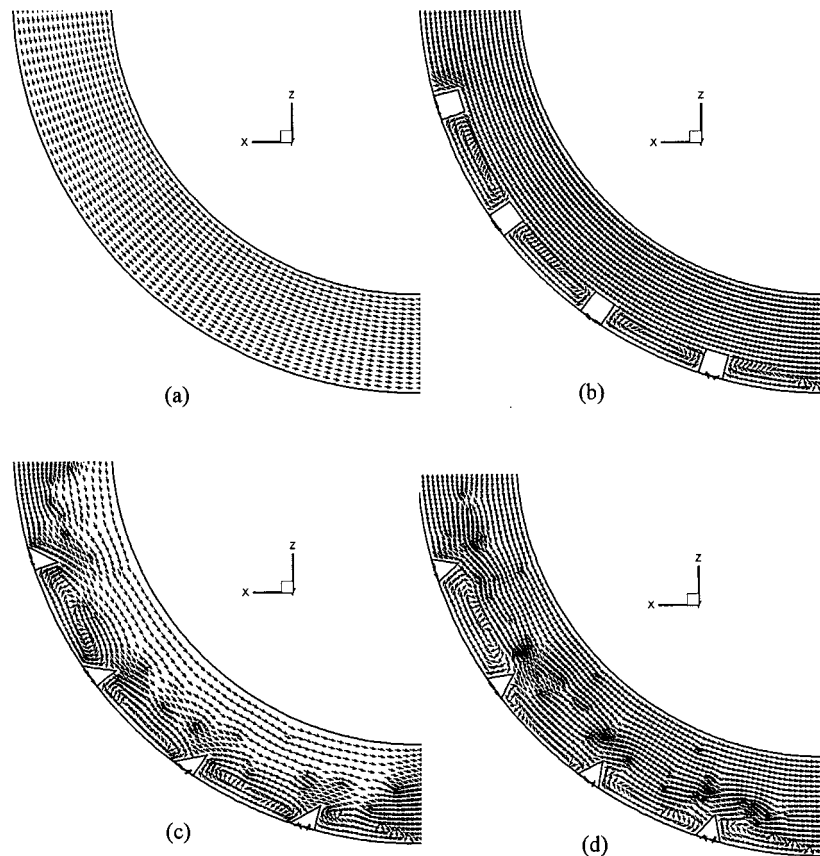


Fig. 3 The velocity vector of flow field at the bend part of $y=0.01$ m plane for in cases A, B, C, and D. (a) Case A. (b) Case B. (c) Case C. (d) Case D.

erosion rate of ribs, E_{wb} refers to the erosion rate of the bend walls, E_{ws} refers to the erosion rate of the sidewalls, and E equals to the sum of E_r , E_{wb} , and E_{ws} .

Results and Discussion

Gas-Phase Flow Field. Figure 2 illustrates the velocity vectors of gas flow at 45° cross section in the four cases. One can see that the secondary flows at the cross sections are clearly depicted. In cases A, B, and D, the distributions of the velocity vectors are almost symmetric along $y=0.01$ plane, but in case C, the velocity vector is not well regulated because of the greater disturbance caused by the right-angled triangle ribs. In common, the secondary flows in the four cases are all mainly located in the edge of the sidewalls.

Figure 3 shows the velocity vector of the flow field at the bend part of $y=0.01$ plane for four cases. It is obvious that there exists a recirculating area behind every rib. The recirculating area just likes a rolling wheel, which plays a very important role in protecting erosion from the particle impact. In particular, the recirculating areas behind the isosceles right-angled triangle ribs affect the flow field greater, which can protect the bend wall from erosion more efficiently.

Particle Trajectories. Figure 4 depicts all particle trajectories in the bend parts of the four bends. In case A, particles not only slide and roll along the walls but also impact on the outer bend wall with higher speed due to their inertial and centrifugal forces (Fig. 4(a)). Smaller particles may not impact on the wall throughout the bend, but big particles collide with the wall more than once. They may first impact on the outer bend soon after entering the bend, then rebound from the wall with sufficient speed and moves into the main flow. After gaining enough momentum there,

they undergo the second impact on the outer bend near the end of the bend. In addition, there are a few particles impact on the two sidewalls, but the impact is lighter.

The trajectories of particles in the ribbed bends are far more complicated than those in the bare bend (Figs. 4(b), (c), (d)). Some particles impact on the first rib by the primary or the secondary impacts, then rebound into the bulk flow with considerable velocity. While the other part particles may collide with the second rib directly or indirectly and then rebound. After colliding with the first rib or the second rib, some particles will get across the third rib and impact on the wall or the fourth rib directly, then enter the developed airflow and flow away quickly. But the other particles will impact on the third and latter ribs sequentially or rebound between the ribs and the wall. The particles with the higher rebounding speed may impact on the bend wall and rebound, and hit the back of the previous rib again. The particles with the lower remnant speed will probably jump upon the rib surface again and again with decaying speed, and eventually slide down to the bend wall.

Most particles in the ribbed bends experience more than once collision with the metal surface. So their trajectories in the bends are likely to be a series of rib-wall-rib impacts before they finally jump out of the ribbed area. Moreover, because the ribs changed the particle incidence angle and impact velocity, erosion caused by rib-wall-rib impacts is not significant.

The typical single particle trajectory for each case is shown in Fig. 5, in which the particle trajectories are more clearly demonstrated.

