

Transactions of the ASME®

Editor
L. S. LANGSTON (2006)
Associate Editors
Advanced Energy Systems
G. REISTAD (2002)
Fuels and Combustion Technologies
S. GOLLAHALLI (2001)
P. MALTE (2003)
Gas Turbine (Review Chair)
D. WISLER (2001)
Gas Turbine
M. MIGNOLET (2002)
Internal Combustion Engines
D. ASSANIS (2002)
Nuclear
R. DUFFEY (2001)
Power
D. LOU (2002)
Structures and Dynamics
N. ARAKERE (2004)
BOARD ON COMMUNICATIONS
Chair and Vice-President
OZDEN OCHOA
OFFICERS OF THE ASME
President, **W. A. WEIBLEN**
Executive Director, **D. L. BELDEN**
Treasurer, **R. E. NICKELL**
PUBLISHING STAFF
Managing Director, Engineering
THOMAS G. LOUGHLIN
Director, Technical Publishing
PHILIP DI VIETRO
Managing Editor, Technical Publishing
CYNTHIA B. CLARK
Managing Editor, Transactions
CORNELIA MONAHAN
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 © 2001 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

Journal of Engineering for Gas Turbines and Power

Published Quarterly by The American Society of Mechanical Engineers

VOLUME 123 • NUMBER 4 • OCTOBER 2001

TECHNICAL PAPERS

Gas Turbines: Advanced Energy Systems

- 717 The Thermoeconomic and Environomic Modeling and Optimization of the Synthesis, Design, and Operation of Combined Cycles With Advanced Options
S. Pelster, D. Favrat, and M. R. von Spakovsky
- 727 Development of Methanol Steam Reformer for Chemical Recuperation
T. Nakagaki, T. Ogawa, K. Murata, and Y. Nakata

Gas Turbines: Coal, Biomass, and Alternative Fuels

- 734 Integration of Advanced Gas Turbines in Pulp and Paper Mills for Increased Power Generation (00-GT-020)
K. Maunsbach, A. Isaksson, J. Yan, G. Svedberg, and L. Eidensten

Gas Turbines: Combustion and Fuels

- 741 Deposit Formation and Mitigation in Aircraft Fuels (99-GT-217)
L. J. Spadaccini, D. R. Sobel, and H. Huang
- 747 In Situ Detailed Chemistry Calculations in Combustor Flow Analyses (99-GT-271)
S. James, M. S. Anand, M. K. Razdan, and S. B. Pope
- 757 Reduced NO_x Diffusion Flame Combustors for Industrial Gas Turbines (00-GT-085)
A. S. Feitelberg, V. E. Tangirala, R. A. Elliott, R. E. Pavri, and R. B. Schiefer
- 766 Acoustic Resonances of an Industrial Gas Turbine Combustion System (00-GT-094)
S. Hubbard and A. P. Dowling
- 774 Flamelet Model of NO_x in a Diffusion Flame Combustor (00-GT-099)
D. V. Volkov, A. A. Belokin, D. A. Lyubimov, V. M. Zakharov, and G. Opdyke, Jr.
- 779 Self-Excited Oscillations in Combustors With Spray Atomizers (00-GT-108)
M. Zhu, A. P. Dowling, and K. N. C. Bray
- 787 Research on a Methane-Fueled Low NO_x Combustor for a Mach 3 Supersonic Transporter Turbojet Engine (00-GT-113)
Y. Kinoshita, T. Oda, and J. Kitajima
- 796 Application of Macrolamination Technology to Lean, Premixed Combustion (00-GT-115)
A. Mansour, M. Benjamin, D. L. Straub, and G. A. Richards
- 803 Integrated Experimental and Numerical Approach for Fuel-Air Mixing Prediction in a Heavy-Duty Gas Turbine LP Burner (00-GT-122)
G. Mori, S. Razore, M. Ubaldi, and P. Zunino
- 810 Experimental and Numerical Investigation of a Planar Combustor Sector at Realistic Operating Conditions (00-GT-123)
M. Carl, T. Behrendt, C. Fleing, M. Frodermann, J. Heinze, C. Hassa, U. Meier, D. Wolff-Gassmann, S. Hohmann, and N. Zarzalis

(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.

- 817 Evaluation of Advanced Two-Phase Flow and Combustion Models for Predicting Low Emission Combustors (00-GT-133)

G. Klose, R. Schmehl, R. Meier, G. Maier, R. Koch, S. Wittig, M. Hettel, W. Leuckel, and N. Zarzalis

- 824 Response of a Model Gas Turbine Combustor to Variation in Gaseous Fuel Composition (00-GT-141)

R. M. Flores, M. M. Miyasato, V. G. McDonell, and G. S. Samuelsen

- 832 Numerical Simulation of Swirl-Stabilized Premixed Flames With a Turbulent Combustion Model Based on a Systematically Reduced Six-Step Reaction Mechanism (00-GT-143)

D. E. Bohn and J. Lepers

Gas Turbines: Structures and Dynamics

- 839 The Nonlinear Response of Multimode Rotors Supported on Squeeze Film Dampers

R. Y. Yakoub and A. El-Shafei

- 849 Reliable Rotor Dynamic Design of High-Pressure Compressors Based on Test Rig Data (99-GT-150)

N. G. Wagner

- 857 Comparison of the Dynamic Characteristics of Smooth Annular Seals and Damping Seals (99-GT-177)

J. M. Darden, E. M. Earhart, and G. T. Flowers

- 864 Self-Sensing in Fault Tolerant Magnetic Bearings (99-GT-178)

D. Montie and E. Maslen

- 871 Air Entrainment Versus Lubricant Vaporization in Squeeze Film Dampers: An Experimental Assessment of Their Fundamental Differences (99-GT-187)

S. E. Diaz and L. A. San Andrés

- 878 Sliding Mode Control of Magnetic Bearings: A Hardware Perspective (99-GT-205)

E. Maslen and D. Montie

- 886 Blade Life: A Comparison by Cumulative Damage Theories (99-GT-287)

J. S. Rao, A. Pathak, and A. Chawla

- 893 A Reduced-Order Model of Mistuning Using a Subset of Nominal System Modes (99-GT-288)

M.-T. Yang and J. H. Griffin

- 901 Periodic Response of Blades Having Three-Dimensional Nonlinear Shroud Constraints (99-GT-289)

J. J. Chen and C. H. Menq

- 910 Rotordynamic Force Prediction of Whirling Centrifugal Impeller Shroud Passages Using Computational Fluid Dynamic Techniques (99-GT-334)

J. J. Moore and A. B. Palazzolo

- 919 Underplatform Dampers for Turbine Blades: Theoretical Modeling, Analysis, and Comparison With Experimental Data (99-GT-335)

K. Y. Sanliturk, D. J. Ewins, and A. B. Stanbridge

- 930 Friction Damper Optimization: Simulation of Rainbow Tests (99-GT-336)

K. Y. Sanliturk, D. J. Ewins, R. Elliott, and J. S. Green

- 940 Experimental Investigation of Mode Localization and Forced Response Amplitude Magnification for a Mistuned Bladed Disk (00-GT-358)

J. Judge, C. Pierre, and O. Mehmed

Internal Combustion Engines

- 951 A Nonlinear, Transient, Single-Cylinder Diesel Engine Simulation for Predictions of Instantaneous Engine Speed and Torque

Z. S. Filipi and D. N. Assanis

- 960 Author Index

ANNOUNCEMENTS AND SPECIAL NOTES

- 967 Information for Authors

- 968 Preparing and Submitting a Manuscript for Journal Production and Publication

- 969 Preparation of Graphics for ASME Journal Production and Publication

S. Pelster¹

D. Favrat

e-mail: daniel.favrat@epfl.ch

Laboratory for Industrial Energetics,
Swiss Federal Institute of Technology
of Lausanne,
CH-1015 Lausanne, Switzerland

M. R. von Spakovsky

Energy Management Institute,
Mechanical Engineering Department,
Virginia Polytechnic Institute
and State University,
Blacksburg, VA 24061-0238

The Thermoeconomic and Environomic Modeling and Optimization of the Synthesis, Design, and Operation of Combined Cycles With Advanced Options

Combined cycle power plants are currently one of the most important options for the construction of new generating capacity as well as for the replacement and repowering of existing units. Due to the complexity and the large number of options and parameters available to such plants, finding optimized solutions for system synthesis, design, and operation is very difficult if not impossible with these traditional methods such as case and parametric tradeoff studies. This is especially true when advanced options as well as thermodynamic, economic, and environmental criteria are considered. A thermoeconomic environomic methodology to deal with these difficulties is presented here. Results for the application of this methodology to a 50 MW cogeneration combined cycle power plant are presented and discussed. [DOI: 10.1115/1.1366323]

Introduction

In spite of the momentary oversupply of electricity and of the uncertainties linked to the liberalization of the European and U.S. energy markets, medium and long term perspectives will be increasingly influenced by a strong worldwide growth in demand for fossil fuel from the developing countries, potential global environmental problems, and more competitive economic forces in the energy sector. The need for energy efficient and economic technologies with reduced environmental impact will, thus, only strengthen in this century.

The methodology proposed in this paper is a step towards a coherent framework for improving the tools used in developing such technologies. This framework deals with the synthesis, design, and operation of energy technologies, and is intended to facilitate not only handling the complexity and large number of degrees-of-freedom involved with technology development but sensitivity studies as well. The latter can be done on any number of factors including the external costs associated with environmental impact as an attempt to address questions such as:

- What is the influence of internalizing external costs on the choice of technology and on its synthesis, design and operation?
- For a given reduction of environmental burden (for example, a certain percentage of CO₂ reduction), what is the cheapest way this can be achieved within the constraint of a viable, competitive technology? For the particular case presented here, what is the best natural gas technology for power generation which could be applied in a range of outputs higher than a few tens of megawatts? Alternatives go from simple gas turbines to complex combined cycles with or without cogeneration.

Combined cycle power plants that combine a gas turbine cycle with a steam turbine cycle are currently one of the most important

options for the construction of new generating capacity as well as the replacement and repowering of old units ([1]). From an engineering point of view, combined cycle power plants represent complex systems with a large number of options and parameters for development and improvement. This not only includes a determining of system synthesis (configuration) but component designs (capacities, nominal performance) and operation (off-design performance) as well. Finding optimized solutions for such systems is very difficult if not impossible with traditional methods (case and parametric tradeoff studies). This is especially true when advanced solutions such as steam-injected and intercooled gas turbines are considered or systems for CO₂ separation become an option and compete with measures for efficiency improvement.

As with other energy systems, combined cycle power plants need to respond to a variety of criteria. Thermodynamic criteria need to be completed by economic criteria including the costs of investment and not just operation. In addition, environmental criteria have become increasingly important through regulations and emission taxes. Traditionally, all such criteria have been dealt with in a nonintegrated fashion due to the lack of a unified approach, which can simultaneously handle them all. A methodology, which addresses this gap, is presented in this paper and offers to the technical decision-maker a tool for system development and operation in the context of the criteria mentioned above.

The methodology is applied to develop an environomic model² of a combined cycle power plant with advanced options as well as a corresponding objective function which can then be optimized to simultaneously determine the optimum synthesis, design, and operational mode for the system.³ Such a model is formed from economic and engineering/physical (or thermodynamic) simulation models of the system and its components as well as from a quantification of the system's environmental characteristics.

The optimization problem solved represented a highly nonlinear, mixed integer programming problem with a high degree of

¹Now with Bayernwerk A.G., Nymphenburger Strasse 39, Munich, Germany. e-mail: stefan.pelster@bayernwerk.de.

Contributed by the Advanced Energy Systems 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 AES Division, June 2000; final revision received by the ASME Headquarters Dec. 2000. Editor: H. D. Nelson.

²The thermoeconomic model of the system is a special case of the environomic one.

³The time-dependency of operation is not considered here but would be handled as described in Olsommer [2] and Olsommer et al. [3–5].

complexity, requiring the development of an adequate optimization algorithm. A heuristic instead of a traditional operations research approach was successfully used.

We begin now with a brief background perspective followed by a description of the formulation of the environomic model and of the algorithm used to optimize it.

Brief Background Perspective

The best practical design of an energy system, the best operation of an existing one, or the synthesis of either which best meets a specific goal or goals may be found based not only on thermodynamic criteria but on economic and environmental ones as well. An approach which mathematically models one or more of these aspects in a unified fashion would permit the use of optimization algorithms for searching the entire solution space for the synthesis, design and/or operation of a new or an existing system.

The development of such an approach falls within the domain of thermoeconomics. Thermoeconomic models when expanded to include environmental criteria associated with long-term resource use and pollution have been called environomic models ([6,7]). When fully developed, they include those thermodynamic, economic, and environmental aspects associated with the entire life cycle⁴ of a system (see [8–11]).

The Environomic Model: General Formulation

The unified “environomic” approach is based on the fact that thermodynamic, economic, and environmental criteria can be expressed as costs of fuel, investment, and pollution (Fig. 1). Including the latter is an attempt to quantify in monetary terms the negative impact of pollution on the environment. Frequently referred to as external costs or externalities, they are costs in the form of damages, prevention, increased health insurance, environmental cleanups, etc. Charged to the user of technology as, e.g., emission taxes, they can be a positive force in developing economically sensible technologies with a lower impact on the environment.

The centerpiece of an environomic model is the environomic objective expressed below as a rate function. This function is optimized with respect to a set of independent (decision) variables and subject to a set of equality and inequality constraints. In general terms, this is expressed as

$$\min \dot{C}_{\text{prod}}(\mathbf{x}, \mathbf{y}) = \sum_{i=1}^N \dot{Z}_i(\mathbf{x}, \mathbf{y}) + \sum_{i=1}^R \dot{\Gamma}_{ri}(\mathbf{x}, \mathbf{y}) + \sum_{i=1}^P \dot{\Gamma}_{pi}(\mathbf{x}, \mathbf{y}) + \dot{K} \quad (1)$$

w.r.t. $\mathbf{x}$ and subject to

$$h_j(\mathbf{x}, \mathbf{y}) = 0 \quad j = 1, \dots, J \quad (2)$$

$$g_k(\mathbf{x}, \mathbf{y}) \geq 0 \quad k = 1, \dots, K \quad (3)$$

where

$$\mathbf{x} = (x_1, \dots, x_I) \quad (4)$$

$$\mathbf{y} = (y_1, \dots, y_J) \quad (5)$$

$$x_{i_{\min}} \leq x_i \leq x_{i_{\max}} \quad i = 1, \dots, I \quad (6)$$

$$y_{j_{\min}} \leq y_j \leq y_{j_{\max}} \quad j = 1, \dots, J \quad (7)$$

and where

$\dot{C}_{\text{prod}}$ = cost rate of products (electricity and heat)

$\dot{Z}_i$ = capital cost rate of component i ;

$\dot{\Gamma}_{ri}$ = cost rate of resource i bought and consumed;

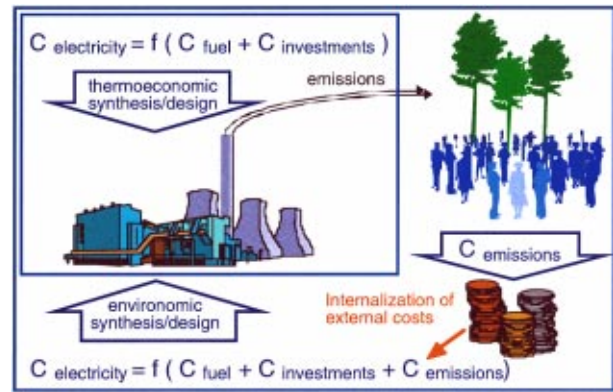


Fig. 1 Schematic view of thermoeconomic and environomic modeling and optimization

$\dot{\Gamma}_{pi}$ = cost rate of pollutant i due to operation;

K = constant surcharge cost rate;

and $\mathbf{x}$ and $\mathbf{y}$ are the independent and dependent variable sets, respectively.

With electricity and heat as the principal products of the energy system and each of the summations in Eq. (1) expressed as $\dot{Z}$, $\dot{\Gamma}_r$, and $\dot{\Gamma}_p$, respectively, the objective function (Eq. (1)) can be rewritten as

$$\min c_{ex \text{ total}} = \frac{\dot{Z} + \dot{\Gamma}_r + \dot{\Gamma}_p + \dot{K}}{\dot{E}_{\text{elec}} + \dot{E}_q} \quad (8)$$

where $c_{ex \text{ total}}$ is the total or combined unit exergy cost of electricity and heat produced. Here $\dot{E}_{\text{elec}}$ is the electricity produced by the system while $\dot{E}_q$ is the exergy equivalent of the heat supplied by the system. Note that if zero pollution costs are assumed, the environomic objective becomes a thermoeconomic one. A more extensive expression and discussion of the environomic objective (Eq. (1)) can be found in Pelster [12], Curti [10], and von Spakovsky and Frangopoulos [8].

The environomic objective can be extended to include life cycle pollution costs, i.e., those associated with the manufacture, installation, dismantling, and removal of a system. Since these are very small when compared with the pollution costs resulting from the plant's operation ([13]) they are neglected here.

The cost functions used to describe the capital cost terms ($\dot{Z}_i$) in Eq. (1) or (8) above are based on standard power functions coupled to cost correction and escalation factors found in the literature and fit to current market data of comparable pieces of equipment. The data were obtained from several of the primary manufacturers involved and resulted in functions, which provide cost predictions that are very representative of actual equipment prices. An example is shown later on in this article.

As to the resource cost functions used to describe the resource cost terms ($\dot{\Gamma}_{ri}$) in Eq. (1) or (8) above, they are based on a simple linear relationship in which the associated unit cost for the resource multiplies the measure used for the resource (e.g., energy, exergy, mass).

The remaining term of significance in Eqs. (1) and (8) is that for the cost rate of pollutant i due to operation ($\dot{\Gamma}_{pi}$). As with the resource cost functions, these functions assume a linear relationship between the unit costs of pollution and the measure (i.e., mass, energy, exergy, etc.) used for the pollutant emission from the plant. In this study, the mass flow rate of pollutant emitted is used. In order to account for changing local conditions related to emission levels already present in the environment and to distinguish effectively between plants whose emission levels approach and even exceed legislated limits and those which are far below

⁴Note that “entire life cycle” includes not only the cycle related to the actual economic lifetime of the system but the cycles as well which deal with the processes both upstream and downstream of the system, i.e., those related to the system's use of fuel resources and the manufacture and removal of the system's equipment.

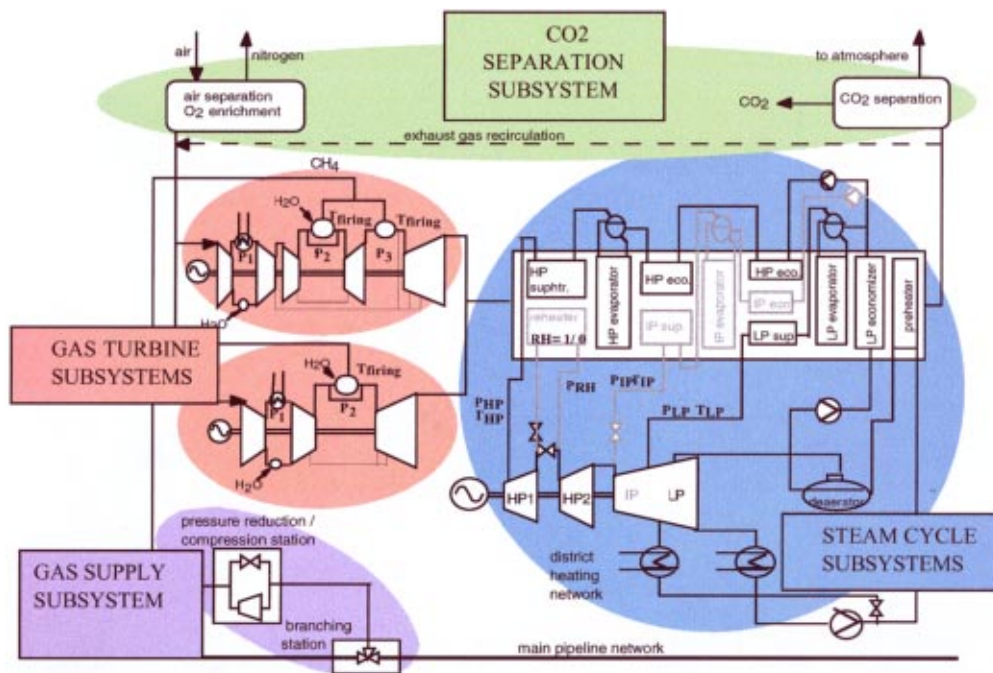


Fig. 2 Modeled super-configuration for the synthesis of cycle options

such limits, Frangopoulos and von Spakovsky ([6–8]) suggested multiplying this linear expression by a penalty factor. The reader may refer to ([11]) for a detailed description of the factor and to ([12]) for taking into account CO₂ national goals for emission reductions in the case of cogeneration power plants.

The Super-Configuration Model

In order to simultaneously model and optimize the synthesis aspects of the cycles considered, the super-configuration depicted in Fig. 2 below was used. It combines, as the name implies, different configurations, both implicitly and explicitly, into a single model. In addition to a range of options that are already available in existing combined cycles, this super-configuration contains various advanced options that can be chosen from. For example, gas turbines with simple or sequential combustion can be selected. The process of “compression” can be accomplished with a compressor (a) without intercooling, (b) with an intercooling heat exchanger or (c) with an evaporative intercooler. In addition, four

different types of steam turbine cycles can be chosen or none at all. With respect to greenhouse gas emissions, CO₂ separation systems including exhaust gas recirculation compete with measures for improvements in cycle efficiency.

Through a combination of all options included in the model, about 110 different configurations are possible. These configuration changes are modeled using six binary/integer variables. The choice of options is summarized in Table 1. In addition, the influence of changing the values of thermodynamic parameters such as pressures, temperatures, flow rates, etc., is modeled by means of 15 continuous variables. Please refer to the Appendix for the complete set of continuous parameters.

The Design Model

Although each of the components in the super-configuration (e.g., a compressor) is physically/thermodynamically modeled as a “black box,” enough detail is introduced to allow for a realistic representation. For example, gas turbine cooling characteristics and the influence of gas composition are included in the component design models.

In addition to the thermodynamic processes present, the design model includes economic and environmental models for component investment costs and system emissions, respectively. While the thermodynamic process and emissions models rely on physical laws and correlations (see [8,12,14]), the investment costs are expressed as functions of thermodynamic parameters. An example is the gas turbine investment cost that depends on pressure ratio, turbine inlet temperature, gas mass flow rate, polytropic efficiency, and the type of NO_x abatement considered.

Another example is the cost for heat exchangers in an HRSG that is calculated as the sum of the cost for the various heat exchange units (e.g., superheaters, evaporators), indicated by the index *i*:

$$C_{\text{heatex}} = c_1 \cdot \sum_i \left(f_{p_i} \cdot f_{T_{i,\text{steam}}} \cdot f_{T_{i,\text{gas}}} \cdot \left(\frac{Q_i}{\Delta T_{lmi}} \right)^{0.8} \right) \quad (9)$$

where ΔT_{lmi} is based on the logarithmic mean temperature difference and $c_1 = 3650 \text{ \$}/(\text{kW}/^\circ\text{K})^{0.8}$. f_p , $f_{T_{\text{steam}}}$, $f_{T_{\text{gas}}}$ are cost cor-

Table 1 Choice of options from discrete variables

Variable for:	Options
CO ₂ separation method	• none • MEA absorption • O₂ enrichment and recirculation
Gas turbine type	• simple combustion • sequential/double combustion
Intercooling method,	• no intercooling • heat exchanger • water injection (evaporative intercooling)
Combustor type for NO _x control	• simple combustor • steam injection • water injection • dry low NO_x combustor
Steam cycle type	• no steam cycle • dual pressure steam cycle • dual pressure reheat steam cycle • triple pressure steam cycle • triple pressure reheat steam cycle
Fast HP turbine	• no separate HP turbine • fast running HP turbine

Table 2 Comparison between modeled and available data for combined cycle plants

	Dual Pressure		Triple Pressure		Dual Press. RH		Triple Press. RH	
	Ref. Plant	Model	Ref. Plant	Model	Ref. Plant	Model	Ref. Plant	Model
Gas Turbine								
Output (MW)	143.6	143.7	161.2	161.8	144.5	146.7	225	226.3
Efficiency (%)	34.1	34.15	35.7	35.8	35.7	35.8	36.3	36.4
M exhaust (kg/s)	497	490	528	510	500	499	624	632
T exhaust (°C)	542	534	529	521	528	524	550	551
Steam Cycle								
M HP (kg/s)	60.1	59.5	57.3	56	52.2	51.5	...	63.8
T HP (°C)	508	519	513	506	493	491	540	539
M IP (kg/s)	...	...	12.5	12.1	...	...	...	14.7
T IP/RH (°C)	...	...	514	506	446	491	315	535
M LP (kg/s)	14.8	13.4	7.9	8.3	14.5	15.2	...	11.9
T LP (°C)	273	285	144	144.5	269	266	230	233
output (MW)	152.8	155.4	269.3	270.5	81.6	83.8	247	244.7
Combined Cycle								
Gross output (MW)	440	442.8	752.9	755.9	226.1	230.5	697	697.3
Gross efficien. (%)	52.3	52.6	55.5	55.7	52.4	53.4	57.1	57.1
Net output (MW)	...	437.6	...	267.8	223.5	227.1	685	686.6
Net efficiency (%)	...	52	...	54.9	51.8	52.6	56.1	56.2

rection factors that introduce a sensitivity of cost to pressure as well as to steam and exhaust gas temperature. $f_{T_{\text{steam}}}$ is shown as an example where the temperatures are in K:

$$f_{T_{\text{steam}}} = 1 + \exp\left(\frac{T_{\text{out,steam}} - 830}{500}\right). \quad (10)$$

The investment cost functions were taken from various sources in the literature ([15–18], etc), as well as from corporate communications. For reasons of space they are not presented here; and instead the reader is referred to Pelster [12] for the complete set.

With all of these various design models, a great deal of emphasis was placed on developing realistic models in order that the inherent uncertainties in this type of modeling be reduced. This was accomplished through calibrations of the models used with existing reference gas turbines and combined cycles as can be seen in Table 2.

The model for NO_x formation is based on a model developed by von Spakovsky and Frangopoulos [8] who used measurement data given by Kehlhofer [19] to model NO_x formation as a function of λ and the inlet air temperature. The model was adapted in order to take account of the benefits of shorter residence times and more homogeneous temperature distributions in state of the art dry low NO_x technology. The effect of changing gas compositions and water/steam injection were considered through correction factors ([12]).

The costs of resources, capital equipment, and pollution used in this work are in monetary units. The costs of pollution include those for emissions of NO_x and CO_2 due to operation of the power plant as well as emissions of CO_2 and CH_4 caused by the production and transportation of natural gas. The unit costs for NO_x used are pollution damage costs taken from a report on the external costs of energy usage ([20,21]). For the case of CO_2 , damage costs, which might occur in the future, could be potentially very high. To avoid these costs, the concept of so-called avoidance cost ([20,21]) is used here.⁵ CO_2 taxes can be used as such and are being considered by some European countries in the range of 11 CHF/t for Denmark ([22–24]) to 66 CHF/t for Norway ([25]).

Synthesis and Design Optimization Using a Genetic Algorithm

Figure 3 shows a very simplified representation of the solution space for the synthesis and design optimization of a super-configuration and illustrates the principal challenges incurred. Common features of such a space are highly nonlinear, continu-

ous, and noncontinuous surfaces; nonconvexity; noncontiguous surfaces; a single global optimum but a number of possible, very different, near-optimum solutions; difficulties in generating feasible parameter sets, etc. This space and its associated optimization problem represent a mixed-integer, nonlinear programming problem (MINLP) for which no standard algorithm is available.

In Fig. 3, the objective function, here a minimization of production costs, is represented by the vertical axis. The left horizontal axis represents structural changes that result in different system configurations and component designs while the right axis represents changes in operational parameters. The combination of axes leads to separate surfaces representing the different configurations modeled with a discrete variable set (binary/integer) while their shapes (here simplified for illustration purposes) describe multiple component designs and operating points modeled with a continuous variable set.

Obviously, finding the global optimum as well as a set of near-optimum solutions for this MINLP poses a number of major challenges. Three outstanding ones are:

- the number of possible configurations may become very large through combinations of choices for the different parts of the cycle. Gradually passing from one configuration to the other is impossible.
- different configurations frequently require different sets of continuous parameter values for good and optimum performance.

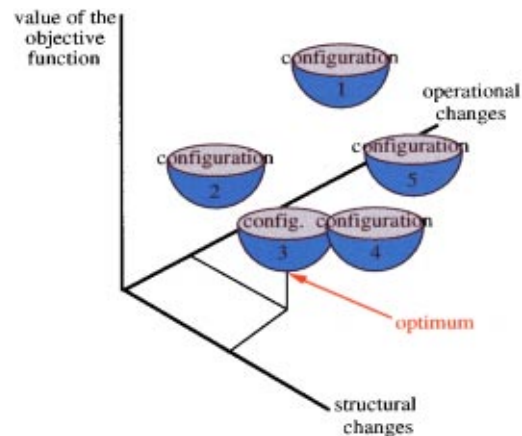


Fig. 3 Simplified representation of the synthesis, design, and operations solution space for energy systems

⁵Not to be confused with “prevention costs” which relate directly to the capital equipment used to prevent emissions.

An example is the gas turbine pressure ratio that needs to take on high values for simple gas turbines cycle, while combined cycles require much lower values.

- certain configurations are harder to optimize than others, e.g., due to the larger number of parameters involved. Even though they may not perform well at the beginning of a search, they may prove to be very good when their potential is developed.

In order to effectively deal with all of the above difficulties a genetic algorithm (GA), in particular a “Struggle GA” ([26]), was used. In contrast to deterministic or mathematically based gradient search methods (e.g., generalized reduced gradient, sequential quadratic programming, etc.), genetic algorithms rely on populations instead of single points. They emulate biological mechanisms such as genes, genomes, crossover, replacement, and mutation ([27,28]).

Every “individual” in the population is described through a “genome,” i.e., a set of parameter values that represent the “genes” that take on either discrete or continuous values. The driving force for improvement is the “survival of the fittest” principle. The “Struggle GA” was modified for improving the capability of converging while maintaining a certain diversity in the population, permitting the development of several “species,” i.e., several promising configurations ([29]). These are developed simultaneously with respect to their continuous variables to converge to the global and near-global optimum solutions mentioned above. Diversity, in general, significantly reduces the risk of becoming trapped in local optima of which there are many. In the end, the user is provided with several good solutions as opposed to a single alternative. The modifications are described in detail in Pelster [12] or Molyneux et al. [30]. The basic outline of the modified “Struggle GA” is summarized in the following seven steps:

- 1 Initially, generate a random population.
- 2 Choose two parents randomly from the population.
- 3 Create an offspring through crossover and apply mutation.
- 4 Calculate a distance measure between the offspring and the rest of the population.
- 5 Compare the offspring to the closest (i.e., most similar) individual.
- 6 If the fitness of the offspring is better than the fitness of the closest individual, the offspring replaces it.
- 7 Return to Step 2.

Note that a binary mapping procedure is only used for integer variables. A blend crossover procedure is used for real variables to prevent a loss of information and maintain a high level of precision when, for example, sizing components or dealing with thermodynamic data.

Application to a Cogeneration Power Plant

Various applications were considered although only one is presented in detail below: determining the synthesis and design for a 50 MW cogeneration power plant.⁶ The base or reference case used for all applications is the thermoeconomic one for which no pollution costs are considered ($\dot{I}_p = 0$) and environmental aspects are only taken into account through emission limit constraints. In addition, a sensitivity analysis provides insight into what changes in system synthesis and component design result from the internalization of pollution costs for the two main substances emitted, NO_x and CO_2 .

The scenarios for these thermoeconomic base cases are shown in Table 3 below. It is supposed that the cogeneration power plant operates 4000 hours per year and supplies 50 MW electric as well as 25 MW thermal power. Since it is located in an urban area, the gas supply pressure is limited to 3.0 bar. Compliance with a NO_x emission limit of 50 mg/m^3 referred to 15 percent oxygen in the

Table 3 Combined scenario for the base or reference case

Electrical power	50 MW
District heating power	25 MW
District heating temp.	70°C–90°C
Annual operating time	4000 h
Gas supply pressure	3.0 bar
Gas price (energy)	2 CHcts/k Wh
Gas price (power)	20 CHF/MW
Interest rate	4.5%
Economic lifetime	15 years
NO_x emission limit	50 mg/m^3
NO_x local concentration	50 $\mu\text{g/m}^3$
NO_x pollution costs	...
CO_2 pollution costs	...

dry gas is imposed through a constraint.⁷ The NO_x emission limit corresponds to the Swiss emission limits proposed for 1998 ([31]).

In order to study the influence of adding pollution costs for NO_x and CO_2 to the base cases, environomic models for both applications were optimized as well. Obviously, for these models, the NO_x emission limit is no longer represented as a constraint but instead is directly incorporated into the cost of pollution through the pollution penalty factor ([11]).

Optimization Results for the Base Case

By means of the modified “Struggle GA” multiple configurations are identified and optimized with respect to their continuous variables in a single run. For the 50 MW base case application, several alternatives with different configurations were found varying from steam injected gas turbines coupled to a heat recovery unit for district heating to combined cycles. For the 50 MW base case, Fig. 4 shows the six best-performing configurations resulting from the optimization. Configurations 1 and 2 are steam injected gas turbines (STIG) with sequential (or double) combustion (DC) and intercooled compressors while configurations 3 and 4 are STIG cycles with simple combustion (SC), one of them with intercooling. Configurations 5 and 6 are combined cycles using DC and SC gas turbines, respectively, as well as dual pressure steam cycles. The most important parameters are summarized in Table A1 in the Appendix. As shown in the same table, the cost of producing electricity, expressed in CHcts/kWh, varies only slightly from one configuration to the other.

As shown in Fig. 5, the optimized continuous parameters for the STIG cycles and combined cycles lead to significant differences in gas turbine design and, thus, illustrate the formation of “solution species.” The optimized pressure ratios and firing tem-

⁷This constraint is formulated through a logarithmic penalty function: The objective function value is multiplied by a penalty factor if the limit is exceeded. This penalty, however, has nothing to do with the pollution penalty factor.

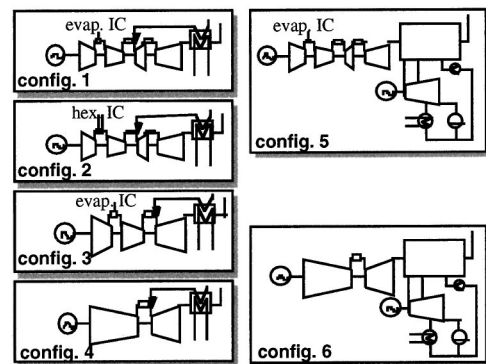


Fig. 4 The six best performing configurations for a 50 MWe/25 MWth cogeneration plant (without pollution cost internalization)

⁶Of all options of the superconfiguration, the results presented here only show those identified through the optimization.

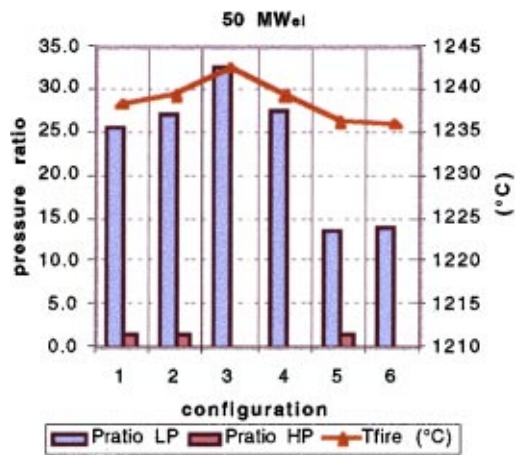


Fig. 5 Gas turbine pressure ratios and firing temperatures

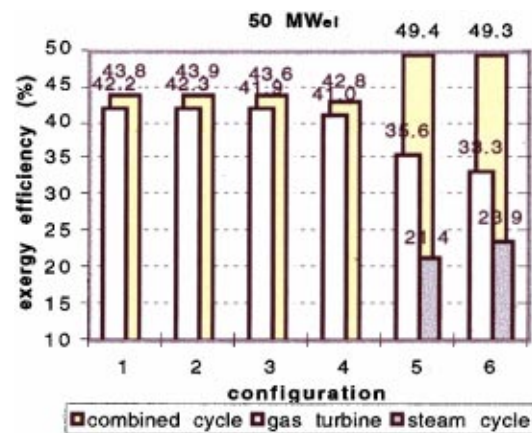


Fig. 7 Combined cycle exergy efficiencies

peratures of the STIG gas turbines in configurations 1 to 4 are significantly higher than for the combined cycle configurations 5 and 6. The intercooled gas turbines (configurations 1, 2, 3, and 5) permit higher overall pressure ratios (the product of Π and Π_{HP}) than without intercooling and the optimized value of Π is higher for cycles with SC than for those with DC. The low overall pressure ratio in the case of combined cycles results in exhaust gas temperatures that are about 100°C higher than for the STIG cycles, permitting an efficient steam cycle operation. This is illustrated in Fig. 6. Further, intercooled DC gas turbines have the lowest air mass flow, resulting in a high power density and low specific investment costs.

Figure 7 clearly demonstrates the efficiency increase of combined cycles compared to STIG cycles as well as the potential of increasing gas turbine electrical efficiency through intercooling and steam injection to almost 44 percent. Of course, this comes with a cost as shown in Fig. 8 where total investment for the combined cycle configurations increases due to the steam turbine cycle as well as the higher civil engineering costs. The small investment for steam cycles in configurations 1 to 4 represents the costs of heat exchangers for steam injection and district heating.

The specific NO_x and CO_2 emissions per unit of exergy produced are given in Fig. 9. With NO_x emissions just below 50 mg NO_x/m^3 , all configurations invest just enough into NO_x reduction measures to comply with the legal limit. Obviously, the combined cycle configurations have the better environmental performance. It

should be noted once more that, for the base case, only NO_x emissions were restricted by legal limits and that no pollution costs were considered.

The composition of the specific (unit) exergy cost of electricity production for each solution is shown in Fig. 10. The specific

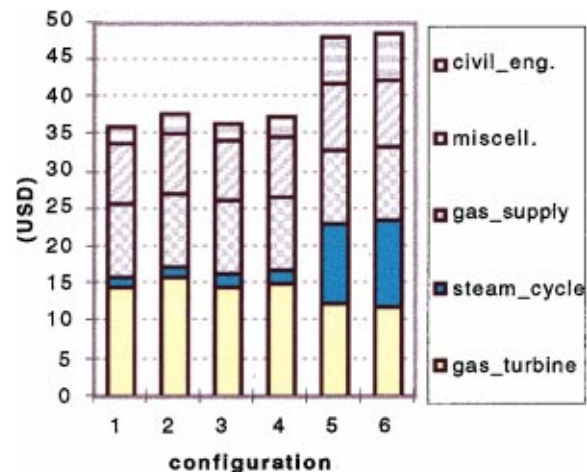


Fig. 8 Investment costs per subsystem in mio. U.S. dollars

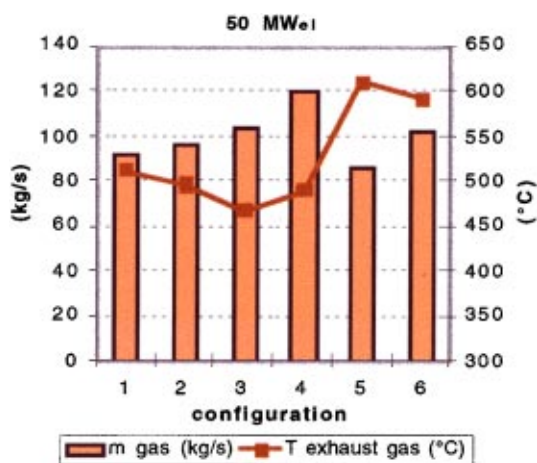


Fig. 6 Gas turbine exhaust gas mass flow rate and temperature

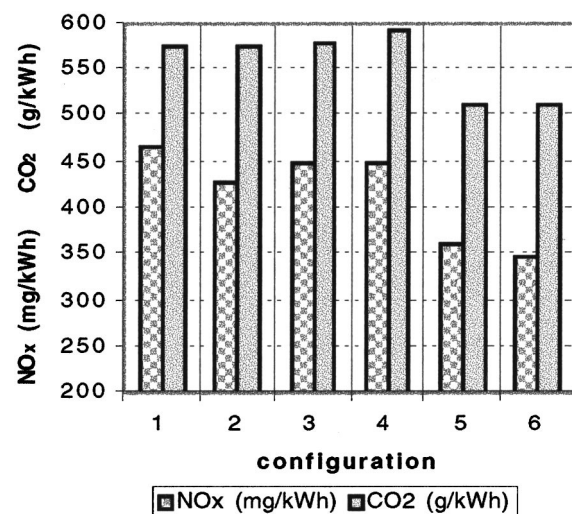


Fig. 9 Specific NO_x and CO_2 emissions

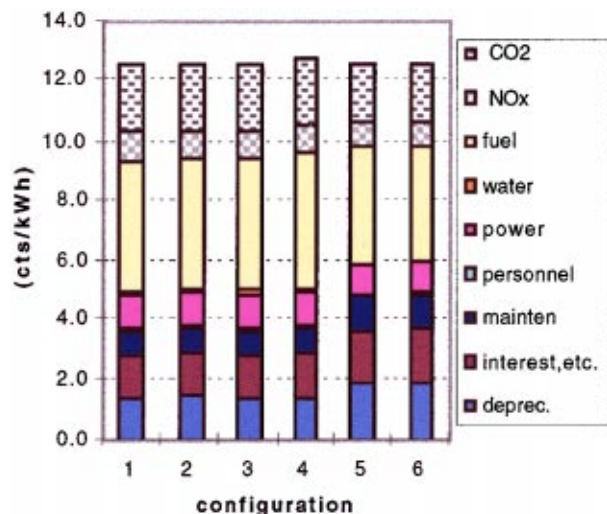


Fig. 10 Specific (unit) exergy cost of electricity production in CHcts/kWh

production costs without those for pollution (solid areas only) are the lowest for the STIG configurations. The capital cost increase of the combined cycle configurations is only partly compensated by reduced resource costs, thus, leading to higher costs for electricity production.