Comparing case C with B and D, an important character of particle trajectories in case C can be found. In case C, most particles impact on the isosceles right-angled triangle ribs rather than the walls and can be taken out of the bend by gas easily. So the

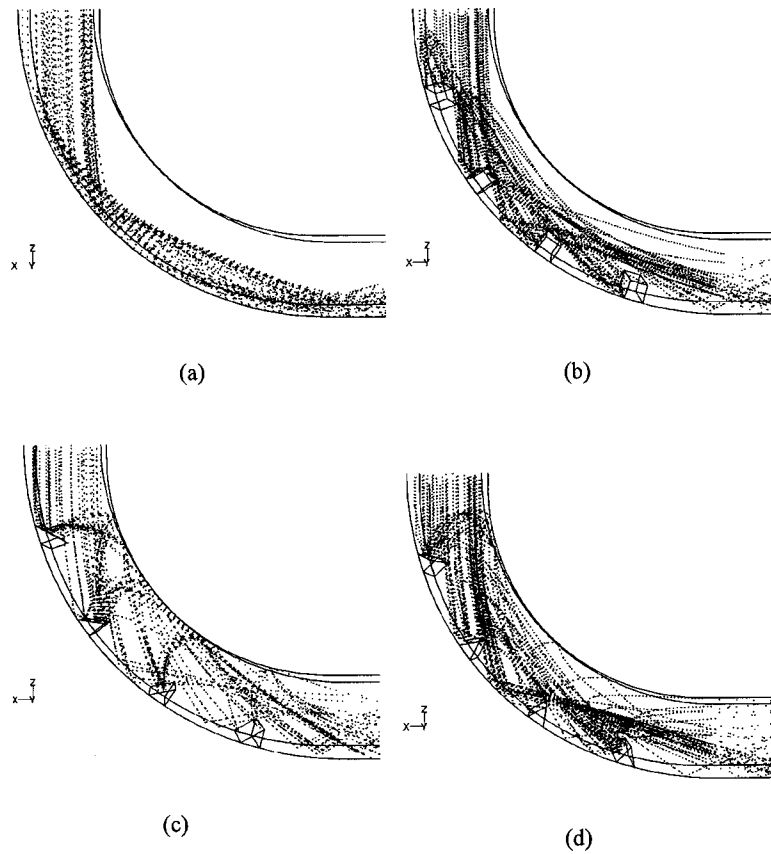


Fig. 4 All the particle trajectories in the bend parts of the four bends, respectively. (a) Case A. (b) Case B. (c) Case C. (d) Case D.

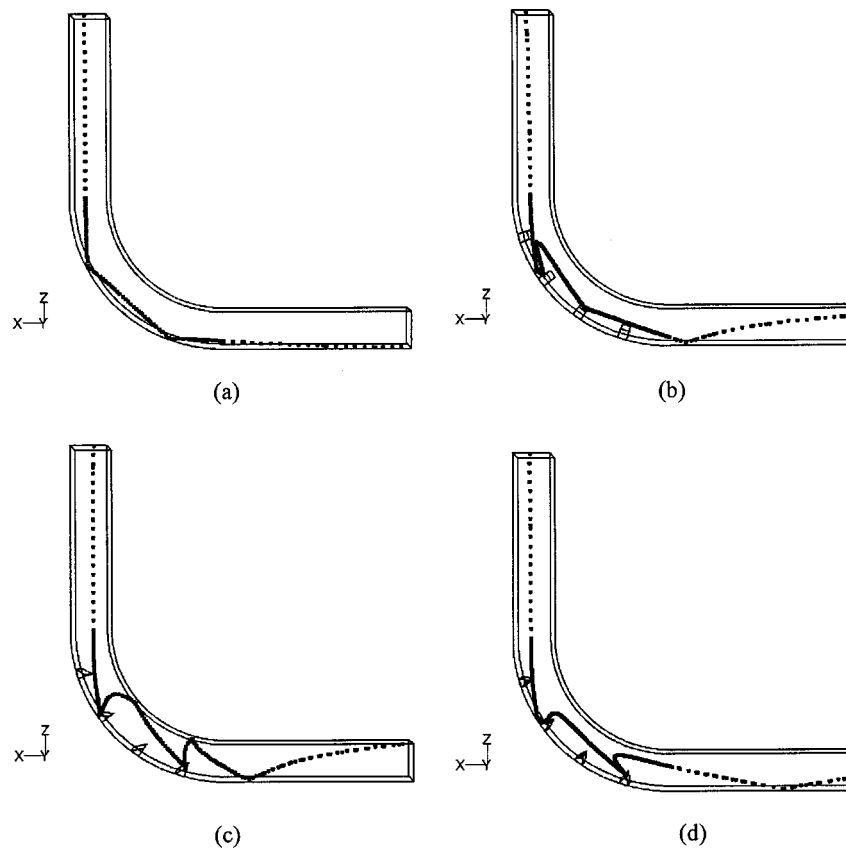


Fig. 5 The typical single particle trajectory in the different cases. (a) Case A. (b) Case B. (c) Case C. (d) Case D.

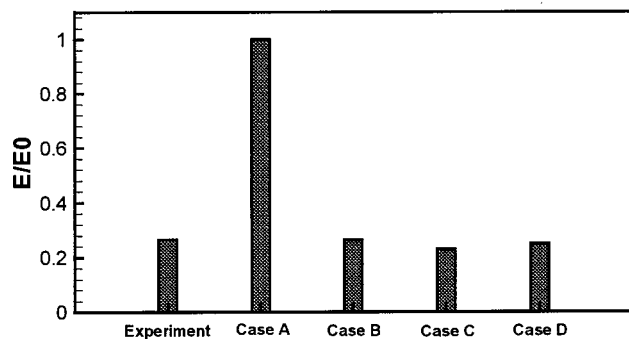


Fig. 6 Comparisons of the experimental erosion rate (Yao et al. [6]) with that of the cases of A, B, C, and D at the same conditions

particle trajectories are relatively simple and the erosion by particle impact may be relatively low. This is very helpful for us to choose the most efficient ribs.

Comparisons of Erosion Rates. Figure 6 compares the experimental erosion rate (Yao et al. [6]) with that of the four cases. One can see that the relative erosion rate of case B is in good agreement with the experimental relative erosion rate, because case B and experimental A3 have the same conditions. While the relative erosion rate of case C is lower than that of case B and D. This indicates that the isosceles right-angled triangle ribs have the largest anti-erosion effect among the three shaped ribs. As a whole, the erosion rate for every ribbed bend is only about 25% of that for the bare bend, indicating a 75% reduction.

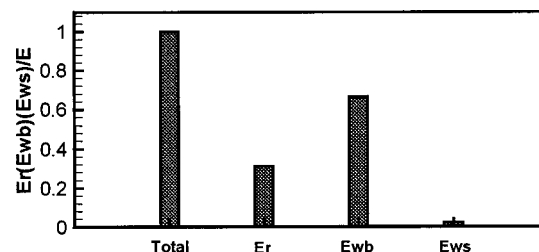
The erosion rates of the ribs, the bend walls and the sidewalls are counted separately in this paper so that we can see the different degree erosion by particle impact between the ribs, the bend walls, and the sidewalls.