If the costs of pollution (dotted areas in Fig. 10) are included for evaluation, the more capital-intensive but more efficient combined cycle configurations 5 and 6 become competitive. This demonstrates that even if the costs of pollution are not directly included in a thermoeconomic optimization but only added afterwards, considering these costs can lead to a different choice of configuration(s) than that indicated by the thermoeconomic optimization results.

Environomic Optimization and Sensitivity Study on the Cost of CO₂

This section summarizes the results of an environomic optimization and sensitivity study on the configuration and design of a 50 MW cogeneration power plant. It demonstrates the influence of changes in the unit cost of CO₂ emissions and of including both NO_x and CO₂ pollution costs directly in the optimization process.

The legal limit on NO_x emissions is no longer a constraint but is instead used in the calculation of the pollution penalty factor that increases exponentially if the legal limit is approached or exceeded. This penalty factor is also a function of the degree of pollution present in the environment. Thus, the NO_x pollution cost for urban areas (50 MW plant scenario) exposed to 50 $\mu\text{g}/\text{m}^3$ NO_x will be generally higher than for rural areas that are only exposed to 20 $\mu\text{g}/\text{m}^3$ NO_x.⁸ The unit cost of CO₂ emissions is varied from 0 to 6 CHcts/kg. The larger number is in the range of the Norwegian CO₂ tax of 6,6 CHcts/kg ([25]).

Table A2 in the appendix shows the optimized parameter values for the environomic optimization. The most important configurational changes caused by variations in the CO₂ costs are shown in Fig. 11. For zero CO₂ costs, the intercooled STIG cycle is the most attractive solution. This corresponds to the optimum configuration identified in the base (thermoeconomic) case. At 2 CHcts/kg CO₂, the global environomic optimum configuration becomes a combined cycle and remains so for all higher CO₂ unit costs. From 2 to 4 CHcts/kg CO₂, the only major configurational change is that the intercooler type changes from an evaporative intercooler to intercooling with a heat exchanger which provides

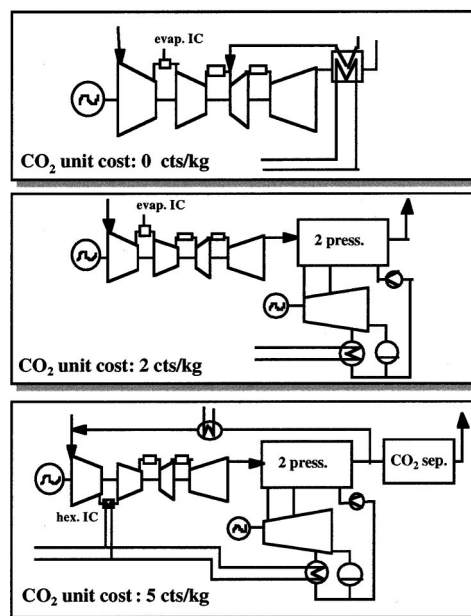


Fig. 11 Most significant global or near global optimum configurational changes as a function of the CO₂ unit pollution costs (cts/kg=CHcts/kg)

heat to the district heating network. A CO₂ separation system is introduced for a unit cost of 5 CHcts/kg. In this case, an exhaust gas recirculation system helps to reduce the size of the CO₂ separation system and to facilitate the separation process through reducing the volume flow rate of gas treated and increasing its CO₂ concentration.

From Table A2 it can be seen that the environomically optimized STIG cycles (at 0 and 1 CHcts/kg CO₂) have higher gas turbine pressure ratios than the global optimum (configuration 1—a STIG cycle) for the base case. The significant drop of the firing temperature between the base case and the environomic cases with 0 and 1 CHcts/kg CO₂ are explained by NO_x reduction measures through a combination of high air equivalence and steam injection ratios.

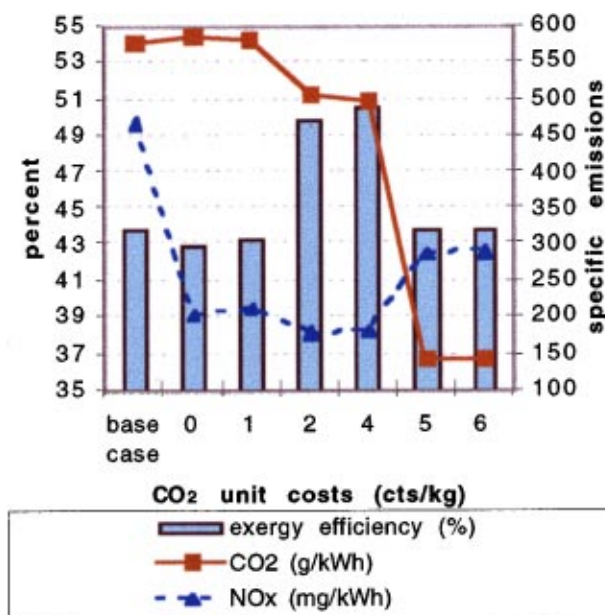


Fig. 12 Exergy efficiency and specific emissions per unit of product produced

⁸Values on an annual average ([21,32]).

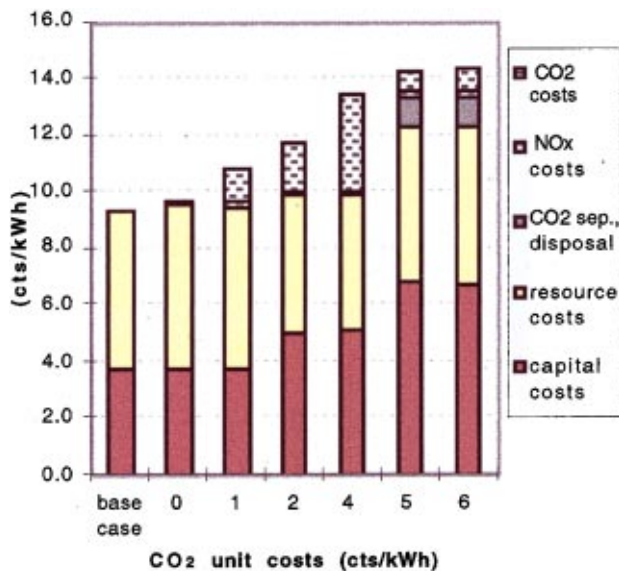


Fig. 13 Optimal specific exergy costs of production (cts =CHcts)

The effect of the different global optimum configurations on efficiency and specific emissions is shown in Fig. 12. These measures are a response to the introduction of NO_x pollution costs in the optimization and reduce the NO_x emissions. At the same time, the cycle exergy efficiency drops slightly. With CO₂ costs above 1 CHcts/kg, the higher efficiency of the combined cycle configuration leads to lower specific emissions and high turbine inlet temperatures become attractive again.

For CO₂ unit costs between 2 and 4 CHcts/kg, the change to combined cycle configurations increases efficiency by about 7.5 percent, thus reducing the specific CO₂ as well as NO_x emissions per unit of exergy supplied by the system. For CO₂ unit costs higher than 5 CHcts/kWh, the use of a CO₂ separation system causes a significant drop in efficiency. Due to the drastic reduction in CO₂ emissions, this type of solution is, nevertheless, attractive for high CO₂ unit costs.

The cost breakdown of the optimum specific exergy costs of production including the costs of pollution are shown in Fig. 13. For the case of 0 CHcts/kg CO₂, only the costs for NO_x are added

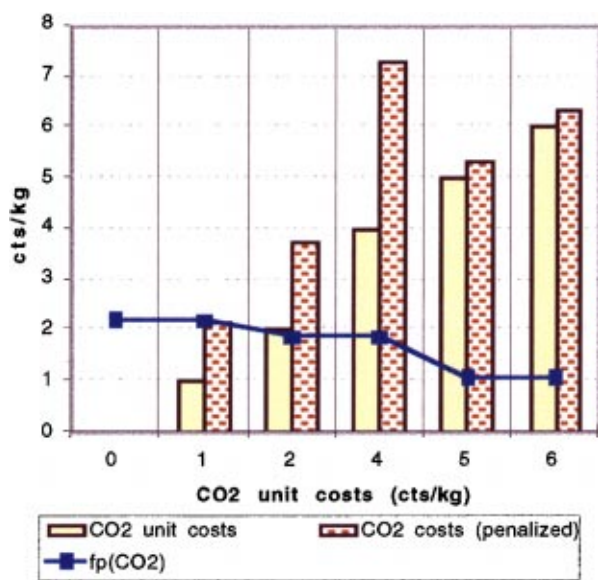


Fig. 14 Unit costs and penalized costs for CO₂ emissions (cts =CHcts)

to the costs of resource use and capital. For the cases of 2 and 4 CHcts/kg CO₂ (the optimal combined cycle configurations without CO₂ separation), the increase in capital costs is partly compensated by a reduction in resource costs.

For the optimal configurations with CO₂ separation at 5 and 6 CHcts/kg CO₂, the large increase in the specific exergy costs of production due to CO₂ separation should be noted. Even though CO₂ emission costs are low, costs for CO₂ separation add significantly to the overall costs.

One final note is that the pollution penalty factor for CO₂ takes on values around 2 for configurations without CO₂ separation and around 1 for configurations using such a system. The effect on the penalized CO₂ unit costs is illustrated in Fig. 14.

Conclusions

The environomic/thermo-economic modeling and optimization methodology for energy was applied to the optimization of the synthesis, design and operation of an advanced power plant super-configuration model. It was shown that the internalization of the external costs of pollution leads to different choices in the optimum configuration and design. The work shows the validity of the method for the development of energy systems consistent with the concept of a sustainable development. Thanks to recent progress in modeling and optimization techniques, engineers should now be able to optimize, from the early development stage on, complex systems such as the advanced power plants presented here. In doing so, they will be able to simultaneously take into account an increased number of parameters including economic and environmental ones through a technology-driven "bottom-up" type of approach. The optimization results will provide knowledge about a number of alternative optimum or near-optimum solutions.

For possible industrial applications, model simplifications may be needed in some cases (e.g., when the primary objective is real-time or pseudo-real-time operation or time-dependent off-design operation) for faster resolution of the problem. For example, one type of simplification may be to replace the general gas turbine model by a catalogue that contains gas turbines available on the market. An illustration of this is the application of the method in a repowering project presented in B  rer et al. [1].

In general, the methodology presented here provides a structure, which can easily incorporate specific industrial knowledge. This is especially important for the cost functions since if not properly formulated, they can introduce unacceptable uncertainties into the modeling process.

Finally, more details of the environomic methodology presented here can be found in Pelster [12], Curti [10], and Curti et al. [11]. In the latter, the methodology is applied to a district heating network with centralized and decentralized heat sources. The extension of this methodology to time-dependent issues and the incorporation of reliability/availability considerations directly into the thermo-economic model of a system are presented in Olsommer et al. [3–5].

Acknowledgments

The authors would like to thank M. Adam Molyneaux for his contribution on the major computer programming issues. Our thanks also go to the Research Fund of the Swiss Gas Industry (FOGA) and to the Swiss National Science Foundation, which sponsored this work, as well as to the Alliance for Global Sustainability between MIT, the University of Tokyo and the Swiss Federal Institutes of Technology.

Nomenclature

- c = unit cost or price (CHcts/kWh, CHF/kg)
- ζ = concentration
- E = exergy rate (kJ/s)
- J, K = number of equality/inequality constraints
- f_p = pollution penalty factor
- K = surcharge rate (CHF/s)

$\dot{m}$ = mass flow rate (kg/s)
 P = pollution penalty function
 Q = heat rate (kJ/s)
 r = specific CO₂ emission (kg/kJ)
 T = absolute temperature (K)
 $\mathbf{x}$ = independent parameter set
 $\mathbf{y}$ = dependent parameter set
 Z = capital cost rate (CHF/s)
 CHF = Swiss francs (at the time of the study: 1.5 CHF=1 U.S. dollar and 1.5 CHcts=1.0 U.S. cent)

Greek Symbols

$\dot{\Gamma}$ = cost rate (CHF/s)
 κ = local degree of environmental pollution (mg/m³, ppm)
 λ = excess air ratio
 ϕ = degree of environmental pollution (mg/m³, ppm)
 Π = pressure ratio

ω = water/steam injection ratio

Subscripts and Superscripts

0 = nominal/design conditions or environmental conditions
 c = critical
 j,k = index of equality/inequality constraints
 elec = electric
 ex = exergy
 p = pollution
 prod = product
 q = heat
 r = resource
 HP = high pressure
 IP = intermediate pressure
 IC = intercooling
 LP = low pressure
 RH = reheat

Appendix

Since these configurations were not competitive for the 50 MW cogeneration application shown here, the variable values are not shown in the table. However, in the case of a 300 MW electrical base load power plant discussed in [12], triple pressure steam cycles proved competitive.

Table A1 Base case scenario—most important parameter values (of a total of 23) for the optimized configurations (50 MW, 4000 h, gas price 2 CHcts/kWh⁹)

	1	2	3	4	5	6
CO ₂ sep.	...	...	...	...	...	...
GT type	DC	DC	SC	SC	DC	SC
GT intercool	evap. IC	hex. IC	evap. IC	...	evap. IC	...
NO _x control	dry	dry	dry	dry	dry	dry
Steam cycle type	...	...	...	...	2p	2p
Fast HP turbine	...	...	...	...	...	...
Π	25.7	27.2	32.9	27.6	13.8	14.2
Π_{HP}	1.78	1.75	...	...	1.77	...
T_{fire}	1238	1239	1243	1239	1236	1236
$\Pi_{IC,pseudo}$	0.39	0.62	0.33	...	0.44	...
ω	3.30	2.25	2.72	3.79	0	0
λ	1.85	2.09	1.71	2.23	1.76	1.80
p_{HP} (bar)	...	...	...	...	48.7	48.2
p_{LP} (bar)	...	...	...	...	5.3	6.0
p_{cond} (bar)	...	...	...	...	0.058	0.058
ΔT_{hot} (°C)	...	...	...	...	69	54
ΔT_{min} (°C)	...	...	...	...	16.1	14.5
C (CHcts/kWh exergy)	9.373	9.459	9.449	9.616	9.816	9.849
C (CHcts/kWh heat)	1.464	1.477	1.476	1.502	1.533	1.538

SC: simple combustion
 DC: double (sequential) combustion
 hex IC: heat exchanger compr. intercooling
 evap. IC: evaporative compr. intercooling
 dry: dry low NO_x combustor

Table A2 Environomic sensitivity study on the unit cost of CO₂—most important parameter values (of a total of 23) for optimized configurations (50 MW, 4000 h, gas price 2 CHcts/kWh⁹)

	Base Case	Unit costs of CO ₂ (CHcts/kg)					
		0	1	2	4	5	6
CO ₂ separation	...	...	...	...	...	MEA+recirc.	MEA+recirc.
GT type	DC	DC	DC	DC	DC	DC	DC
Intercooling	evap. IC	evap. IC	evap. IC	evap. IC	hex. IC	hex. IC	hex. IC
NO _x control	dry	dry	dry	dry	dry	dry	dry
Steam cycle type	...	...	...	2p	2p	2p	2p
fast HP turb.	...	...	...	...	...	...	...
Π	25.8	27.2	29.9	13.7	14.6	15.7	15.3
Π_{HP}	1.78	1.79	1.62	1.78	1.84	2.61	2.69
T_{fire}	1238	1074	1093	1249	1269	1271	1276
ω	3.3	3.84	3.60	0	0	0	0
λ	1.85	2.5	2.5	2.5	2.5	1.65	1.63
p_{HP} (bar)	...	...	...	55.6	50.4	52.1	51.9
p_{LP} (bar)	...	...	...	4.0	4.5	4.7	5.3
p_{cond} (bar)	...	...	...	0.052	0.049	0.051	0.052
ΔT_{hot} (°C)	...	...	...	74	65	78	86
ΔT_{min} (°C)	...	...	...	11.4	8.8	10.8	10.4
C (CHcts/kWh)	9.37	9.63	10.78	11.74	13.45	14.22	14.35

⁹MEA+recirc: MEA absorber and exhaust gas recirculation. Other abbreviations: see Table A1 above
 Referred to the lower heating value

References

- [1] Burer, M., Favrat, D., and Pelster, St., 2000, "Modélisation et optimisation thermoéconomique d'une centrale à vapeur existante transformée en une centrale à cycle combine," *Journal GWA* (Swiss Society for Gas and Water Industries), No. 12, pp. 885–890.
- [2] Olsommer, B., 1998, "Development and Application of a Thermo-economic Optimization Method to a Waste Incineration Plant With Cogeneration and a Topping Cycle," Ph.D. thesis, Swiss Federal Institute of Technology, Lausanne.
- [3] Olsommer, B., von Spakovsky, M. R., and Favrat, D., 1997, "An Approach for the Time-Dependent Thermo-economic Modeling and Optimization of Energy System Synthesis, Design and Operation," *Proc. of the Int. Conf. on Thermodynamic Analysis and Improvement of Energy Systems (TAIES'97)*, Beijing, ASME, New York.
- [4] Olsommer, B., von Spakovsky, M. R., and Favrat, D., 1999, "An Approach for the Time-Dependent Thermo-economic Modeling and Optimization of Energy System Synthesis, Design and Operation," (Part I), *Int. J. of Applied Thermodynamics*, 2, No. 3, pp. 97–113.
- [5] Olsommer, B., von Spakovsky, M. R., and Favrat, D., 1999, "An Approach for the Time-Dependent Thermo-economic Modeling and Optimization of Energy System Synthesis, Design and Operation," (Part II), *Int. J. of Applied Thermodynamics*, 2, No. 4, pp. 177–186.
- [6] Frangopoulos, C. A., and von Spakovsky, M. R., 1993, "The Environomic Analysis and Optimization of Energy Systems (Part I)," *Proc. of the Int. Conf. on Energy Systems and Ecology: ENSEC'93*, Vol. I, ASME, New York, pp. 123–132.
- [7] von Spakovsky, M. R., and Frangopoulos, C. A., 1993, "The Environomic Analysis and Optimization of Energy Systems (Part II)," *Proc. of the Int. Conf. on Energy Systems and Ecology: ENSEC'93*, Vol. I, ASME, New York, pp. 123–132.
- [8] von Spakovsky, M. R., and Frangopoulos, C. A., 1994, "The Environomic Analysis and Optimization of a Gas Turbine Cycle with Cogeneration," *Thermodynamics and the Design, Analysis and Improvement of Energy Systems*, AES-Vol.-33, ASME, New York.
- [9] von Spakovsky, M. R., 1998, "The Thermodynamic, Thermo-economic and Environomic Modeling and Optimization of Energy Systems—Discussion and Applications," *Entropie*, No. 205, pp. 12–26.
- [10] Curti V., 1998, "Modélisation et optimisation environomiques de systèmes de chauffage urbain alimentés par pompes à chaleur", Ph.D. thesis No. 1776, Swiss Federal Institute of Technology, Lausanne.
- [11] Curti, V., von Spakovsky, M. R., and Favrat, D., 2000, "An Environomic Approach for the Modeling and Optimization of a District Heating Network based on Centralized and Decentralized Heat Pumps, Cogeneration and/or Gas Furnace (Parts I and II)," *Int. J. Therm. Sci.*, 39, No. 6, pp. 721–741.
- [12] Pelster, St., 1998, "Environomic Modeling and Optimization of Advanced Combined Cycle Cogeneration Power Plants Including CO₂ Separation Options," Ph.D. thesis No. 1791, EPFL, Lausanne, Switzerland.
- [13] Schwaiger, K., 1997, "Vergleich des Kumulierten Energieaufwandes und der Stoffbilanzen von Kraftwerken," *Proceedings on Ganzheitliche Bilanzierung von Energiesystemen*, Düsseldorf, Ap. 16–17, VDI Bericht 1328, VDI Verlag, Düsseldorf.
- [14] Yokoyama, R., and Ito, K., 1995, "Multi-Objective Optimization in Unit Sizing of a Gas Turbine Cogeneration Plant," *ASME J. Eng. Gas Turbines Power*, 117, pp. 53–59.
- [15] Hendriks, C. A., 1994, *Carbon Dioxide Removal From Coal-Fired Power Plants*, Kluwer, Dordrecht.
- [16] Frangopoulos, C. A., 1991, "Comparison of Thermo-economic and Thermodynamic Optimal Designs of a Combined Cycle Plant," *Proc. of the Int. Conf. on the Analysis of Thermal and Energy Systems*, Athens, Greece, June 3–6.
- [17] Boehm, 1987, *Designing of Thermal Systems*, John Wiley and Sons, New York, App. D: Preliminary Cost Estimation Data.
- [18] Foster-Pegg, R. W., 1986, "Capital Cost of Gas-Turbine Heat-Recovery Boilers," *Chem. Eng.*, July, pp. 73–78.
- [19] Kehlhofer, 1991.
- [20] PROGNO AG, 1992, "Identifizierung und Internalisierung externer Kosten der Energieversorgung," Basel Switzerland.
- [21] INFRAS AG, Zürich/PROGNOS AG, Basel, 1994, "Externe Kosten und kalkulatorische Energiepreiszuschläge für den Strom- und Wärmebereich"; Synthesebericht und Teilbericht 3; Bundesamt für Energiewirtschaft, Bundesamt für Konjunkturfragen, Amt für Bundesbauten, Bern.
- [22] Engström, M., 1993, "Energie- und Umweltabgaben in Schweden," *Umweltabgaben in Europa: Konsequenzen für die Schweiz*, BUWAL, eds., Bern.
- [23] Tang, E., 1993, "Energie-/CO₂-Abgaben in Dänemark" *Umweltabgaben in Europa: Konsequenzen für die Schweiz*, BUWAL, eds., Bern, Switzerland.
- [24] Scherp, J., 1993, "CO₂-Energiesteuer: Die Pläne der EG, Umweltabgaben in Europa: Konsequenzen für die Schweiz," BUWAL, eds. Bern.
- [25] Falk-Pedersen, O., and Dannström, H., 1997, "Separation of Carbon Dioxide from Offshore Gas Turbine Exhaust," Kvaerner Water Systems a.s., Sandefjord.
- [26] Wallace, D. R., 1996, "Design Search under Probabilistic Specifications using Genetic Algorithms," *Computer Aided Design*.
- [27] Tomassini, M., 1996, "Evolutionary Algorithms, Towards Evolvable Hardware," *Lecture Notes in Computer Science*, No. 1062, Springer-Verlag, New York.
- [28] Goldberg, D. E., 1989, *Genetic Algorithms in Search, Optimization and Machine Learning*, Addison-Wesley, Reading, MA.
- [29] Mahfoud, S. W., 1995, "A Comparison of Parallel and Sequential Niching Methods," *Proceedings of the Sixth International Conference on Genetic Algorithms*, pp. 136–143.
- [30] Molyneaux, A. K., Olsommer, B., Pelster S., and Curti, V., 1997, "Development and Evaluation of a Modified Version of the Struggle Genetic Algorithm," Internal Report, LENI, EPFL, Lausanne, Switzerland.
- [31] Opaire, 1997.
- [32] Buwal, 1996.

Development of Methanol Steam Reformer for Chemical Recuperation

T. Nakagaki

e-mail: takao.nakagaki@toshiba.co.jp

T. Ogawa

K. Murata

Y. Nakata

Toshiba Corporation
Power and Industrial Systems
Research and Development Center
1, Toshiba-cho, Fuchu-shi,
Tokyo, 183-8511, Japan

The purpose of the present work is to establish the design method of methanol steam-reformer for application to chemical recuperation in a gas turbine system. The reaction rate of the methanol steam-reforming was measured with a small amount of catalyst using the gaseous mixture of methanol, water, hydrogen, and carbon dioxide as a simulated product gas. The reaction rate equation could be expressed by power law of methanol mole fraction and total pressure. The reaction and heat transfer in the catalyst-packed bed was analyzed numerically using the reaction rate equation. The analytical results of temperature distribution and conversion were compared with the experimental results using a reforming tube. These results agreed well except for the region of high methanol conversion. [DOI: 10.1115/1.1359238]

1 Introduction

In recent years, methanol has attracted attention as an oil-alternative energy source in view of its environmental advantages ([1]). Methanol is superior for transportation and storage to natural gas or hydrogen because it is a liquid fuel at atmospheric temperature and pressure. In addition, by steam-reforming with Cu–Zn catalysts, methanol can be converted to hydrogen and carbon dioxide at 450–600 K, that is, at lower temperature than is required for conversion of methane. When methanol is utilized as a fuel in gas turbine power plants, conversion of methanol into hydrogen-rich fuel using exhaust heat from the gas turbine enables the generating efficiency to be higher than direct combustion ([2]). This is especially remarkable in the simple gas turbine cycle.

A methanol steam reformer is one of the most important components in such a chemically recuperated gas turbine system, and it is essential to predict the reformer performance in practical conditions from the fundamental tests on catalyst performance. Indeed many works on methanol steam reformers have been published. Amphlette et al. have presented experimental data of commercial catalysts, Girdler G66B and CCI C18HC ([3,4]). Jiang et al. have shown the reaction rate equations of BASF S3-85 catalyst, Harshow's catalyst and the catalysts prepared in the laboratory ([5,6]). Idem et al. have described the reaction kinetics of Cu–Al catalysts and shown their reaction rate equations ([7]). But almost all these experiments have been conducted in microreactor at atmospheric pressure using crushed and sifted catalysts. Fujimoto et al. or Thérien et al. have compared the experimental data which were obtained using a catalyst-packed bed reformer, and the analytical data on the temperature distribution and methanol conversion ([8,9]). Imai et al. and Nagaya et al. have reported on the practical tests that were conducted at 1.5 MPa ([10,11]). However, the design methods of practical reformers are not shown. A number of analytical studies of reformer have been reported. Murray et al. or Alatiqi et al. have reported analytical results concerning methane steam reformer ([12,13]).

The reaction rate of reforming catalysts at pressurized condition and analysis of heat transfer kinetics are needed in order to design or analyze the reformer. The purpose of this work is to establish a design method of methanol reformers for chemically recuperated gas turbine systems. First, the tests for evaluating the reaction

rates were conducted with commercial Cu–Zn catalyst pellets and the reaction rate equation was determined. Next, the reforming performance was examined by the reactor-tube tests using a single tube in which the catalysts were packed 300 mm in length.

2 Experimental Apparatus and Procedures

Figure 1 shows a flow diagram of the experimental apparatus. The mixture gases of methanol, water (as material), hydrogen and carbon dioxide (as product gas) were supplied into the reforming section in order to simulate the local reformed gas in a practical reforming tube, and the reforming performance was examined.

The pure methanol and ion exchanged water were blended in steam-carbon ratio 1.0 (methanol/water molar ratio=1:1, abbreviated to S/C), and were fed into the evaporator by a liquid pump. The hydrogen and carbon dioxide gases were mixed with methanol and water in the evaporator and they were heated to a fixed temperature in the superheater. The reforming section was composed of a $\phi 23$ mm $\times$ L500 mm stainless steel tube and sheathed heaters. The sheathed heaters are controlled separately every 100 mm tube length. The reforming pressure was kept at fixed values by a pressure-regulating valve at the exit of the reforming section. Only the gas components separated at the cooling section were led to the measuring section, where the gas flow rate and the gas composition were measured by an accumulating flow meter and a TCD gas chromatograph, respectively.

The catalysts used in the present work were the commercial catalysts, Nissan Girdler G66-B (Cu–Zn alumina support, 1/8 in.–1/8 in. tablets). They were reduced by a mixture gas of hydrogen and nitrogen before the reforming tests.

The following two tests were conducted in the present work.

1 Tests for evaluating the reaction rates

As shown in Fig. 1, 2.0 g catalysts were packed in the reforming tube, and the reaction rates were measured by feeding the simulated gas. The compositions of the simulated gas were arranged so as to be equivalent to the reformed gas for the methanol conversion of 25, 50, and 75 percent. In this report, the reaction rate, r_M , denotes the quantity of methanol converted per second and per catalyst weight

2 Tests with a reforming tube

The catalyst was packed into the 300 mm long and the temperature distribution through the center of tube cross section and the outlet conversion, X_o , were measured. The outlet conversion, X_o ,

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 Power Division, July 2000; final revision received by the ASME Headquarters January 11, 2000. Editor: D. Lou.

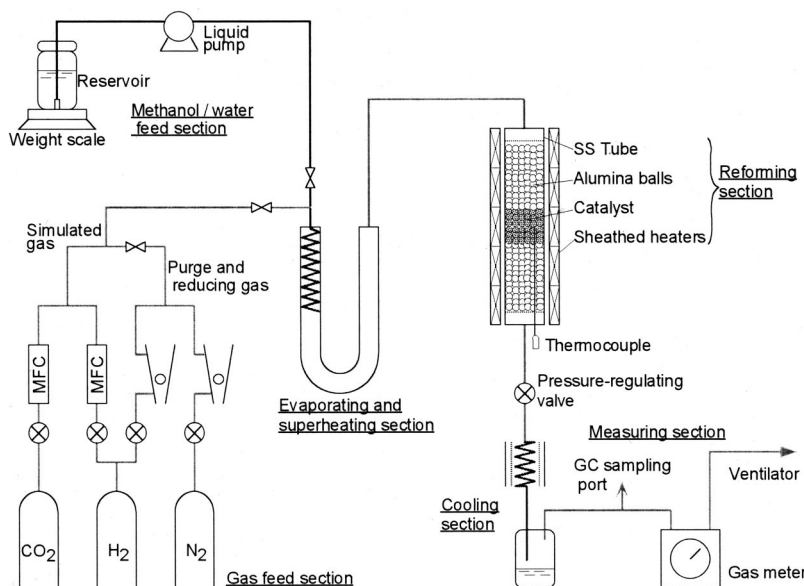


Fig. 1 Experimental apparatus

Table 1 Summary of parameters

Test No.	1	2
Total pressure, atm	1, 5, 10, 15	1, 10
Catalyst temperature, K	473, 498, 513, 538	—
Wall temperature, K	—	518, 548
Mass flux, kg/(sm ²)	0.25	0.25, 0.36
Simulated conversion, percent	0, 10, 20, 50, 75	0, 80

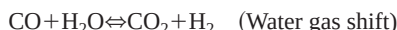
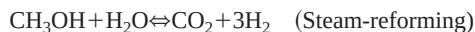
was estimated using Eq. (1) from the feed rate of methanol and water, and the flow rate and composition of product gas.

$$X_o = \frac{(\text{Product gas flow rate}) \times (\text{Mole fraction of CO and CO}_2)}{(\text{Fed liquid quantity}) / (1 + S/C)} \quad (1)$$

Parameters in these two tests are summarized in Table 1.

3 Results and Discussion

Tests for evaluating the reaction rate. It seems that although the following reactions occur in a methanol steam reformer, the steam-reforming reaction is particularly significant at relatively low temperature.



First, the results of tests using a mixture vapor of methanol and water as a reaction gas are shown and discussed in Figs. 2 and 3. In order to examine the influence of mass flux on reaction rate, the reaction rate was measured at a constant temperature and various mass fluxes. Figure 2 shows the relation between the reaction rate, r_M , and the mass flux, G , at $T=513$ K, $P=1$ and 10 atm. As this figure shows, r_M can be regarded as constant for $G > 0.14$ kg/(sm²), but is reduced for $G < 0.14$ kg/(sm²). It is considered that diffusion resistance in gas phase is not negligible when mass flux of the reforming gas becomes lower. Thus, all tests in the present work were conducted for $G > 0.25$ kg/(sm²) because r_M is not dependent on mass flux of reforming gas for $G > 0.14$ kg/(sm²). Besides these, CO concentration did not exceed 1 percent extremely and the molar ratio between hydrogen

and carbon dioxide was almost 77–75/23–25 within pressure and temperature ranges of the present work. Therefore, the influence of CO was neglected and the simulated gas did not contain CO.

Figure 3 shows the relation between reaction rate and total pressure in the form of logarithms. The reaction rate r_M becomes lower when pressure becomes higher, but the tendency does not depend on temperature. This is because the steam-reforming is a reaction where the total molar number increases. As the relation between r_M and P is shown as a straight line in Fig. 4, the reaction rate r_M is expressed by power law of total pressure P .

Figure 4 shows the effect of temperature on reaction rate in the form $\log(r_M)$ versus $1/T$ (Arrhenius plot). For $T=473$ – 513 K, $\log(r_M)$ is almost in proportion to $1/T$ and represented as a straight line in Fig. 4. For $T=513$ – 538 K, however, $\log(r_M)$ is lower than the extrapolated value from the straight line, and the apparent activation energy is smaller. It seems that the influence of the diffusion resistance in micropores of catalyst particles on the reaction rate becomes more significant as the temperature becomes higher. Figure 4 implies that the dependence of r_M on temperature is expressed for $T < 513$ K by Arrhenius's law, but it requires revision for $T > 513$ K. The activation energy, E , was estimated to

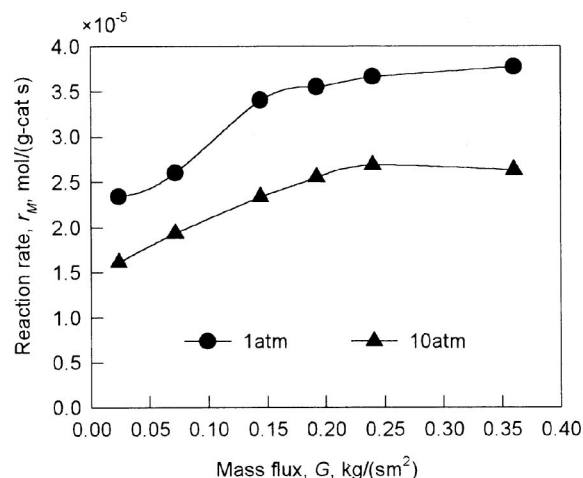


Fig. 2 Influence of flow rate on reaction rate at $T=513$ K and $X_{in}=0$ percent

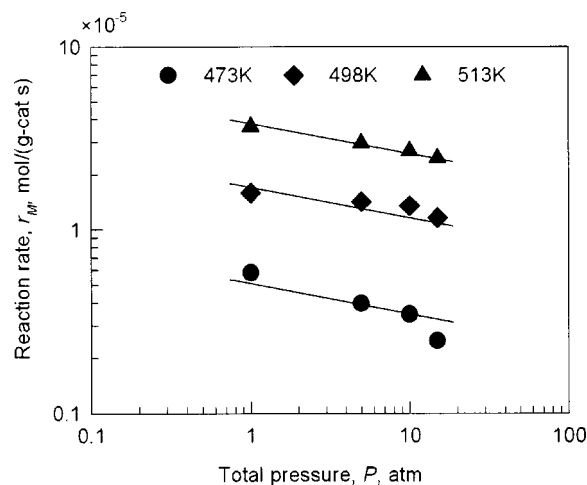


Fig. 3 Dependence of reaction rate on total pressure at $G = 0.25 \text{ kg/(sm}^2\text{)}$ and $X_{in} = 0$ percent

be about 100 kJ/mol for $T = 473\text{--}513 \text{ K}$, which is comparable to the value reported by Jiang et al. using Cu–Zn catalyst (105 kJ/mol) ([14]).

Next, the results of tests conducted with the mixture gases of hydrogen, carbon dioxide, and the methanol and water vapor are described. Figure 5 shows the effects of methanol mole fraction $c_M = P_M/P$ on reaction rate, r_M . The relation between $\log(r_M)$ and $\log(c_M)$ is linear, and the gradients are not influenced by total pressure. Therefore, r_M seems to be expressed by power law of methanol mole fraction c_M . The reaction rate equation is often expressed by power law of the compositions in the reforming gas and the operating pressure and temperature. From the results of these tests, the reaction rate equation was expressed by Eq. (2).

$$r_M = k_0 P^l \left(\frac{T}{513} \right)^m e^{E/RT} c_M^n \quad (2)$$

where $k_0 = 1.35 \times 10^6 \text{ mol/(g-cat s atm}^1\text{)}$, $E = 1.0 \times 10^5 \text{ J/mol}$, $l = -0.13$, $n = 1.3$, $m = -10$ ($T < 513 \text{ K}$, $m = 0$). In this report, S/C is constant ($S/C = 1.0$) and the effects of methanol and water concentrations in reforming gases are not examined individually. We will conduct the tests with the S/C value varied and revise the reaction rate equation if it is necessary to do so.

Figure 6 shows the results at $P = 1$ and 10 atm as relation between reaction rate r_M and conversion X . In this figure, the

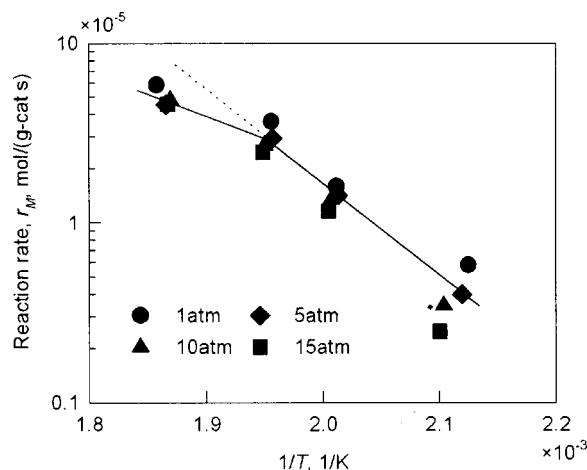


Fig. 4 Effect of temperature on reaction rate at $G = 0.25 \text{ kg/(sm}^2\text{)}$ and $X_{in} = 0$ percent

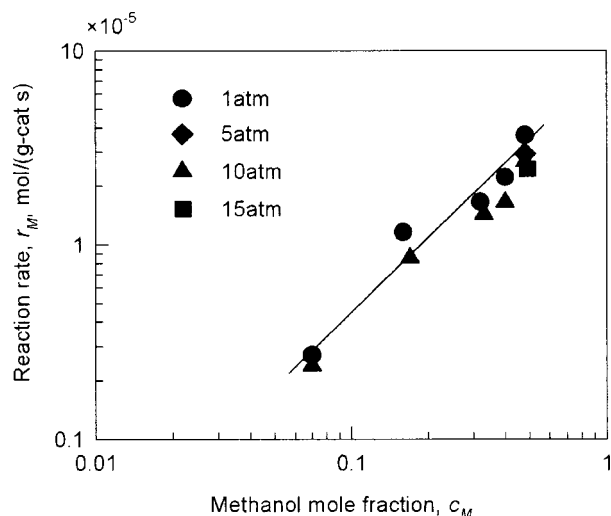


Fig. 5 Correlation between reaction rate and methanol mole fraction at $G = 0.25 \text{ kg/(sm}^2\text{)}$, $T = 513 \text{ K}$ and $X_{in} = 0$ percent

results estimated by Eq. (2) are also shown. As this figure shows, the reaction rate decreases immediately when the conversion increases. The calculated reaction rates agree well with experimental results regardless of total pressure, temperature, and conversion.