Comparisons of the erosion rates of the ribs, the bend walls and the sidewalls for cases B, C, and D are shown in Fig. 7. The erosion of the sidewalls (E_{ws}) for each case is about 2.8% of the total erosion (E), extremely smaller than that of the bend walls (E_{wb}) and the ribs (E_r), but it is bigger than that of the bare bend, 2.2%. This point shows that the ribs change the particle trajectories and make the particles collide with the sidewalls more often. As a whole, to examine the erosion of the ribs (E_r) and the bends (E_{wb}) for the three cases, we find that the rib erosion rate (E_r) contributes only about 34.8% of the total rate, but the bend wall erosion rate (E_{wb}) accounts for the rest 62.4%, almost twice of the rib erosion rate.

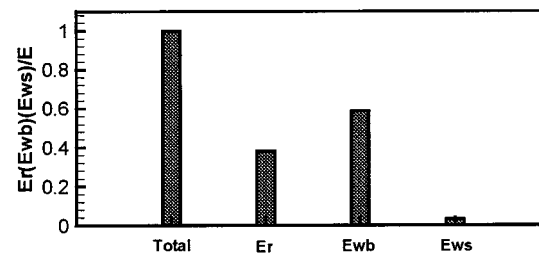
From the comparisons, we can conclude that the impact and the erosion by the particles mainly occur on the bend walls. If we choose the apt ribs, which reduce the bend wall erosion largely, then we can obtain the better anti-erosion effect. Among cases B, C, and D, the erosion rate of the bend wall (E_{wb}) in case C is the lowest, suggesting again that the isosceles right-angled triangle ribs have the best contribution to the erosion reduce. It can also be pointed out that the rib erosion rate and the sidewall erosion rate are the largest in case C.

Comparisons of Particle Impact Velocity. It has been found that the particle impact velocity is a key parameter to the material erosion. From the definition of the erosion rate above, we can see that the effect of particle impact velocity on erosion rate is described by a power-law correlation, (particle velocity)ⁿ, in which the power exponent is usually in the range $2 < n < 2.5$ for ductile materials. While for brittle materials it can be much larger, [13].

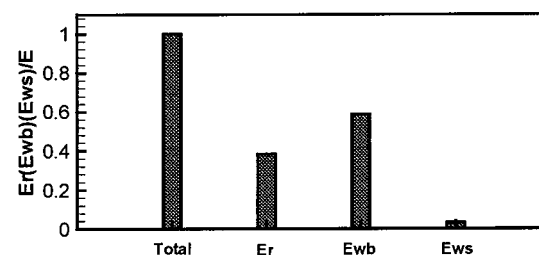
For the convenience of comparison, all impact velocities are divided into four different ranges: 0–20 m/s, 20–30 m/s, 30–40 m/s, and 40 m/s or higher. Figure 8 compares the normalized impact velocity distribution of particle-wall impacts (computa-



(a)



(b)



(c)

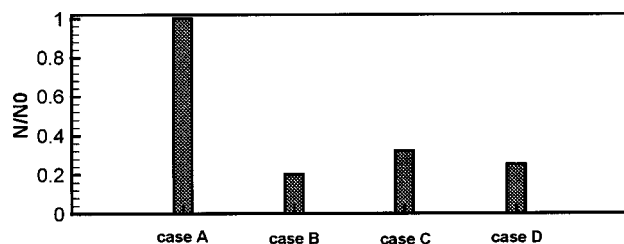
Fig. 7 Comparisons of the erosion rates of the ribs, the bend walls and the sidewalls for the cases of B, C, and D. (a) Case B. (b) Case C. (c) Case D.

tional unit time) between four cases, where N is the occurrence (computational unit time) of particle-wall impacts with an impact velocity that falls into a certain range, while N_0 refers to that of the bare bend. The ratio N/N_0 can reflect the difference of impact velocity distribution between different cases. Because no collision happens with a velocity lower than 20 m/s in case A, the comparison of velocity trivial 0–20 m/s for all cases is not plotted.

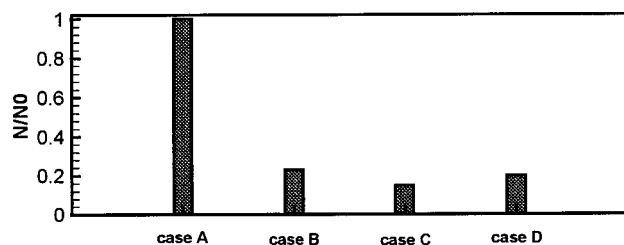
Figure 8 shows that the ratio N/N_0 for the bare bend is far higher than that of the ribbed bends. It exhibits again that the particle-wall impacts in the bare bend are mainly occurred with a higher impact velocity (>20 m/s) than that in the ribbed bends. Among cases B, C, and D, in the velocity trivial 20–30 m/s, case C has the highest frequency of impact; But in the higher velocity 30–40 m/s and 40 m/s or higher, case C has the fewest impact occurrence, indicating that the impact velocity of case C is smaller than that of cases B and D. As a result, case C may possess the lowest erosion rate.

As a whole, the impact velocity between particles and walls tends to be pretty low in cases B, C, and D due to the action of the ribs. It is a significant factor to decrease the erosion of the ribbed bends.

Comparisons of Particle Impact Angle. As above mentioned, the isosceles right-angled triangle ribs have the best anti-erosion



(a)



(b)



(c)

Fig. 8 Comparisons of the normalized impact velocity distribution of particle-wall impact (computational unit time) between four cases. (a) 20–30 m/s; (b) 30–40 m/s; (c) >40 m/s.

efficiency among cases B, C, and D. This can also be found from the comparisons of the particle impact angle between the four cases. It has been found that the impact angle also strongly affects the erosion rates. Studies show that for ductile materials, the maximum erosion occurs at a relatively low angle of incidence. In the present paper, the material involved is medium carbon steel that can be considered as ductile material. Previous investigations, [14,15], have shown the maximum erosion occurs at an incidence angle of 20 deg–30 deg for such materials.