Tests with a reforming tube. The outlet conversion X_o and temperature distribution along the center axis of tube cross section using a mixture vapor of methanol and water ($X_{in} = 0$) are shown in Fig. 7, 8, and 9. In these figures, equilibrium conversion at wall temperature and analytical results mentioned in the following section are also shown. Figure 7 shows outlet conversion X_o and temperature distribution for wall temperature $T_w = 548 \text{ K}$, gas inlet temperature $T_{in} = 518 \text{ K}$, and mass flux $G = 0.25 \text{ kg/(sm}^2\text{)}$. The parameter is total pressure, $P = 1$ and 10 atm. The following analytical results are also shown in Fig. 7. Both outlet conversion and the difference between the lowest temperature around 50 mm from inlet and wall temperature decrease when total pressure becomes high. It is considered that reaction rate becomes lower when total pressure becomes high, as shown in Fig. 3. However, the difference of X_o with total pressure is not so large as the variance of reaction rate with total pressure shown in Fig. 7. Be-

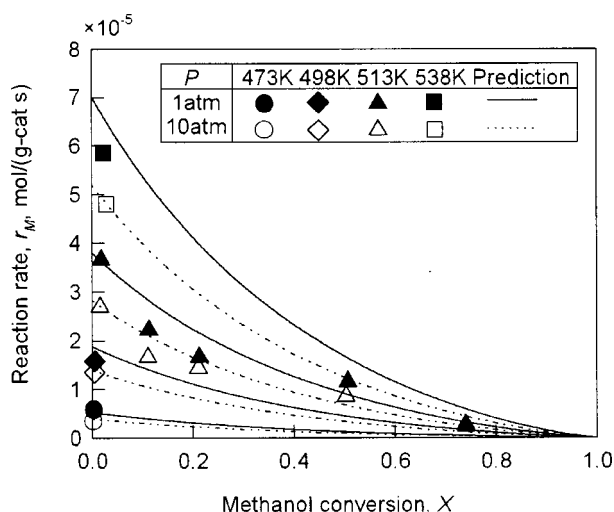


Fig. 6 Relation between reaction rate and methanol conversion at $G = 0.25 \text{ kg/(sm}^2\text{)}$

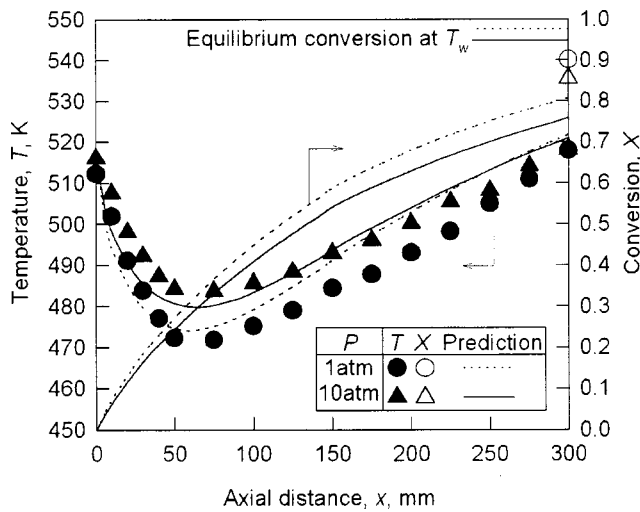


Fig. 7 Effect of pressure on temperature and conversion profiles at $T_w=548$ K, $T_{in}=518$ K, $X_{in}=0$ percent and $G=0.25$ kg/(sm²)

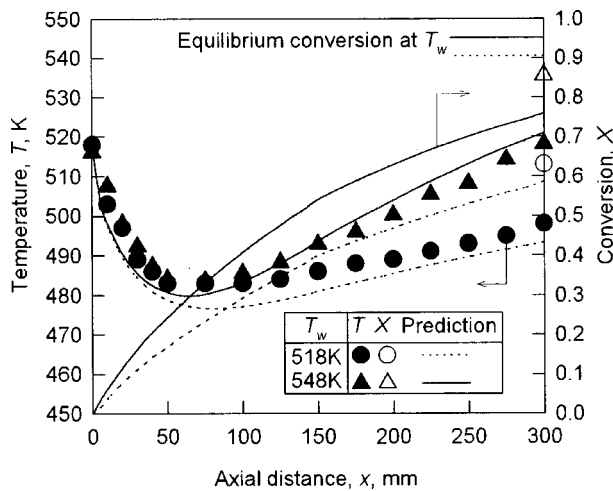


Fig. 8 Effect of T_w on temperature and conversion profiles at $P=10$ atm, $T_{in}=518$ K, $X_{in}=0$ percent, and $G=0.25$ kg/(sm²)

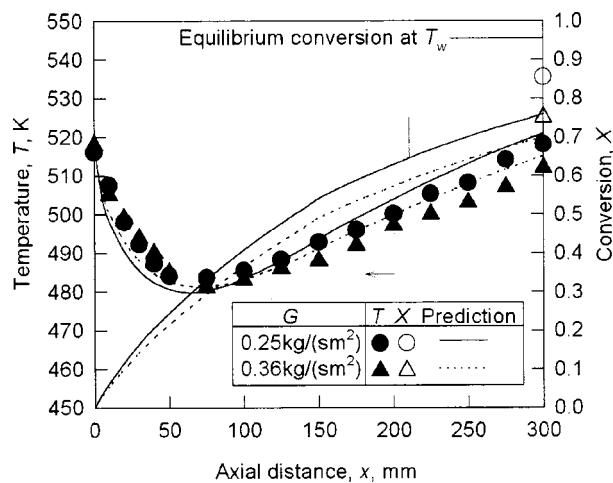


Fig. 9 Effect of mass flux on temperature and conversion profiles at $P=10$ atm, $T_w=548$ K, $T_{in}=518$ K and $X_{in}=0$ percent

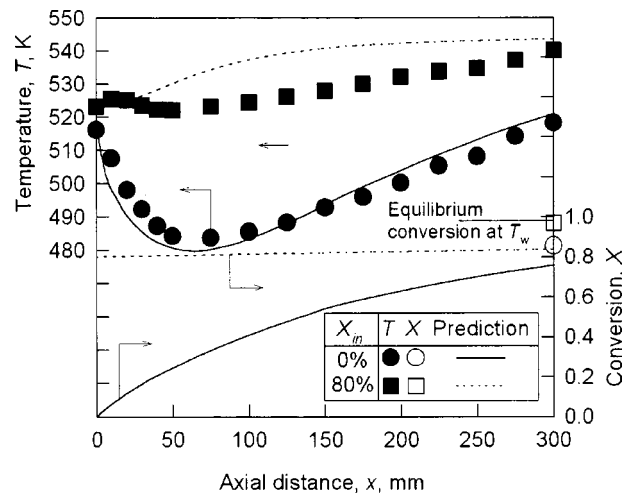


Fig. 10 Effect of X_{in} on temperature and conversion profiles at $P=10$ atm, $T_w=548$ K, $T_{in}=518$ K and $G=0.25$ kg/(sm²)

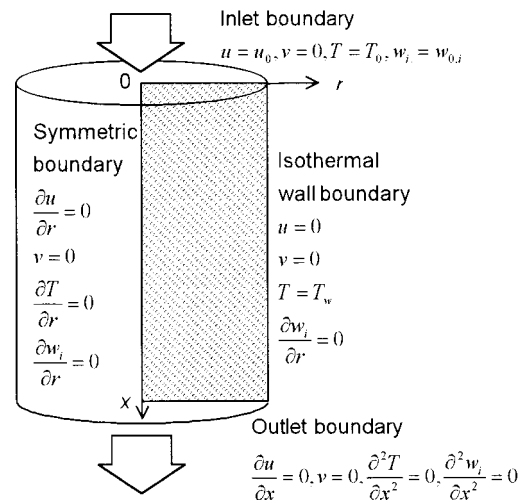


Fig. 11 Schematic illustration of geometry, boundary conditions, and coordinate system

cause drops in the reaction rate and temperature around the inlet are lower when total pressure becomes higher, the reaction rate decreases relatively little.

Figure 8 shows the results for $P=10$ atm, $T_{in}=518$ K and $G=0.25$ kg/(sm²). The parameter is wall temperature, $T_w=518$ and 548 K. The difference of X_0 between $T_w=518$ and 548 K is no less than 20 percent. It is considered that temperature influences reaction rate remarkably.

Figure 9 shows the results for $P=10$ atm, $T_{in}=518$ K and $T_w=548$ K. The parameter is mass flux, $G=0.25$ and 0.36 kg/(sm²). As shown in Fig. 9, the lowest temperature appears around 50 mm from inlet, but temperature does not increase toward outlet in the case of $G=0.36$ kg/(sm²). The temperature distribution seems not to be influenced by mass flux until the lowest temperature because the change of reaction rate at $G=0.3$ kg/(sm²) is the same as at $G=0.25$ kg/(sm²). But the temperature in the case of $G=0.36$ kg/(sm²) does not increase so much in the lower reaches because the remaining methanol is more than ($i=0.25$ kg/(sm²)).

Figure 10 shows the result using a mixture gas blended with methanol, water, hydrogen, and carbon dioxide. The composition of simulated gas is arranged so as to be equivalent to the reformed

gas for methanol conversion $X_{in}=80$ percent (mole fraction $\text{CH}_3\text{OH}/\text{H}_2\text{O}/\text{H}_2/\text{CO}_2=5.6/5.6/22.2/66.6$). The conditions of Fig. 11 are $P=10$ atm, $T_{in}=518$ K, $T_w=548$ K, and $G=0.25$ kg/(sm²). Figure 10 also shows the result for $X_{in}=0$ percent. The temperature in the case of $X_{in}=80$ percent is a little down at around 40 mm from inlet and does not change so much as $X_{in}=0$ percent. For this reason, the reaction rate above $X=80$ percent seems to be very small. The outlet conversion X_o is about 95 percent and the reaction seemed to reach equilibrium in the 300 mm tube.

4 Numerical analysis

Governing equations and analytical procedure. In order to simulate concentration of each gas component, velocity and temperature profile, the two-dimensional numerical analysis was conducted. Streamwise velocity; u , radial velocity; v , pressure; p , density; ρ , temperature; T , methanol mass fraction; w_M , hydrogen mass fraction; w_{H_2} and water mass fraction; w_{H_2O} (any three of four components are available) are defined as independent variables. Governing equations are shown as follows ([15]). Also, the coordinate system and boundary conditions in this analysis are shown in Fig. 11. As mentioned above, CO concentration is very small and neglected in this analysis.

Continuity

$$\frac{\partial(\rho ru)}{\partial x} + \frac{\partial(\rho rv)}{\partial r} = 0 \quad (3)$$

Momentum (modified Darcian equation)

$$\begin{aligned} \frac{\partial}{\partial x} \left(r \frac{\rho uu}{\varepsilon} \right) + \frac{\partial}{\partial r} \left(r \frac{\rho uv}{\varepsilon} \right) = & -r \frac{\partial p}{\partial x} \varepsilon - r \frac{\mu}{K} u \varepsilon - r \rho C \sqrt{u^2 + v^2} u \varepsilon \\ & + r \rho g \varepsilon + \frac{\partial}{\partial x} \left(r \varepsilon \mu \frac{\partial}{\partial x} \left(\frac{u}{\varepsilon} \right) \right) \\ & + \frac{\partial}{\partial r} \left(r \varepsilon \mu \frac{\partial}{\partial r} \left(\frac{u}{\varepsilon} \right) \right) + \frac{\partial}{\partial x} \left(r \varepsilon \mu \frac{\partial}{\partial x} \left(\frac{v}{\varepsilon} \right) \right) \\ & + \frac{\partial}{\partial r} \left(r \varepsilon \mu \frac{\partial}{\partial r} \left(\frac{v}{\varepsilon} \right) \right) \end{aligned} \quad (4)$$

$$\begin{aligned} \frac{\partial}{\partial x} \left(r \frac{\rho vu}{\varepsilon} \right) + \frac{\partial}{\partial r} \left(r \frac{\rho vv}{\varepsilon} \right) = & -r \frac{\partial p}{\partial r} \varepsilon - r \frac{\mu}{K} v \varepsilon - r \rho C \sqrt{u^2 + v^2} v \varepsilon \\ & + \frac{\partial}{\partial x} \left(r \varepsilon \mu \frac{\partial}{\partial x} \left(\frac{v}{\varepsilon} \right) \right) \\ & + \frac{\partial}{\partial r} \left(r \varepsilon \mu \frac{\partial}{\partial r} \left(\frac{v}{\varepsilon} \right) \right) + \frac{\partial}{\partial x} \left(r \varepsilon \mu \frac{\partial}{\partial x} \left(\frac{u}{\varepsilon} \right) \right) \\ & + \frac{\partial}{\partial r} \left(r \varepsilon \mu \frac{\partial}{\partial r} \left(\frac{u}{\varepsilon} \right) \right) \end{aligned} \quad (5)$$

Energy

$$\frac{\partial(r\rho uh)}{\partial x} + \frac{\partial(r\rho vh)}{\partial r} = \frac{\partial}{\partial x} \left(r \lambda_e \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial r} \left(r \lambda_e \frac{\partial T}{\partial r} \right) \quad (6)$$

Component i

$$\frac{\partial(r\rho u w_i)}{\partial x} + \frac{\partial(r\rho v w_i)}{\partial r} = \frac{\partial}{\partial x} \left(r \rho D_e \frac{\partial w_i}{\partial x} \right) + \frac{\partial}{\partial r} \left(r \rho D_e \frac{\partial w_i}{\partial r} \right) + r R_i \quad (7)$$

$$i = M, \text{ H}_2\text{O}, \text{ H}_2$$

$$w_{\text{CO}_2} = 1 - \sum_i w_i$$

Gas state

$$P = \frac{\rho}{\sum_i w_i m_i} R T \quad (8)$$

where R_i is source term of component i and estimated product of r_M calculated by Eq. (2) and bulk density. K and C are expressed as Eqs. (9) and (10) by Ergun [16].

$$K = \frac{d_s^2 \varepsilon^3}{150(1 - \varepsilon)^2} \quad (9)$$

$$C = \frac{1.75(1 - \varepsilon)}{d_s \varepsilon^3} \quad (10)$$

λ_e and D_e are given by Eqs. (11) and (12), respectively ([17]).

$$\frac{\lambda_e}{\lambda_g} = \frac{\lambda_e^0}{\lambda_g} + \gamma_\lambda \text{Re}_s \text{Pr} \quad (11)$$

$$\frac{D_e}{D_g^0} = \gamma_D \text{Re}_s \text{Sc} \quad (12)$$

The enthalpy h in Eq. (6) is determined by gas composition and temperature in Eq. (13).

$$h = \sum_i w_i \int C_{p_i} dT + \sum_i w_i h_{0,i} \quad i = M, \text{ H}_2\text{O}, \text{ H}_2, \text{ CO}_2 \quad (13)$$

Equations (3)–(8) are rearranged to difference equations by the control volume method and converted into nondimensional form and solved for u , v , P , ρ , T , and w_i numerically.

The methanol conversion X of analysis is calculated by Eq. (14)

$$X = 1 - \frac{\int_0^{d_t/2} u 2\pi r \rho w_M dr}{\frac{\pi d_t^2}{4} u_0 \rho_0 w_{0,M}} \quad (14)$$

Analysis results. Figure 12 shows profiles of temperature and methanol mole fraction in the whole region for the case shown in Fig. 8. The analytical conditions are $T_w=548$ K, $T_{in}=518$ K, $G=0.25$ kg/(sm²), and $P=10$ atm. As this figure shows, temperature is not uniform in the region and the maximum difference is about 65 K. The lowest temperature is located on symmetric axis at around 50 mm from inlet boundary, and it increases radially. The change of methanol mole fraction along r coordinate is relatively small, it is 8 percent maximum around $x=50$ mm and it decreases along the downstream. For this reason, it is considered that high wall temperature causes much methanol conversion, but decrease of methanol causes reduction of reaction rate. And it is also considered that reaction rate increases with temperature after $x=50$ mm. In the case of catalyst-packed bed, inertial, and viscosity term in Eqs. (4) and (5) are negligibly small. Therefore, radial velocity v is much lower than streamwise velocity u . The streamwise velocity u varies with gas density but it is approximately uniform in a cross section. In the present work, the graphs about velocity are omitted.

Figures 7–10 show analysis results with experimental results. In the case of $X_{in}=0$, it is clear that predicted T and X_o is a little

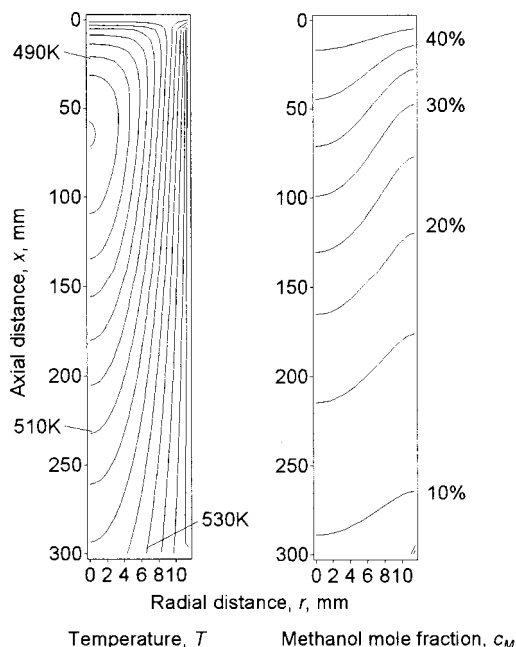


Fig. 12 Temperature and methanol mole fraction profile at $P = 10$ atm, $T_w = 548$ K, $T_{in} = 518$ K, $G = 0.25$ kg/(sm²) and $X_{in} = 0$ percent

different from experimental results, but they agree approximately. However, Fig. 11 shows that the predicted temperature does not become lower, but rises gradually from the inlet to outlet. The predicted conversion also does not change so much and the predicted conversion at outlet is smaller than experimental result. For this reason, the estimated reaction rate seems to be smaller when the conversion X is above 80 percent. The reaction rate at low methanol concentration (methanol conversion X is high) needs further studies.

5 Conclusions

In the present work, the reaction rate of the methanol steam-reforming was measured with a small amount of catalyst using the gaseous mixture of methanol, water, hydrogen, and carbon dioxide as a simulated product gas. Also, the reaction and heat transfer in the catalyst-packed bed was analyzed numerically using the reaction rate equation. The analysis was compared with the experimental results using a reforming tube of 300 mm in length. The summary is as follows:

- 1 The reaction rate equation was expressed by the power law of total pressure and methanol mole fraction, Eq. (2).
- 2 The analytical results using the reaction rate equation agreed well with the experimental results using a reforming tube in the region of $X_o \leq 80$ percent, but the prediction was not adequate for $X_o > 80$ percent.

In order to predict the reformer performance in the region of $X_o > 80$ percent adequately, the reaction rate needs further studies.

Nomenclature

- C = inertial coefficient, m⁻¹
 c_M = methanol mole fraction
 C_p = specific heat, J/(kg K)
 D_e = effective diffusivity, m²/s
 D_g^0 = stagnant diffusivity, m²/s
 d_s = equivalent diameter of catalyst particle, m
 d_t = inner diameter of tube, m
 E = activation energy, J/mol

- G = mass flux, kg/(sm²)
 g = gravity, 9.8 m/s²
 h = enthalpy, J/kg
 K = permeability, m²
 k_0 = reaction rate constant, mol/(g-cat s atm¹)
 m_i = molecular weight of component i
 P = (total) pressure, atm
 p = pressure, Pa ($P = p/101,325$)
 Pr = Prandtl number
 R = gas constant, 8.3143 J/(mol K)
 R_i = reaction rate of component i , kmol/(sm³)
 Re_s = Reynolds number, defined with d_s
 r = radial coordinate, m
 r_M = reaction rate, mol/(g-cat s)
 Sc = Schmidt number ($= \mu/\rho D$)
 T = temperature, K
 u = streamwise velocity, m/s
 u_0 = inlet velocity, m/s
 v = radial velocity, m/s
 w_i = mass fraction of component i
 $w_{0,M}$ = inlet mass fraction of methanol
 X = methanol conversion
 X_{in} = inlet methanol conversion
 x = axial coordinate, m

Greek Letters

- γ_D = coefficient in dispersion diffusivity
 γ_λ = coefficient in dispersion conductivity
 ε = porosity
 λ_e = effective conductivity, W/(m K)
 λ_g = gas conductivity, W/(m K)
 λ_s = solid (catalyst particle) conductivity, W/(m K)
 λ_e^0 = stagnant effective conductivity, W/(m K)
 μ = viscosity, Pa s
 ρ = density, kg/m³
 ρ_0 = inlet density, kg/m³

Superscripts and Subscripts

- e = effective
 g = gas
 i = CH₃OH(M), H₂O, H₂, CO₂
 in = inlet
 M = methanol
 o = outlet
 s = solid (catalyst particle)
 t = tube
 w = wall
 0 = inlet, stagnant, specific

References

- [1] Ogawa, T., Hori, M., Ohashi, Y., Murata, K., Fukuda, M., and Tamaura, Y., 1998, "Environment-Friendly Energy System," *Proc.*, 33rd IECEC, No. 179.
- [2] Carapellucci, R., Risalvato, V., Bruno, C., and Cau, G., 1996, "Performance and Emissions of CRGT Power Generation Systems with Reformed Methanol," *Proc.*, IECEC 96, Vol. 2, pp. 707–712.
- [3] Amphlett, J. C., Mann, R. F., and Weir, R. D., 1985, "Hydrogen Production by the Catalytic Steam Reforming of Methanol. Part 2: Kinetics of Methanol Decomposition Using Girdler G66B Catalyst," *Can. J. Chem. Eng.*, **63**, pp. 605–611.
- [4] Amphlett, J. C., Mann, R. F., and Weir, R. D., 1988, "Hydrogen Production by the Catalytic Steam Reforming of Methanol. Part 3: Kinetics of Methanol Decomposition Using C18HC Catalyst," *Can. J. Chem. Eng.*, **66**, pp. 950–956.
- [5] Jiang, C. J., Trimm, D. L., and Wainwright, M. S., 1993, "Kinetic mechanism for reaction between methanol and water over a Cu-ZnO-Al₂O₃ catalyst," *Appl. Catal.*, **97**, pp. 145–158.
- [6] Jiang, C. J., Trimm, D. L., and Wainwright, M. S., 1993, "Kinetic study of steam reforming of methanol over copper-based catalysts," *Appl. Catal.*, **93**, pp. 245–255.
- [7] Idem, R. O., and Bakhshi, N. N., 1996, "Kinetic Modeling of the Production of Hydrogen from the Methanol-steam Reforming Process over Mn-promoted Coprecipitated Cu-Al Catalyst," *Chem. Eng. Sci.*, **51**, No. 14, pp. 3697–3708.
- [8] Fujimoto, S., Ishihara, H., and Tsuruno, S., 1987, "A study on a steam reformer for methanol," *JSME Int. J.*, **30**, No. 267, pp. 1437–1442.

- [9] Thérien, N., and Tessier, P., 1987, "Modélisation et simulation de la décomposition catalytique du méthanol dans un réacteur à lit fixe," *Can. J. Chem. Eng.*, **65**, pp. 950–957.
- [10] Kuroda, K., Imai, T., and Yanagi, M., 1989, "Development of Methanol Reformer," *Mitsubishi Jukon Gihou*, **26**, No. 5, pp. 408–412.
- [11] Nagaya, K., Oshiro, H., Ito, H., and Yoshino, N., 1993, "Demonstration Test of Reformed Methanol Gas-Turbine Power Generation," *Hitachi Zousen Gihou*, **54**, No. 3, pp. 180–184.
- [12] Murray, A. P., and Snyder, T. S., 1985, "Steam-Methane Reformer Kinetic Computer Model with Heat Transfer and Geometry Options," *Ind. Eng. Chem. Process Des. Dev.*, **24**, pp. 286–294.
- [13] Alatiq, I. M., Meziou, A. M., and Gasmelseed, G. A., 1989, "Modeling, Simulation and Sensitivity Analysis of Steam-Methane Reformers," *Int. J. Hydrogen Energy*, **14**, pp. 241–256.
- [14] Jiang, C. J., Trimm, D. L., and Wainwright, M. S., 1993, "Kinetic mechanism for the reaction between methanol and water over $\text{Cu-ZnO-Al}_2\text{O}_3$ catalyst," *Appl. Catal.*, **97**, pp. 145–158.
- [15] Tien, C. L., and Hunt, M. L., 1987, "Boundary-Layer Flow and Heat Transfer in Porous Beds," *Chem. Eng. Prog.*, **21**, pp. 53–63.
- [16] Ergun, S., 1952, "Fluid Flow Through Packed Columns," *Chem. Eng. Prog.*, **48**, pp. 89–94.
- [17] Hunt, M. L., and Tien, C. L., 1988, "Non-Darcian Convection in Cylindrical Packed Beds," *Trans. ASME*, **110**, pp. 378–378.

K. Maunsbach

A. Isaksson

J. Yan

G. Svedberg

Department of Chemical Engineering
and Technology/Energy Processes,
Royal Institute of Technology,
Stockholm, Sweden

L. Eidensten

Vattenfall Utveckling AB,
Stockholm, Sweden

Integration of Advanced Gas Turbines in Pulp and Paper Mills for Increased Power Generation

The pulp and paper industry handles large amounts of energy and today produces the steam needed for the process and some of the required electricity. Several studies have shown that black liquor gasification and combined cycles increase the power production significantly compared to the traditional processes used today. It is of interest to investigate the performance when advanced gas turbines are integrated with next-generation pulp and paper mills. The present study focused on comparing the combined cycle with the integration of advanced gas turbines such as steam injected gas turbine (STIG) and evaporative gas turbine (EvGT) in pulp and paper mills. Two categories of simulations have been performed: (1) comparison of gasification of both black liquor and biomass connected to either a combined cycle or steam injected gas turbine with a heat recovery steam generator; (2) externally fired gas turbine in combination with the traditional recovery boiler. The energy demand of the pulp and paper mills is satisfied in all cases and the possibility to deliver a power surplus for external use is verified. The study investigates new system combinations of applications for advanced gas turbines.

[DOI: 10.1115/1.1359773]

Introduction

In the Swedish pulp and paper industry today, only 20 percent of the power demand is internally generated and internal fuels represent 74 percent of the total fuel consumption. Additional power and fossil fuels are therefore needed to satisfy the energy demand ([1]). While recovering important chemicals by burning black liquor, steam and power are produced at the same time. The traditional technology for recovery is the Tomlinson recovery boiler, and bark is normally fired in a bark boiler. In the future, the improvement of the pulp and paper process by, for example, an increased closure of the process, will result in a reduction of the process steam demand. This will give an opportunity to increase the power generation for both internal and external use.

Several studies have shown that replacing the conventional Tomlinson recovery boiler with black liquor gasification and a combined cycle increases the power production significantly ([2–6]). In addition to the combined cycle, another interesting alternative is to use a gas turbine with water/steam addition without a steam turbine. The present study has focused on comparing the integration of a combined cycle with the integration of advanced gas turbines such as the STIG (Steam Injected Gas Turbine) cycle connected to a process steam generator and the EvGT (Evaporative Gas Turbine) also called HAT (Humid Air Turbine) cycle. These cycles use steam injection and water evaporation respectively.

The study was performed as a part of a Swedish research program called “The Ecocyclic Pulp Mill.” The aim of the program is to develop realistic and environmentally optimal solutions for the production of high-quality pulp and paper products which minimize or eliminate the use of non-renewable resources and efficiently utilize the energy potential of biomass.

The studied gas turbine processes are connected to reference pulp and paper mills. The reference kraft pulp mill is defined by

the research program “The Ecocyclic Pulp Mill”. The integrated pulp and paper mill is based on the average mill of 1994 and adjusted to the reference kraft pulp mill.

Advanced gas turbine cycles have been built in pulp and paper mills fired with premium fuels as for example natural gas. These systems have been developed for very practical reasons, namely to ensure availability of reliable, cost-efficient electric power ([7]). The gasification technology or external firing with heat transfer through a heat exchanger creates the opportunity to fire efficient gas turbines with internal fuels in pulp and paper mills, eliminating the need for fossil fuels.

Advanced Gas Turbine Cycles

Gas turbines with water/steam addition without a steam turbine is an interesting alternative to the combined cycle for integration in pulp and paper mills. The two different kinds of advanced gas turbines applied in the present study, the STIG cycle and the EvGT cycle, are described below. Some previous work and status on these systems are also discussed.

STIG Cycle and Its Application in Pulp and Paper Industry. In a steam injected gas turbine, heat in the exhaust gas is used for generation of steam which is injected into the combustion chamber as shown in Fig. 1. Complementary injection can also be made downstream the combustion chamber. Steam injection in a gas turbine can be used to increase the net power efficiency and also to reduce the NO_x emission ([8]).

Steam injection into gas turbines for power generation has been proven to provide exceptional flexibility of operation in cogeneration applications ([9]). The part load performance of the STIG cycle deteriorates at a slower rate with reduced load when compared to the combined cycle. This could be a valuable characteristic in connection to the pulp and paper industry with fluctuations in steam demand caused by, for example, one of several pulping lines closed ([10]).

Several steam injected gas turbines were built during the middle of the 1980s. The STIG cycle has since then been commercially available from a few companies in several sizes, for example GE LM2500 and GE LM5000 (26.4 and 51 MW) and Allison 501KH (5.3 MW) ([9]).

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-020. Manuscript received by IGTI Feb. 2000; final revision received by ASME Headquarters Jan. 2001. Associate Editor: M. Magnolet.

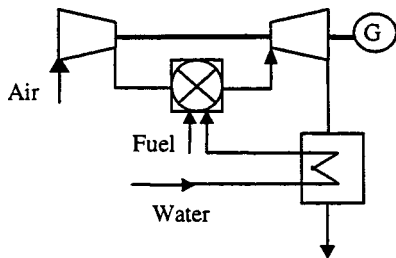


Fig. 1 A simple steam injected gas turbine (STIG)

Many of the natural gas fired gas turbines applied in the paper industry during the 80's were aero-derivative gas turbines such as GE's LM2500 and LM5000, because of high efficiency and availability. During 1985, the Simpson Paper Co. in California USA retrofitted an existing LM 2500 with partial steam injection, which increased the net power efficiency from 37 percent to 41 percent with a turbine reliability of 99.5 percent. Based on this, Simpson installed a full LM 2500 STIG cogeneration facility at another Californian mill during 1987 ([11,12]).

The Steyrermühl Paper Mill in Austria built a combined cycle plant during 1989–1990 provided with a steam injection system to reduce the NO_x emissions to required levels. The downstream heat recovery system was especially adapted to meet the requirements of the paper mill by using a supplemental firing system with a possibility to increase the steam production by 80 percent beyond pure heat recovery ([13]).

Ihrén and Svedberg [5] studied some configurations of steam injected gas turbines in cogeneration with pulp and paper mills. An air-blown black liquor gasifier was integrated to the system which resulted in a low heating value of the syngas and bark was not gasified. The present investigation also considers gasification of black liquor but with bark included too. The gasifier is oxygen-blown which renders possible an increased heating value of the syngas and therefore increased net power efficiencies. Ihrén and Svedberg [5] showed that aero-derivative gas turbines with high pressure ratios and high gas turbine inlet temperature are best suited for the case with STIG and process steam generation. Similar turbine performance are used in the present study.

EvGT Cycle. The evaporative gas turbine cycle is a type of advanced gas turbine that uses water to increase the thermal efficiency and the power output without including a steam turbine cycle. In the EvGT cycle the water is added into compressed air by means of humidification. The humidification method is thermodynamically more favorable than steam injection because it makes possible to recover lower temperature heat than STIG ([14]).

The evaporative gas turbine cycle is normally fired with natural gas. However, in this study, the focus has been on using a solid fuel (bark) for generating electricity in an externally fired evaporative gas turbine cycle, where heat is transferred to the working fluid in the gas turbine through a high temperature heat exchanger. This technology has been shown to have the possibility to increase power generation without a bottoming cycle and also to be able to use biomass without gasification and gas cleaning. The key technology is the development of ceramic heat exchangers allowing higher temperatures, which leads to higher turbine inlet temperatures and higher net power efficiencies ([15,16]).

The evaporative gas turbine cycle technology is in an earlier development stage compared to the STIG cycle. The technology is not yet commercially available, but a pilot plant has been built and operated at the Lund Institute of Technology, Sweden, in 1998 ([17]).

The Reference Pulp and Paper Mills

In the present study, advanced gas turbine cycles are integrated in reference pulp and paper mills. The conditions, energy demand, etc., for the reference pulp and paper mills are described below.

The Reference Kraft Pulp Mill. The research program, "The Ecocyclic Pulp Mill," has defined a reference kraft pulp mill. The reference mill is a kraft pulp mill based on the best pulping technology of today, combined with techniques that generate more power. In this case the calculations are based on a production level of 1000 tonne pulp/day.

In the kraft pulping process, slightly more than half of the material entering the pulp mill is dissolved in the cooking liquor which after cooking is called black liquor. As more lignin than carbohydrates is dissolved in the cooking liquor a large share of the calorimetric heat content is found in the black liquor, thus representing a major energy source. While recovering important chemicals from the black liquor, steam and electricity could be produced at the same time. Black liquor together with bark is the available biomass used for electricity and steam generation. The fuel amount is 1797 tonne black liquor DS/day and 182 tonne bark DS/day, of which more than half is used in the lime kiln.

The traditional Tomlinson Recovery boiler and bark boiler are used for steam production in the reference pulp mill. Steam conditions are 79 bars and 485°C as compared to the conventional level of 60 bars and 450°C. A back-pressure turbine and a condensing turbine are used for the generation of electricity and process steam. Medium pressure (MP) steam of 12 bar and low pressure (LP) steam of 4.5 bars, which is needed for the pulping process, are supplied. Steam of 25–30 bars is used for sootblowing in the recovery boiler. The steam demand of the reference mill is 11 GJ/ADt of which around 1 GJ/ADt is used for the sootblowing. The required process steam production is 78.5 tonne MP steam/h and 137.4 tonne LP steam/h. The power demand of the reference kraft pulp mill is 791 kWh/ADt ([18]).

The energy demand of the reference kraft pulp mill is lower compared to the Swedish average market kraft pulp mill in 1994 with a steam demand of 15.4 GJ/ADt and power demand of 851 kWh/ADt ([1]). This is caused by, for example, increased closure of the pulping process, and more efficient process units etc. The simplified energy system of the reference pulp mill is shown in Fig. 2.

The Reference Integrated Pulp and Paper Mill. In the reference integrated pulp and paper mill, a kraft pulp mill is connected to a paper mill. The amount of fuel is the same as for the pulp mill, but the energy demand is higher because the paper production also has to be supplied with steam and power.

Power and steam demands for the average integrated kraft pulp and paper mill 1994 were 1346 kWh/ADt and 17 GJ/ADt respec-

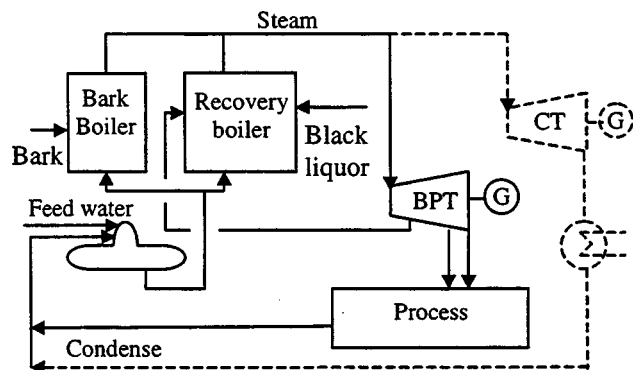


Fig. 2 The reference pulp mill energy system. (The dashed line represents the condensing turbine which is not used in the average mills of 1994.)

tively ([1]). The energy system of the reference integrated pulp and paper mill is also shown in Fig. 2, where the condensing turbine in dashed line is not used in the average mills of 1994.

If the power and steam demand for the average integrated mill are assumed to be reduced to the same extent as for the reference kraft pulp mill, the steam and power demand levels would, according to the average data for 1994, increase with factors of respectively 17/15.4 and 1346/851 compared to the reference kraft pulp mill. This will result in a steam demand of 12 GJ/ADt of which around 1 GJ/ADt is used for sootblowing and a power demand of 1251 kWh/ADt.

Steam Demand Variation. The steam demand could be reduced by an improved heat exchange network in a pulp mill, increased closure and with changes in process design. Theoretically, the steam demand in a kraft pulp mill could be reduced to around half of the reference kraft pulp mill steam demand, around 5 GJ/ADt (not included sootblowing) ([19]). The steam demand can also be reduced temporarily because of, for example, process failure, which can force the mill to close one of several or all pulping lines. This will create the need for a system with favorable part load performance. The present study has used different steam demand levels compared to the steam demand of the reference pulp mills.

Modeling of Advanced Gas Turbine Systems

Two different systems are simulated:

- 1 Gasification of black liquor and bark with a combined cycle or a steam injected gas turbine cycle connected to a process steam generator.
- 2 Evaporative gas turbine cycle externally fired with bark, in combination with a Tomlinson recovery boiler.

The two systems are described below.

Part 1—Comparison Between the Combined Cycle and the Steam Injected Gas Turbine Cycle. The chemical recovery is in this part performed in a black liquor gasifier. The simulation model is based on a black liquor gasifier and a biomass gasifier connected to either a combined cycle or a steam injected gas turbine. More than half of the produced gas from the biomass gasifier is used in the lime kiln. The simulated system is shown in Fig. 3 where the shaded area is replaced with Fig. 4 in the case

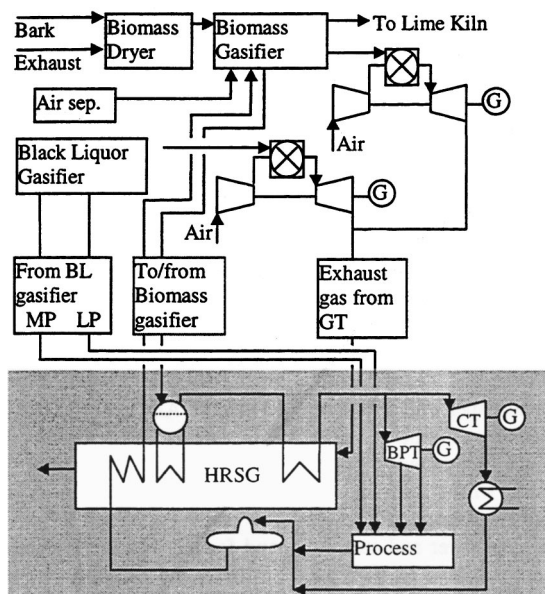


Fig. 3 The simulated model

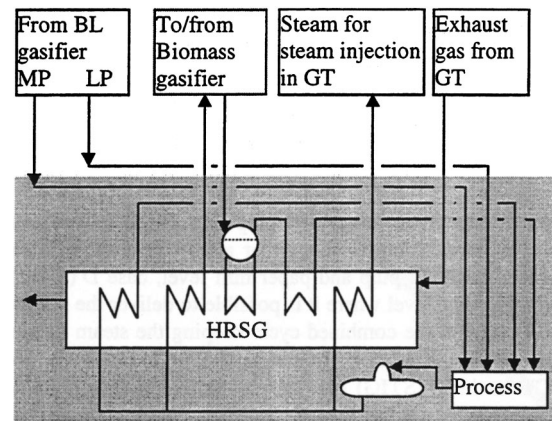


Fig. 4 Part in Fig. 3 replaced for steam injection

with steam injection. In the combined cycle, a back pressure steam turbine and in some cases a condensing steam turbine are used for power and process steam production.

The black liquor gasifier model used was developed within the “Ecocyclic Pulp Mill” research program ([4,20]). The gasifier is an oxygen blown pressurized gasifier which gasifies 1797 tonne black liquor DS/day. Gasification of bark is made in a separate pressurized oxygen blown gasifier, which is based on a combination of literature data and simulations. Data for the gasifiers are presented in Table 1 ([6]).

The product gas is used in two separate gas turbine modules with similarities to the General Electric LM2500. However the gas turbines are assumed to be adapted to low heating value gas with, for example, an allowed mass flow increase of 15 percent from the compressor to the expander. Assumptions for the gas turbines are presented in Table 2.

The steam injected gas turbine has similarities to GE LM 2500 STIG and is based on the assumptions in Table 2. Steam of 30 bars and 450°C is injected in the combustion chamber. The

Table 1 Black liquor and biomass gasifier conditions

Gasifier conditions:	Black liquor gasifier	Biomass gasifier
Temperature [°C]	950	900
Pressure [bar]	25	27
Gas to GT:		
Temperature [°C]	38	350
Pressure [bar]	29	27
Composition [mol %]		
CO	39.2	30.0
H ₂	41.1	24.1
CO ₂	16.7	24.2
CH ₄	1.23	5.6
H ₂ O	0.34	15.9
LHV [MJ/kg]	11.3	8.1
Steam generation: [ton/h]		
Net MP Steam (12 bar)	55.5	
Net LP Steam (4.5 bar)	49.9	
Saturated steam at the pressure of the HRSG		4.2–4.5

Table 2 Assumptions for gas turbines

Parameter	Assumption
Turbine inlet temperature	1200°C
Pressure ratio	25.8
Mechanical efficiency	0.98
Isentropic efficiency, expander	0.92
Isentropic efficiency, compressor	0.90
Mass flow increase exp/comp	15–30 %

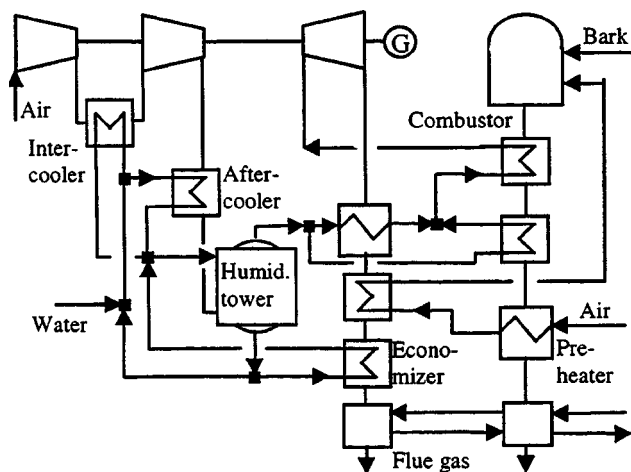


Fig. 5 The externally fired evaporative gas turbine cycle

Table 3 Assumptions for the externally fired evaporative gas turbine cycle

Parameter	Assumption
Turbine inlet temperature	900/1100°C
Pressure ratio	12
Mechanical efficiency	0.98
Isentropic efficiency, expander	0.90
Isentropic efficiency, compressor	0.88
Mass flow increase	20 percent

amount of injected steam (water/air inlet) varies between 0–20 percent and the mass flow increase is around 30 percent in the case with 20 percent steam injection. The exhaust gas from the gas turbines is used for generation of steam for the pulp and/or paper process at 4.5 bars and 12 bars, as well as steam for steam injection at 30 bars. The shaded parts in Fig. 3 that are replaced for the steam injection is shown in Fig. 4.