So, it is useful to examine the difference in the distribution of impact angle between all cases, especially in the range of 20 deg–30 deg. Figure 9 shows that for case A, more than 80% of the impact angle falls into the range of 20 deg–30 deg and collisions almost never occur at range of 40 deg–90 deg. All the ribbed bends have a higher probability of impact angle occurring in two intervals of incidence angle 0 deg–20 deg and 50 deg–60 deg, but the probability for each case is different. For case C, the probability of impact angle in the range of 0 deg–20 deg is the smallest compared with that of cases B and D, while in the range of 50 deg–60 deg it becomes the largest, about 70%. These differences are associated with the different shapes of the ribs. In the key

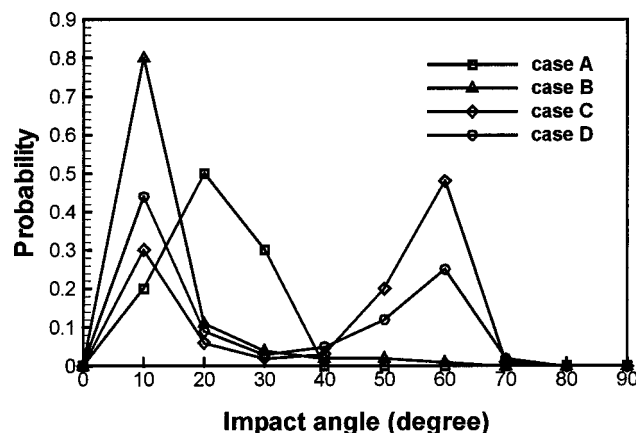


Fig. 9 Comparison of particle-wall impact angle distribution between different cases

range of 20 deg–30 deg, it is obvious that the probability of case A is far higher than that of cases B, C, and D, and the probability of case C is the lowest, which helps to understand the best anti-erosion efficiency of isosceles right-angled triangle ribs.

Conclusions

Large eddy simulation method is used to study the erosion by particle impact in a bare duct bend and three ribbed bends with different geometry shape at the same dimensions and arrangement. The gas-solid flow is dilute and the interaction between gas and particles is taken into account. The results show that the erosion of the bends by particle impact can be greatly reduced by adding ribs to the outer wall, and the isosceles right-angled triangle ribs have the best anti-erosion efficiency among three different shape ribs. Numerical predictions of gas flow and particle trajectories help to understand the underlying mechanism of erosion reduction by ribs. The gas flow between the ribs forms a recirculation area, which looks just like a rolling wheel and plays a very important role in the erosion protection from particle impact. A series of rib-wall-rib impacts constitute the particle trajectories. The changes of the gas flow manners near the outer wall and the particle impact manners in the ribbed bends decrease the particle impact velocity and alter the particle incidence angle, which reduce the erosion of the bends primarily.

For the isosceles right-angled triangle ribs, the impact velocity is reduced more greatly and the impact angle is changed more properly due to its special geometry structure. So the erosion of the bend with the isosceles right-angled triangle ribs is the lowest among the three different kinds of ribs. In addition, the volume of this kind of ribs is only half of the volume of the square shape ribs with the same dimensions, meaning that half of the rib material is saved. Obviously, the isosceles right-angled triangle shape ribs are the best choice for anti-erosion in the gas-solid bend flows among three different shape ribs at the same dimensions and arrangement. This conclusion can give direction to anti-erosion technique in the engineering application systems.

Acknowledgment

This paper is supported by the National Natural Science Foundations (No. 50236030 and 50076038).

Nomenclature

- C_D = drag coefficient
- d_p = particle diameter
- $\dot{E}$ = erosion rate
- E_0 = erosion rate of bare bend
- E/E_0 = relative erosion rate

E_r = erosion rate of ribs
 E_{wb} = erosion rate of the bend walls
 E_{ws} = erosion rate of the sidewalls
 F_{pi} = interaction term between the gas and the particles in control volume
 f = modification factor for the Stokes drag coefficient
 M_a = mass of impacting abrasive particles
 p = gas pressure
 Re_p = relative Reynolds number of particle
 u_g, v_g, w_g = instantaneous velocity of the gas in the x, y, z -direction, respectively
 u_p, v_p, w_p = instantaneous velocity of a particle in the x, y, z -direction, respectively
 $\bar{u}_i$ = gas average velocity
 v_n, v_t = particle velocity impact component normal and tangential to the wall, respectively.
 w_1 = impact velocity
 x_p, y_p, z_p = location of a particle in the x, y, z -direction, respectively
 $\Delta x, \Delta y, \Delta z$ = grid space in the x, y, z -direction, respectively
 β_1 = angle between the incident velocity and the tangent to the surface
 ρ = gas density
 ρ_p = particle density
 ν = kinematic viscosity of the gas
 ν_t = vortex viscosity coefficient of the gas
 τ = particle relaxation time

References

- [1] Tilly, G. P., 1979, "Erosion Caused by Impact of Solid Particles," *Treat. Mat. Sci. Tech.*, **13**, p. 287.
- [2] Finnie, I., 1972, "Some Observations on the Erosion of Ductile Metals," *Wear*, **19**, pp. 81–90.
- [3] Humphrey, J. A. C., 1990, "Fundamentals of Fluid Motion in Erosion by Solid Particle Impact," *Int. J. Heat Fluid Flow*, **11**, pp. 170–195.
- [4] Fan, J. R., Zhou, D. D., Zeng, K. L., and Cen, K. F., 1992, "Numerical and Experimental Study of Finned Tube Erosion Protection Methods," *Wear*, **152**, pp. 1–19.
- [5] Song, X. Q., Lin, J. Z., Zhao, J. F., and Shen, T. Y., 1996, "Research on Reducing Erosion by Adding Ribs on the Wall in Particulate Two-Phase Flows," *Wear*, **193**, pp. 1–7.
- [6] Yao, J., Zhang, B. Z., and Fan, J. R., 2000, "An Experimental Investigation of a New Method for Protecting Bends From Erosion in Gas-Solid Flows," *Wear*, **240**, pp. 215–222.
- [7] Smagorinsky, J., 1963, "General Circulation Experiments With the Primitive Equations. I. The Basic Experiment," *Mon. Weather Rev.*, **91**(3), pp. 91–164.
- [8] Squires, K. D., and Eaton, J. K., 1991, "Preferential Concentration of Particles by Turbulence," *Phys. Fluids A*, **3**, pp. 169–1178.
- [9] Sommerfeld, M., and Zivkovic, G., 1992, "Recent Advances in the Numerical Simulation of Pneumatic Conveying Through Tube Systems," *Comput. Meth. Appl. Sci.*, **21**, pp. 201–212.
- [10] Fan, J. R., Zhang, X. Y., Chen, L. H., and Cen, K. F., 1997, "New Stochastic Particle Dispersion Modeling of a Turbulent Particle-Laden Round Jet," *Chem. Eng. J.*, **66**, pp. 207–215.
- [11] Grant, G., and Tabakoff, W., 1975, "Erosion Prediction in Turbomachinery Resulting From Environmental Solid Particles," *J. Aircr.*, **12**, pp. 471–487.
- [12] Tabakoff, W., Kotwal, R., and Hamed, A., 1979, "Erosion Study of Different Materials Affected by Coal Ash Particles," *Wear*, **52**, pp. 161–173.
- [13] Mason, J. S., and Smith, B. V., 1972, "The Erosion of Bends by Pneumatically Conveyed Suspension of Abrasive Particles," *Powder Technol.*, **6**, pp. 323–335.
- [14] Finnie, I., 1960, "Erosion of Surfaces by Solid Particles," *Wear*, **3**, p. 87.
- [15] Cousen, A. K., and Hutchings, I. M., 1983, *Proc. 6th Int. Conf. on Erosion by Liquid and Solid Impact*, No. 41.