Part 2—Tomlinson Recovery Boiler Together with an Externally Fired Evaporative Gas Turbine Cycle. This part examines an alternative to increase the power production without gasification. The gas turbine technology presented has not yet been used commercially. Both closed cycles and evaporative cycles have individually shown to be more economic compared to the combined cycle, especially at small sizes. The chemical recovery in this part is performed in the conventional Tomlinson recovery boiler. The steam data is similar to the reference kraft pulp mill energy system of 79 bars and 485°C. Bark is fired in a solid fuel combustion system and heat is transferred to an evaporative gas turbine cycle through a heat exchanger. The lime kiln is fired with external fuels since the internal bark is used for the externally fired evaporative gas turbine. Two turbine inlet temperatures are used according to the type of heat exchanger, 900°C or 1100°C in a metallic and a ceramic heat exchanger respectively. In the evaporative gas turbine cycle the compressed air is brought into contact with heated water in a humidification tower, causing the water to evaporate and air to become humid. After the humidification tower, the compressed air has a water content between 15–20 percent. The simulated gas turbine system is shown in Fig. 5.

Assumptions for the externally fired gas turbine are presented in Table 3.

Simulation—Conditions and Results

The two different parts are simulated with variation of process steam demand and water to air ratio in the gas turbines. The conditions and results are presented below.

Table 4 Conditions—comparison of combined cycle and STIG cycle in the kraft pulp mill

Mill/system type	Ref.	B ² CC	C ³ CC	A ¹ STIG	B ² STIG	C ³ STIG
BL [MW]	252	252	252	252	252	252
Bark [MW]	38.5	38.5	38.5	38.5	38.5	38.5
Process steam [MW]	115.7	86.8	57.9	115.7	86.8	57.9
Steam data in HRSG/steam injection:						
-Pressure [bar]		60	60	30	30	30
-Temp. [°C]		450	450	450	450	450

¹The reference pulp mill steam demand is produced.

²Three quarters of the reference pulp mill steam demand is produced.

³Half of the reference pulp mill steam demand is produced.

Part 1—Comparison Between the Combined Cycle and the Steam Injected Gas Turbine Cycle. By reducing the process steam demand, the amount of steam available for use in the condensing turbine in the combined cycle is increased. An alternative use of excess steam could be to increase the steam injection in the STIG cycle. Conditions and results for the comparison of the combined cycle and the STIG cycle integrated in a kraft pulp mill or an integrated pulp and paper mill are described below.

Integration in a Kraft Pulp Mill. Steam demand is reduced to three quarters of the reference pulp mill steam demand in case B (CC) and B (STIG), and half of the reference pulp mill steam demand in case C (CC) and C (STIG). When using the combined cycle, the process steam generation cannot be kept at the reference pulp mill steam demand level and still generate steam in the HRSG at the level of 60 bars and 450°C. In the present process configuration, which includes the gasification of all the available biofuel, the steam demand has to be reduced to 14 bars and 268°C to still be able to produce the reference pulp mill process steam demand ([6]). However in the case using a STIG, the same steam demand level is possible to achieve with a lower steam injection level, case A (STIG). The HRSG steam pressure level in the combined cycle is kept at the level of 60 bars and 450°C to receive a more realistic comparison between the combined cycle and the STIG cycle. Conditions and results for the comparison of the combined cycle and the STIG cycle integrated in a kraft pulp mill is presented in Tables 4 and 5.

The net power efficiency is somewhat higher for the combined cycle compared to the steam injected gas turbine. This is caused by an increased heat loss with the exhaust gas that contains a higher water content in the STIG case. The surplus of power for the cases B-C (CC) and A-C (STIG) versus the reference kraft pulp mill steam demand is shown in Fig. 6.

Table 5 Results—comparison of combined cycle and STIG cycle in the kraft pulp mill. Based on LHV and a pulp production of 1000 ADt/d.

Mill/system type	Ref.	B CC	C CC	A STIG	B STIG	C STIG
Process steam [GJ/ADt]	10	7.5	5	10	7.5	5
Total power [MW]	57.0	93.2	102	79.0	88.3	97.0
Power surplus [kWh/ADt]	576	144	165	1104	132	153
Condensing turbine [MW]	23	7.1	24	0	0	0
Steam injection [water/air inlet percent]	0	0	0	1.9	9.8	20
Net power eff. [percent]	19.6	32.1	35.1	27.2	30.4	33.4
Total eff. [percent]	59.5	62.0	55.0	67.0	60.3	53.3

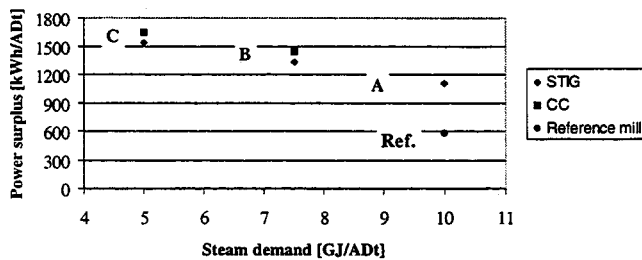


Fig. 6 Steam demand vs power surplus in the kraft pulp mill

Integration in an Integrated Pulp and Paper Mill. Steam demand is, in this comparison, either almost kept same as the reference integrated pulp and paper mill level, case D (STIG), or reduced to the level where it is possible to deliver the needed process steam in the combined cycle keeping the steam data in the HRSG at the level of 60 bars and 450°C, E (STIG) and E (CC). In the D (STIG) case, no steam injection is possible since all heat is needed for process steam production. It was found that the use of a STIG turbine was not possible in this case where a gas turbine without steam injection is needed. Conditions and results for the comparison of combined cycle and STIG cycle integrated in an integrated pulp and paper mill is presented in Tables 6 and 7.

The reference energy system will satisfy the steam and power demand for the integrated pulp and paper mill. The D (STIG) case will deliver a significant power surplus with only a slightly reduced steam demand. The electricity surplus is reduced by more than 400 kWh/ADt compared to the integration in the reference kraft pulp mill. The surplus of power for the cases D-E (STIG) and E (CC) versus variation of the reference pulp and paper mill steam demand is shown in Fig. 7.

Part 2—Tomlinson Recovery Boiler Together With Externally Fired Evaporative Gas Turbine.

By varying the turbine

Table 6 Conditions—comparison of combined cycle and STIG cycle in the integrated pulp and paper mill

Mill/system type	Ref.	D ¹ STIG	E ² STIG	E ² CC
BL [MW]	252	252	252	252
Bark [MW]	38.5	38.5	38.5	38.5
Process steam [MW]	127.3	126.0	98.9	98.9
Steam data in HRSG/steam injection:				
-Pressure [bar]		30	30	60
-Temp. [°C]		450	450	450

¹99 percent of the reference pulp and paper mill steam demand is produced. No steam for injection in GT is available.

²The largest amount of steam possible to deliver in connection to the combined cycle is produced.

Table 7 Results—comparison of combined cycle and STIG cycle in the integrated pulp and paper mill. Based on LHV and a pulp production of 1000 ADt/d.

Mill/system type	Ref.	D SGIG	E STIG	E CC
Process steam [GJ/ADt]	11	10.9	8.5	8.5
Total power [MW]	54.5	76.4	84.4	89.3
Power surplus [kWh/ADt]	55.0	581	773	890
Steam injection [water/air inlet percent]	0	0	6.2	0
Net power eff. [percent]	18.8	26.3	29.1	30.7
Total eff. [percent]	62.6	69.7	62.9	64.6

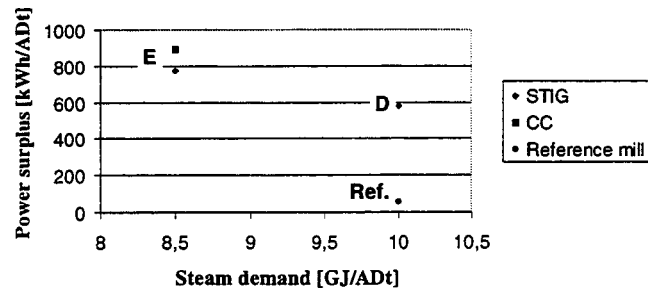


Fig. 7 Steam demand vs power surplus in the integrated pulp and paper mill

inlet temperature and water to air ratio the net power efficiency for the evaporative gas turbine cycle will change. The EvGT cycle is combined with the reference kraft pulp mill energy system with the withdrawal of the bark inlet. The bark inlet to the EvGT cycle is dependent on the fuel supply to the lime kiln. To integrate an evaporative gas turbine into a kraft pulp mill it is assumed that the whole bark amount is needed to justify the integration. The lime kiln is here assumed to be fired with for example extra intake of felling wood waste and the available amount of bark is used in the EvGT cycle. To make a realistic comparison the whole amount of bark is used for steam and power generation in the reference energy system also. The power output from the reference steam turbine system is thereby increased to 61.1 MW in the case with the whole bark amount of which 53.0 MW corresponds to production from black liquor. Two turbine inlet temperatures (900°C and 1100°C) are presented which result in different possibilities to reach high water to air ratios (0.15 and 0.20), case F (EvGT) and G (EvGT), respectively. The conditions and results for the simulation of externally fired evaporative gas turbines are presented in Tables 8 and 9.

The net power efficiency (bark fuel) for the externally fired gas turbine is higher compared to the reference kraft pulp mill. Compared to the bark boiler in the reference mill the net power output for the EvGT cycle is increased considerably. However, the fuel

Table 8 Conditions—externally fired evaporative gas turbine cycle in the kraft pulp mill

Mill/system type	Ref.	F (EvGT)	G (EvGT)
BL [MW]	252	252	252
Bark [MW]	38.5	38.5	38.5
Process steam [MW]	115.7	115.7	115.7
TIT [°C]		900	1100
Water-air ratio [percent]		0.15	0.20

The reference kraft pulp steam demand is produced in all cases.

Table 9 Results—externally fired evaporative gas turbine cycle in the kraft pulp mill. Based on LHV and a pulp production of 1000 ADt/d.

Mill/system type	Ref.	F (EvGT)	G (EvGT)
Bark fuel			
Power EvGT/BPT [MW]	8.10	13.7	14.8
Net power eff. [percent]	21.0	35.6	38.4
Black liquor			
Power BTP [MW]	53.0	53.0	53.0
Total fuel (bark and BL)			
Total power [MW]	61.1	66.7	67.8
Power surplus [kWh/ADt]	674	809	835
Net power eff. [percent]	21.0	23.0	23.3
Total eff. [percent]	60.9	62.8	63.2

supply to the gas turbine is only 13 percent of the total fuel amount which results in an only slightly increased total net power efficiency. The total power output increase is between 9.2 percent and 11 percent.

System Modeling. For the calculations in this study, the sequential process simulator ASPEN PLUSTM has been used. Advantages of the simulator are its large databases of physical properties, components, unit operations and convergence methods. The system contains streams with different components and physical properties. Therefore, different databases and convergence methods were used, for example steam data and combustion data bases. Pressure losses are neglected in all the simulated cases. An estimation of the expected pressure losses has been made, for the externally fired evaporative gas turbine, according to data from Yan et al. [16], where pressure losses are assumed to be between one percent for the hot side of a heat exchanger to four percent for the humidification tower. The total effect on the net power efficiency in the present study would be less than two percentage points.

Economic Considerations

A thorough cost analysis has not been performed in the present study. However, a few economic considerations are presented from other studies, which indicate how costs for different advanced gas turbine cycles relate to each other.

The Nth-plant capital investment cost per unit of black liquor processing at a kraft pulp mill is estimated to be around the same for black liquor gasification together with combined cycle compared to the traditional Tomlinson recovery boiler and back pressure steam turbine. Since the gasification process generates significantly more power the cost per produced kW electricity will be about half compared to the Tomlinson recovery process ([2,3]).

In the present study the compared concepts with black liquor and biomass gasification connected either to the combined cycle or the STIG cycle contains the same gasification processes and the same mill size and energy demand. Therefore the different kinds of turbines will result in a variation in the investment cost. Equipment price levels for these gas turbines are \$404 per kW for LM 2500 without steam injection and \$357 per kW for the LM 2500 STIG. The turnkey plant equipment cost is for the combined cycle based on LM 2500 \$800 per kW ([21]).

An economical comparison between an evaporative gas turbine cycle and a combined cycle shows that the specific capital investment (SEK/kWe) for a plant of the size 70–80 MWe is significantly lower (15–30 percent) for the evaporative gas turbine ([22]). The economic advantage of the evaporative gas turbine cycle compared to the combined cycle is most likely even larger at the smaller sizes used in the present study.

Discussion

The STIG cycle has been proven to provide a better fit to the power and steam demand of the reference mills compared to the combined cycle. This means that while generating the maximum amount of power in the gas turbine, the exhaust gas heat is not enough for a back pressure steam turbine, but for process steam generation and steam injection in a STIG cycle in the kraft pulp mill. In the integrated mill though, the steam demand has to be slightly reduced and no steam injection is possible without reducing the steam demand. The STIG cycle has also in literature showed on favorable part load performance. This is another reason for the advantageous fit of a STIG cycle in a pulp and paper mill where the steam demand can vary for several reasons. The possibility to fulfill the required process steam demand without the traditional energy system could be questioned. However, the Steyermühl plant, for example, has shown that the process steam demand can be guaranteed with an additional firing system.

While the relationship between the quantity of available fuel and the energy demand is specific for the reference mills, the

results can also be extended to today's average mill. As the energy demand of the reference mill is lower than today's average mill, yet its fuel availability is similar, the STIG cycle will also be more suitable to the average mill of today than the combined cycle.

The economics for the STIG cycle appear to be more favorable compared to the combined cycle for the system considered. The combined cycle contributes to a lower investment cost compared to the STIG cycle only at very large net plant outputs.

The externally fired EvGT cycle has a very high net power efficiency, but the system studied in the present study does not show the full potential of the cycle. This is caused by the low share of the fuel amount that is used in the cycle and the relatively high efficiency of the reference system. If integrating an externally fired evaporative gas turbine within a system that has a net power efficiency of the average kraft pulp mill of 1994 (9.4 percent) and the same fuel amount as in the reference pulp mill, the total power generation will increase by 40 percent compared to around 10 percent in the present study. Externally fired turbines, along with a high efficiency, will give a possibility to increase power generation without the need for gasification.

The net power efficiency is higher for the externally fired gas turbine compared to the STIG cycle and the combined cycle with gasification of biomass. This could be due to two reasons; no process steam generation is included in the process, or higher heat losses occur in the gasification processes than in the externally fired boiler and heat exchanger.

The STIG cycle has higher heat losses in the exhaust gas compared to the combined cycle. However, the EvGT cycle has a more efficient heat recovery system for the exhaust gas. An interesting alternative would be a combination of STIG and EvGT, which uses the exhaust gas for a combination of process steam generation, steam injection and humidification of compressed air. This is a future work.

Conclusions

According to energy demand, costs and part load performance, the STIG cycle has shown to be more suitable for use in the reference mills compared to the combined cycle. The STIG cycle can double the 576 kWh/ADt power surplus of the reference kraft pulp mill while still satisfying the steam demand. A gas turbine without steam injection can deliver 5 times the 115 kWh/ADt power surplus of the integrated reference mill, with only a slightly reduced steam demand.

The externally fired EvGT cycle has a high net power efficiency. The effect on the total system although is modest due to the low fraction of the total fuel amount used in the EvGT system.

The presented study shows that a great potential exists to increase the power generation for external use in the pulp and paper industry. However, some of the studied technologies still need to be tested and verified in practical use.

Acknowledgments

This work is a part of the "Ecocyclic Pulp Mill" research program financed by MISTRA, the Swedish Foundation for Strategic Environmental Research.

Nomenclature

ADt	=	Air Dry Tonne
BL	=	Black Liquor
BPT	=	Back Pressure Turbine
CC	=	Combined Cycle
CT	=	Condensing Turbine
DS	=	Dry Substance
EvGT	=	Evaporative Gas Turbine
HRSG	=	Heat Recovery Steam Generator
LP	=	Low pressure
MP	=	Medium pressure
ST	=	Steam Turbine

STIG = Steam Injected Gas Turbine
TTT = Turbine Inlet Temperature

References

- [1] Wiberg, R., 1995, "Energiförbrukning i massa-och pappersindustrin 1994," "Energy Consumption in the Pulp and Paper Industry 1994," ÅF-IPK, Stockholm (in Swedish).
- [2] Larson, E. D., Kreutz, T. G., and Consonni, S., 1998, "Combined Biomass and Black Liquor Gasifier/Gas Turbine Cogeneration at Pulp and Paper Mills," ASME Paper No. 98-GT-339.
- [3] Larson, E. D., Consonni, S., and Kreutz, T. G., 1998, "Preliminary Economics of Black Liquor Gasifier/Gas Turbine Cogeneration at Pulp and Paper Mills," The 43rd ASME Gas Turbine and Aeroengine Congress, Stockholm, Sweden, June.
- [4] Berglin, N., and Berntsson, T., 1997, "Energisystemkonsekvenser och process-integrationsaspekter av svartlutsförgasning," "Energy System Consequences and Process Integration-Aspects of Black Liquor Gasification," Slutrapport till forskningsprogram Sulfatåtervinning /Svartluts-förgasning, ISSN 1103-2952, Stockholm (in Swedish).
- [5] Ihrén, N., and Svedberg, G., 1994, "Simulation of Combined Cycles with Black Liquor Gasification," ASME Paper No. 94-GT-447.
- [6] Maunsbach, K., Svedberg, G., and Martin, V., 1999, "Gasification of Black Liquor and Biomass in a Pulp Mill for Increased Power Generation," The 6th New Available Technologies Conference, SPCI'99, Stockholm, June.
- [7] Lensink, M., 1997, "Small Cogen System Solves Energy Reliability Dilemma at Sonoco Mill," Pulp & Paper, August.
- [8] Kellerer, A., and Spangenberg, Ch., 1998, "Operating Experience with a Cheng-Cycle Unit," VGB PowerTech, No. 11, pp. 16–22.
- [9] Tuzson, J., 1991, "Status of Steam injected Gas Turbines," ASME-COGEN-TURBO, IGTI, Vol. 6.
- [10] Harvey, S., Abdallah, H., and Faccini, B., 1998, "Part Load Performance of Chemically Recuperated Gas Turbine Cycles Compared to Other Advanced Cycles," ASME Paper No. 98-GT-037.
- [11] Brown, R. M., and Gray, R. H., 1987, "Aero-derivative gas turbines gaining industry favor," Paper Industry Management Association, v69, n7, July, pp. 17–21.
- [12] Kasseborn, B., 1991, "Retrofitted Gas Turbine to a Steam Electric Unit of a Paper Mill Solves Environmental and Economic Problems," ASME-COGEN-TURBO, IGTI, Vol. 6.
- [13] Schobesberger, P., Gasteiger, G., and Budin, K., 1991, "Combined Cycle power Plants with Special Respect to the Heat Recovery Steam Generator on the Basis of the Steyrermühl Power Plant," ASME-COGEN-TURBO, IGTI, Vol. 6.
- [14] Ågren, N., 1997, "Simulation and Design of Advanced Air-Water Mixture Gas Turbine Cycles," Licentiate Treatise, TRITA-KET R74, ISSN 1104-3466, Royal Institute of Technology, Stockholm, Sweden.
- [15] Yan, J., and Eidensten, L., 1994, "Performance Evaluation of Biomass Externally Fired Evaporative Gas Turbine System," ASME-COGEN-TURBO, IGTI, Vol. 9.
- [16] Yan, J., Eidensten, L., and Svedberg, G., 1996, "Externally Fired Evaporative Gas Turbine with a Condensing Heat Exchanger," ASME Paper No. 96-GT-77.
- [17] Ågren, N., Westermark, M., Bartlett, M., and Lindquist, T., 2000, "First Experiments on an Evaporative Gas Turbine Pilot Power Plant-Water Circuit Chemistry and Humidification Evaluation," ASME Paper No. 00-GT-168.
- [18] Delin, L., 1998, "Energibalanser för referensfabrik," "Energy Balances for the Reference Mill," KAM-report B26, ÅF-IPK AB, Stockholm (in Swedish).
- [19] Strömberg, J., Berglin, N., and Berntsson, T., 1997, "Using Process Integration to Approach the Minimum Impact Pulp Mill," Proceedings, Environmental Conference and Exhibit, Minneapolis, May 5–7.
- [20] Berglin, N., and Berntsson, T., 1998, Appl. Therm. Eng., **18**, No. 11, pp. 947–961.
- [21] *Gas Turbine World, 1998–1999*, Handbook, Vol. 19, a Pequot Publication.
- [22] Nilson, P. A., 1996, "Ekonomisk studie av EvGT för kraft-resp kraftvärmeanläggning," "Economics of EvGT for Heat and Power Plants," EvGT-projektet, Block 3 (in Swedish).

Deposit Formation and Mitigation in Aircraft Fuels

L. J. Spadaccini

D. R. Sobel

H. Huang

United Technologies Research Center,
East Hartford, CT

The development of a viable strategy for limiting coke deposition involves combining synergistic approaches for suppressing deposit buildup and reducing its impact on performance. Candidate approaches, including selection of favorable operating conditions (viz., pressure, temperature, heat flux, residence time, and passage size) and coke-tolerant heat exchanger designs, were investigated to evaluate their effectiveness and provide a basis for combining them into a single design philosophy. These approaches were evaluated through testing of current jet fuels in single-tubes and segments of heat exchanger configurations at temperatures up to 1000°F, pressures up to 1200 psi and liquid hourly space velocities up to 40,000/h. A key result of this work is the ranking of the importance of heat exchanger operating conditions on carbon deposition, with fuel temperature and those parameters that control species diffusion having the most pronounced impact. Residence time and pressure are of lesser importance. Alternative coke-tolerant heat exchanger designs featuring interchannel communication were evaluated and ranked, with several of these concepts demonstrating improvement over continuous passages.

[DOI: 10.1115/1.1383772]

Introduction

High heat sink fuel cooling technology can be applied to enhance engine performance over the entire spectrum of flight regimes. For hypersonic flight, it provides the only means for meeting the cooling requirements with storable fuels; for advanced fighter aircraft, it provides a path to enhance performance with current materials; and, for lower-speed military and commercial aircraft, it can increase growth potential and play a key role in emissions-reduction strategies. However, utilization of this technology will require heating the fuel to supercritical temperatures—well above the maximum temperature ($\sim 325^\circ\text{F}$) allowable in conventional fuel systems.

The principal engine operability issue that will affect hydrocarbon fuel cooling technology is coke formation. In gas turbine applications, where long duty cycles are required, even low rates of deposition will accumulate, degrading performance and, if left unchecked, leading to system failure. In hypersonic missile applications, duty cycles are short, but requirements for maximizing heat sink lead to very high fuel temperature operation and the potential for accelerated coking. The extent to which the benefits of this technology can be realized is directly related to our ability to mitigate against coke formation.

The objective of this study is to investigate coke deposition in aircraft fuels and to identify design concepts for minimizing its impact. Candidate approaches were investigated to evaluate their effectiveness and provide a basis for combining them into a single design philosophy. The individual elements examined include: (1) definition of guidelines for specifying operating conditions to minimize coke deposition in both the subcritical and supercritical temperature regimes, and (2) identification and evaluation of heat exchanger design concepts that have the potential for accommodating moderate coke buildup without progressive degradation.

The key variables and component operating conditions examined correspond to gas turbine and scramjet applications. Tests were conducted using Jet A and JP-8 fuels that were supplied by the Air Force Research Laboratory (batches POSF-2926 and -3405, respectively) and with JP-8+100, which includes special

thermal stability additives. The test results can be used to define guidelines to support fuel heat exchanger/thermal management system design and implementation, as well as to broaden the database available for validating and upgrading coke deposition models.

Background

In the temperature regime below approximately 700°F , oxygen dissolved in fuel is the major contributor to thermal decomposition and coke deposition. When air-saturated fuel is heated to temperatures above approximately 350°F , the dissolved oxygen reacts to form free-radical species (coke precursors) which initiate and propagate other autoxidation reactions leading to deposit formation. These reaction paths are dominant at temperatures up to approximately 700°F , but become insignificant when the concentration of the dissolved oxygen is reduced from its saturated value (~ 70 ppm) to very low levels (< 5 ppm). It has been shown that the dissolved oxygen is consumed as coke deposit is formed, and that increasing fuel temperature within the autoxidation regime accelerates oxygen depletion ([1]).

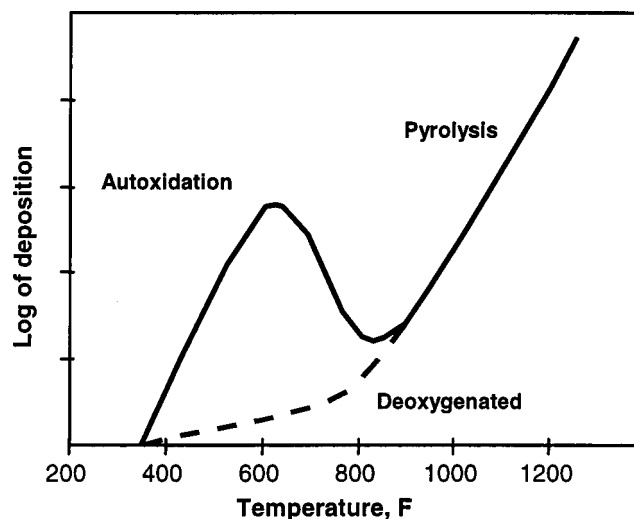


Fig. 1 Deposition rate for jet fuel

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-217. Manuscript received by IGTI Oct. 1998; final revision received by the ASME Headquarters Mar. 1999. Associate Editor: D. Wisler.

At temperatures above approximately 800°F, the deposition mechanism is characterized by pyrolysis, wherein chemical bonds are broken and large alkanes are converted into smaller alkanes, alkenes, and some hydrogen. This mechanism for coke production can be initiated by thermal (homogeneous) and/or catalytic (heterogeneous) cracking reactions, and followed by polymerization.

The rate of coke deposit buildup with increasing temperature is illustrated in Fig. 1 for a typical jet fuel. As indicated in the figure, fuel deoxygenation reduces coke deposition at low and intermediate temperatures. At higher temperatures, deposit buildup from aerated fuel approaches deposit buildup from deoxygenated fuel because the dissolved oxygen is consumed and the coke precursors are decomposed ([2]).

Test Apparatus

Most of the testing was performed with the multiple heated-tube coking rig shown in Fig. 2. The individual tubes were manufactured from type 304 stainless steel and sized to simulate single passages in a heat exchanger. The parallel flow paths enable simultaneous evaluation of up to five configurations that may have different geometries or operating conditions, and are particularly efficient for long-duration testing. A nearly uniform surface temperature distribution is obtained by clamping the tubes between isothermal copper blocks (8 in.×8 in.×1 in.) that are heated electrically by strip heaters attached to the outer surface. In some tests, coupons consisting of multiple flow passages were used.

Prior to a test, 1.5 gal piston-type accumulators are filled with fuel by a gear pump. The 1.5 gal capacity allows a full day of testing at moderate flow rates. Longer/higher flow rate tests can be accommodated by refilling fuel on-line, during tests. By applying

high-pressure nitrogen to the back-side of the pistons in the accumulators, the fuel is then pressurized to 150 to 200 psig higher than the predetermined test pressures. The five flowpaths are essentially independent; the construction of the rig requires only that the test sections operate at the same back pressure. The flowpaths may have different fuels, flow rates, additives, coatings, and/or tube sizes. In each of the separate flowpaths, the fuel is filtered and the flow rate is measured by a flow meter and controlled by a metering valve. Depending on the desired simulation conditions, the fuel can be preheated up to about 350°F, which is below its autoxidative coke deposition temperature, before entering the first of two successive test sections. The test sections can be maintained at different temperatures (up to approximately 1200°F), simulating different parts of a thermal management system. Downstream of the heat exchanger tubes, the processed fuel can be sampled for compositional analysis and is quenched in a cooler. System pressure is maintained by a back-pressure regulator.

During a test, key measurements include the inlet and exit fuel temperatures and pressures, and copper block and heat exchanger wall temperatures. These measurements provide important information to correlate performance and coke deposition. In preparation for determining the time-averaged carbon deposition rate, the heat exchanger tube is removed from the rig at the test completion, rinsed with hexane and dried in a vacuum oven. The tubes are then cut into 1/2–2 in. lengths and the carbon accumulated in each section is determined using a LECO RC-412 Carbon Determinator which quantifies carbon deposition (precision of ± 3 percent) by measuring carbon dioxide produced in a controlled carbon burnoff. Typically, the deposit consists of more than 75 percent carbon, with the remainder being oxygen and hydrogen. For the most part, test results are correlated using the total carbon deposit in the tube, i.e., the sum of the individual sections. However, the axial profiles are also examined to explain local phenomena.

Operating Conditions

The range of parameters investigated was selected to reflect that suitable for practical heat exchanger systems and is summarized in Table 1. Because the flow passages are small, fuel temperatures were measured only at the inlet, the midpoint (between the first and second heated copper blocks), and the exit of the tubes. Fuel temperature profiles were calculated between the measured values based on a uniform surface temperature within each block and heat transfer coefficients determined using the Dittus-Boelter correlation. In the analysis, the complex jet fuel composition is simulated by a mixture of 12 hydrocarbons ([3]) and the physical properties of the mixture at various temperatures and pressures are estimated using the NIST SUPERTRAPP program ([4]). A representative fuel temperature profile in successive heat exchanger tubes is shown in Fig. 3.

Fuel Temperature. Representative coke deposit levels as a function of fuel temperature are plotted in Fig. 4. The data are correlated in terms of parts per million (ppm) of carbon deposit

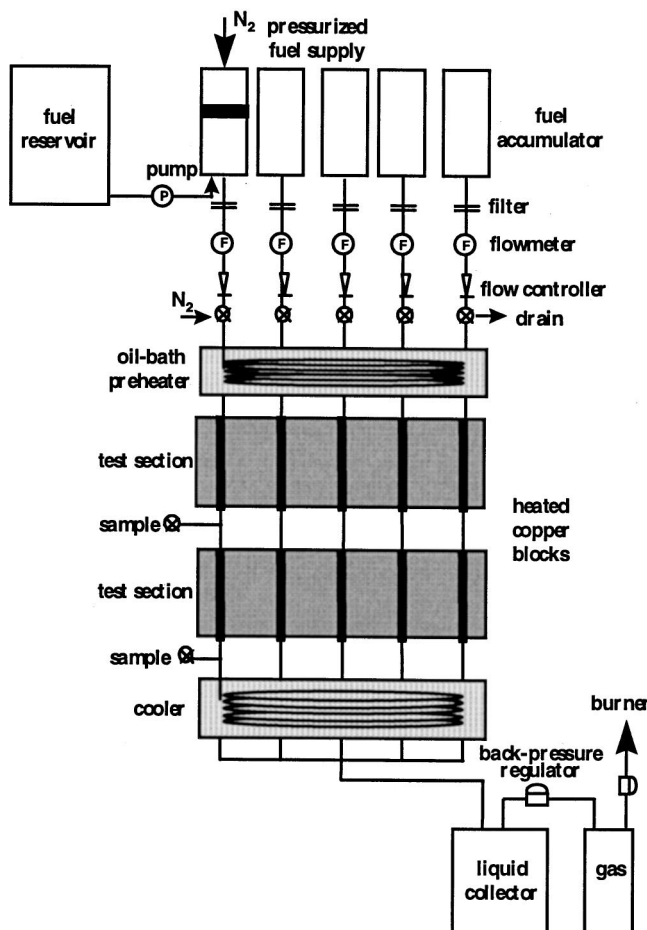


Fig. 2 Multiple heated-tube coking rig

Table 1 Test conditions

Key Variables	Range
Fuel	JP-8+100
Temperature, F	300–1000
Flow rate, lbm/h	1.8–7.1
Tube ID, in.	0.03, 0.058, 0.085
Pressure, psig	600, 1200
lhsv*, 1/h	860–9200
Residence time, s	0.14–1.40
Test duration, h	7–25
Tube material	stainless steel–304

*lhsv=volumetric flow rate/tube volume

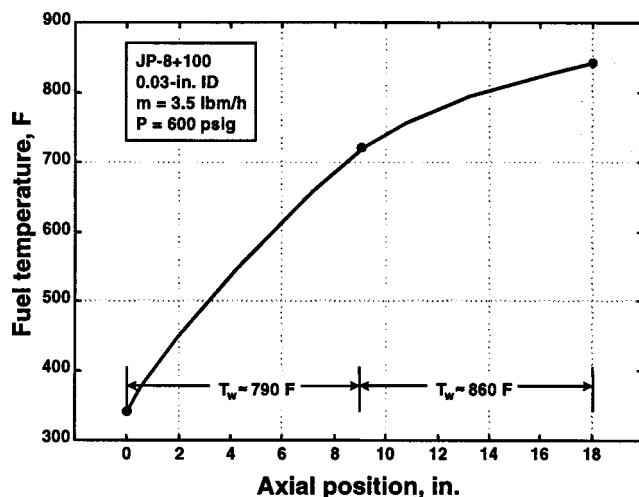


Fig. 3 Fuel temperature distribution

relative to fuel flow (i.e., $\text{mass}_{\text{carbon}}/\text{mass}_{\text{fuel}}$) and normalized by surface area for each of the two inch segments analyzed in the LECO carbon determinator. As shown in the figure, the carbon deposition rate increases with increasing fuel temperature up to a point ($\sim 600^\circ\text{F}$) beyond which any further increase in fuel temperature causes the deposition rate to decline. As discussed earlier, the coke deposition over this temperature range comes from the reaction of oxygen dissolved in the jet fuel. The oxidation rate of the fuel is accelerated by increasing the fuel temperature, thereby increasing the coke deposition rate. However, as the reaction proceeds along the heat exchanger tube, dissolved oxygen is consumed and its concentration decreases. In addition, the decomposition of coke precursors is also accelerated by increasing fuel temperature. Thus, the depletion of both dissolved oxygen and coke precursors results in the decreasing rate of deposition.

Reynolds Number and Passage Size. Both heat and mass transfer rates are dependent on Reynolds number (Re). The influence on coke deposition of a change in Reynolds number and flow passage diameter at constant flowrate is shown in Fig. 5 in terms of total accumulated carbon. The results indicate that increasing the passage diameter (i.e., decreasing Re) reduces carbon deposition, a trend that has also been reported by Chin and Lefebvre [5] and Jones et al. [6]. Moreover, if the results were normalized by surface area, the effect of passage size on the deposition would be even more pronounced. The trend of decreasing deposit rate with increasing diameter reflects a balance of mass-diffusion length scale, turbulence-enhanced mass-diffusion rates and residence time influences. When air-saturated fuel is heated to temperatures

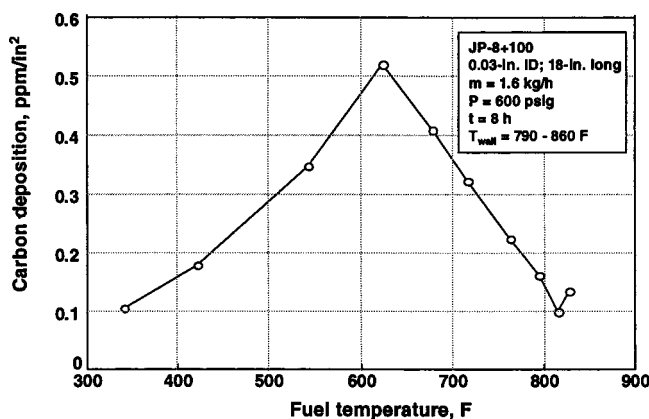


Fig. 4 Representative deposition profile

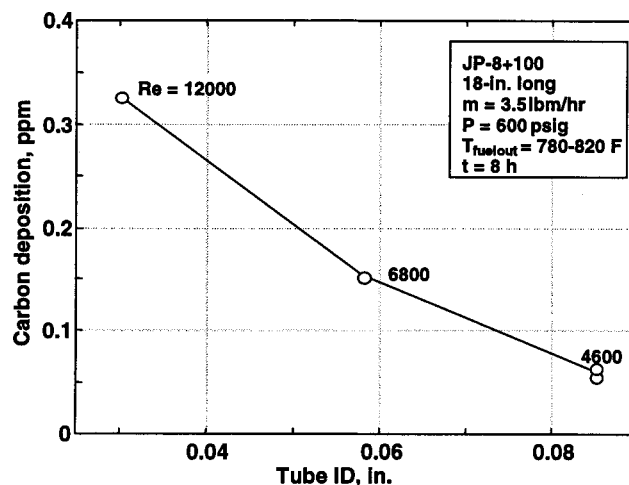
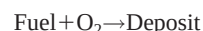


Fig. 5 Effect of passage size on deposition

above approximately 350°F , the dissolved oxygen reacts to form free-radical species (coke precursors) which participate in other autoxidation reactions leading to deposit formation. The surface deposition mechanisms are inextricably tied to the mass transfer process. For example, the surface deposition mechanisms



involve diffusion of dissolved oxygen, coke precursor, and deposit formed in the bulk flow to the wall surface. The mass transfer length scale, i.e., the mean distance between molecules and the wall surface, is linearly proportional to the passage diameter, and shorter length scales facilitate diffusion. Turbulence of the fuel flow is also affected by the passage size. Smaller ID tubes have larger Reynolds numbers (i.e., $Re = 4\dot{m}/\pi d\mu$) and therefore higher mass transfer coefficients, also promoting diffusion and thereby deposition. As Reynolds number is increased, heat transfer coefficients also increase. Deposition at very high heating rates is discussed below.

Residence Time and Space Velocity. Increasing the residence time of fuel in the heat exchanger tubes would be expected to increase coke deposition. For this reason, a smaller diameter tube would be expected to give less coke deposition than a larger diameter tube with the same flow rate. However, as discussed above, diffusion is the controlling factor for autoxidative coke deposition at the conditions tested. The influence of the fuel flow rate on the autoxidative coke deposition is shown in Fig. 6. Carbon deposition results are normalized by the fuel flow rate and indicate that, for a constant fuel outlet temperature, carbon deposition increases with increasing fuel flow rate. The mean Reynolds numbers (calculated based on film temperature) and residence times in the heated tubes at the three fuel flow rates plotted are also shown in the figure. As can be seen, although residence time was reduced 50 percent by increasing the flow rate, the deposition doubled. From this and the discussion above, it can be concluded that the enhanced diffusion effect is stronger than the impact of residence time.

To isolate the influence of residence time at a constant Reynolds number, results from two tests were compared, namely, a high fuel flow rate in a large-diameter tube (3.5 lbm/h, 0.058 in. ID, Fig. 5) and a low fuel flow rate in a small-diameter tube (1.8 lbm/h, 0.030 in. ID, Fig. 6). As the Reynolds numbers are equivalent (i.e., $Re=6800$) and the carbon depositions were approximately equal in the two tubes, the comparison indicates a balance

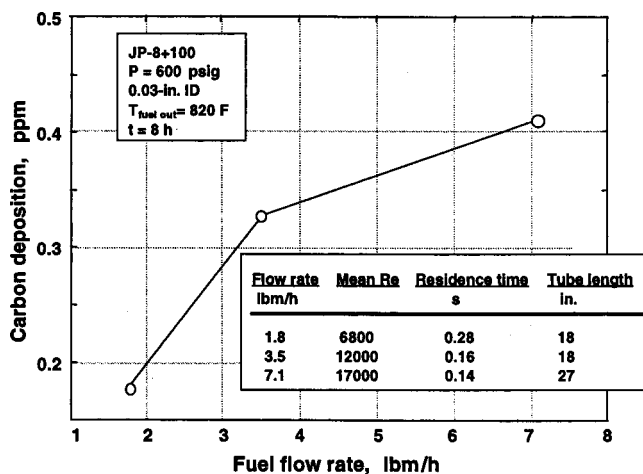


Fig. 6 Effect of flow rate on deposition

between the smaller diameter (length scale) tending to increase deposition and the larger diameter (high residence time) tending to decrease deposition. Also, the axial location of the peak deposit was shifted downstream to higher temperature in the smaller-diameter tube, reflecting the effect of reaction kinetics on deposition.