Analytical Investigations of Rotating Disks With and Without Incorporating Rotating Heat Pipes

J. Ling and Y. Cao

Department of Mechanical Engineering, Florida
International University, Miami, FL 33199

R. Rivir and C. MacArthur

Propulsion Directorate, Air Force Research Laboratory,
Wright Patterson AFB, OH 45433

[DOI: 10.1115/1.1760528]

Introduction

The temperature limitation is one of the most crucial limiting factors to the efficiency of military gas turbine engines. An increased turbine inlet gas temperature decreases both specific fuel and air consumption while increasing efficiency. This desire for a high turbine inlet gas temperature, however, is often in conflict with materials available to withstand this high temperature. For example, the maximum disk rim temperature of some advanced development engines would exceed 1000°C, which is approaching the creep limitation of the commonly used disk materials, such as an Inconel. A literature survey regarding the turbine disk cooling reveals that although the average air-cooling heat transfer coefficient is generally high, the local heat transfer coefficient at the disk rim is relatively low. Based on the understanding that one of the major causes of the high disk rim temperature is the low thermal conductivity associated with the turbine disk material (on the order of 20 W/m-K), the concept of cooling the disk by employing the heat pipe cooling technique was proposed (Cao et al. [1]). With the incorporation of the heat pipe, the effective heat dissipation area of the disk is substantially increased and the problem associated with the accumulation of the excessive heat at the disk rim is alleviated. The proposed disk with the heat pipe was numerically analyzed with excellent results (Cao et al. [2]). In this paper, closed-form analytical solutions are sought for turbine disks cooled through both the conventional air-cooling and heat pipe cooling techniques. The closed-form solution has the advantages of simplicity and convenience for practical design of a turbine disk employing a specific cooling technique. The solutions could also be useful for cooling the rearmost stages of an ad-

vanced turbine engine compressor where the temperature could be substantially high due to a very high-pressure ratio across the compressor.

Analytical Solution of a Rotating Disk Having Homogenous Thermal Properties

A rotating disk subject to a heat input at the disk rim and cooled at the disk lateral surfaces through convection is schematically shown in Fig. 1. With reference to a two-dimensional heat conduction model and the operation of a gas turbine, intense heat from high-temperature gas and turbine blades is transferred to the disk rim surface at $r=r_2$. The two lateral surfaces of the disk are subject to strong thermal convection due to the high rotating speed of the disk, and serve as heat dissipating surfaces. It should be noted that it is assumed that the two lateral surfaces of the disk would have the same cooling condition, so that only one half of the disk is studied due to the symmetry. As a result, the dimension L in Fig. 1 represents one-half thickness of the disk, and an adiabatic boundary condition is specified at the bottom surface of the disk model. Additionally, the disk is assumed to have homogenous thermal properties throughout the one-half disk thickness of L . The inner surface of the disk at $r=r_1$ is approximately treated as an adiabatic surface because of a small temperature gradient there. If an excess temperature, $\theta=T-T_c$, is used, where T_c is the cooling air (surrounding) temperature, the heat conduction equation and boundary conditions for the disk are as follows:

$$\frac{\partial^2 \theta}{\partial r^2} + \frac{1}{r} \frac{\partial \theta}{\partial r} + \frac{\partial^2 \theta}{\partial z^2} = 0 \quad r_1 < r < r_2 \quad \text{and} \quad 0 < z < L \quad (1.1)$$

$$\frac{\partial \theta}{\partial r} = 0 \quad \text{at} \quad r = r_1 \quad (1.2)$$

$$\frac{\partial \theta}{\partial r} = \frac{q''}{k} \quad \text{at} \quad r = r_2 \quad (1.3)$$

$$\frac{\partial \theta}{\partial z} = 0 \quad \text{at} \quad z = 0 \quad (1.4)$$

$$\frac{\partial \theta}{\partial z} + H\theta = 0 \quad \text{at} \quad z = L \quad (1.5)$$

where $H=h/k$. By employing the method of the separation of variables (Arpaci [3] and Ozisik [4]), the temperature distribution in the disk is obtained:

$$\theta(r, z) = \frac{q''}{k} \sum_{m=1}^{\infty} \frac{\sin(\eta_m L)}{N(\eta_m) \eta_m^2} \cdot \frac{K_1(\eta_m r_1) I_0(\eta_m r) + I_1(\eta_m r_1) K_0(\eta_m r)}{K_1(\eta_m r_1) I_1(\eta_m r_2) - I_1(\eta_m r_1) K_1(\eta_m r_2)} \cos(\eta_m z). \quad (2)$$

Contributed by the Heat Transfer Committee of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for publication in the ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received by the HT Committee October 2002; final revision received September 2003. IGTI Committee Chair: Ron Bunker.