Liquid hourly space velocity, or $lhsv$, is defined as the ratio of the volumetric flow rate of liquid fuel to the passage volume. This scaling parameter incorporates the effects of the tube ID, length, and residence time. A smaller tube or higher fuel flow rate gives larger space velocities, higher mass transfer rates and lower residence times. The influence of the space velocity on the autoxidative coke deposition is shown in Fig. 7, which illustrates that, for the turbulent flow conditions tested (i.e., $4500 < Re < 17000$), the autoxidative coke deposit increases with increasing space velocity. At higher Reynolds numbers, where turbulent mixing is enhanced and residence time is significantly decreased, it is anticipated that deposit formation will no longer be controlled by mass transfer, but by the reaction kinetics.

Pressure. The influence of the fuel pressure on autoxidative coke deposition was investigated using a 0.085 in. ID tube that was 18 in. long. At pressures of 600 and 1200 psig, 3.5 lbm/h fuel flow and 810°F fuel outlet temperature, the total cumulative carbon depositions were relatively low, i.e., 0.062 and 0.050 ppm, respectively, after eight hours duration. Since the fuel is in the liquid state over a significant portion of the tube, increasing the fuel pressure from 600 to 1200 psig effects only a very small

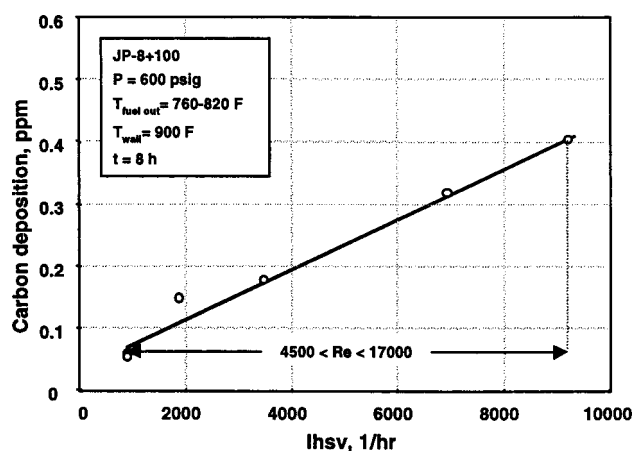


Fig. 7 Effect of space velocity on deposition

decrease in the Reynolds number, as a consequence of increases in fuel viscosity and density. Therefore, fuel pressure should have only a small effect on mass transfer and residence time at these conditions. Thus, as observed in the tests, the variation of fuel pressure in the liquid range has a minor impact on the autoxidative coke deposition. At higher temperatures, where the fuel is a supercritical vapor, increasing pressure will increase residence time and pyrolytic reactions may be more significantly affected.

High Heating Rate. To determine the coking characteristics under very high heat flux and low residence time conditions, a series of tests was performed with resistively heated tubes in the single-element Bench-Scale Reactor Test Rig described in Sobel and Spadaccini [7]. (These high heat fluxes cannot be achieved over a substantial length of the tube using the multiple-heated tube coking rig.) In the single-element bench-scale reactor test rig, the heat flux is uniform and the wall temperature increases from inlet to exit (typically from 800°F to 1400°F). The resistively heated tubes were 12 in. long and 0.030 in. ID, and were selected to maximize the heat flux, by reducing wetted area at a given power input, while avoiding excessive end effects and maintaining readily manageable flow rates with substantial residence times. In these tests, the fuel was not preheated upstream of the tube.

Tests were run with heat fluxes from 45 to 240 Btu/ft²s at 600 psi for a duration of seven hours (one run day). For each test, the fuel exit temperature was maintained at 1000°F and both autoxidative and pyrolytic coke was formed. It should be noted that the space velocity increases by a factor of approximately six over the range of flow conditions examined; the residence time is reduced by essentially the same factor (from 0.4 to 0.07 s).

The carbon deposition results are shown in Fig. 8. Since the tests were run with the same outlet temperature but different flow rates, this analysis allows a direct comparison of the amount of carbon deposited for a specific fuel heat sink (heat sink/mass_{fuel}). These results show that carbon deposition can be reduced by significantly increasing heat flux, while increasing flow rate (and Reynolds number) and decreasing the residence time. This trend reflects the changing balance between heating, mixing, and kinetics, and occurs in spite of the higher wall temperatures associated with the higher heat flux. As heat flux is increased substantially with the same fuel exit temperature, the wall temperature will also increase. At heat fluxes of 45 and 240 Btu/ft²s, the exit (peak) wall temperatures were 1200 and 1450°F, respectively—well into the pyrolytic regime. (The carbon distribution indicates approximately equal amounts of autoxidative and pyrolytic deposition.)

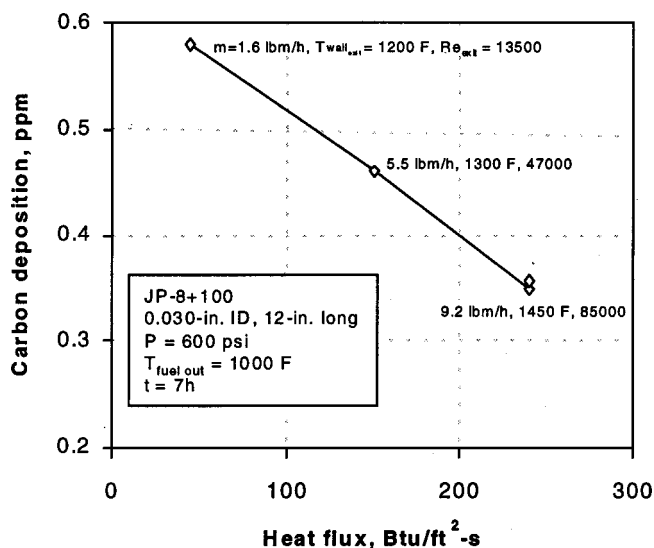


Fig. 8 Effect of heat flux on deposition

These results support the hypothesis that at high Reynolds numbers species diffusion is very rapid and the deposition process is limited by the reaction kinetics.

Coke-Tolerant Designs

Understanding the impact of operating conditions on deposit formation enables effective flow passage design. Conventional heat exchanger designs, whether co, counter, or cross-flow, are characterized by individual parallel fuel passages. In this type of design, carbon deposits may build up over time, causing a restriction in a given passage. The restriction, in turn, can reduce the coolant flow in that passage, increasing local surface and fuel temperatures and thereby potentially increasing the coking rate itself. Moreover, the increased surface temperature will also result in higher temperatures in neighboring passages and the fouling problem may propagate across the heat exchanger, resulting in failure. Therefore, candidate configurations for coke-tolerant designs, i.e., designs which have the potential for accommodating moderate coking rates without progressive degradation, were identified, fabricated as sectors and tested in the multi-tube coking rig. Concepts were evaluated that allow for interchannel communication, permitting isolation of coke buildup and thereby coke accommodation, and enhanced heat transfer, due to turbulence generation and frequent restarting of the boundary layer.

A standard test element design, shown in Fig. 9, was developed and five different fluid passage geometries were created by machining or chemical milling type 304 stainless steel. Although chemical etching increases surface roughness (from $\sim 0.4 \mu\text{m}$ to $\sim 4.0 \mu\text{m}$) which can promote deposition (by increasing surface area or turbulence), the effect is greatly reduced or eliminated as deposits accumulate ([8]). As shown in the figure, the overall size of all coupons was 8.0 in. $\times$ 0.55 in. $\times$ 0.25 in. The internal patterns were as follows: (1) Parallel channels—four 0.050 in. $\times$ 0.050 in. straight flow channels separated by 0.025 in. ribs; (2) Offset parallel strip fins—same as above, except that the ribs had 0.100 in. long staggered openings cross-connecting the channels each 1.00 in. along the length; (3) Circular pin fins—0.010 in. dia. $\times$ 0.030 in. high fins in a 0.045 in. $\times$ 0.067 in. staggered grid array; (4) Small teardrop pin fins—0.010 in. $\times$ 0.030 in. teardrop fins, 0.030 in. high, in a 0.033 in. $\times$ 0.120 in. staggered grid array; and (5) Large teardrop pin fins—0.010 in. $\times$ 0.090 in. teardrop fins, 0.030 in. high, in a 0.033 in. $\times$ 0.352 in. staggered grid array. Additional key geometric parameters of the test elements are summarized in Table 2. The same parameters for three cylindrical tubes are also listed in the table for comparison. The differences in these parameters can have significant effects on the heat transfer (i.e., the transverse temperature gradient) and coke deposition. The open volumes of all of the test elements listed in the table are

Table 2 Test element geometric patterns

Coupon Type	No. of Pins	V_{flow} in. ³	A_{surface} in. ²	A_g/V_f 1/in.
Parallel channels	-	0.0706	5.63	80
Offset strip fins	-	0.0715	5.63	79
Circular pin fins	1586	0.0745	6.87	92
Small teardrop pin fins	1155	0.0725	8.06	111
Large teardrop pin fins	394	0.0727	7.64	105
Tubes:				
0.085-in. ID	-	0.0454	2.14	47
0.058-in. ID	-	0.0211	1.46	69
0.030-in. ID	-	0.0057	0.75	133

quite similar; therefore, the mean linear flow velocity of fuel through the coupon and the fuel pressure drop should be comparable for the same mass flow rate.

Tests were conducted using JP-8 fuel instead of JP-8+100 to accelerate coke deposition and, thereby, reduce the test time required to evaluate the design concepts. A 0.058 in. ID tubular preheater was installed in the first copper block (to raise the fuel inlet temperature to 420°F) and the second copper block, containing the test elements, was heated to 1150°F. The tests were run until the fuel pressure drop across the test element increased significantly, indicating that complete blockage was imminent. Since the coke deposit buildup introduced an additional thermal resistance at the wall, causing the fuel temperature to decrease as the tests proceed, the fuel exit temperatures were time averaged.

Carbon deposit profiles for the large-teardrop pin-fin sector aligned in two different directions of flow are shown in Fig. 10. The data correspond to sixteen 1/2 in. segments of the test element. Reversing the orientation (i.e., flow from the sharp end of the teardrop to the blunt end) facilitates evaluation of the effects of recirculation and turbulence. More carbon deposit was found with the fuel flow direction from sharp end to blunt end, which produces a larger and more turbulent wake flow. To reduce the coke deposition, the fuel flow direction should be arranged from blunt end to sharp end for the teardrop pin fins. Consequently, only fuel flow from blunt end to sharp end was tested for the small teardrop element.

Carbon deposit profiles from tests with each of the five different fuel passages are compared in Fig. 11. Despite the differences in geometry, carbon deposition always peaks at an axial position corresponding to a fuel temperature of 550–650°F. This behavior is consistent with cylindrical tubes and is due to the consumption of dissolved oxygen and coke precursors. The test data, consisting

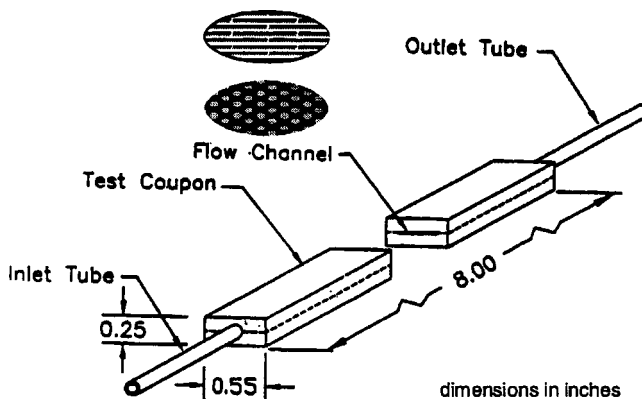


Fig. 9 Test element design

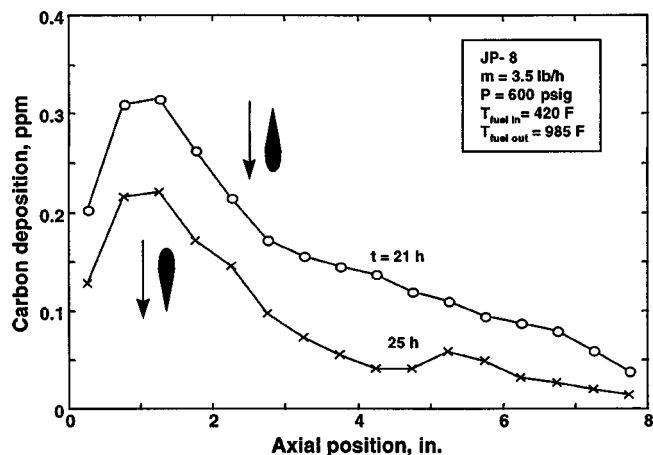


Fig. 10 Deposit profiles in large-teardrop pin fins

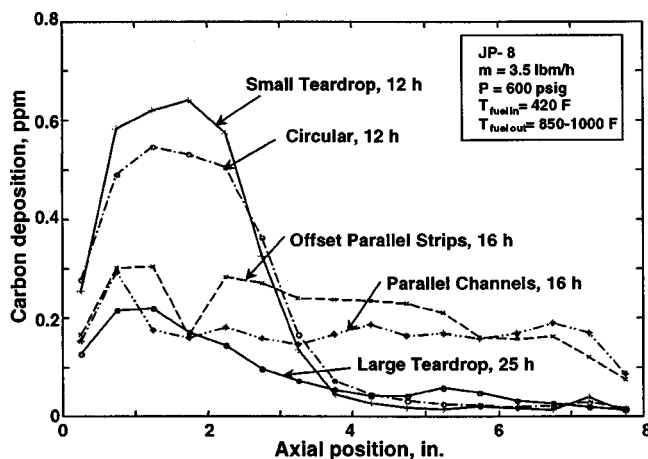


Fig. 11 Deposit profiles in coke-tolerant designs

of total carbon deposit, maximum carbon deposit rate and pressure loss, indicate that the order of increasing coke deposition for these designs is

- large teardrop with flow from blunt end to sharp end
- large teardrop with flow from sharp end to blunt end
- parallel channels
- offset parallel strip fins
- circular pin fins $\approx$ small teardrop with flow from blunt end to sharp end.

Fuel passage configuration and pin-fin geometric size can significantly impact both the autoxidative coke deposition and heat transfer. In general, for a given shape, the more turbulent configurations enhance heat transfer, but yield more coke deposit as a result of the increased mass transfer. Also, because the fins can provide higher surface area, they can increase heat transfer, thereby increasing the bulk fuel temperature and simultaneously reducing the wall temperature. The best configuration of the five designs tested is the large teardrop pin fins with the fuel flow direction from blunt end to sharp end, as it afforded the longest duration test, smallest coke deposition, and significant heat transfer enhancement.

Concluding Remarks

The rate of coke deposition results from a balance among the controlling chemical kinetics rates, the fuel residence time, and the diffusion rate of reactants and products to the surface. In turn, the kinetics rates are dependent primarily on fuel temperature. Based on the results of the current research, the following conclusions may be made regarding coke deposition/mitigation and guidelines for the design of heat exchangers for gas turbine and hypersonic applications.

- Temperature is the most important factor affecting coke deposit formation. Autoxidative deposition peaks at approximately 600°F and then decreases as a result of the consumption of dis-

solved oxygen and coke precursors. Pyrolytic deposition begins at approximately 800°F and increases monotonically.

- The transport/diffusion of reactive species to the wall (generally the location of maximum temperature and residence time) is the next most important factor affecting coke deposition for low and moderate turbulent Reynolds numbers. Increasing turbulence enhances heat transfer but also increases autoxidative coke deposition as a result of improved mass transfer. Consequently, increasing flow rate or space velocity usually increases deposit buildup in the autoxidative regime, while enlarging the flow passage (surface-to-volume ratio) reduces deposit buildup.

- Residence time is a less important factor in high-temperature heat exchangers where the coke deposition process is controlled by diffusion (i.e., low and moderate Re). However, coke deposition can be reduced by significantly increasing heat flux, while increasing flow rate at high Reynolds numbers and decreasing the residence time, the resultant higher wall temperatures notwithstanding.

- For the range of pressures examined (600 to 1200 psi), pressure is not a significant factor affecting autoxidative coke deposition.

- Techniques for providing interchannel communication and reducing the transverse temperature gradient (e.g., fins) can reduce deposit buildup. High aspect ratio streamline-shaped pin fins offer potential for lowering coke deposition, significantly enhancing heat transfer and increasing lifetime relative to continuous channels.

Acknowledgments

This paper is based on work performed for the Air Force Research Laboratory, Propulsion Directorate under Contract F33615-92-C-2295, administered by Mr. Edwin Corporan. The authors gratefully acknowledge the test support provided by Mr. John Hawkins, and the fabrication of alternative fuel passage designs by SPARTA, Inc.

References

- [1] Balster, W. J., and Jones, E. G., 1998, "Effect of Temperature on Formation of Insolubles in Aviation Fuels," *ASME J. Eng. Gas Turbines Power*, **120**, pp. 289–293.
- [2] Hazlett, R. N., 1991, *Thermal Oxidation Stability of Aviation Turbine Fuels*, ASTM Monograph 1, American Society for Testing and Materials, Philadelphia, PA.
- [3] Heneghan, S. P., Locklear, S. L., Geiger, D. L., Anderson, S. D., and Schultz, W. D., 1993, "Static Tests of Jet Fuel Thermal and Oxidative Stability," *AIAA J. of Propulsion and Power*, **9**, No. 1, pp. 5–9.
- [4] Ely, J. F., and Huber, M. L., 1990, *NIST Standard Reference Database 4—NIST Thermophysical Properties of Hydrocarbon Mixtures*, U.S. Dept. of Commerce, National Institute of Standards and Technology, Gaithersburg, MD.
- [5] Chin, J. S., and Lefebvre, A. H., 1993, "Influence of Flow Conditions on Deposits From Heated Hydrocarbon Fuels," *ASME J. Eng. Gas Turbines Power*, **115**, pp. 433–438.
- [6] Jones, E. G., Balster, W. J., and Pickard, J. M., 1996, "Surface Fouling in Aviation Fuels: An Isothermal Chemical Study," *ASME J. Eng. Gas Turbines Power*, **118**, pp. 286–291.
- [7] Sobel, D. R., and Spadaccini, L. J., 1997, "Hydrocarbon Fuel Cooling Technologies for Advanced Propulsion," *ASME J. Eng. Gas Turbines Power*, **119**, pp. 344–351.
- [8] Bradley, R., Bankhead, R., and Bucher, W., 1974, "High Temperature Hydrocarbon Fuels Research in an Advanced Aircraft Fuel System Simulator on Fuel AFFB-14-70," Report AFAPL-TR-73-95, Air Force Aero Propulsion Laboratory, WPAFB, OH.

In Situ Detailed Chemistry Calculations in Combustor Flow Analyses

S. James

M. S. Anand

M. K. Razdan

Rolls-Royce Allison,
P.O. Box 420, Speed Code T-14,
Indianapolis, IN 46206

S. B. Pope

Sibley School of Mechanical
and Aerospace Engineering,
Cornell University,
Ithaca, NY 14853

In the numerical simulation of turbulent reacting flows, the high computational cost of integrating the reaction equations precludes the inclusion of detailed chemistry schemes, therefore reduced reaction mechanisms have been the more popular route for describing combustion chemistry, albeit at the loss of generality. The in situ adaptive tabulation scheme (ISAT) has significantly alleviated this problem by facilitating the efficient integration of the reaction equations via a unique combination of direct integration and dynamic creation of a look-up table, thus allowing for the implementation of detailed chemistry schemes in turbulent reacting flow calculations. In the present paper, the probability density function (PDF) method for turbulent combustion modeling is combined with the ISAT in a combustor design system, and calculations of a piloted jet diffusion flame and a low-emissions premixed gas turbine combustor are performed. It is demonstrated that the results are in good agreement with experimental data and computations of practical turbulent reacting flows with detailed chemistry schemes are affordable.
[DOI: 10.1115/1.1384878]

Introduction

The use of detailed kinetic schemes in combustor flow calculations is crucial for the accurate prediction of emissions, ignition, lean blow-out, and other performance characteristics. A priori simplification of the detailed chemistry (typically consisting of hundreds of reactions and tens of species) into reduced chemistry schemes consisting of a few steps and species, while computationally efficient, is inadequate for these predictions. Additionally, these combustor performance characteristics are strongly influenced by turbulence/chemistry interactions and accurate prediction of these interactions is equally important. The probability density function (PDF) method offers several advantages over conventional Reynolds-averaged combustion models for the computations of turbulent reacting flows due to the fact that turbulence/chemistry interactions are described exactly by the PDF and chemical kinetics of arbitrary complexity can be incorporated without modeling assumptions. However, due to the enormous computational expense involved in calculations with detailed chemistry schemes, the majority of studies involving PDF methods employ reduced kinetic schemes ([1]) or other techniques to simplify the chemistry such as the intrinsic low-dimensional manifold (ILDM) technique ([2,3]).

The in situ adaptive tabulation scheme (ISAT) ([4]) has significantly relaxed this restriction and detailed chemistry schemes can be economically implemented in PDF calculations. Typically, calculations with reduced chemistry schemes are performed via look-ups from a precalculated table of the results of integration of the reaction rate equations. However, for detailed chemistry calculations, not only will the size of the table be prohibitively large, but also the structure of the table (i.e., independent variables and table intervals) will be too complex to accurately and conveniently represent the chemistry. On the other hand, the direct integration (DI) of the detailed reaction schemes each time it is needed in the calculations is prohibitively expensive. In the in situ tabulation procedure, the table is created as needed during the calculations,

so that only the compositional region that is appropriate to the flowfield is dynamically generated, stored, and accessed repeatedly. Calculations of a homogeneous turbulent reactor have demonstrated that the ISAT scheme provides a speed-up factor of 1000 compared to DI ([4]). ISAT has also been implemented in calculations using the joint PDF of velocity-frequency-composition ([5]) with resulting speed-ups around 50. For immediate practical applications, the scalar PDF method in which the PDF of composition is solved in conjunction with conventional finite-volume flow-solvers is advantageous, since these flow solvers are computationally affordable and are well developed and matured for complex geometries.

The present study combines the ISAT scheme with a scalar PDF method in a practical combustor design system. The design system is based on a finite-volume solver, which is a conventional pressure-based low-speed variable-density Navier-Stokes solver with a standard $k-\epsilon$ model. The scalar PDF transport equation is solved by an Eulerian Monte-Carlo method, wherein, the PDF at each node is represented by an ensemble of notional particles. This design system in conjunction with reduced chemical kinetics has been applied to diffusion flame swirl combustors and premixed low-emissions combustors ([6,7]). In the present paper, PDF/ISAT calculations of a jet diffusion flame are compared to detailed experimental data including those for the species compositions. Similar calculations are performed on a premixed low-emissions combustor. The successful application of this method for practical gas turbine concepts, such as a low emissions combustor, demonstrates the capability to make detailed species predictions and to optimize combustor design to meet stringent emissions and performance requirements of current and future gas turbine engines. In the following sections, first we briefly describe the PDF method, followed by a description of the ISAT scheme. The results and discussions are presented next, followed by summary and conclusions.

The Scalar Probability Density Function (PDF) Method

The governing equation for the transport of the joint probability density function $P(\Psi; \mathbf{x}, t)$ of the scalars Φ at position $\mathbf{x}$ and t is ([8–10])

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-271. Manuscript received by IGTI Oct. 1998; final revision received by the ASME Headquarters Mar. 1999. Associate Editor: D. Wisler.

$$\frac{\partial}{\partial t}(\langle \rho \rangle P) + \frac{\partial}{\partial x_i}(\langle \rho \rangle U_i P) = \frac{\partial}{\partial \psi_\alpha} \left[\langle \rho \rangle \left\langle \frac{1}{\rho} \frac{\partial J_i^\alpha}{\partial x_i} \middle| \Psi \right\rangle P \right] - \frac{\partial}{\partial x_i}[\langle \rho \rangle \times \langle u_i'' | \Psi \rangle P] - \frac{\partial}{\partial \psi_\alpha}[\langle \rho \rangle S_\alpha P] \quad (1)$$

where t is the time, x_i is the i th component of the position vector, ρ is the density, U_i is the i th component of the velocity vector, and ψ_α is the mass fraction of species α . J_i^α is the i th component of the diffusional flux for species α , u_i'' is the i th component of the fluctuating velocity, S_α is the source term of species α , Ψ is the set of scalar variables in composition space, and Φ is the set of scalar variables in physical space. The notations, $\langle A \rangle$ and $\langle A | B \rangle$, denote the expected value of A and the conditional expectation of A given B , respectively. In the above equation, the terms on the left-hand side are the time rate of change and convection by the mean velocity, respectively, of the PDF and are in a closed form. The first term on the right-hand side is the influence of molecular mixing on the transport of the PDF in scalar space. The second term on the right-hand side represents the transport of the PDF in physical space due to convection by turbulent velocities. These two terms need to be modeled. The last term, which is the effect of reaction source term on the evolution of the PDF in scalar space, is also closed.

Modeling and Discretization. The transport of the PDF in physical space due to turbulent convection is modeled by the gradient diffusion hypothesis ([8])

$$\langle \rho \rangle \langle u_i'' | \Psi \rangle P = -\langle \rho \rangle D_T \frac{\partial P}{\partial x_i}, \quad (2)$$

where D_T is the turbulent diffusivity. Molecular mixing is modeled by the “interaction by exchange with the mean” (IEM) model:

$$\langle \rho \rangle \left\langle \frac{1}{\rho} \frac{\partial J_i^\alpha}{\partial x_i} \middle| \Psi \right\rangle P = \langle \rho \rangle C_\phi \omega (\psi_\alpha - \langle \phi_\alpha \rangle) P \quad (3)$$

where C_ϕ is the mixing model constant and ω is the turbulent frequency, defined as the ratio of the turbulent dissipation rate and the turbulent kinetic energy. Past studies on jet diffusion flames (in particular, Flame L of Masri et al., [11]) have shown that the IEM model fails to sustain the flame. Efforts are in progress to develop advanced models ([12]) to better represent the mixing process. In the present study, it is noted that the IEM model is adequate to sustain the flame and is therefore, adopted in all the simulations. The final modeled form of the PDF transport equation is

$$\begin{aligned} \frac{\partial}{\partial t}(\langle \rho \rangle P) + \frac{\partial}{\partial x_i}(\langle \rho \rangle U_i P) = & \frac{\partial}{\partial \psi_\alpha} [\langle \rho \rangle C_\phi \omega (\psi_\alpha - \langle \phi_\alpha \rangle) P] \\ & + \frac{\partial}{\partial x_i} \left(\langle \rho \rangle D_T \frac{\partial P}{\partial x_i} \right) \\ & - \frac{\partial}{\partial \psi_\alpha} (\langle \rho \rangle S_\alpha P). \end{aligned} \quad (4)$$

For obtaining the discretized form of the above equation in the finite-volume context, the volume integrals for convection and diffusion terms are first converted to surface integrals using the Gauss theorem and the PDF in each cell is discretized by a hybrid differencing scheme that varies smoothly from an upwind scheme to a central scheme. The resulting equation is

$$\begin{aligned} P_i(t + \Delta t_i) = & P_i(t) + \sum (C_{iK} + D_{iK})(P_K - P_i) \\ & + \Delta t_i \frac{M_i(t)}{M_i(t + \Delta t_i)} \frac{\partial}{\partial \psi_\alpha} [C_\phi \omega (\psi_\alpha - \langle \phi_\alpha \rangle) P_i] \\ & - \Delta t_i \frac{M_i(t)}{M_i(t + \Delta t_i)} \frac{\partial}{\partial \psi_\alpha} (\langle \rho \rangle S_\alpha P_i), \end{aligned} \quad (5)$$

where the index K denotes the faces of cell i and the index k refers to the cell adjacent to cell i across face K . The terms D_{iK} and C_{iK} are the diffusive and convective coefficients and are evaluated as discussed in Anand et al. [2]. M_i is the mass in cell i and Δt_i is the time-step in cell i . Equation (5) is solved by a fractional step method in which the transport in physical space is first implemented followed by molecular mixing and chemical reaction. The PDF in each cell is represented by an ensemble of notional particles, each having identical mass and a set of scalar values. The Favre average value of a scalar in a cell is computed by performing an ensemble average of the scalar value on all the particles in the cell. In the present calculations, each cell has an equal number of particles. A variable time-step algorithm Anand et al. [2] is used to accelerate convergence of the solution.

The In Situ Adaptive Tabulation (ISAT) Scheme

The objective of the ISAT scheme ([4]) is to efficiently integrate the coupled set of reaction rate equations:

$$\frac{d\phi_\alpha(t)}{dt} = S_\alpha[\Phi(t)], \quad (6)$$

i.e., given the initial condition Φ^0 , the reaction rate equation is integrated for a time Δt , to obtain $\Phi(t_0 + \Delta t)$. A table is built as the reactive flow calculation progresses, in such a way that only the accessed region of the composition space is tabulated. Since the accessed region is a small subset of the realizable region, the dynamic creation of the table enables efficient integration of the reaction equations than when the table is created a priori and the entire realizable region is tabulated. A table entry consists of a composition Φ^0 ; the mapping $R(\Phi^0)$; the gradient of the mapping $A(\Phi^0)$; and the specification of the ellipsoid of accuracy (EOA). The gradient of the mapping is used in the linear approximation of the mapping

$$R(\Phi^q) \approx R(\Phi^0) + A(\Phi^0)(\Phi^q - \Phi^0) \quad (7)$$

for a query Φ^q . The EOA is an ellipsoidal region, centered at Φ^0 , within which the above linear approximation is accurate. As the calculation progresses, the EOA expands such that the error involved in Eq. (7) is within a user specified tolerance (ϵ_{sp}). The entries in the table are stored in a binary tree which, given a query composition, Φ^q , can be traversed to obtain a table entry Φ^0 , which is close to Φ^q . The essential ingredients in the algorithm are summarized as follows:

- 1 Given a query composition Φ^q , the binary tree is traversed until Φ^0 is reached.
- 2 If Φ^q is within the EOA, then the linear approximation Eq. (7) is returned. This is a *retrieve* (R).
- 3 Otherwise, a direct integration is performed to determine the mapping $R(\Phi^q)$, and the error ϵ that would result from using Eq. (7) is determined.
- 4 If the error is within the tolerance, then the EOA is grown and $R(\Phi^q)$ is returned. This outcome is *growth* (G).
- 5 Otherwise a new record is generated based on Φ^q . This outcome is an *addition* (A).

Results and Discussions

The two-dimensional finite-volume solver used in the present calculations is a conventional pressure-based low-speed variable-

Table 1 16-species 41-step mechanism

Reaction-Step	A mole, cm ³ , sec	n	E cal/mole
H+O ₂ =OH+O	1.59E+17	-0.927	16874.
O+H ₂ =OH+H	3.87E+04	2.70	6262.
OH+H ₂ =H ₂ O+H	2.16E+08	1.51	3430.
OH+OH=O+H ₂ O	2.10E+08	1.40	-397.
H+H+M=H ₂ +M	6.40E+17	-1.0	0.
H+OH+M=H ₂ O+M	8.40E+21	-2.00	0.
H+O ₂ +M=HO ₂ +M	7.00E+17	-0.80	0.
HO ₂ +H=OH+OH	1.50E+14	0.0	1004.
HO ₂ +H=H ₂ +O ₂	2.50E+13	0.0	693.
HO ₂ +O=O ₂ +OH	2.00E+13	0.0	0.
HO ₂ +OH=H ₂ O+O ₂	6.02E+13	0.0	0.
H ₂ O ₂ +M=OH+OH+M	1.00E+17	0.0	45411.
CO+OH=CO ₂ +H	1.51E7	1.3	-758.
CO+O+M=CO ₂ +M	3.01E+14	0.0	3011.
HCO+H=H ₂ +CO	7.23E+13	0.0	0.
HCO+O=OH+CO	3.00E+13	0.0	0.
HCO+OH=H ₂ O+CO	1.00E+14	0.0	0.
HCO+O ₂ =HO ₂ +CO	4.20E+12	0.0	0.
HCO+M=H+CO+M	1.86E+17	-1.0	16993.
CH ₂ O+H=HCO+H ₂	1.26E+08	1.62	2175.
CH ₂ O+O=HCO+OH	3.50E+13	0.0	3513.
CH ₂ O+OH=HCO+H ₂ O	7.23E+05	2.46	-970.
CH ₂ O+O ₂ =HCO+HO ₂	1.00E+14	0.0	39914.
CH ₂ O+CH ₃ =HCO+CH ₄	8.91E-13	7.40	-956.
CH ₂ O+M=HCO+H+M	5.00E+16	0.0	76482.
CH ₃ +O=CH ₂ O+H	8.43E+13	0.0	0.
CH ₃ +OH=CH ₂ O+H ₂	8.00E+12	0.0	0.
CH ₃ +O ₂ =CH ₃ O+O	4.30E+13	0.0	30808.
CH ₃ +O ₂ =CH ₃ O+OH	5.20E+13	0.0	34895.
CH ₃ +HO ₂ =CH ₃ O+OH	2.28E+13	0.0	0.
CH ₃ +HCO=CH ₄ +CO	3.20E+11	0.50	0.
CH ₄ (+M)=CH ₃ +H(+M)	6.3E14	0.0	10 4000.
CH ₄ +H=CH ₃ +H ₂	7.80E+06	2.11	7744.
CH ₄ +O=CH ₃ +OH	1.90E+09	1.44	8676.
CH ₄ +O ₂ =CH ₃ +HO ₂	5.60E+12	0.0	55999.
CH ₄ +OH=CH ₃ +H ₂ O	1.50E+06	2.13	2438.
CH ₄ +HO ₂ =CH ₃ +H ₂ O ₂	4.60E+12	0.0	17997.
CH ₃ O+H=CH ₂ O+H ₂	2.00E+13	0.0	0.
CH ₃ O+OH=CH ₂ O+H ₂ O	5.00E+12	0.0	0.
CH ₃ O+O ₂ =CH ₂ O+HO ₂	4.28E-13	7.60	-3528.
CH ₃ O+M=CH ₂ O+H+M	1.00E+14	0.0	25096.

density Navier-Stokes solver ([13]) with a standard $k-\epsilon$ model. The solver employs a staggered grid, with the scalar quantities located at the centers of the cells and the velocities at the cell boundaries. A power-law differencing scheme is used to discretize the governing equations. The finite-volume solver provides the flux to the PDF module which simulates the transport of the scalars. The density change due to turbulent combustion is computed within the PDF module and supplied to the flow-solver and calculations are performed until convergence is achieved. In all the cases studied, the PDF solver employs 50 particles per cell and the value of the mixing model constant is 2.0. Calculations are performed using two chemistry mechanisms. Table 1 shows a C_1 mechanism involving 16 species and 41 reactions ([14]), where A , n , and E are the frequency factor, the pre-exponential exponent and the activation energy, respectively. Table 2 shows a 12-step,

Table 2 16-species 12-step augmented reduced mechanism

O ₂ +2CO=2CO ₂
H+O ₂ +CO=OH+CO ₂
H ₂ +O ₂ +CO=H+OH+CO ₂
HO ₂ +CO=OH+CO ₂
O ₂ +H ₂ O ₂ +CO=OH+HO ₂ +CO ₂
O ₂ +0.50C ₂ H ₂ =H+CO ₂
O ₂ +CH ₃ +CO+C ₂ H ₄ =CH ₄ +CO ₂ +CH ₂ O+0.50C ₂ H ₂
O ₂ +2CH ₃ =H ₂ +CH ₄ +CO ₂
O ₂ +2CH ₃ +CO=CH ₄ +CO ₂ +CH ₂ O
O ₂ +CH ₃ +CO=H+CO ₂ +CH ₂ O
O ₂ +CO+C ₂ H ₆ =CH ₄ +CO ₂ +CH ₂ O
H+OH=H ₂ O

16-species augmented C_2 reduced mechanism (ARM) ([15]) which is derived from the GRI mechanism ([16]). The flow configurations studied are a piloted jet diffusion flame and a premixed low-emissions gas turbine combustor.

Piloted Jet Diffusion Flame. The flow configuration considered is the piloted jet diffusion flame configuration (Flame D) ([17]) where the radius of the central jet is 3.6 mm and that of the outer annulus of the pilot is 9.1 mm. The fuel jet (having a mixture fraction (Z) value equal to 1) consists of 25 percent methane and 75 percent air by volume and the outer coflow is air ($Z = 0$). The pilot burns a mixture of C₂H₂, H₂, CO₂, N₂ and air, having nominally the same equilibrium composition and enthalpy as CH₄/air. The co-flow velocity and temperature are 0.9 m/s and 291 K, respectively. The main jet bulk velocity and temperature are 49.6 m/s and 294K, respectively. The pilot stream has a velocity of 11.4 m/s and a mixture fraction value of 0.27. Calculations are performed on a rectangular domain with 152 cells in the x -direction and 72 cells in the y -direction. The inlet conditions for the velocity and the scalars are specified from the measurements. Free slip boundary conditions are imposed on the top boundary and zero derivative outflow boundary conditions are imposed at the outflow. Symmetry boundary conditions are imposed at the lower boundary (centerline).

A parametric study on the effect of the ISAT error tolerance on the flowfield indicated that a value of 0.008 is adequate to resolve the reacting field of the jet flame. Another study of the turbulent Schmidt number indicates that a value of 2.0 produced the best predictions. For the $k-\epsilon$ constants, the standard values were used for C_{μ} (=0.09) and $C_{\epsilon 2}$ (=1.92), but for $C_{\epsilon 1}$ the standard value of 1.44 as well as a modified value of 1.6 were used for reasons explained later.

Figure 1 presents the contour plot of the Favre temperature in the flowfield with the C_1 chemistry and $C_{\epsilon 1} = 1.6$. It can be noted that just downstream of the pilot, the reaction zone thickness is decreased due to finite-rate chemistry effects. Further downstream, these effects relax and the reaction zone thickness increases. Radial profiles of the mixture fraction f at $x/d = 7.5$, 15, and 30 are presented in Fig. 2. As expected, computations with the standard values of the $k-\epsilon$ constants, produce a higher spread rate. This is a common feature observed in computations of round jets and a modified value of $C_{\epsilon 1} = 1.6$ ([18]) is recommended for this type of flow as a special case. It can be observed that calculations with the modified value produce the correct spread rate. It is also noted that the ARM calculations (C_2 chemistry) with $C_{\epsilon 1} = 1.6$ accurately predict the mean mixture fraction.

Figure 3 presents the radial profiles of the Favre temperature. Consistent with the results from Fig. 2, results for $C_{\epsilon 1} = 1.44$ show a higher spread rate at all the three axial locations. In contrast, calculations with the modified value for both chemistry mechanisms are in excellent agreement with the data. Similar features are also observed in the radial profiles of the major species. Figure 4 presents the radial profiles of CO₂ mass fraction. Calculations with the modified value of $C_{\epsilon 1}$ are in reasonable agreement with the data at all the three axial locations for both reaction mechanisms. Figure 5 presents the radial profiles of the CO mass fraction. It can be observed that the computation with the C_1 mechanism and $C_{\epsilon 1} = 1.44$ overpredicts the peak value and displays a high spread rate at all the three axial locations. In contrast, the C_1 mechanism calculations with the modified value produce the correct spread rate but significantly overpredict the peak value, and at $x/d = 30$ the computed peak value occurs incorrectly at the centerline. The cause of this inaccuracy may be due to the deficiency of the C_1 mechanism for correctly predicting the flame structure in the fuel-rich zones. Calculations performed with the C_2 chemistry (ARM) ($C_{\epsilon 1} = 1.6$) are in reasonable agreement with the data, which indicates that C_2 chemistry is essential to accurately predict CO emissions. Other sources of error in the computations may be due to the neglected influence of radiation heat loss

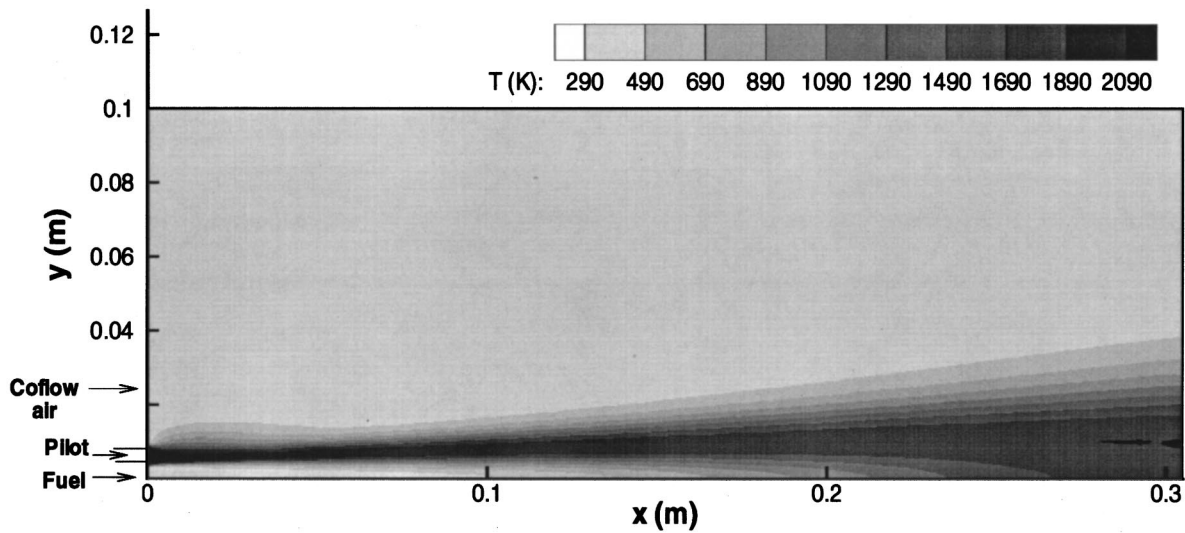


Fig. 1 Contours of the Favre mean temperature in the flowfield

(which reduces the temperature and the CO level), the mixing model used, and the deficiencies in the flow solution (spread rate/turbulent transport).