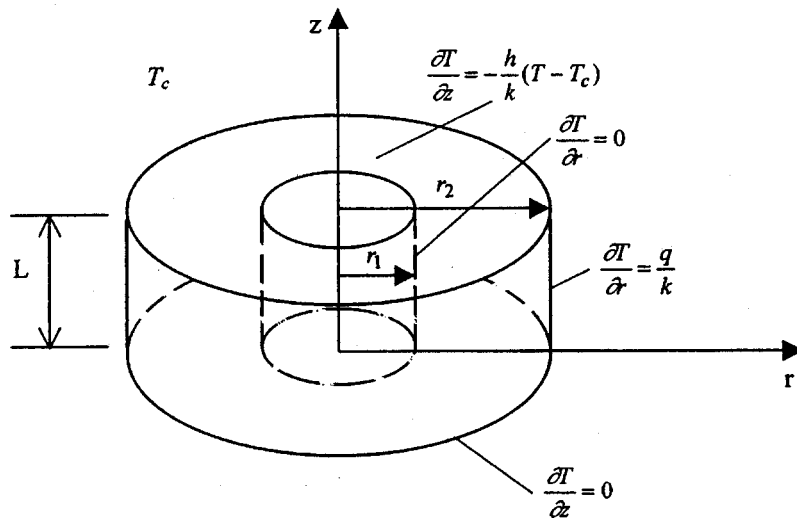


Fig. 1 Schematic of a rotating disk under a cylindrical coordinate system

It is clear from Eq. (2) that the temperature distribution in the disk is a function of heat flux, thermal conductivity, heat transfer coefficient, and geometrical dimensions of the disk. The same problem is also solved through a three-dimensional numerical calculation employing the commercial software FLUENT/GAMBIT. A comparison between the analytical and numerical results under typical heating condition at the disk rim and cooling condition at the disk lateral surfaces indicated that the analytical solution is in good agreement with the numerical calculation. After the comparison is made, the closed-form solution is employed to illustrate the effects of various parameters on the temperature distribution in the disk. Figure 2 shows the temperature distributions with different heat transfer coefficients associated with the lateral surfaces of the disk. It can be seen from Fig. 2 that as the heat transfer coefficient increases from 150 to 350 W/m²°C, the temperature level in the disk would decrease gradually. It should be noted that the increase of the heat transfer coefficient would require a substantial increase in the cooling-air flow rate, which would be limited by the capacity of the compressor and fuel consumption of the gas turbine engine. Figure 3 shows temperature distributions with different values of the thermal conductivity under the same geometry and boundary conditions.

It is evident from Fig. 3 that as the thermal conductivity is increased from 24 to 105 W/m °C, the temperature at the disk rim, $r=r_2$, is reduced by about 250°C and that at the inner surface of the disk, $r=r_1$, is increased by only about 50°C. Figure 4 shows the comparison of temperature distributions with different disk sizes. It can be seen from Fig. 4 that the temperature level in the entire disk could remain high if the size of the disk is small. Finally, it is evident from Eq. (2) that the temperature distribution in the disk is significantly influenced by the disk thickness and is directly proportional to the heat flux on the disk rim. With an increase in disk thickness or the heat flux, the temperature in the disk will be increased accordingly.

Modified Closed-Form Solution for a Turbine Disk Incorporating Heat Pipes

Figure 5(a) illustrates schematically a proposed turbine disk incorporating a number of radially rotating heat pipes (Cao et al. [1,2]). As discussed earlier, the disk rim surface is subject to the worst heating condition, which would result in a maximum temperature there. The prediction of the maximum temperature on the

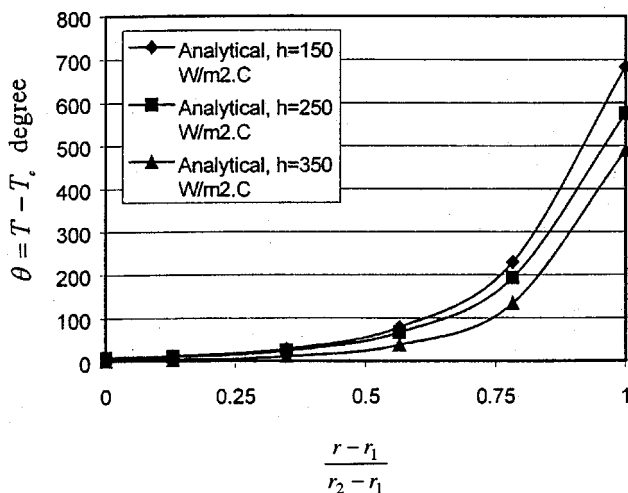


Fig. 2 Comparison of the temperature distributions with different heat transfer coefficients under the same geometry and boundary conditions ($q''=3 \times 10^5$ W/m², $z=0$, $k=24$ W/m °C, and $T_c=525^\circ\text{C}$)

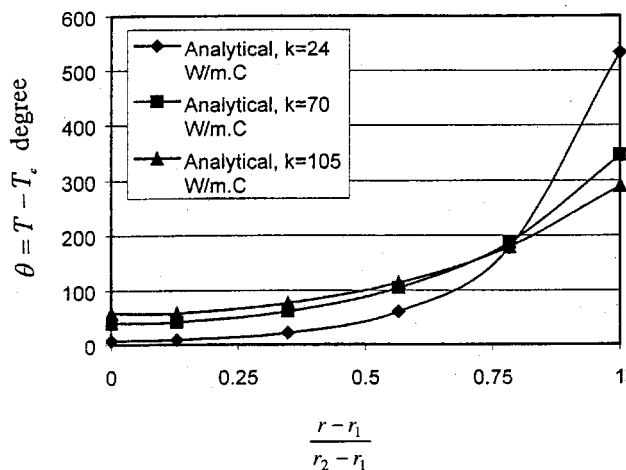


Fig. 3 Comparison of the temperature distributions with different thermal conductivity under the same geometry and boundary conditions ($h=250$ W/m² °C, $q''=3 \times 10^5$ W/m², $z=0$, and $T_c=525^\circ\text{C}$)