Figure 6 presents the variance of the mixture fraction at three axial locations. Although the calculations with $C_{\epsilon 1}=1.6$ produce the correct spread rate, the peak values are not in good agreement with the data at $x/d=15$ and 30. This may be due to deficiencies in the mixing model and the errors/uncertainties involved in the

prescription of the inlet turbulent kinetic energy and the turbulent dissipation profiles. The root mean square (RMS) value of temperature is presented in Fig. 7. As with the other quantities the calculation with $C_{\epsilon 1}=1.6$ is in excellent agreement with the data in regard to the spread rate. Additionally, the peak values are also in good agreement with the data. In comparison, the calculation with the standard value of $C_{\epsilon 1}$ highly overpredicts the spread rate and at $x/d=30$, the RMS profile does not even have the qualita-

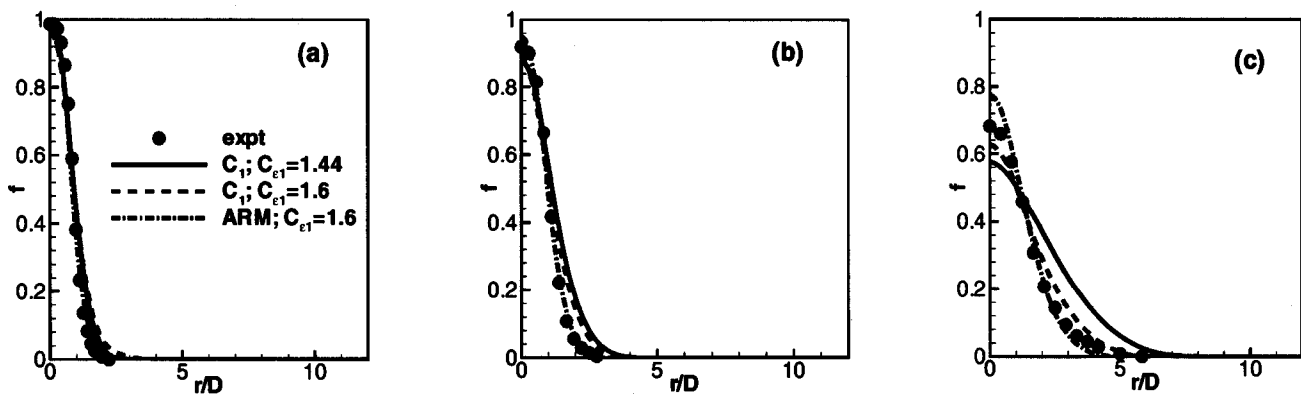


Fig. 2 Radial profiles of mean mixture fraction from calculations with the C_1 and the arm mechanism; (a) $X/D=7.5$, (b) $X/D=15$, and (c) $X/D=30$

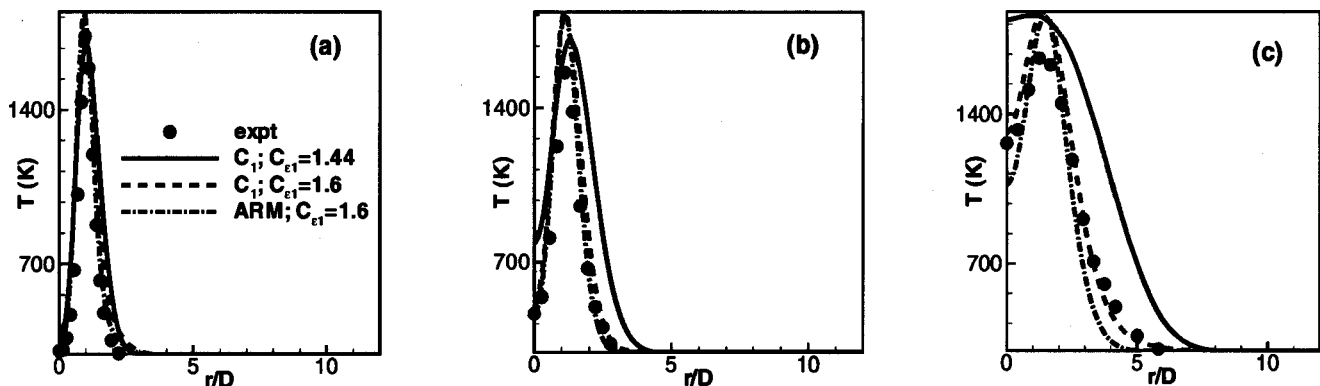


Fig. 3 Radial profiles of mean temperature; (a) $X/D=7.5$, (b) $X/D=15$, and (c) $X/D=30$

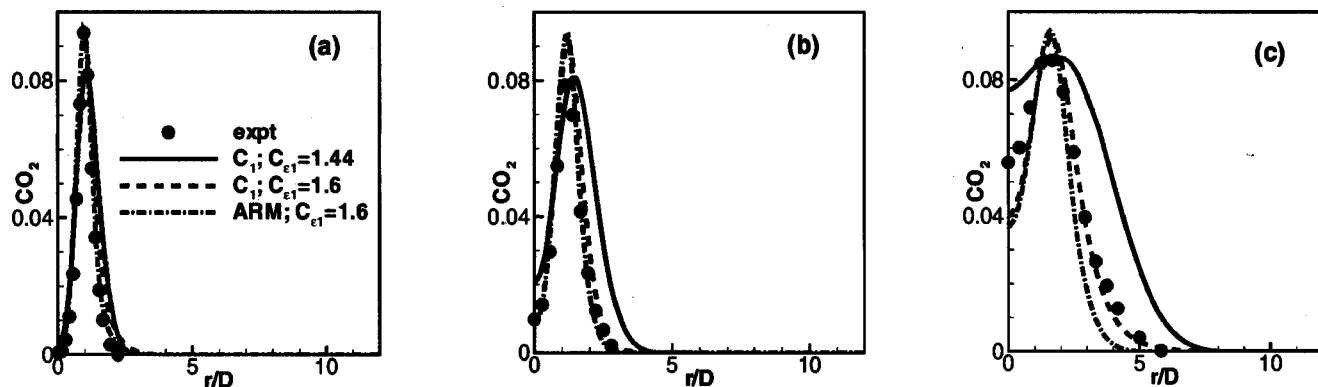


Fig. 4 Radial profiles of mean CO_2 mass fraction; (a) $X/D=7.5$, (b) $X/D=15$, and (c) $X/D=30$

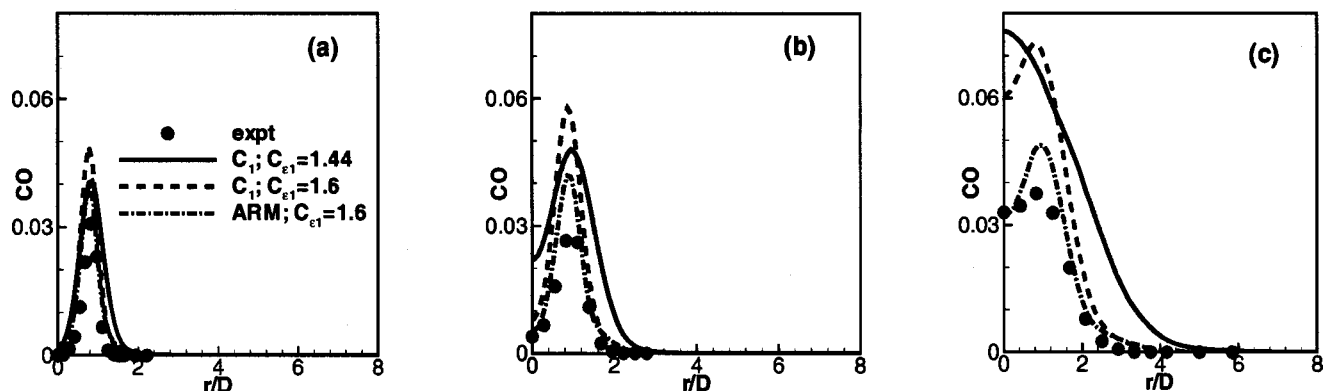


Fig. 5 Radial profiles of mean CO mass fraction; (a) $X/D=7.5$, (b) $X/D=15$, and (c) $X/D=30$

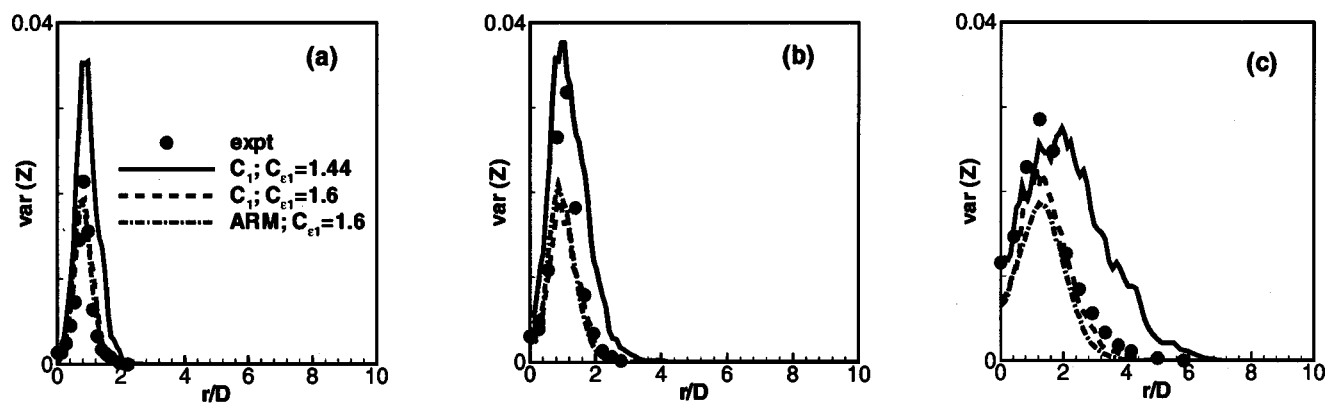


Fig. 6 Radial profiles of the variance of mixture fraction; (a) $X/D=7.5$, (b) $X/D=15$, and (c) $X/D=30$

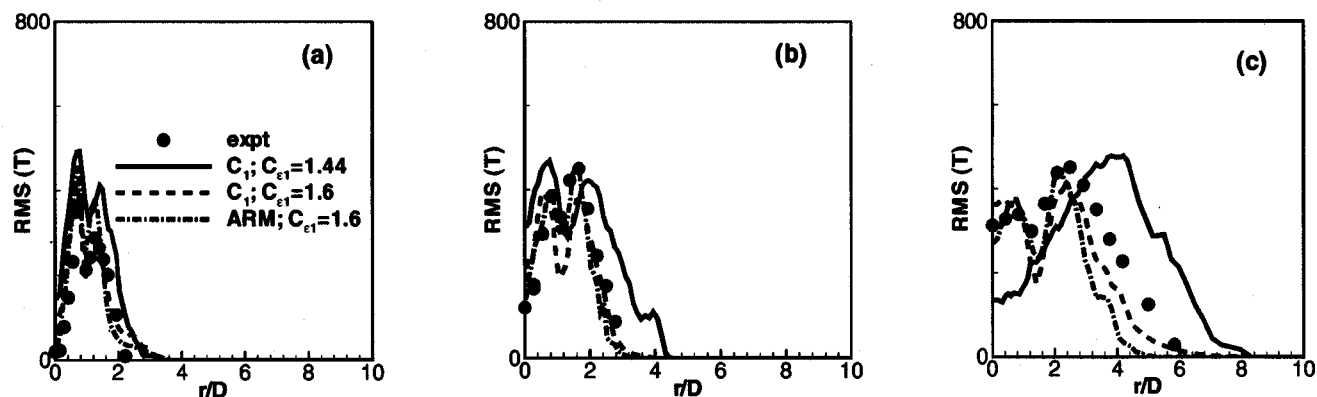


Fig. 7 Radial profiles of the rms of T ; (a) $X/D=7.5$, (b) $X/D=15$, and (c) $X/D=30$

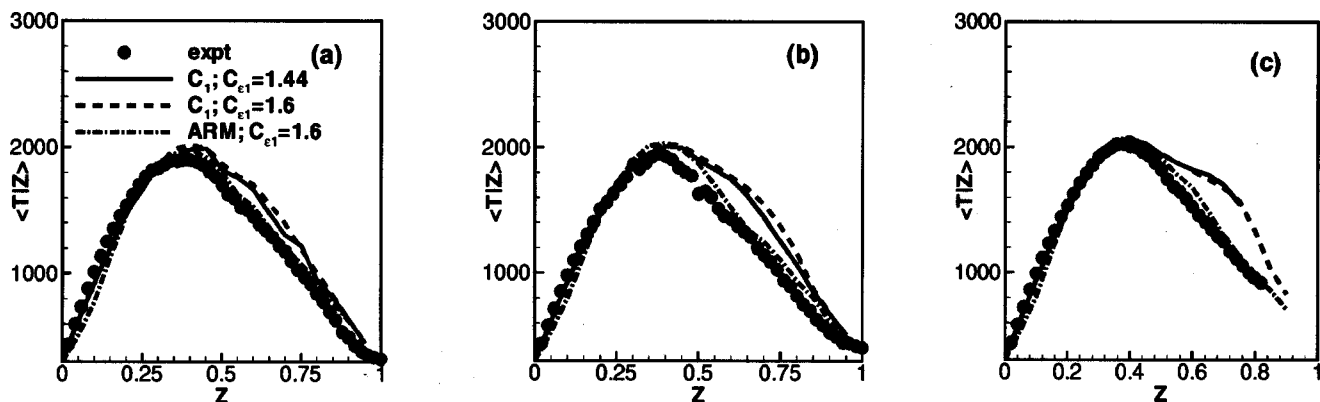


Fig. 8 Conditional expectation of T ; (a) $X/D=7.5$, (b) $X/D=15$, and (c) $X/D=30$

tive feature of the data. The results from the C_2 mechanism are similar to those from the C_1 mechanism with the modified value of $C_{\epsilon 1}$.

The main differences between the C_1 and the C_2 chemistry can be clearly observed from the expected values of species mass fraction and temperature conditioned on mixture fraction. Figure 8 shows the conditional expectation of temperature versus mixture fraction at three axial locations. It can be observed that all the three computations predict the fuel-lean region quite accurately. The fuel-rich region is poorly predicted in the calculation with the C_1 chemistry mechanism, however, the predictions with the C_2 mechanism is excellent. This feature of the two mechanisms is clearly portrayed in Fig. 9 which shows the conditional expectation of CO mass fraction. The C_1 mechanism highly overpredicts the fuel-rich regime; in contrast, the prediction of the C_2 mechanism is reasonable. These results clearly indicate that accurate prediction of CO emissions can be possible only if C_2 chemistry is used in the computations. The prediction of the FLAME D composition structure has been the subject of various investigations which have been summarized in a recent workshop ([19]). A comparison of the results from PDF calculations presented in the workshop with those from the present calculations indicates that overall the flame structure is better predicted by the PDF/ISAT method with the ARM mechanism and $C_{\epsilon 1}=1.6$.

Premixed Low-Emissions Gas Turbine Combustor. The PDF method with detailed chemistry is applied to a practical gas turbine combustor to demonstrate the feasibility of making detailed performance and emissions calculations. Further, a premixed combustor is chosen since conventional turbulent combustion models fail to predict premixed combustion behavior, and

specialized models need to be used. In contrast, the PDF method is equally applicable to nonpremixed and premixed flames.

A radial swirl premixed combustor ([20]) consists of a radial swirl inlet with fuel injectors, a premixer module and the combustor. The flame is stabilized by a recirculating zone outside of the premixer nozzle induced by the swirl motion of the fluid. Bench tests have been performed on this radial swirl combustor and measurements have been obtained for velocity and temperature in the combustor section. The tests were performed with an air flow rate of 0.0635 Kg/s, and an inflow fuel to air ratio of 0.037. The inflow air was preheated to 373 K. The numerical simulation of the combustor is carried out from the exit of the swirler vane and the fuel injector to the exit of the combustor. The fuel injectors are simulated using six discrete fuel injection ports at the inlet and calculations are performed on a 151×31 grid. In the calculations, the ignition heat source is simulated by introducing high-temperature combustion products downstream of the premixer nozzle. In order to establish a stable flame, ignition is switched on after a swirl induced recirculating zone is established in the flow calculations and turned off after one subsequent timestep of the calculations. The standard values of the $k-\epsilon$ constants are used in these computations. Calculations indicated that an ISAT error tolerance value of 0.008 results in approximately 18 table entries, which is not adequate for an accurate resolution of the flame structure. Therefore, calculations are performed with a tolerance value of 0.001. The C_1 chemistry mechanism is used to demonstrate the capability of ISAT for making computations of practical gas turbines combustors with detailed mechanisms.

Analysis of the mixture fraction field shows that mixing is complete by the time the flow reaches the exit of the premixer where

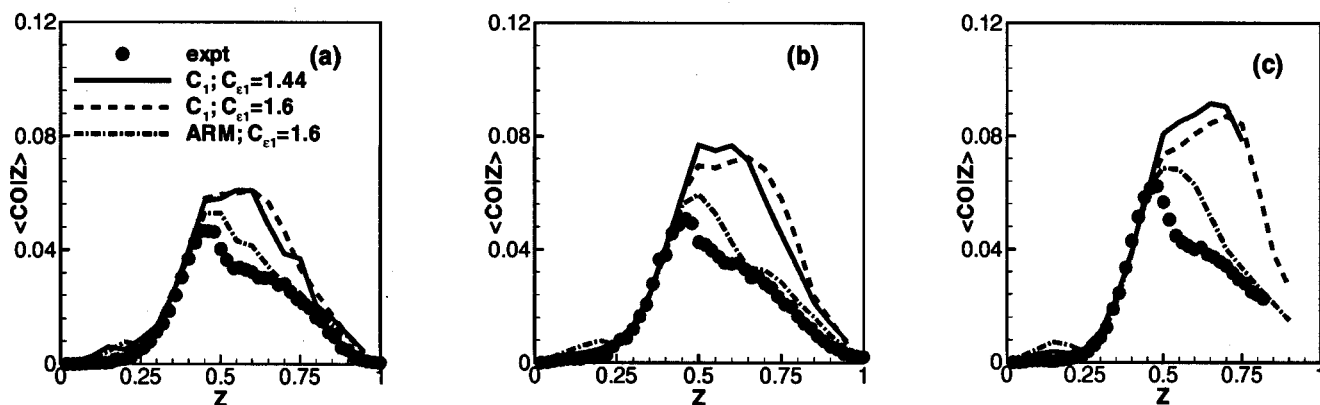


Fig. 9 Conditional expectation of CO; (a) $X/D=7.5$, (b) $X/D=15$, and (c) $X/D=30$

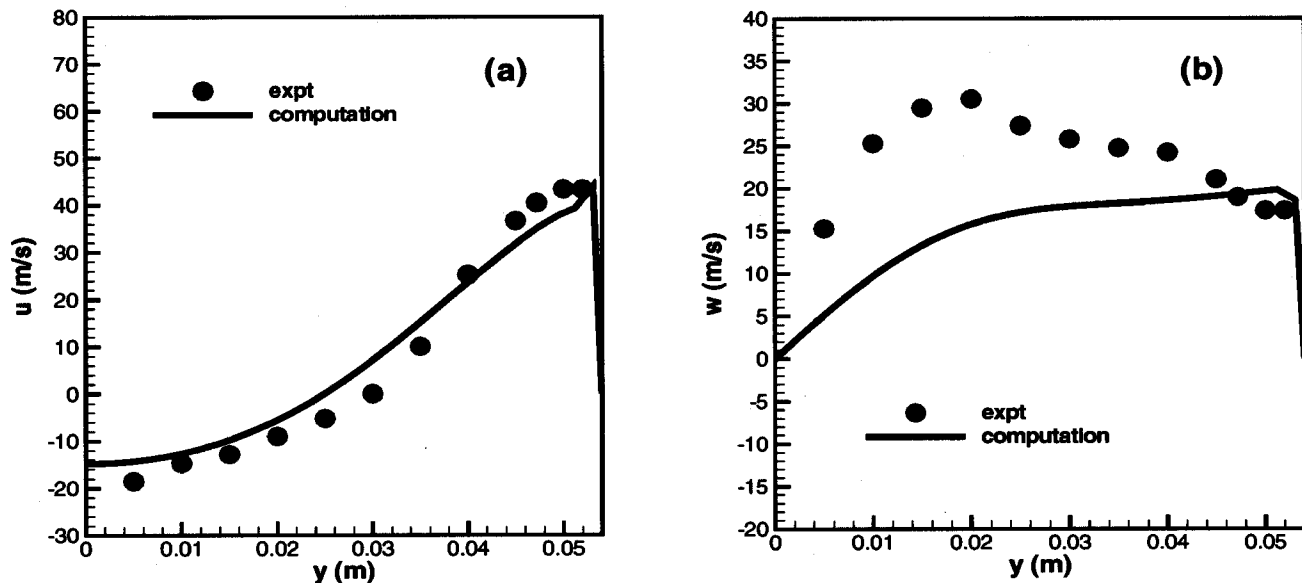


Fig. 10 Cross-stream profiles of (a) mean axial velocity and (b) mean swirl velocity at $X=0.221$ m

a nearly uniform fuel distribution is found. The swirling motion of the fluid induces a recirculating region near the centerline, downstream of the premixer. Figures 10 and 11 show the cross-stream profiles of the axial and swirl velocities at $x=221$ and 297 mm, respectively. Although the axial velocities are in good agreement with data, discrepancies in the prediction of the swirl velocities are observed. This is due to the inability of the $k-\epsilon$ model to correctly simulate recirculating flows ([21]) indicating that improved turbulence models are needed to accurately predict these flows. The temperature contours of Fig. 12 show that the flame front is stabilized outside of the premixer nozzle by the recirculating zone and stands in front of the stagnation point of this zone. Comparison of the predicted temperature with experimental data shows excellent agreement at $x=221$ mm (Fig. 13(a)) and at $x=297$ mm (Fig. 13(b)). These results indicate that the present methodology is capable of accurately predicting realistic combustor characteristics, including turbulent mixing, flow and flame

structure and the flame stabilization location. In contrast, calculations using the eddy breakup model (EBU) fail to even qualitatively predict the behavior of the combustor—the EBU model failed to stabilize the flame outside the premixer and always showed flashback ([7]). The detailed mechanism implemented in the present calculations allows the evaluation of minor species in the flowfield. Due to the lack of experimental data, the accuracy of the predictions of minor species cannot be verified.

ISAT Performance. The efficiency of the ISAT in the scalar PDF calculations is investigated in this section. Results of the regression analyses of the performance data for the two flow configurations studied here are presented in Table 3, where, Δt_{ret} is the retrieve time, Δt_{gro} is the time for growth of the EOA and Δt_{add} is time for the addition of a new record. The analysis is performed for an ISAT error tolerance of 0.008. It can be observed that for both cases studied here, Δt_{ret} is the lowest and Δt_{add} is the

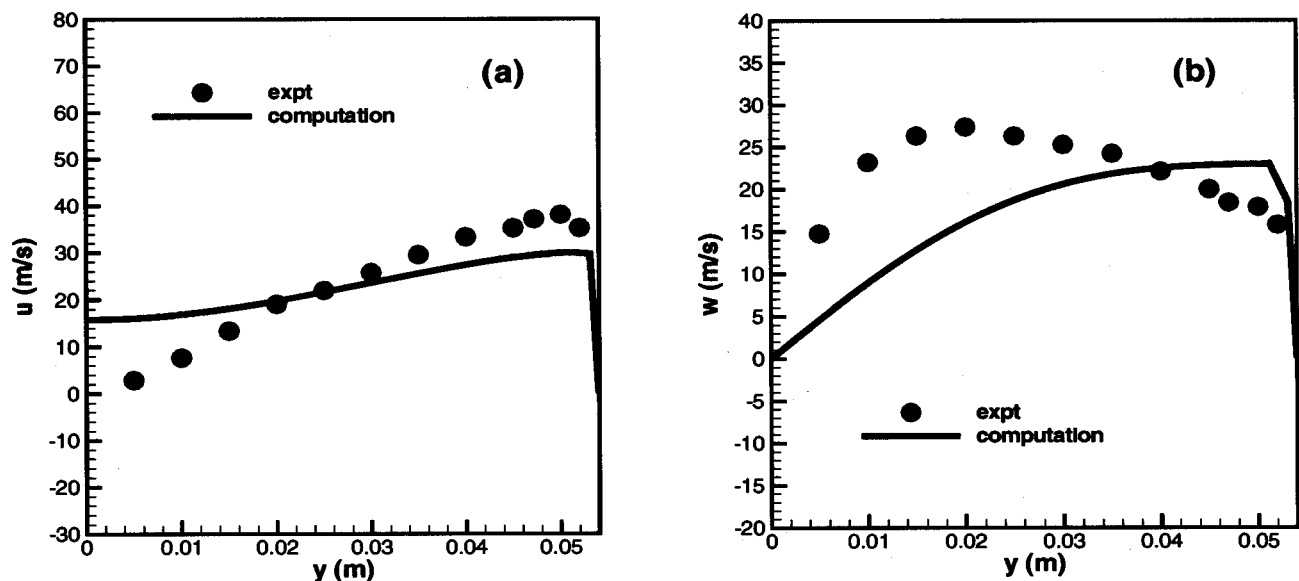


Fig. 11 Cross-stream profiles of (a) mean axial velocity and (b) mean swirl velocity at $X=0.297$ m

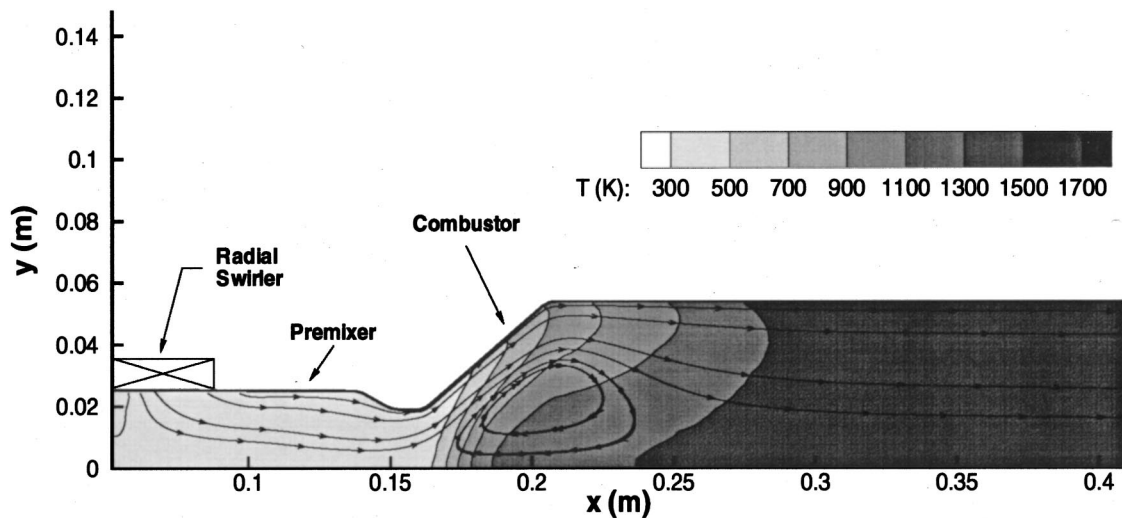


Fig. 12 Contours of the Favre mean temperature and sample steamtraces in the premix combustor

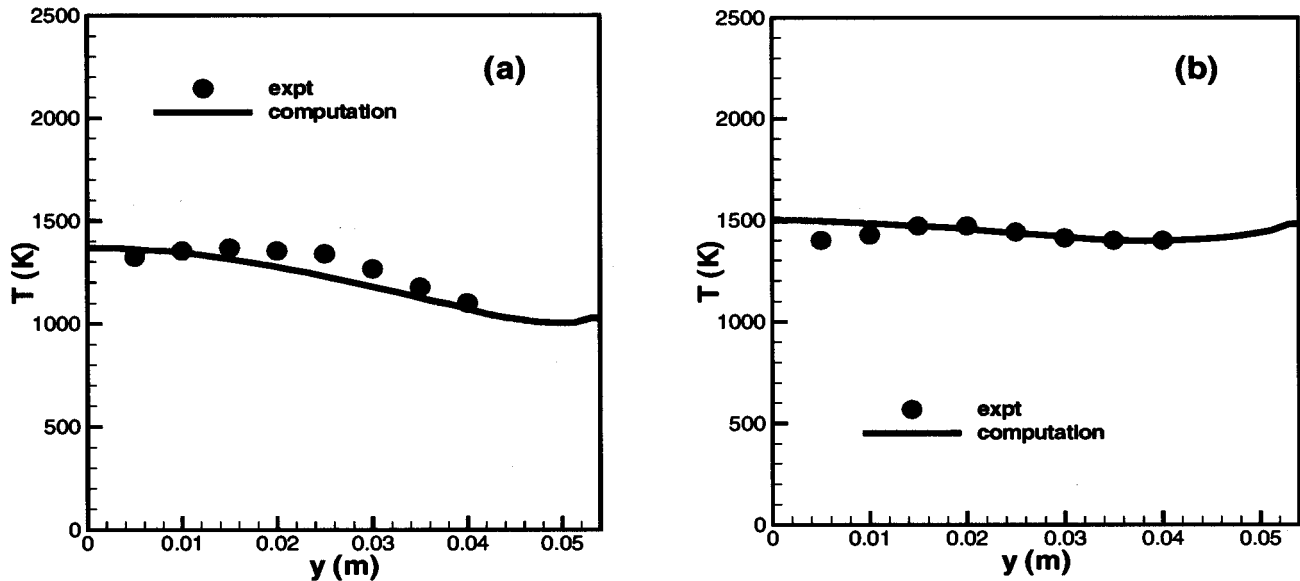


Fig. 13 Cross-stream profiles of mean temperature at (a) $X=0.221$ m and (b) $X=0.297$ m

highest. The asymptotic value of the speed-up factor, defined as the ratio of growth to retrieve times, is at least two orders of magnitude higher for the premix combustor calculations.

Figure 14 shows the evolution of the number of records created versus calls to ISAT. It is observed that for an ISAT error tolerance value of 0.008, approximately, 1400 table entries are created in the jet flame computations. In contrast, for the same tolerance value, only 18 entries are created in the combustor calculations. Decreasing the tolerance to 0.001, increases the table entries to 288. Due to the premixed nature of the combustor, the region downstream of the nozzle has an almost uniform composition, therefore the accessed region of the compositional space is significantly smaller than that in the jet diffusion flame.

A measure of the cumulative CPU time spent in ISAT for the jet flame calculations can be obtained from Fig. 15 which shows the fraction of time spent in ISAT versus the number of calls. It can be observed that after 10^6 calls, almost 60 percent of the time is spent in evaluating chemistry. For the premix combustor, approximately, 80 percent of the time is spent in chemistry. The evolution of the

speed-up factor which is the ratio of the time taken to perform the calculations using DI to that using ISAT, is presented in Fig. 16. It can be observed that after the initial transients, the curve flattens to a value of approximately 117. Values close to 200 have been observed in previous joint velocity-scalar PDF calculations. Although the storage requirement for the coarse tolerance level used in the jet flame calculations is not significant, it can become a serious issue when lower tolerance levels are required. Correspondingly, the CPU time will also increase for simulations with a lower tolerance value.

Table 3 Regression statistics

Geometry	Δt_{ret} sec	Δt_{gro} sec	Δt_{add} sec	Speed-up
Diffusion flame	0.000103	0.012	0.428	117
Premix combustor	0.000109	0.018	0.784	165

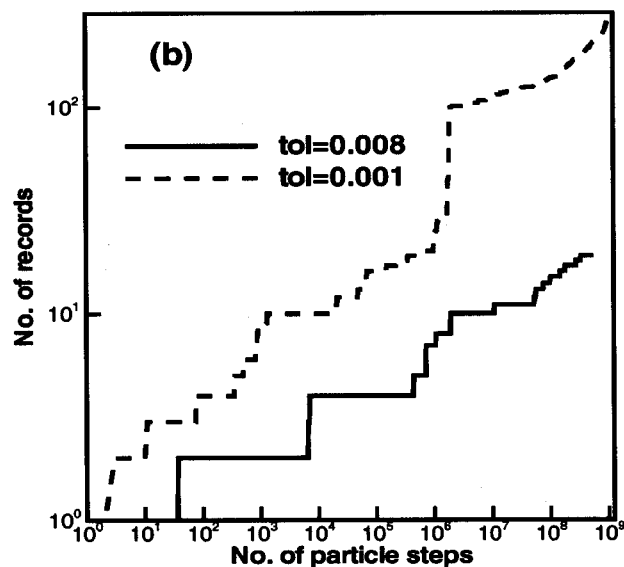
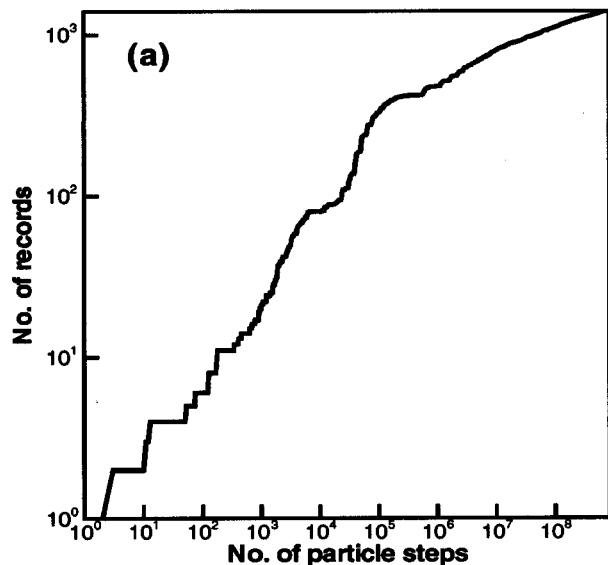


Fig. 14 Evolution of the number of records; (a) jet diffusion flame and (b) premix combustor

Summary and Conclusions

The present paper demonstrates the application of detailed chemical kinetics in scalar PDF computations of a jet diffusion flame and a practical premixed low-emissions gas turbine combustor. The jet diffusion flame calculations with a modified value of one of the $k-\epsilon$ constants ($C_{\epsilon 1}=1.6$) and C_1 chemistry, produce good agreement of the major species mass fractions and temperatures with the experimental data. For accurate prediction of CO emissions, it has been shown that C_2 chemistry schemes have to be used. Computations of the premixed swirl combustor also indicate good agreement with experimental data for temperatures and velocities. The behavior of the combustor including the location and extent of the flame are well predicted.

Performance analyses of ISAT show that speed-up factors of 117 and 165 are achieved in the computations for the jet diffusion flame and the premix combustor, respectively. The cumulative fraction of time spent in evaluating chemistry is 0.6 and 0.8 for the jet diffusion flame and the premixed combustor, respectively. The number of records created in the premixed combustor calcu-

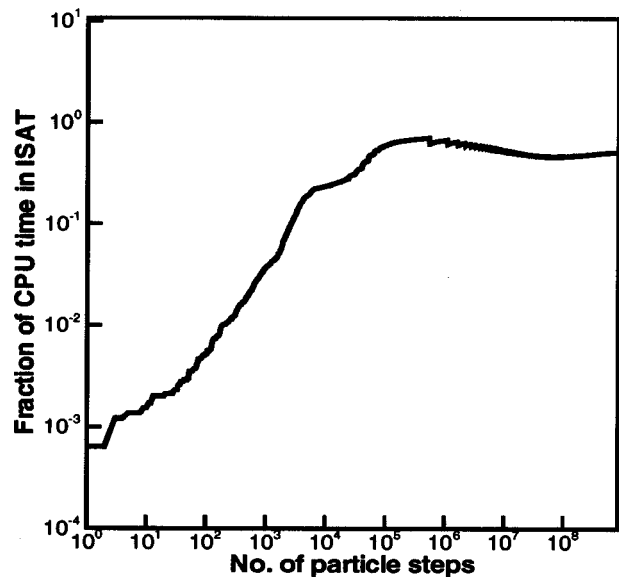


Fig. 15 Fraction of the CPU time spent in ISAT for the jet diffusion flame

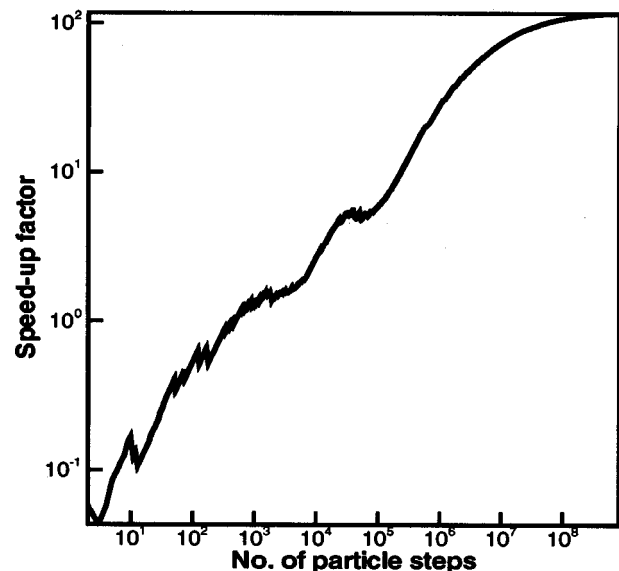


Fig. 16 Speed-up factor of ISAT over DI for the jet diffusion flame

lations is significantly lower than the jet calculations, which indicates that the accessed region of the scalar space for the premix flame is significantly smaller than that for the diffusion flame. These calculations indicate that using ISAT, details of the major and minor species in practical combustors can be obtained at an affordable cost. Consideration of minor species in the detailed mechanism allows the prediction of emissions. Accurate prediction of emissions for practical combustors is the subject of ongoing investigations.

Acknowledgments

This work is sponsored in part by the U.S. Department of Energy, Federal Energy Technology Center under co-operative agreement # DE-FC21-92-MC29061 through the Advance Gas Turbine Systems Research (AGTSR) with Dr. Daniel Fant as the program manager.