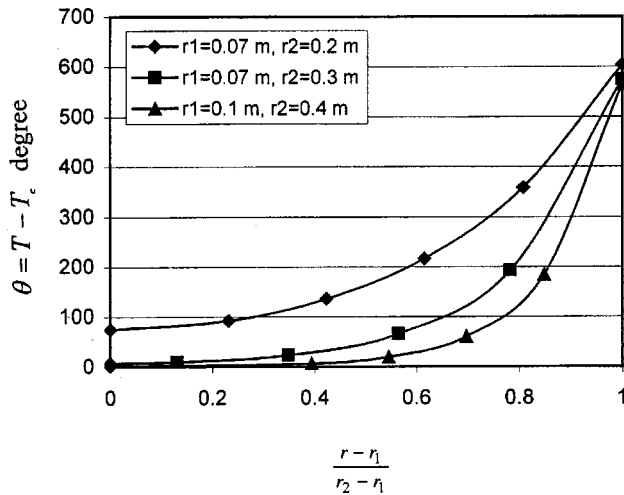


Fig. 4 Comparison of the temperature distributions with different inner and outer radii ($q''=3 \times 10^5 \text{ W/m}^2$, $k=24 \text{ W/m}^\circ\text{C}$, $h=250 \text{ W/m}^2^\circ\text{C}$, $z=0$, $T_c=525^\circ\text{C}$)

disk rim is the major task of this section. Since Eq. (2) can be applied only to a disk having homogenous thermal properties, such as that shown in Fig. 1, it must be modified before it can be used to predict the maximum temperature of a turbine disk incorporating heat pipes that is shown in Fig. 5. In the remainder of this paper, a modification correlation, which takes into account the influence of additional parameters, is established with the assistance of numerical calculations. These parameters would include the distance between the heat pipe top and the disk rim surface, s , the heat pipe length, L_{hp} , and the apparent thermal conductivity, k_a , of the heat pipe cooled disk, which includes the influence of both the disk material thermal conductivity and the heat pipe effective thermal conductance. The apparent thermal conductivity should be used to replace the material thermal conductivity k in Eq. (2) and is defined by the following relation:

$$k_a = k \left(1 - \frac{A_{hp}}{A_a} \right) + k_{eff} \frac{A_{hp}}{A_a} \quad (3)$$

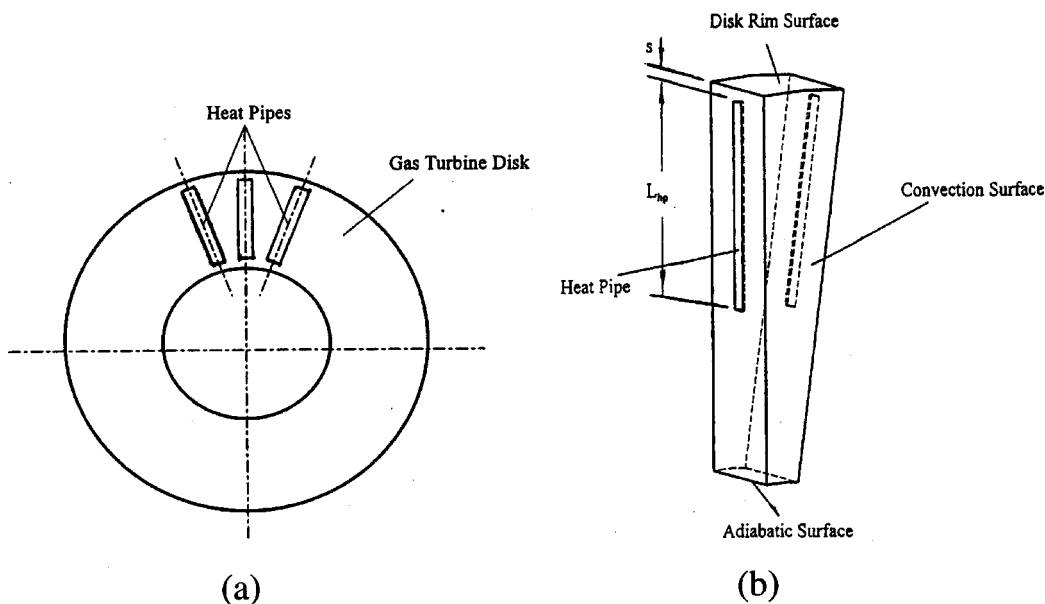


Fig. 5 Schematics of a turbine disk (a), and a simulated turbine disk sector (b), incorporating radially rotating heat pipes

where $A_{hp} = n\pi r_{hp}^2$ and $A_a = 2\pi r_a L$. A_{hp} is the total cross-sectional area of the heat pipes incorporated, n is the number of heat pipes incorporated in the disk, r_a is the average radius of the disk, $r_a = (r_1 + r_2)/2$, and A_a is the sectional area of the disk at the average radius, r_a . Having introduced the foregoing influential parameters, the ratio of the maximum temperature at the disk rim surface, which usually occurs at $z=0$, to that from Eq. (2) can be expressed by the following equation:

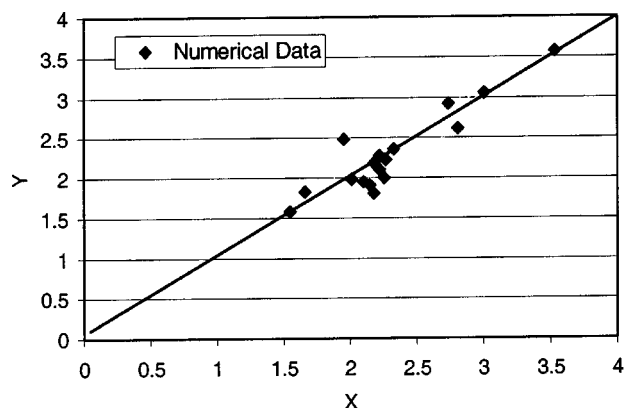
$$\frac{\theta_{\max}(r_2, 0)}{\theta(r_2, 0)} = c_1 \left(\frac{s}{L_{hp}} \right)^{c_2} \left(\frac{k_a}{hr_2} \right)^{c_3} \quad (4)$$

where

$$\theta(r_2, 0) = \frac{q''}{k_a} \sum_{m=1}^{\infty} \frac{\sin(\eta_m L)}{N(\eta_m) \eta_m^2} \cdot \frac{K_1(\eta_m r_1) I_0(\eta_m r_2) + I_1(\eta_m r_1) K_0(\eta_m r_2)}{K_1(\eta_m r_1) I_1(\eta_m r_2) - I_1(\eta_m r_1) K_1(\eta_m r_2)}$$

c_1 , c_2 , and c_3 are constants to be determined.