References

- [1] Chen, J. Y., Kollmann, W., and Dibble, R. W., 1989, "PDF Modeling of Turbulent Nonpremixed Methane Jet Flames," *Combust. Sci. Technol.*, **64**, pp. 315–346.
- [2] Anand, M. S., James, S., and Razdan, M. K., 1998, "A Scalar PDF Combustion Model for the National Combustion Code," AIAA paper AIAA-98-3856.
- [3] Norris, A. T., 1998, "Automatic Simplification of Full Chemical Mechanisms: Implementation in National Combustion Code," AIAA paper AIAA-98-3987.
- [4] Pope, S. B., 1997, "Computationally Efficient Implementation of Combustion Chemistry Using in situ Adaptive Tabulation," *Combust. Theory Modell.*, **1**, pp. 41–63.
- [5] Saxena, V., and Pope, S. B., 1998, "PDF Simulations of Turbulent Combustion Incorporating Detailed Chemistry," *Combust. Flame*, to be published.
- [6] Hsu, A. T., Anand, M. S., and Razdan, M. K., 1996, "An Assessment of PDF Versus Finite-Volume Methods for Turbulent Reacting Flow Calculations," AIAA paper AIAA-96-0523.
- [7] Hsu, A. T., Anand, M. S., and Razdan, M. K., 1997, "Calculation of a Premixed Swirl Combustor Using the PDF Method," 42nd ASME Gas Turbine and Aero Engines Congress, Orlando, FL.
- [8] Pope, S. B., 1976, "The Probability Approach to Modeling of Turbulent Reacting Flows," *Combust. Flame*, **27**, pp. 299–312.
- [9] Pope, S. B., 1981, "A Monte-Carlo Method for the PDF Equation of Turbulent Reactive Flow," *Combust. Sci. Technol.*, **25**.
- [10] Pope, S. B., 1985, "PDF Methods for Turbulent Reacting Flows," *Prog. Energy Combust. Sci.*, **11**, pp. 119–192.
- [11] Masri, A. R., Bilger, R. W., and Dibble, R. W., 1988, "Turbulent Nonpremixed Flames of Methane Near Extinction: Mean Structure From Raman Measurements," *Combust. Flame*, **71**, pp. 245–266.
- [12] Masri, A. R., Subramaniam, S., and Pope, S. B., 1996, "A Mixing Model to Improve the PDF Simulations of Turbulent Diffusion Flames," *Proceedings of 26rd Symp. (Int.) on Combustion*, The Combustion Institute, Pittsburgh, PA.
- [13] Karki, K. C., and Patankar, S. V., 1988, "Calculation Procedure for Viscous Incompressible Flows in Complex Geometries," *Numer. Heat Transfer*, **14**.
- [14] Yang, B., and Pope, S. B., 1998, "An Investigation of the Accuracy of Manifold Methods and Splitting Schemes in the Computational Implementation of Combustion Chemistry," *Combust. Flame*, **112**, pp. 16–32.
- [15] Chen, J. Y., and Law, C. K., 1998, poster, *Proceedings of 27th Symp. (Int.) on Combustion*, The Combustion Institute, Pittsburgh, PA.
- [16] Bowman, C. T., Hanson, R. K., Davidson, D. F., Gardiner, Jr., W. C., Lissianski, V., Smith, G. P., Golden, D. M., Frenklach, M., and Goldenberg, M., 1995. http://www.me.berkeley.edu/gri_mech.
- [17] Barlow, R. S., and Frank, J. H., 1998, "Effects of Turbulence on Species Mass Fractions in Methane/Air Jet Flames," *Proceedings of 27th Symp. (Int.) on Combustion*, The Combustion Institute, Pittsburgh, PA.
- [18] Pope, S. B., 1978, "An Explanation of the Turbulent Round-Jet/Plane-Jet Anomaly," *AIAA J.*, **16**, pp. 279–281.
- [19] Barlow, R. S., and Chen, J. Y., 1998, "Third International Workshop on Measurement and Computation for Turbulent Nonpremixed Flames," <http://www.ca.sandia.gov/tdf/3rdWorkshop/Boulder.html>.
- [20] Puri, R., Stansel, D. M., Smith, D. A., and Razdan, M. K., 1995, "Dry Ultra-Low NO_x Green Thumb Combustor for Allison's 501-K Series Industrial Engines," ASME paper 95-GT-406.
- [21] Shih, T. H., Zhu, J., and Lumley, J. L., 1996, "Calculation of Wall-Bounded Complex Flows and Free Shear Flows," *Int. J. Numer. Methods Fluids*, **23**, pp. 1–12.

Reduced NO_x Diffusion Flame Combustors for Industrial Gas Turbines

A. S. Feitelberg

V. E. Tangirala

GE Corporate Research and Development,
One Research Circle,
Nishayuna, NY 12309

R. A. Elliott

R. E. Pavri

R. B. Schiefer

GE Power Systems,
Schenectady, NY 12301

This paper describes reduced NO_x diffusion flame combustors that have been developed for both simple cycle and regenerative cycle MS3002 and MS5002 gas turbines. Laboratory tests have shown that when firing with natural gas, without water or steam injection, NO_x emissions from the new combustors are about 40 percent lower than NO_x emissions from the standard combustors. CO emissions are virtually unchanged at base load, but increase at part load conditions. Commercial demonstration tests have confirmed the laboratory results. The standard combustors on both the MS3002 and MS5002 gas turbine are cylindrical cans, approximately 10.5 inches (27 cm) in diameter. A single fuel nozzle is centered at the inlet to each can and produces a swirl stabilized diffusion flame. The walls of the cans are louvered for cooling, and contain an array of mixing and dilution holes that provide the air needed to complete combustion and dilute the burned gas to the desired turbine inlet temperature. The MS3002 turbine is equipped with six combustor cans, while the MS5002 turbine is equipped with twelve combustors. The new, reduced NO_x emissions combustors (referred to as a "lean head end," or LHE, combustors) retain all of the key features of the conventional combustors; the only major difference is the arrangement of the mixing and dilution holes in the cylindrical combustor cans. By optimizing the number, diameter, and location of these holes, NO_x emissions can be reduced considerably. Minor changes are also sometimes made to the combustor cap. The materials of construction, pressure drop, and fuel nozzle are all unchanged. The differences in NO_x emissions between the standard and LHE combustors, as well as the variations in NO_x emissions with firing temperature, are well correlated using turbulent flame length arguments. Details of this correlation are presented.

[DOI: 10.1115/1.1376722]

Introduction

The MS3002 and MS5002 gas turbines are mechanical drive models that are widely used in pumping applications. Both turbines have two shafts: a high-pressure turbine powers the air compressor and a low-pressure turbine drives the desired load. Both gas turbines are also available in simple cycle and regenerative cycle configurations. In the regenerative cycle configurations, the hot turbine exhaust gas is used to heat the compressor discharge air (via a heat exchanger) to temperatures as high as 900°F (480°C) before combustion. The pressure ratio, compressor discharge temperature, and net power output of selected MS3002 and MS5002 turbines are listed in Table 1.

Environmental regulations continue to drive emissions reductions from almost all combustion turbines. Dry low NO_x (DLN) combustors (that is, combustors that do not require steam or water injection) are a very popular option, because of their demonstrated potential for extremely low emissions. However, there are applications that do not require the low emissions (and relatively high cost) of lean premixed combustors. For example, when modifying an existing turbine to increase power output and/or efficiency (often by increasing the pressure ratio and/or turbine inlet temperature), frequently the only regulatory requirement is that modifications do not result in increased emissions. Increasing the pressure ratio and/or turbine inlet temperature tends to increase NO_x emissions. Since the available modifications to existing units are usually relatively minor, emissions often increase only 10–15 percent

after a turbine modification. In these situations, there is a need for low-cost combustion system modifications with relatively modest emissions reductions.

GE has developed lean head end (LHE) combustors for the simple cycle and regenerative cycle MS3002 and MS5002 turbines to fill this need. NO_x emissions from LHE combustors are about 40 percent lower than the emissions from the standard combustors they replace. LHE combustors are swirl-stabilized diffusion flame combustors with the same overall length, diameter, and pressure drop as the corresponding standard combustors. The key differences between LHE and standard combustors are best illustrated by comparing the combustors for a specific turbine model.

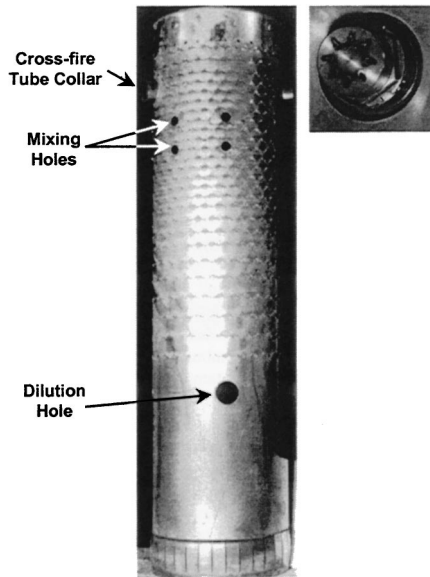
A standard simple cycle MS5002 combustor liner and fuel nozzle are shown in Fig. 1. The MS5002 turbine is equipped with 12 cylindrical can-type combustors. Both the standard and LHE combustors are approximately 27 cm (10.5 in.) in diameter and 112 cm (44 in.) long. An annular vortex generator is mounted in the combustor cap, and surrounds the single fuel nozzle that is centered at the inlet to each can. The walls of the can and the combustor cap are louvered for cooling. An array of mixing and dilution holes provides the air needed to complete combustion and dilute the burned gas to the desired combustor exit temperature. A flexible seal at the downstream end of the combustor liner fits in to the transition piece on assembly. Cross-fire tubes at the upstream end of the combustor are used to equalize the pressure between the combustion chambers and propagate the flame upon ignition.

The only significant difference between the standard and LHE combustors for the simple cycle MS5002 turbine is the arrangement of the mixing and dilution holes. In the LHE combustor, the mixing and dilution holes have been moved upstream (closer to the fuel nozzle). In addition, the diameter and number of the mixing holes have been changed. Put simply, air is redistributed from

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-085. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

Table 1 Operating parameters for selected MS3002 and MS5002 turbines at ISO conditions

Turbine Model	Pressure Ratio	Compressor Discharge Temperature (°F/°C)	Regenerator Discharge Temperature (°F/°C)	Combustor Discharge Temperature (°F/°C)	Net Power Output (MW)
Simple cycle MS3002J A/T	7.0	513/267	N/A	1841/1005	11
Simple cycle MS5002B	8.6	553/290	N/A	1747/953	27
Regen cycle MS5002C	9.0	588/309	893/479	1842/1006	27

**Fig. 1 Standard MS5002 combustor liner (left) and gas fuel nozzle (right)**

the downstream end of the combustor towards the upstream end of the combustor, resulting in decreased NO_x formation. These modifications produce a more fully aerated diffusion flame, reducing the turbulent flame length and decreasing the amount of time spent in the stoichiometric flame zone, thus reducing NO_x emissions. For a given combustor exit temperature, the total combustor air flow and the overall fuel/air ratio are unchanged.

The standard and LHE combustors for the regenerative cycle MS5002 turbine are very similar to the standard and LHE combustors for the simple cycle MS5002 turbine. The only major

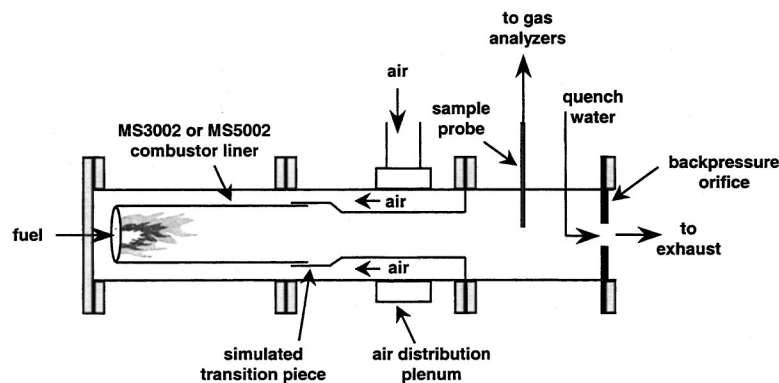
difference is the spacing of the rows of louvers. The combustor liner for the regenerative cycle MS5002 turbine has a larger number of rows of louvers, and the rows are closer together. Tighter louver spacing is required to cool the regenerative cycle liners because the combustion air temperature is much higher (see Table 1), making the cooling air less effective. There are also minor differences in the precise location of the mixing and dilution holes, as well as the effective open area of the combustor cap.

The standard and LHE combustors for the simple cycle MS3002J A/T are also similar to the standard and LHE combustors for the simple cycle MS5002. The MS3002J A/T combustor liners are slightly longer than MS5002 liners. Since the MS3002 turbine has only six combustors, the cross-fire tube collars are located in different circumferential positions. The standard combustor liner lacks a vortex generator; swirl is provided only by louvers in the cap. The LHE liner design includes a vortex generator in the cap.

Development of LHE combustors for the MS3002 and MS5002 turbines began in 1997. After a series of laboratory tests at GE Corporate Research and Development, the first machine test of LHE combustors for the simple cycle MS5002 turbine was conducted at a British Petroleum site in Prudhoe Bay, AK. Details of these tests can be found in the literature ([1]). The first machine test of LHE combustors for the simple cycle MS3002J A/T turbine was conducted in 1999 at a customer site in Durango, CO. The LHE combustors for the regenerative cycle MS5002 turbine have not been machine tested, although laboratory development is complete.

Laboratory Development and Results

The laboratory test stand used for development of the LHE combustors described in this paper is shown in Fig. 2 and has been described previously ([1,2]). The combustor liner is housed in a 40.5 cm (16 in.) diameter pressure vessel which can be supplied with up to 7.3 kg/s (16 lb_m/s) of preheated, nonvitiated, oil-free air. The air enters the vessel through 24 holes uniformly spaced around the circumference of an air distribution plenum.

**Fig. 2 Laboratory test stand for evaluating both simple cycle and regenerative cycle combustors**

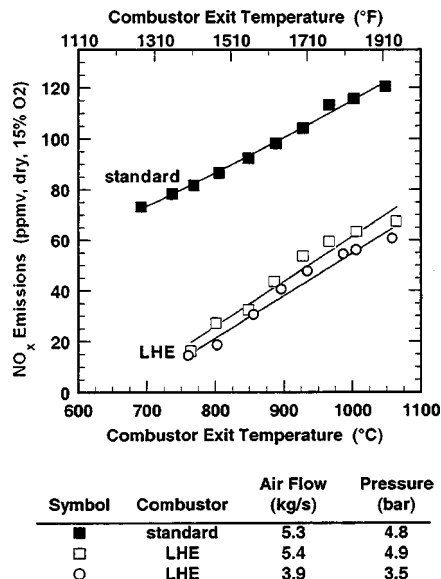


Fig. 3 Laboratory measurements of NO_x emissions from standard and lean head end (LHE) simple cycle MS3002J combustors. Combustion air temperature = $269 \pm 1^\circ\text{C}$ for all points. At ISO conditions, the base load combustor exit temperature for the simple cycle MS3002J is 1005°C .

The air flows over a simulated transition piece and continues upstream, towards the combustor liner and fuel nozzle, mimicking the reverse flow configuration of the MS3002J and MS5002 turbines. The pressure inside the combustor is established with an orifice plate with the appropriate effective area. The backpressure orifice is cooled by a water spray injected into the hot exhaust gas. The exhaust gas is sampled with an uncooled stainless steel probe located upstream of the water spray and analyzed for CO, CO_2 , O_2 , NO_x , and unburned hydrocarbons with continuous emissions analyzers. The gas sample probe extends across a diameter of the duct and has inlet holes equally spaced across its entire length. In this manner the collected sample gas represents an integrated average across the duct.

Simple Cycle MS3002J A/T. Results from laboratory tests of the standard and LHE simple cycle MS3002J A/T combustors are shown in Figs. 3, 4, and 5. Overall, these results are very similar to laboratory tests of the standard and LHE combustors for the simple cycle MS5002 turbine which were reported previously ([1]). Combustors are usually tested at two different total air flow rates and pressures because the total air flow through a single combustor exceeds the capacity of the laboratory air compressors. Variations in the air flow between the combustors during different tests are due to the day-to-day variations in the settings of the laboratory air compressors. Since lab emissions must be scaled to full pressure regardless of the air flow, no attempt was made to exactly duplicate air flows from day to day.

We find, in general, that diffusion flame combustor NO_x emissions are proportional to P^n , where P is the absolute pressure and $0.3 < n < 0.6$. This is a common observation, and a recent summary of similar findings is available ([3]). Through tests at two different air flow rates (and, therefore, pressures) the pressure exponent n can be determined for each combustor, and laboratory NO_x emissions can be scaled to full flow and pressure. We have found in the past ([1]), and as will be shown below, that the pressure exponent n is about the same for standard and LHE combustors operating at the same inlet air temperature. Laboratory CO and unburned hydrocarbon (UHC) emissions from these combustors

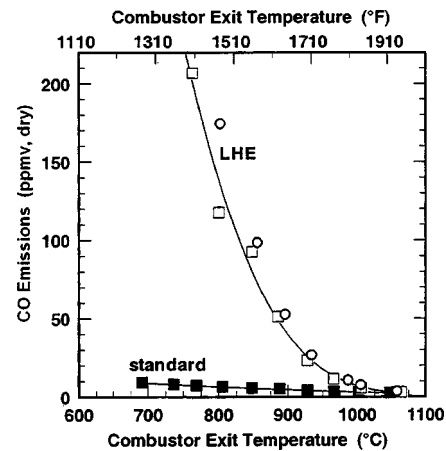


Fig. 4 Laboratory measurements of CO emissions from standard and lean head end (LHE) simple cycle MS3002J combustors at the same conditions as shown in Fig. 3. The legend shown in Fig. 3 also applies to Figs. 4 and 5.

tors are generally found to be insensitive to total flow rate and pressure, so emissions measurements of these species do not require scaling.

Figure 3 shows that NO_x emissions decrease as the fuel flow and combustor exit temperature decrease, as expected for a diffusion flame combustor. At the base load combustor exit temperature for an MS3002J A/T turbine (1005°C , or 1841°F) and at similar pressures, laboratory NO_x emissions decreased from about 116 ppmv (on a dry, 15 percent O_2 basis) for the standard combustor to 63 ppmv for the LHE combustor, a decrease of about 45 percent. The percent reduction in NO_x emissions was larger at part load conditions. A regression analysis of the data in Fig. 3 showed that NO_x emissions from the LHE combustor were proportional to $P^{0.39}$.

CO and unburned hydrocarbon (UHC) emissions from the laboratory tests of the standard and LHE simple cycle MS3002J A/T combustors are shown in Figs. 4 and 5, respectively. CO and UHC emissions increase monotonically as the fuel flow rate and combustor exit temperature decrease, as expected. CO and UHC emissions from the LHE combustor were higher than emissions from the standard combustor at part load, but were similar at base load MS3002J A/T conditions. Figures 4 and 5 also show that labora-

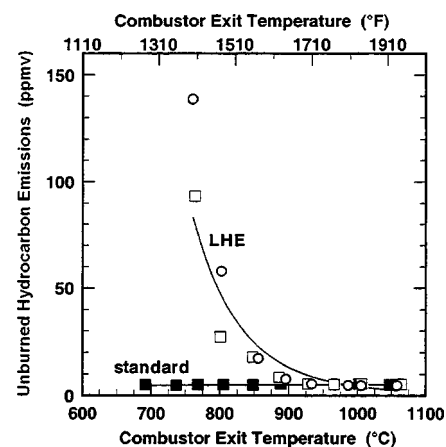
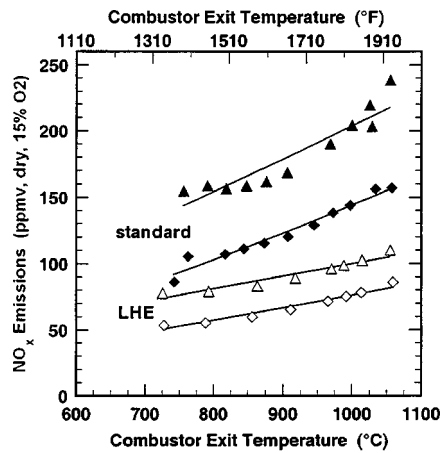


Fig. 5 Laboratory measurements of unburned hydrocarbon emissions from standard and lean head end (LHE) simple cycle MS3002J combustors at the same conditions as shown in Figs. 3 and 4. The legend shown in Fig. 3 also applies to Figs. 4 and 5.



Symbol	Combustor	Air Flow (kg/s)	Pressure (bar)
▲	standard	5.3	5.1
◆	standard	2.8	2.6
△	LHE	3.9	3.6
◇	LHE	2.4	2.1

Fig. 6 Laboratory measurements of NO_x emissions from standard and lean head end (LHE) regenerative cycle MS5002C combustors. Combustion air temperature was $492 \pm 3^\circ\text{C}$ for all points. At ISO conditions, the base load combustor exit temperature for the simple cycle MS5002C is 1006°C .

tory CO and UHC emissions from these diffusion flame combustors do not depend upon the total air flow rate and pressure.

Although increasing primary zone aeration might be expected to have a negative impact on the lean blow-out characteristics of the combustor, the LHE combustor for the simple cycle MS3002J A/T turbine actually has slightly better lean blow out characteristics than the standard combustor. This improvement is due to the addition of a vortex generator to the combustor cap, a feature not present in the standard combustor. The additional swirl provided by the vortex generator enhances flame stability and offsets the effect of increased primary zone aeration. For one particular set of laboratory test conditions, the combustor temperature rise at lean blow out decreased from 120°C for the standard combustor to 100°C for the LHE combustor.

Regenerative Cycle MS5002C. Results from laboratory tests of the standard combustor for the regenerative cycle MS5002C turbine and a prototype LHE combustor for the same machine are shown in Figs. 6 and 7. NO_x emissions from regenerative cycle turbines are considerably greater than NO_x emissions from simple cycle turbines, due to the higher combustion air temperature (compare Figs. 3 and 6). The increased air temperature increases the stoichiometric flame temperature, which is one of the dominant factors governing NO_x emissions from diffusion flame combustors.

Regression analyses of the data in Fig. 6 shows that NO_x emissions from the standard liner are proportional to $P^{0.55}$, while NO_x emissions from the LHE liner are proportional to $P^{0.58}$. This difference in the NO_x pressure dependence is within the experimental uncertainty of the data, and is not considered significant. In general, we find that the same pressure scaling applies for the standard and LHE combustors for a given turbine, most probably because the inlet air temperature and velocity are the same for both liners. However, the pressure scaling factor is different for different turbines. In a previous study ([1]), NO_x emissions from the standard and LHE liners for the simple cycle MS5002B turbine were found to be proportional to about $P^{0.38}$, and as shown above, the pressure scaling for the simple cycle MS3002J A/T

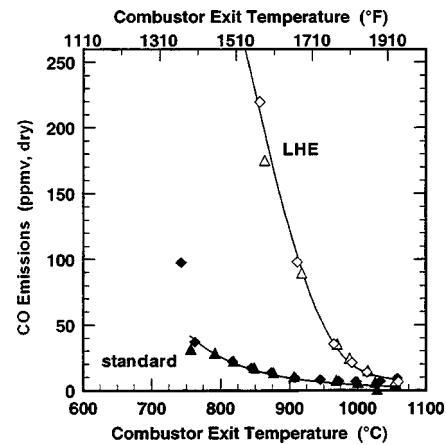


Fig. 7 Laboratory measurements of CO emissions from standard and lean head end (LHE) regenerative cycle MS5002C combustors for the same conditions as shown in Fig. 6. The legend shown in Fig. 6 also applies to this figure.

turbine was found to be about $P^{0.39}$. The similarity in pressure scaling for the simple cycle MS5002B and MS3002J A/T is probably due to the great similarity in the operating conditions for these two machines (see Table 1).

Scaling the data in Fig. 6 to a single pressure shows that at base load regenerative cycle MS5002C conditions (combustor discharge temperature = 1006°C), NO_x emissions from the prototype LHE combustor were about 50 percent lower than NO_x emissions from the standard combustor. Figure 7 shows CO emissions increased from about 10 ppmv (dry) to about 20 ppmv (dry) at base load. Figure 7 also shows that laboratory CO emissions from these diffusion flame combustors were insensitive to total air flow rate and pressure.

Design Methodology. As mentioned previously, LHE combustors have lower NO_x emissions than the standard combustors they replace because the size, number, and location of the mixing and dilution holes in the combustor liner have been altered, along with other minor modifications. Figures 8 and 9 compare the axial distributions of effective area (A_{eff}), expressed as a percentage of the total effective area, for the simple cycle MS3002J A/T and regenerative cycle MS5002C combustors, respectively. Effective areas of individual liner features (mixing holes, louvers, etc.) were measured separately in an atmospheric pressure and temperature test stand. The cumulative A_{eff} at each axial location was then determined by a simple linear combination of the liner features

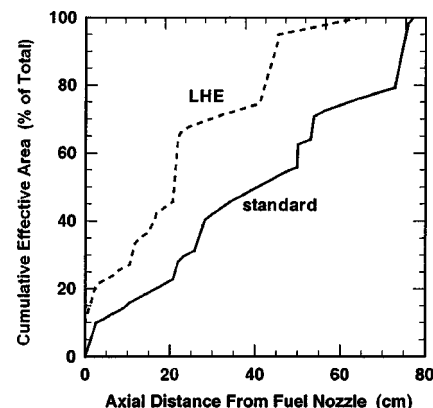


Fig. 8 Axial distribution of effective area in the standard and lean head end (LHE) simple cycle MS3002J A/T combustor liners

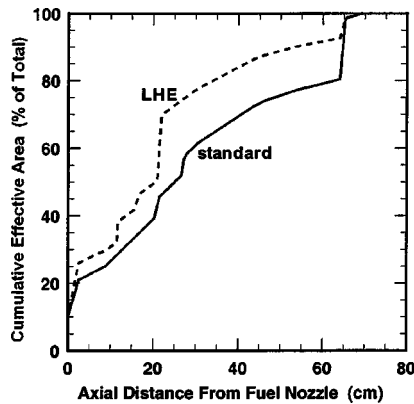


Fig. 9 Axial distribution of effective area in the standard combustor for the regenerative cycle MS5002C turbine and the prototype lean head end (LHE) combustor for the same machine

and open areas upstream of that location. Abrupt, large increases in A_{eff} indicate the location of mixing and dilution holes. Gradual increases in effective area correspond to regions of the liner with cooling louvers, but no holes.

We have developed a semi-empirical model that relates liner features to NO_x emissions. This model served as a tool which aided in the design of the LHE combustor effective area distributions shown in Figs. 8 and 9. The model has been discussed in some detail elsewhere ([1]). Briefly, the model assumes the confined turbulent diffusion flame length (L_f) is related to the fuel flow rate (W_{fuel}) and primary air flow rate ($W_{primary\ air}$) by

$$L_f \propto \frac{W_{fuel}}{W_{primary\ air}} \quad (1)$$

as has been shown by, for example, Beer and Chigier [4]. The other two key assumptions are (1) most of the NO_x is formed at the stoichiometric fuel/air interface, and (2) residence time in the stoichiometric flame zone is proportional to flame length.

With these three assumptions, a stoichiometric flame zone residence time (τ) can be inferred from the measured NO_x emissions, and this τ is proportional to fuel flow rate and inversely proportional to the primary air flow rate. In practice, we determined this proportionality for the standard (existing) combustors by varying W_{fuel} at a fixed $W_{primary\ air}$ and measuring NO_x (thus determining τ). The standard combustor emissions shown in Figures 3 and 6 are examples of these measurements. Once this proportionality was determined for a standard combustor, the correlation could be used to interpolate and/or extrapolate to new LHE liner designs with different primary air flow rates (due to different distributions of effective area). The new LHE liner designs were then fabricated and evaluated in the laboratory. In general, the proportionality constant varies with inlet air temperature and combustor pressure, and must be determined experimentally for each new set of conditions.

The stoichiometric flame zone residence time corresponding to any particular NO_x measurement was estimated by calculating the NO_x concentration in a stoichiometric perfectly stirred reactor (PSR) at the same pressure and inlet temperature. PSR calculations were performed using the Chemkin II package of programs and subroutines ([5,6]), and a ~ 250 step elementary chemical reaction set ([7]). Previous work ([8]) has shown there is little difference between the selected reaction set and two other popular reaction sets ([9,10]) at least when calculating PSR NO_x concentrations.

The only model parameter which has not been defined is the "primary air." Previous work ([1]) showed that for simple cycle conditions, the best correlation between τ and $W_{fuel}/W_{primary\ air}$ was obtained when the primary air was defined as the air entering

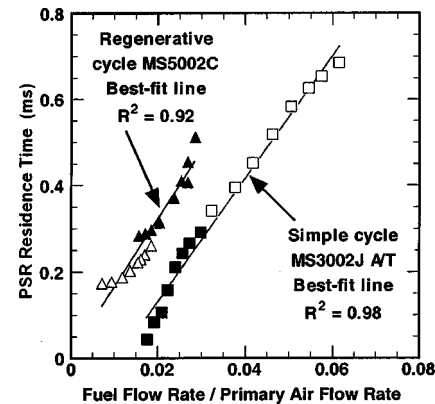


Fig. 10 Correlation between the perfectly stirred reactor (PSR) residence time and $W_{fuel}/W_{primary\ air}$ when the primary air is defined as the air entering the combustor through the first 27 cm of the combustor liner. Data points correspond to the high air flow rate, standard and lean head end (LHE) combustor NO_x measurements in Figs. 3 and 6 (the legends to Figs. 3 and 6 apply to this figure as well). With this definition of primary zone, R^2 is greater than 0.9 for both simple cycle and regenerative cycle conditions.

the combustor within one combustor diameter (27 cm, or 10.5 in.) of the fuel nozzle. The present work will show that this is also the best definition of primary air for regenerative cycle conditions, with much higher combustion air temperatures. Using this definition of primary zone, primary aeration in the MS3002J A/T turbine increased from about 32 percent for the standard combustor to 68 percent for the LHE combustor. As a result, the overall equivalence ratio at the end of the primary zone at base load decreased from about 0.97 for the standard combustor to about 0.45 for the LHE combustor. For the regenerative cycle MS5002C turbine, replacing the standard combustor with the prototype LHE combustor increases primary aeration from about 57 percent to 75 percent, and decreases the overall equivalence ratio at the end of the primary zone from about 0.38 to 0.29.

Figure 10 shows the resulting linear correlation between residence time τ and $W_{fuel}/W_{primary\ air}$ for the high flow rate, standard and LHE combustor data shown in Figs. 3 and 6. With the primary zone defined as the first 27 cm of the combustor liner, the correlation is fairly good ($R^2 > 0.91$) for both the simple cycle and the regenerative cycle conditions. Two interesting conclusions can be drawn from this result. First, a single definition of primary zone is suitable for a wide range of combustion air temperatures. From Table 1, the combustion air temperature is $\sim 265^\circ C$ for the simple cycle MS3002J A/T, and $\sim 480^\circ C$ for the regenerative cycle MS5002C. Second, a single definition of primary zone can be used even when there is a significant change in the swirl strength. As mentioned above, the standard simple cycle MS3002J A/T combustor does not have a vortex generator in the cap, and relies upon the relatively weak swirl imparted by angled louvers in the cap. The LHE combustor for this machine has a vortex generator in the cap, and significantly stronger swirl.

Figure 11 shows the values of R^2 that would have appeared in Fig. 10 for a range of different definitions of the primary zone. For completeness, the same data for standard and LHE combustors developed for the simple cycle MS5002B has been reproduced from [1]. The model is most often, most nearly linear, as predicted by the hypotheses above, when the primary zone is defined as the region of the combustor within one combustor diameter of the fuel nozzle.

Computational Fluid Dynamics Modeling. Alterations to the pattern of the mixing and dilution holes of a combustor liner will, in general, result in changes to the temperature field at the

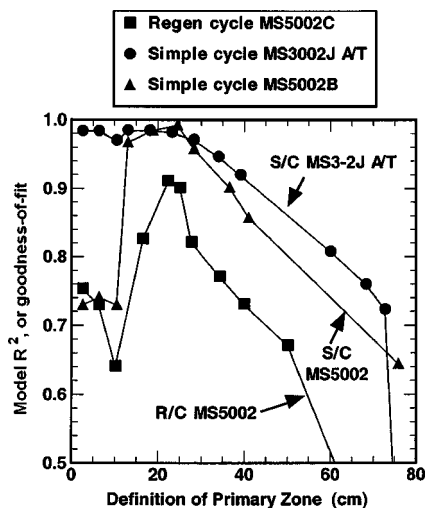


Fig. 11 Model goodness-of-fit for various definitions of the primary zone for three different pairs of standard and lean head end (LHE) combustors. Comparing the three cases shows that the correlation between perfectly stirred reactor (PSR) residence time and $W_{\text{fuel}}/W_{\text{primary air}}$ is most often most nearly linear when the primary air is defined as the air entering the combustor through the first ~27 cm (10.5 in.) of the combustor liner. The simple cycle MS5002B points have been reproduced from [1].

exit of the transition piece. Since changes in the temperature field can have an effect on turbine nozzle and bucket life, a computational fluid dynamics (CFD) study was conducted to predict the temperature field at the exit of the MS3002J A/T transition piece. Because the MS3002J A/T combustors are oriented at a 90-deg angle with respect to the axis of the turbine, the burned gas exiting each combustor flows through an elbow before entering the transition piece (see Fig. 12). A computational study was required because the laboratory test stand cannot accommodate either the 90-deg elbow or the transition piece.

Three-dimensional reacting flow CFD calculations were performed using STAR-CD, a commercially available CFD code. An unstructured grid with 540,000 nodes was used to simulate the



Fig. 12 Geometry of 90-deg elbow and transition piece downstream of MS3002J combustor

combustor geometry. Due to the symmetry in the combustor liner, the combustor was modeled with only a 90-deg wedge-shaped sector and periodic boundary conditions. Louvers were modeled as interrupted slots. Fuel injection was modeled explicitly by including individual fuel nozzle holes in the model geometry. A complete 360-deg geometry (with 450,000 nodes) was used to model the elbow and transition piece shown in Fig. 12. The chemical reaction kinetics of methane combustion in air were handled using fast chemistry (a common assumption for turbulent nonpremixed flames) and an assumed probability density function (PDF) model ([11]). A standard $k-\epsilon$ turbulence model was used.

The CFD calculations were “anchored” by comparing the predicted temperature field at the exit of the combustor with radiation-corrected laboratory thermocouple measurements. Laboratory temperature measurements were collected using thermocouples mounted on two traversing mechanisms positioned at the same axial location, but along two different combustor diameters oriented 90 deg apart. Measurements were taken at the base load temperature of 1105°C (1841°F) for both the standard combustor and the LHE combustor at the high flow rate conditions of Figs. 3–5. The measured temperature profiles were roughly parabolic in shape, with the highest temperatures near the centerline and the lowest temperatures near the walls of the combustor. The CFD model was considered anchored when the calculated temperatures were within 25°C of the laboratory data.

The two factors which had the largest impact on the agreement between the laboratory temperature measurements and the CFD model temperature predictions were (1) the geometrical model of the combustor liner louvers used in the CFD calculations, and (2) the assumed Schmidt number. Modeling louvers as interrupted slots was critical to anchoring the CFD predictions. A Schmidt number (Sc) of 0.5 gave the best agreement between the laboratory temperature measurements and the CFD predictions. Calculated temperatures were generally lower than the radiation-corrected thermocouple measurements, but still within 25°C of the lab data using this value for Sc number. Valachovic [12] also found 0.5 to be the best choice for Sc number.

Figures 13 and 14 compare the calculated temperature contours at the exit of the transition piece for models incorporating the standard and LHE combustors, respectively. The selected conditions for these two cases are slightly below base load conditions. Overall, predicted temperature fields at the transition piece exit are remarkably similar in magnitude and shape. The small differences in peak temperature are not significant, and result from the difficulty in specifying precisely the same inlet conditions in the CFD calculations for both cases. In summary, replacing standard combustors with LHE combustors is not expected to decrease turbine nozzle or bucket life.

Field Testing and Results

The first field tests of simple cycle MS3002J A/T turbines equipped with LHE combustors were conducted at a customer site in Durango, CO during 1999. Two MS3002J turbines equipped with standard combustors were tested just before a scheduled outage. During the outage LHE combustors were installed, the turbines were uprated to the MS3002J A/T configuration, and steam injection for power augmentation capabilities were added to both turbines. The turbines were subsequently restarted and tested both with and without steam injection. Turbine operating parameters were collected from the turbine data logging system. After the tests were complete, these operating parameters were used to calculate the combustor exit temperature for each data point.

A gas sample probe was inserted into an existing port on each turbine exhaust stack and positioned along a stack diameter. Each sample probe had 12 inlet holes equally spaced along its entire length, to facilitate collection of an integrated average gas sample across the exhaust stack. The sample gas was dried and sent to a

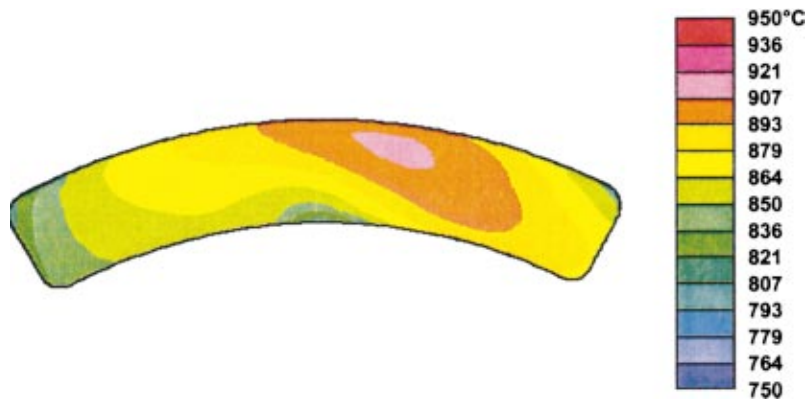


Fig. 13 Calculated temperature contours at the transition piece exit for a simple cycle MS3002J turbine equipped with standard combustors

portable gas analyzer (an Environmental Equipment, Inc., IMR 2800P) which measured concentrations of CO, O₂, NO, and NO₂.

Dynamic pressure fluctuations in each combustion chamber were monitored with piezoelectric sensors (PCB Piezotronics, Inc., Models 112A21 and 102M43). A portable data acquisition system scanned each sensor and performed a fast Fourier transform (FFT) to convert the signal from the time domain to the frequency domain. In this manner the amplitudes and frequencies of dynamic pressure fluctuations were observed and recorded in real time during the test. The dynamic pressure transducers could not be installed while the turbines were operating, so dynamic pressures were only recorded after the units restarted with LHE liners installed.

The LHE liners installed in these units were slightly different from the LHE liner evaluated in the laboratory, due to the addition of steam injection for power augmentation on the field units. Each combustion chamber was equipped with four steam nozzles, located 90 deg apart, that injected steam directly into the combustor through a steam hole in the combustor liner. Steam holes were located at single axial location, well downstream of the dilution holes in the LHE liner, to exclude steam from the reaction zone. These steam holes were not present in the LHE liner evaluated in the laboratory. Thus, the LHE liners installed in the field units had a different axial distribution of effective area than shown in Fig. 8.

When steam is not being injected, the steam holes redistribute air from the upstream end of the LHE liner towards the downstream end. This has the effect of reducing primary zone acration, so the LHE liners with steam injection holes were expected to produce higher NO_x emissions than the laboratory LHE liner.

When steam is being injected, the steam prevents air from entering the combustor through the steam holes. This forces more air into the primary zone. Thus, steam injection for power augmentation is expected to reduce NO_x emissions, in comparison to the same LHE liner running without steam injection, even if the steam does not enter the primary combustion zone.

Field test emissions measured without steam injection are shown in Figs. 15 and 16. As expected from the laboratory tests, the LHE combustors reduced NO_x emissions 25 percent to 35 percent and increased CO emissions slightly. To reduce the effect of varying ambient conditions, NO_x emissions shown in Fig. 15 were corrected to a constant pressure of 5.1 bar using the $P^{0.39}$ pressure scaling found in the laboratory. The range of the measurements was limited by the operating limits of the load compressors being driven by the MS3002J A/T turbines, not by the turbines themselves. The unit-to-unit variations shown in Figs. 15 and 16 are not unexpected, and reflect in part the difference in maintenance history and condition between the two machines at the time of the test. Unit-to-unit differences shown in Figs. 15 and 16 are smaller than differences between the standard and LHE combustor liners.

Figures 17 and 18 show the effect of steam injection for power augmentation on NO_x and CO emissions in Unit 1, equipped with LHE liners. Detailed emissions data was not collected from Unit 2 when operating with steam injection due to time limitations. As expected, steam injection forces more air into the primary zone and decreases NO_x emissions. The change in CO emissions is more complex, and may be complicated by a chemical effect. If the only effect of steam injection was to force more air into the

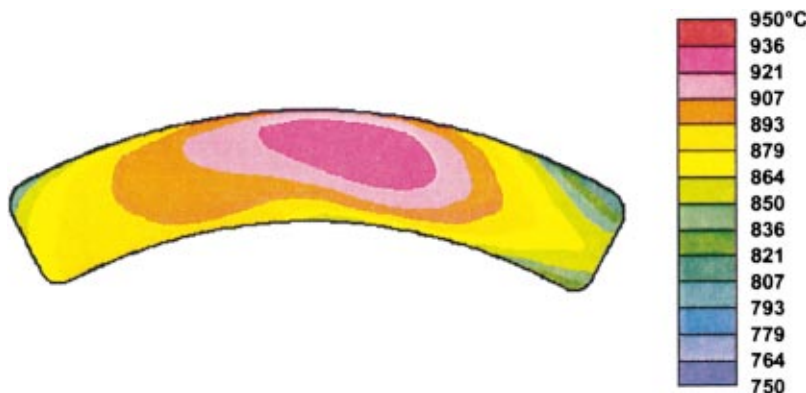


Fig. 14 Calculated temperature contours at the transition piece exit for a simple cycle MS3002J turbine equipped with lean head end (LHE) combustors

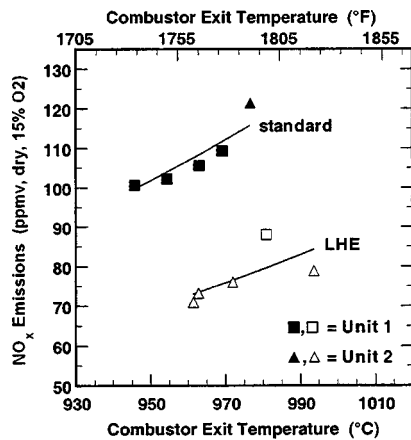
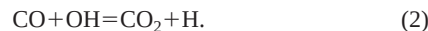


Fig. 15 NO_x emissions from two MS3002J A/T turbines equipped with standard combustors and lean head end (LHE) combustors with steam injection for power augmentation. Steam was not being injected for these test points. Field data has been corrected to $P=5.1$ bar using the $P^{0.39}$ pressure scaling found in the laboratory.

primary zone, then CO emissions should increase monotonically with increasing steam injection. The added steam may alter CO burnout by changing the OH radical concentration. In the presence of moisture, the primary route for CO destruction is



If adding steam increases the OH radical concentration, this would tend to offset the increase in CO expected from the increase in primary zone aeration. Another possibility is that some of the steam does not enter the combustor through the steam injection holes, but becomes entrained in the air flowing around the outside of the combustor liner, towards the fuel nozzle. This steam could enter the primary reaction zone and act as a diluent and reduce the stoichiometric flame temperature.