To determine the three constants in Eq. (4), extensive three-dimensional numerical calculations were undertaken. Because of the structural symmetry, only a disk sector containing two halves of the heat pipes was numerically modeled (Fig. 5(b)). To simplify the analysis, the heat pipes installed in the disk are treated as solid bars having a high effective thermal conductance, k_{eff} . This simplification is the same as that adopted by Cao et al. [2] and could avoid solving the complicated two-phase flow in the heat pipe. Numerical simulations were made by employing the commercially available computer software FLUENT. The disk and heat pipes were meshed into hexahedral cells. The total number of cells is about 26,000, and the convergent criterion is 10^{-8} . During the simulations, parameters in connection with the heat input, air cooling condition, material properties, disk geometry, and heat pipe dimensions were varied within the following ranges: $10^5 \text{ W/m}^2 \leq q'' \leq 5 \times 10^5 \text{ W/m}^2$; $k_{eff} = 10,000 \text{ W/m}^2^\circ\text{C}$; $k = 24 \text{ W/m}^\circ\text{C}$; $3 \text{ mm} \leq s \leq 20 \text{ mm}$; $T_c = 525^\circ\text{C} = 798 \text{ K}$; $0.04 \text{ m} \leq L_{hp} \leq 0.2 \text{ m}$; $1.5 \text{ mm} \leq r_{hp} \leq 3.5 \text{ mm}$; $100 \text{ W/m}^2^\circ\text{C} \leq h \leq 500 \text{ W/m}^2^\circ\text{C}$; $0.07 \text{ m} \leq r_1 \leq 0.1 \text{ m}$; $0.25 \text{ m} \leq r_2 \leq 0.5 \text{ m}$; $L = 0.017 \text{ m}$.



where $Y = \frac{\theta_{\max}(r_2, 0)}{\theta(r_2, 0)}$, $X = 1.66 \left(\frac{s}{L_{hp}} \right)^{0.03} \left(\frac{k}{hr} \right)^{-0.37}$

Fig. 6 Comparison of the closed-form analytical solution and numerical data for the maximum disk temperatures under different conditions

A large number of numerical data were generated through the numerical simulation, which were subsequently employed to determine the three constants, c_1 , c_2 , and c_3 , in the correlation through a multivariable regression analysis. The three constants thus determined are as follows:

$$c_1 = 1.66, \quad c_2 = 0.03, \quad c_3 = -0.37.$$

Substituting all these constants into Eq. (4), a modified closed-form solution for the maximum temperature at the disk rim incorporating the heat pipe is obtained:

$$\theta_{\max}(r_2, 0) = 1.66 \left(\frac{s}{L_{hp}} \right)^{0.03} \left(\frac{k_a}{hr_2} \right)^{-0.37} \frac{q''}{k_a} \sum_{m=1}^{\infty} \frac{\sin(\eta_m L)}{N(\eta_m) \eta_m^2} \cdot \frac{K_1(\eta_m r_1) I_0(\eta_m r_2) + I_1(\eta_m r_1) K_0(\eta_m r_2)}{K_1(\eta_m r_1) I_1(\eta_m r_2) - I_1(\eta_m r_1) K_1(\eta_m r_2)}. \quad (5)$$

Equation (5) is compared with the corresponding numerical results in Fig. 6. Apparently, Eq. (5) is in good agreement with the numerical results, and most of the numerical data fall within 10% of the values predicted by Eq. (5). As a result, it can be used within the uncertainty range to predict the maximum temperature at the disk rim incorporating radially rotating heat pipes. The deviation between Eq. (5) and the numerical results may be attributable to the errors in connection with numerical simulation, computational errors from Eq. (5), which would include the truncation errors of the infinite series, and the defect associated with the mathematical structure of the modification correlation.

Conclusions

The closed-form solution for a rotating disk having homogeneous thermal properties is derived. The analytical solution is then

used to illustrate the effects of various parameters on the temperature distribution in the disk. To accommodate the disk incorporating the heat pipe cooling technique, the closed-form solution is modified through a modification correlation. The correlation is subsequently established using numerical data from extensive three-dimensional numerical simulation. The modified closed-form solution thus obtained is compared with corresponding numerical data with good agreement. In summary, the closed-form solutions generated through this study can provide a useful and convenient tool for simulating the performance of a turbine disk with and without incorporating the heat pipe cooling technique.

Acknowledgments

The authors would like to thank the Air Force Research Laboratory, Wright-Patterson AFB, for the financial support and their interest in this research topic.

Nomenclature

- c_1, c_2 , and c_3 = constants
- I, K = modified Bessel functions of the first and second kinds
- k = thermal conductivity, W/m °C
- k_a = apparent thermal conductivity of the disk, W/m °C
- k_{eff} = effective thermal conductance of the heat pipe, W/m °C
- h = heat transfer coefficient, W/m² °C
- L = half disk thickness, m
- L_{hp} = heat pipe length, m
- r_1, r_2 = inner and outer radii of the disk, m
- r_{hp} = heat pipe radius, m
- q'' = heat flux, W/m²
- s = distance between the disk rim and heat pipe top end, m
- T = temperature, °C
- T_c = cooling air temperature, °C
- z = axial coordinate, m
- η_m = eigenvalues
- θ = temperature difference, $\theta = T - T_c$, deg
- $\theta_{\max}$ = maximum temperature difference, $\theta_{\max} = T_{\max} - T_c$, deg

References

- [1] Cao, Y., Chang, W. S., and MacArthur, C., 1998, "An Analytical Study of Turbine Disks Incorporating Radially Rotating Heat Pipes," ASME, New York.
- [2] Cao, Y., Ling, J., Rivir, R., and MacArthur, C., 2000, "A Numerical Analysis of Gas Turbine Disks Incorporating Rotating Heat Pipes," ASME, New York, 3, pp. 61–67.
- [3] Arpacı, V. S., 1966, *Conduction Heat Transfer*, Addison-Wesley Company, Menlo, CA.
- [4] Ozisik, M. N., 1993, *Heat Conduction*, 2nd Ed., John Wiley and Sons, New York.