Dynamic pressure fluctuations were remarkably low under all conditions. Without steam injection, the amplitude of the maximum discrete peak observed under any condition in any combustion chamber was only 0.2 psi peak to peak. With the maximum steam injection rate (four percent of the air flow rate), the amplitude increased to only 0.24 psi peak to peak. The frequency of the maximum peak remained in the 100–300 Hz range under all conditions. These amplitudes are well below the allowable levels for louvered combustor liners.

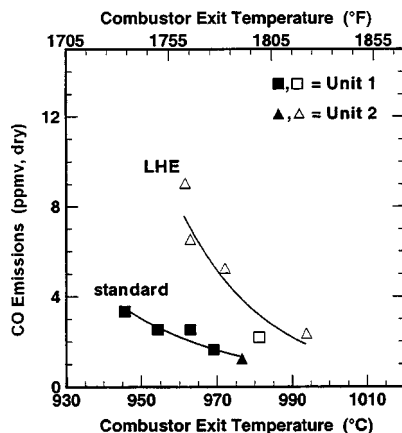


Fig. 16 CO emissions from the same two MS3002J A/T turbines, and at the same conditions, as shown in Fig. 15

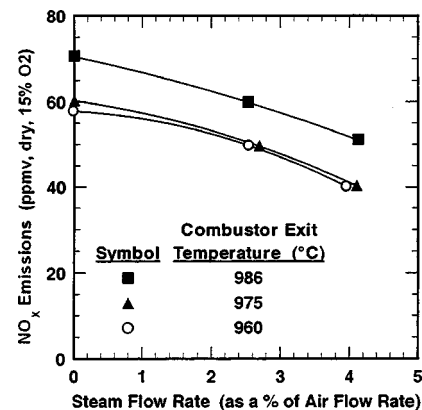


Fig. 17 Effect of steam injection for power augmentation on NO_x emissions from Unit 1

Discussion

One of the motivations for developing the semi-empirical NO_x model presented here is the relative scarcity of published correlations which accurately predict the effect of primary zone aeration on emissions from nonpremixed, industrial, natural gas fired turbines. For example, one recent study ([13]) found NO_x emissions from five different natural gas fired industrial turbines was well correlated by the expression

$$\text{NO}_x = 18.1P^{1.42} \dot{m}_A^{0.3} q^{0.72}$$

where the NO_x concentration is given in ppmv, P is the absolute pressure [atm], $\dot{m}_A$ is the combustor air flow rate [kg/s], and q is the fuel/air mass ratio. Since the total air flow rate, pressure, and overall fuel/air ratio are the same for a given turbine model equipped with either standard or LHE liners, this correlation cannot predict the ~40 percent reduction in NO_x achieved with LHE liners.

The value of the model described here resides in its predictive capability. By measuring NO_x emissions from a single combustor liner over a range of fuel flow rates, the effect of varying hole diameters and locations (and, therefore, the primary zone aeration) can also be evaluated. This was, in fact, the method used to develop the LHE liner for the regenerative cycle MS5002C turbine.

GE has employed a variety of steam injection techniques for increasing power output and/or reducing NO_x emissions. In some cases steam has been injected into the turbine wrapper which, due to the reverse flow configuration of most machines, results in steam entering both the head end and the dilution zone of the

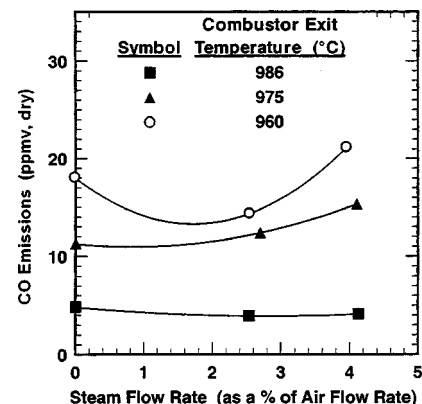


Fig. 18 Effect of steam injection for power augmentation of NO_x emissions from Unit 1. Conditions are the same as shown in Fig. 17.

combustor. In these installations steam injection both reduces NO_x emissions and increases power output. Other options involve steam injection through the fuel nozzle directly into the head end of the combustor. Head-end steam injection can also be achieved via direct injection of steam into the combustion casing, with steam injection directed towards the head end of the combustor liner. These options result in a relatively larger reduction in NO_x emissions with relatively minimal power augmentation. More recent applications for a significant effect on power augmentation have employed steam injection into the combustion casing directed to the liner dilution zone only.

The MS3002J LHE liner was developed primarily for customers desiring a NO_x emissions reduction without resorting to diluent injection. Market analysis has shown that few MS3002J customers desire steam injection or have the required steam capability. The first application of the MS3002J LHE combustor coincided with the rare customer who both had steam injection capability and a desire to use the steam for power augmentation. Given that there was not a significant demand for a steam injected MS3002J LHE combustor, this particular application was developed with steam injected only into the dilution zone. This design choice avoided the unnecessary development of a steam injected MS3002J fuel nozzle.

Conclusions

Diffusion flame combustors have been developed for both simple cycle and regenerative cycle MS3002 and MS5002 turbines which reduced NO_x emissions by about 40 percent when fired with natural gas. These new lean head end (LHE) combustors offer modest NO_x reductions with only a small increase in base load CO emissions, and at a substantially lower price than lean premixed combustors. LHE combustors are intended primarily for retrofit applications, where moderate NO_x reductions are required to offset NO_x increases that might otherwise occur when uprating an existing unit. LHE combustors offer significant NO_x reductions without diluent injection, and are also compatible with steam injection for power augmentation. With proper design, LHE combustors can have better lean blow out characteristics than the combustors they replace. The amplitudes of dynamic pressure fluctuations in LHE combustors are well below recommended levels. CFD calculations indicate that any perturbations in the transition piece exit temperature field caused by replacing standard combustors with LHE combustors are small, and should not have an effect on hot gas path component lifetimes.

The differences in NO_x emissions between standard and LHE combustors, as well as the variation in NO_x emissions with firing

temperature, were well correlated with a semi-empirical model based upon turbulent flame length theory. This model has been used as a rapid design tool to aid in the development of new LHE combustors. The model has been shown to apply over a wide range of combustion air temperatures and modest changes in swirl strength.

Acknowledgments

The authors gratefully acknowledge James F. Dalpe and Steven G. Goebel for their assistance in completing the simple cycle MS3002J A/T field test in Durango, CO. The cooperation of Bruce Chrisman and all of the staff at Williams Field Services is also greatly appreciated.

References

- [1] Feitelberg, A. S., Starkey, M. D., Schiefer, R. B., Pavri, R. E., Bender, M., Booth, J. L., and Schmidt, G. R., 2000, "Performance of a Reduced NO_x Diffusion Flame Combustor for the MS5002 Gas Turbine," *J. Eng. Gas Turbines Power*, **122**, pp. 301–306.
- [2] Maughan, J. R., Luts, A., and Bautista, P. J., 1994, "A Dry Low NO_x Combustor for the MS3002 Regenerative Gas Turbine," ASME Paper No. 94-GT-252.
- [3] Lefebvre, A. H., 1999, *Gas Turbine Combustion*, 2nd Ed., Taylor and Francis, London.
- [4] Beer, J. M., and Chigier, N. A., 1983, *Combustion Aerodynamics*, Krieger, Malabar, FL.
- [5] Glarborg, P. R., Kee, R. J., Grcar, F., and Miller, J. A., 1986, "PSR: A FORTRAN Program for Modeling Well-Stirred Reactors," Sandia National Laboratories Report SAND86-8209.
- [6] Kee, R. J., Rupley, F. M., and Miller, J. A., 1989, "Chemkin-II: A Fortran Chemical Kinetics Package for the Analysis of Gas-Phase Chemical Kinetics," Sandia National Laboratories Report SAND89-8009.
- [7] Michaud, M. G., Westmoreland, P. R., and Feitelberg, A. S., 1992, *Twenty-Fourth Symposium (International) on Combustion*, The Combustion Institute, Pittsburgh, PA, pp. 879–887.
- [8] Feitelberg, A. S., and Correa, S. M., 2000, "The Role of Carbon Monoxide in NO_2 Plume Formation," *J. Eng. Gas Turbines Power*, **122**, pp. 287–292.
- [9] Glarborg, P., Kubel, D., Kristensen, P. G., Hansen, J., and Dam-Johansen, K., 1995, *Combust. Sci. Technol.*, **110–111**, pp. 461–485.
- [10] Bowman, C. T., Hanson, R. K., Davidson, D. F., Gardiner, W. C., Jr., Lissianski, V., Smith, G. P., Golden, D. M., Frenklach, M., and Goldenberg, M., http://www.me.berkeley.edu/gri_mech/.
- [11] Jones, W. P., and Pridin, C., 1978, "Predictions of the Flow Field and Local Gas Composition in Gas Turbine Combustors," *Seventeenth Symposium (International) on Combustion*, p. 399.
- [12] Valachovic, T. G., 1993, "Numerical Predictions of Idle Power Emissions from Gas Turbine Combustors," ASME Paper No. 93-GT-175.
- [13] Rokke, N. A., Hustad, J. E., and Berg, S., 1993, "Pollutant Emissions From Gas Fired Turbine Engines in Offshore Practice—Measurements and Scaling," ASME Paper No. 93-GT-170.

Acoustic Resonances of an Industrial Gas Turbine Combustion System

S. Hubbard
Research Associate

A. P. Dowling
Professor

e-mail: apd1@eng.cam.ac.uk

Department of Engineering,
University of Cambridge,
Cambridge CB2 1PZ, UK

A theory is developed to describe low-frequency acoustic waves in the complicated diffuser/combustor geometry of a typical industrial gas turbine. This is applied to the RB211-DLE geometry to give predictions for the frequencies of the acoustic resonances at a range of operating conditions. The main resonant frequencies are to be found around 605 Hz (associated with the plenum) and around 461 Hz and 823 Hz (associated with the combustion chamber), as well as one at around 22 Hz (a bulk mode associated with the system as a whole). The stabilizing effects of a Helmholtz resonator, which models damping through nonlinear effects, are included, together with effects of coupled pressure waves in the fuel supply system. [DOI: 10.1115/1.1370975]

1 Introduction

Thermoacoustic oscillations can occur whenever there is combustion within an acoustic resonator. This happens as the unsteady heat release generates sound waves, which produce pressure and velocity fluctuations. In an acoustic resonator these fluctuations can in turn perturb the rate of heat input. If the phase relation is suitable it is possible for the system to become unstable, as, while the acoustic waves perturb the heat release, the unsteady heat release generates yet more sound. Premix burners are sensitive to noise, and self-excited combustion oscillations can become so intense that they are of concern. Noise in gas turbine combustors has been of great enough amplitude to be attributed to excessive wear, and even to the failure of some components.

Rayleigh [1] has given a physical description of the phenomenon of self-excited thermoacoustic oscillations. Acoustic waves gain energy when the unsteady rate of heat input within an acoustic resonator is in phase with the pressure perturbation. If this energy gain exceeds that lost on reflection from the boundaries of the resonator, linear acoustic waves grow in strength and the system is unstable. In practice, the amplitudes of the pressure waves are limited by nonlinear effects. Nevertheless, they can become so intense that structural damage is done.

The geometry of an industrial gas turbine combustor is complex and some simplification is required. A theory has been developed [2,3] to predict the low-frequency acoustic resonances in a simplified diffuser/combustor geometry with a mean flow and combustion, for the case when combustion is considered as confined to a single plane. The theoretical model, developed to be generic and readily applicable to most industrial gas turbines, can predict the frequencies and mode shapes of these resonances.

There are many practical devices that are susceptible to destructive thermoacoustic oscillations. Emphasis has been placed on investigating the frequency of oscillations in lean premixed pre-vaporized (LPP) burners. In Section 2 we summarize the essential features of the geometry of a LPP industrial gas turbine. Section 3 builds on previous work and covers the governing equations for the flow in the gas turbine combustion system and its solution, in the form of a mean part and perturbations of the flow quantities.

We consider two different heat release models, and the acoustic

resonances that occur within the gas turbine combustion system. In Section 4 we apply a simple model used previously in the literature [4,5], in which the rate of heat release is taken to be linearly proportional to the instantaneous fuel-air ratio. The results from applying this model to the RB211-DLE are presented. Predictions are given for the frequency and stability of oscillations. Comparison is made between including the influence of entropy waves in the downstream boundary condition and assuming that the "hot spots" have diffused and mixed before the combustor exit.

In Section 5 we use an alternative analytical model of the linear flame motion, based on the concept of a turbulent flame speed, to calculate the instantaneous rate of heat release. At lean conditions, the flame speed varies rapidly with changes in fuel-air ratio and leads to significant perturbations of the reaction rate and of the position and surface area of the flame. The results of using this flame model are presented.

The addition of an acoustic damper to the plenum chamber, in the form of a Helmholtz resonator, is considered in Section 6, together with the subsequent variation of the resonant frequencies and their stability. This section extends previous work [2] by using a nonlinear analysis to describe the large amplitude velocity perturbation in the neck of the Helmholtz resonator near resonance. In Section 7 we extend the flame model to investigate the importance of the acoustics of the fuel supply system on the stability of the plenum/combustor resonances.

2 Geometry

The geometry of an industrial gas turbine is complex; it is unrealistic to model the geometry precisely. A model which has a much simpler geometry than the gas turbine is constructed, but it is one which still contains the main features of the combustion system, see Fig. 1. The simpler geometry contains two large chambers, representing the plenum and the combustion chamber, which are connected by short premix ducts in which fuel is added to the air.

There is also an acoustic resonator connected to the plenum chamber, this is to be investigated as a possible acoustic damper. It consists of a neck and small cavity and is acoustically equivalent to a Helmholtz resonator (see Dowling and Ffowcs Williams [6]). Low-frequency acoustic waves are not influenced by bends and it is convenient to conceptually simplify the geometry to that shown in Fig. 1. The validity of this simple model has been confirmed by measuring the response to loudspeaker forcing in a full scale rig ([2]).

Even this simplified geometry leads to a system of relatively complex acoustics. This model has the added advantage of not

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-094. Manuscript received by IGTI October 1999; final revision received by ASME Headquarters October 2000. Associate Editor: D. Wisler.

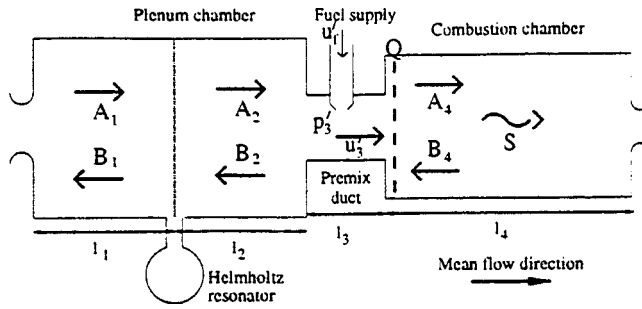


Fig. 1 Schematic of wave amplitudes, and the sections of the gas turbine

being combustion system specific, and can be readily applied to most industrial gas turbines. But even this simplified geometry leads to a system of relatively complex acoustics.

3 Mathematical Model

Most practical thermoacoustic oscillations involve mean flow, and it is this that we need to calculate before it is possible to investigate the unsteady flow. The flow quantities, pressure (p), velocity (u), density (ρ) and temperature (T) are divided into their mean quantities, denoted by an overbar ($\bar{}$), and perturbations, denoted by a prime ($'$), e.g., $p(x,t) = \bar{p}(x) + p'(x,t)$.

The geometry of the turbine system is split into four parts, before and after the position of the Helmholtz resonator in the plenum chamber, the premix duct and the combustion chamber. These four sections are shown in Fig. 1, suffices 1–4 are used to denote the flow in the different sections. Three mean flow quantities at the entrance of the plenum chamber are needed to start the calculation of the mean flow for the remainder of the geometry. These are usually given as pressure, temperature and mass flow rate, and from these the remaining mean flow quantities in the first section, and then subsequently in all of the sections, can be calculated. This is done by use of conservation of mass and energy fluxes, applied between the different sections of the plenum/combustor geometry of the gas turbine.

$$\delta(S\bar{\rho u}) = 0, \quad \delta \left([S\bar{\rho u}] \left[c_p \bar{T} + \frac{1}{2} \bar{u}^2 \right] \right) = \bar{Q}. \quad (1)$$

$\bar{Q}$ is the mean heat release per unit mass of premixed gas, S is the duct cross-sectional area, and c_p is the specific heat at constant pressure. Across area reductions, we assume the flow is isentropic and at the abrupt area increase at the combustor inlet we assume that the flow separates and apply a momentum balance. The details are given in Hubbard and Dowling [2].

Once the mean flow has been calculated throughout the gas turbine the perturbations in the flow quantities can be considered. Inside the system acoustic waves will propagate in both directions. We are interested in low-frequency disturbances, where only plane waves carry acoustic energy, and travel up and downstream with speeds $c - \bar{u}$ and $c + \bar{u}$. Following the methodology in Hubbard and Dowling [2], disturbances with time-dependence $e^{i\omega t}$ are considered, and the perturbations in the flow quantities in each section 1–4 can be written as

$$p'(x,t) = e^{i\omega t} (Ae^{-i\omega x/c(1+M)} + Be^{i\omega x/c(1-M)}),$$

$$u'(x,t) = \frac{e^{i\omega t}}{\rho c} (Ae^{-i\omega x/c(1+M)} - Be^{i\omega x/c(1-M)}),$$

$$\rho'(x,t) = \frac{p'(x,t)}{c^2} - \frac{S\bar{\rho}}{c_p} e^{i\omega(t-x/\bar{u})},$$

$$c_p T'(x,t) = \frac{p'(x,t)}{\bar{\rho}} + \frac{Sc^2}{(\gamma-1)c_p} e^{i\omega(t-x/\bar{u})}, \quad (2)$$

where A and B are the amplitudes of the downstream and upstream travelling acoustic plane waves and $M = \bar{u}/c$. S is the amplitude of the entropy wave or convected hot spot, and is nonzero only downstream of the separated flow and the heat release within the combustor.

Application of appropriate boundary conditions at the junctions between zones 1–4 leads to a system of equations for the amplitudes of the plane acoustic and entropy waves, heat release Q and flow quantities in the premix duct. The details are given in Ref. [2] and will not be repeated here. Essentially they are constructed by use of the standard continuity of mass and energy fluxes, together with isentropic flow across area reductions and conservation of momentum flux across the short combustion zone. The premix ducts are sufficiently short in comparison with c/ω that the flow in them can be taken to be incompressible.

The particle velocity u_0 into the Helmholtz resonator is a function of the pressure perturbation of p'_1 at the neck, i.e.,

$$S_0 \bar{\rho} u'_0 = H_0 p'_1, \quad (3)$$

where S_0 is the cross-sectional area of the resonator neck. The form of H_0 for linear oscillations is well known (see Dowling and Ffowcs Williams [6]), but contains no damping. In Section 6 we develop a nonlinear model for a Helmholtz resonator which includes dissipation at the neck exit. Then H_0 is a function of the SPL p'_1 .

The inlet and outlet flows are choked, so that the nondimensional mass flow rates are constant. At exit this condition is [7]

$$2 \frac{u'}{\bar{u}} + \frac{\rho'}{\bar{\rho}} = \frac{p'}{\bar{p}}. \quad (4)$$

Before the frequency of the thermoacoustic oscillations can be determined the rate of heat input per unit cross-sectional area must be described. Two different forms of relationship between the unsteady rate of heat release $Q'(t)$ and the flow will be considered.

4 Fixed Flame Model

Initially we consider the fuel in the premix duct to be injected at a steady rate, while the air flow past the fuel bars, in the premix duct, has an unsteady speed $u(t)$. The convection time delay from the fuel injector to a position $l+x$ downstream is taken as $(l+x)/\bar{u}$, where $x=0$ is at combustor entry and l is the distance from the fuelling bars to the entrance to the combustor. For linear harmonic perturbations, fluctuations in equivalence ratio are given by

$$\frac{\hat{\phi}}{\bar{\phi}} = \frac{\hat{u}}{\bar{u}} e^{-i\omega(l+x)/\bar{u}}, \quad (5)$$

where the circumflex denotes a complex amplitude. Schuermans et al. [5] take the rate of volume consumption of the unburnt gas to be constant. Then the instantaneous rate of heat release $Q(t)$ is proportional to $\rho \Delta H$, where ΔH is defined as the heat release per unit mass of premixed gas and ρ is the density of the gas. ΔH is a function of the local equivalence ratio ϕ , and for methane is given by

$$\Delta H(\phi) = \begin{cases} \frac{\phi 2.9125 \times 10^6}{1 + \phi 0.05825} & J \text{ for } \phi \leq 1, \\ \frac{2.9125 \times 10^6}{1 + \phi 0.05825} & J \text{ for } \phi > 1 \end{cases}. \quad (6)$$

Acoustic perturbations cause ϕ and hence ΔH to vary significantly. Fluctuations in ρ are $O(M)$ smaller. These assumptions then lead to the following equation for the fluctuations in heat release:

$$\frac{\hat{Q}}{Q} = -\frac{\hat{u}}{u} e^{-i\omega(l+x)/\bar{u}} \quad (7)$$

Lieuwen and Zinn [4] derive a similar expression for the unsteady rate of heat released based on a well-stirred reactor model.

This flame model (7), together with boundary conditions between sections 1–4 and at combustor inlet and outlet, leads to ten homogeneous equations which can be written in the matrix form

$$\mathbf{X} \begin{pmatrix} A_1 \\ B_1 \\ A_2 \\ B_2 \\ A_4 \\ B_4 \\ S(\bar{\rho}_4 \bar{c}_4^2 / c_p) e^{-i\omega(l_1+l_2+l_3+l_4)/\bar{u}_4} \\ p'_3 \\ u'_u \\ \hat{Q} \end{pmatrix} = 0, \quad (8)$$

where $\mathbf{X}$ is a 10×10 matrix, and $\hat{Q}e^{i\omega t} = Q'(t)$, and the entries of $\mathbf{X}$ given in the Appendix.

For given mean flow properties, Eq. (8) only has nontrivial solutions when $\det \mathbf{X} = 0$ which becomes an eigenvalue equation for ω and, in general, it has complex roots. The sign of imaginary ω determines whether disturbances grow or decay, and real ω gives the frequency of the mode.

It is a straightforward matter to determine the zeros of $\det \mathbf{X}$ numerically. Figure 2 shows the variation of the frequencies of oscillation with combustor temperature. There is one frequency which is more or less constant as the combustor temperature changes. This is an oscillation associated with the plenum chamber and so it is influenced primarily by plenum chamber temperature and is expected to be approximately independent of the combustor temperature. The other oscillations are associated with the combustor, and vary with its temperature. The unstable roots (imaginary $\omega < 0$) are shown as dotted lines. Many modes are predicted because of the influence of the entropy fluctuations. When these are accelerated through the area contraction at the

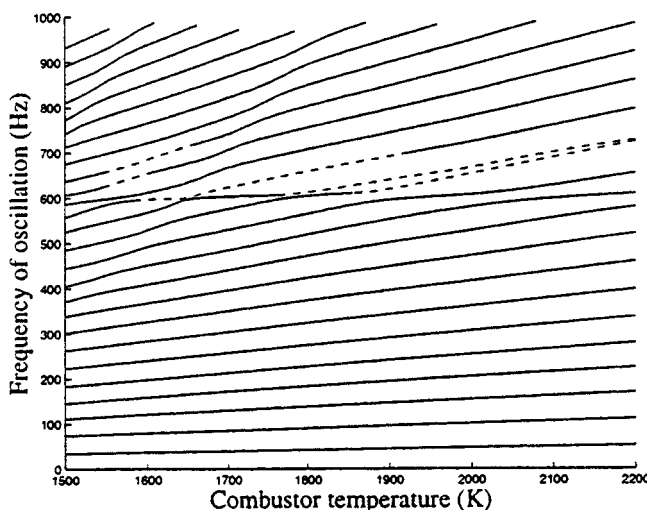


Fig. 2 Frequencies of oscillation, fixed flame model; — stable, - - - unstable

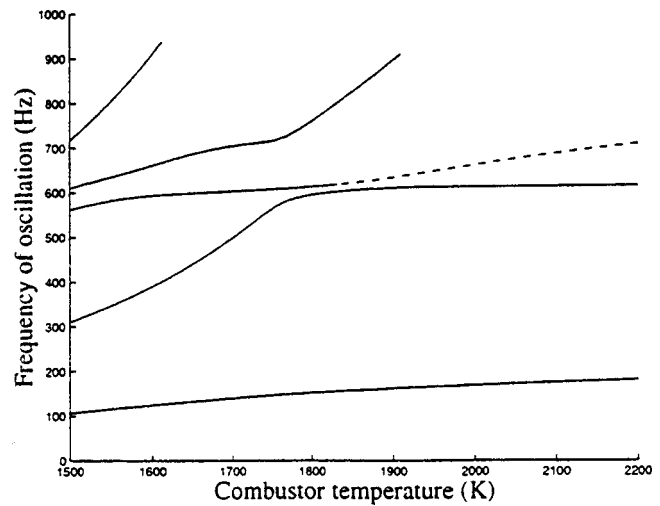


Fig. 3 Frequencies of oscillation, fixed flame model with entropy diffusion; — stable, - - - unstable

combustor exit they generate an upstream propagating pressure wave according to Eq. (4), and lead to a low-frequency oscillation and many harmonics.

For low frequencies, the entropy waves have a long wavelength, and once produced “hot spots” convect without a change in amplitude along the combustion chamber. For higher frequencies, the frequencies of interest in LPP combustors, the entropy waves have very short wavelengths. Heat diffusion and turbulent mixing are then significant and tend to attenuate the entropy waves by the time they have been convected downstream to the combustor exit. Moreover with the heat release confined to a single plane, at the combustion chamber entrance, the initial strength of the entropy waves is overpredicted. To investigate the effects of this dissipation the contributions from the entropy wave in the downstream boundary condition have been neglected, i.e., ρ' has been set equal to p'/c^2 in (4). Figure 3 shows the variation of the frequencies of oscillation with combustor temperature when there is no entropy wave in the downstream boundary condition. The plenum mode is unchanged, but now only a few combustion modes occur; much more representative of the engine experiences.

5 Flame Model

Hubbard and Dowling [2] derived an alternative flame model based on a turbulent flame speed S_u . In that work the fuel flow rate was assumed to be constant and the entropy waves convect without a change of amplitude from the combustion zone to the turbine entry. We will investigate how predictions are modified by the attenuation of the entropy waves by mixing and diffusion. Since in Section 7 we will extend the flame model to include the effects of coupled variations in the fuel supply, it is appropriate to summarize the main elements of the model.

As in the previous section we will take the fluctuations in the equivalence ratio to convect without change. So that

$$\frac{\hat{\phi}(x)}{\phi} = \frac{\hat{\phi}_0}{\phi} e^{-i\omega x/\bar{u}}, \quad (9)$$

where $\hat{\phi}_0$ is the complex amplitude of equivalence ratio fluctuations at $x=0$.

The flame surface is assumed to propagate into the unburnt fluid with a turbulent flame speed S_u , which is a function of the local equivalence ratio ϕ . As in Ref. [2], we assume the flame speed S_u to have the same dependence on ϕ as a laminar flame,

i.e., S_u taken to be a fixed constant $k \times$ the laminar flame speed which, for methane, is (from Abu-Off and Cant [8])

$$S_u(\phi) = kA\phi^B e^{-C(\phi-D)^2} \left(\frac{T}{T_R}\right)^\alpha \left(\frac{p}{p_R}\right)^\beta, \quad (10)$$

where

$$\begin{aligned} A &= 0.6079 \text{ ms}^{-1}, & p_R &= 1 \text{ bar}, \\ B &= -2.554, & T_R &= 300 \text{ K}, \\ C &= 7.31, & \alpha &= 2, \\ D &= 1.23, & \beta &= -0.5, \end{aligned} \quad (11)$$

and $k=20$ in our results.

Suppose the inlet to the combustion chamber is an annulus of inner radius a , outer radius b . If we model the flame front by an axisymmetric surface extending from $r=a$ to b , whose axial location at radius r is $x=\xi(r,t)$. Then the instantaneous heat release is given by

$$\begin{aligned} Q(t) &= \int_a^b \Delta H(\phi) S_u(\phi) \rho \times \text{element of flame area}, \\ &= \int_a^b \Delta H(\phi) S_u(\phi) \rho 2\pi r dr \left(\left(\frac{d\xi}{dr} \right)^2 + 1 \right)^{1/2}, \end{aligned} \quad (12)$$

and the flame position satisfies

$$\frac{\partial \xi}{\partial t} = u - S_u \left(\left(\frac{d\xi}{dr} \right)^2 + 1 \right)^{1/2}. \quad (13)$$

After some algebraic manipulation, for linear fluctuations of frequency ω , Eqs. (9), (12), and (13) lead to

$$\begin{aligned} \frac{\hat{Q}}{\bar{Q}} &= \frac{\hat{\phi}_0}{\bar{\phi}} \left[\left(\frac{\phi}{\Delta H S_u} \frac{d}{d\phi} (\Delta H S_u) \right)_{\bar{\phi}} \times \frac{1 - e^{-i\Omega_1}}{i\Omega_1} - \left(\frac{\phi}{S_u} \frac{dS_u}{d\phi} \right)_{\bar{\phi}} e^{-i\Omega_1} \right] \\ &+ \frac{\hat{u}}{\bar{u}} \frac{1 - e^{-i\Omega_1}}{i\Omega_1}, \end{aligned} \quad (14)$$

where $\Omega_1 = \bar{\xi}(1/2(a+b)/2)\omega/\bar{u}$. The method of solution is similar to that in Hubbard and Dowling [2] except that the effects of changes in equivalence ratio and air velocity are displayed separately. The first term on the right-hand side of (14) describes fluctuations in the rate of heat release driven by equivalence ratio changes: These affect both the rate of consumption of fuel/unit flame area (i.e., change $\Delta H S_u$) and cause the flame to move thereby changing the flame area (described by the last term within the square brackets). In addition, velocity fluctuations, accounted for by the last term in Eq. (14), also drive unsteady flame motion, changing the instantaneous flame area and leading to fluctuations in the rate of heat release. Once differences in geometry are accounted for, the last term is equivalent to the flame model derived by Dowling [9] for a premixed flame of constant fuel-air ratio. When fluctuations in equivalence ratio are due only to air velocity changes as in Eq. (5), the flame model in (14) is identical to that derived by Hubbard and Dowling [2]. In Sec. 7 we will consider also the effect of fuel flow rate changes.

When $\hat{\phi}_0/\bar{\phi}$ is rewritten in terms of $\hat{u}/\bar{u}$ from Eq. (5) and substituted into (14), the equations are again combined into matrix form, as in Eq. (8), with a modified form of the matrix $\mathbf{X}$, again given in appendix.

It is a straightforward matter to determine the zeros of $\det \mathbf{X}$ numerically. Figure 4 shows the variation of the frequencies of oscillation with combustor temperature, assuming that the entropy waves have diffused and mixed before the choked exit. It shows the same features as Fig. 3. There is again one constant frequency

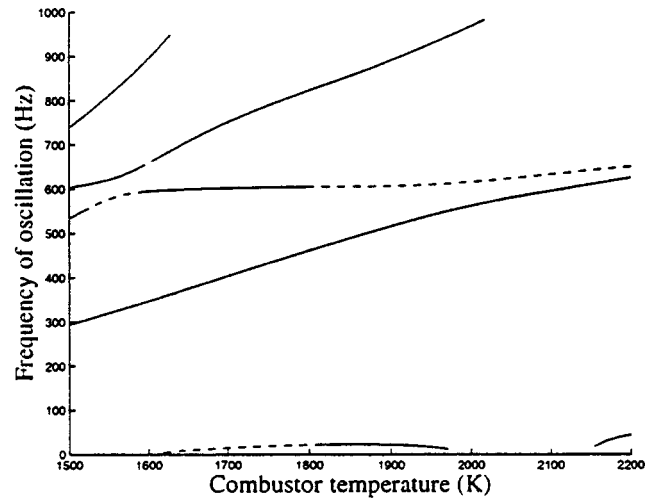


Fig. 4 Frequencies of oscillation, derived flame model with entropy diffusion; — stable, - - - unstable

associated with the plenum chamber, other oscillations are associated with the combustor. The frequencies of the unstable modes are again shown as dotted lines.

A comparison of Figs. 3 and 4 shows that with the exception of the lowest frequency mode, the resonant frequencies are only slightly influenced by these differences in the flame model. However, the growth rates are affected. For the flame model in Fig. 4 there are bands of instability at lean conditions, very reminiscent of engine operations. Engine tests have given the resonant frequencies of the system at around 450 Hz and 600 Hz, which compare well with the predicted frequencies gained from the model. For a combustor temperature of 1800 K, the predicted frequencies are 461 Hz and 823 Hz (associated with the combustion chamber), 605 Hz (associated with the plenum), and 22 Hz (associated with a bulk mode of the system as a whole).

The mode shapes of pressure, for the resonant frequencies shown Fig. 4, at a combustion chamber temperature of 1900 K, are given in Fig. 5. The pressure is referenced to the pressure at the plenum entrance. Figure 5(a) shows the bulk mode associated with the lowest frequency of oscillation. Figure 5(c) shows the pressure mode shape for the resonance associated with the ple-

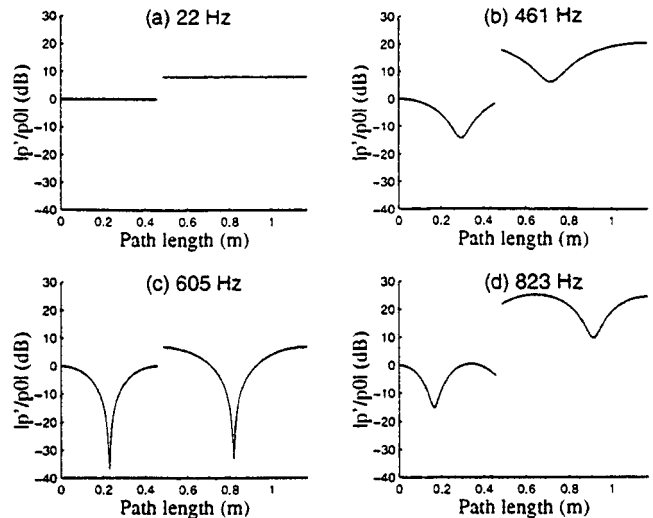


Fig. 5 Pressure mode shape, derived flame model with entropy diffusion

num, and from the figure it can be seen that the mode shape in the plenum is a half wavelength, which confirms this association, at this combustor temperature.

6 Helmholtz Resonator

In this section we consider the effects of the addition of an acoustic damper, in the form of a Helmholtz resonator, to the plenum. Hubbard and Dowling [2] included a standard linear model of a Helmholtz resonator (see, for example, Dowling and Ffowcs Williams [6]) in their analysis of combustion oscillations. This linear model contains no damping and so can only affect a self-excited combustion instability by altering the mode shape and hence the phase relationship between pressure and heat release. Unphysically, it does not describe the attenuation of acoustic energy by the resonator. We investigate a nonlinear model, based on that developed by Cummings [10] for dissipation by an orifice plate.

There is no mean flow through the neck of the Helmholtz resonator but there is an alternating acoustic flow of velocity $u'_0(t)$. According to linear theory this velocity is infinite at the Helmholtz resonance frequency. Therefore although we linearize other parts of the flow field we retain terms nonlinear in this velocity fluctuation. In the nonlinear model we consider a loss of dynamic head, associated with the dissipation of the kinetic energy in the vortex ring shed each half-cycle from alternating sides of the neck. The stagnation pressure in the plenum is therefore less than that in the jet, and the pressure drop across the resonator neck is given by

$$\Delta p = \bar{\rho} l_0 \frac{\partial u'_0}{\partial t} + \frac{\bar{\rho} u'_0 |u'_0|}{0.92}. \quad (15)$$

l_0 is the effective neck length and the factor 0.92 takes account of the venae contractae formed in the jet.

The internal pressure within the Helmholtz resonator can be related to the density and hence to the mass flux through the neck in the usual way ([6]). When this is combined with Eq. (15) it leads to

$$p'_1 = \rho \left(\frac{c^2 S_0}{i \omega V_0} + i \omega l_0 \right) u'_0 + \frac{\rho u'_0 |u'_0|}{0.92}, \quad (16)$$

for the pressure perturbation p'_1 at the neck of the resonator. S_0 and V_0 are the cross-sectional area of the neck and the volume of the bulb of the Helmholtz resonator respectively.

Equation (16) must be solved iteratively to find u_0 for a prescribed p'_1 , and then H_0 defined in Eq. (3) is given by

$$H_0 = \frac{i V_0 S_0 \omega}{c^2 S_0 - l_0 V_0 \omega^2 + \frac{i \omega V_0}{0.92} |u_0|}. \quad (17)$$

Again substitution for the unsteady flow into the boundary conditions and heat release equation leads to ten homogeneous equations for the ten unknowns. These can be written in matrix form, as in the previous section.

It is a straightforward matter to determine the zeros of $\det X$ numerically. Figure 6 shows the variation of the frequencies and growth rate with changes in the area of the neck, S_0 , for a fixed SPL p'_1 . The resonant frequency of the resonator is kept constant, $\omega_0 = 3770$ rad/s (600 Hz), by varying the volume of the resonator, once the neck area has been specified, according to

$$V_0 = \frac{S_0 c^2}{l_0 \omega_0^2}, \quad (18)$$

where the resonator neck has no physical length, but is taken to have an effective length of $1.2 \times$ neck radius, after accounting for the acoustic end corrections.

Figure 6 is for a SPL level corresponding to a rms pressure oscillation of one percent of the mean pressure in the plenum,

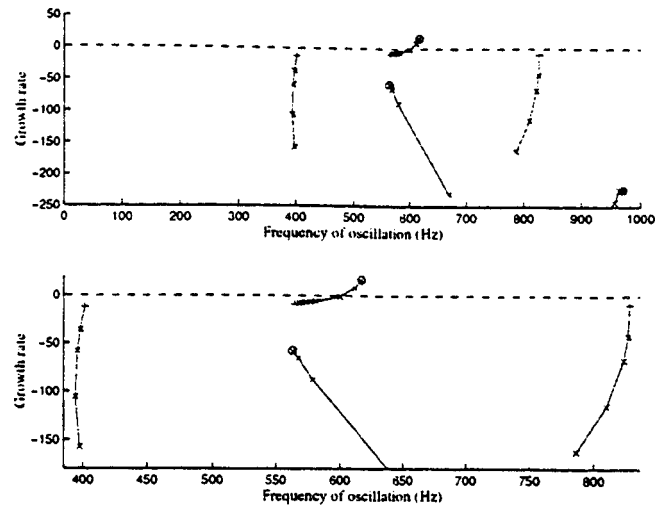


Fig. 6 Frequencies of oscillation and their growth rates, with a Helmholtz resonator, for flame model derived in Section 5, with entropy diffusion before combustor exit. Combustor temperature of 2000 K. Neck SPL=169 db, $\circ: r_0=0$ mm, $\times: r_0=5, 10, 20, 50$ mm, $+: r_0=100$ mm.

around 169 dB, for a mean pressure of 0.55 MPa. It can be seen from this figure that a Helmholtz resonator is able to stabilize modes near its design frequency of 600 Hz. Not unsurprisingly it has most effect on the mode associated with the plenum, but we see that it also modifies the stability of some of the combustor modes. Although some oscillations are stabilized, others are actually destabilized.

Figure 7 shows the variation of the frequencies of oscillation with combustor temperature, with the diffusion of entropy, when a Helmholtz resonator is connected to the plenum. The resonator has a neck radius of 20 mm. Comparing Figs. 7 and 4 the plenum mode now changes frequency with combustor temperature, although it is still around 600 Hz. With the addition of the resonator the mode is now stable over a large range of temperatures. As might be expected for a Helmholtz resonator connected to the plenum, the frequencies associated with the combustor have not significantly altered, in frequency or stability.

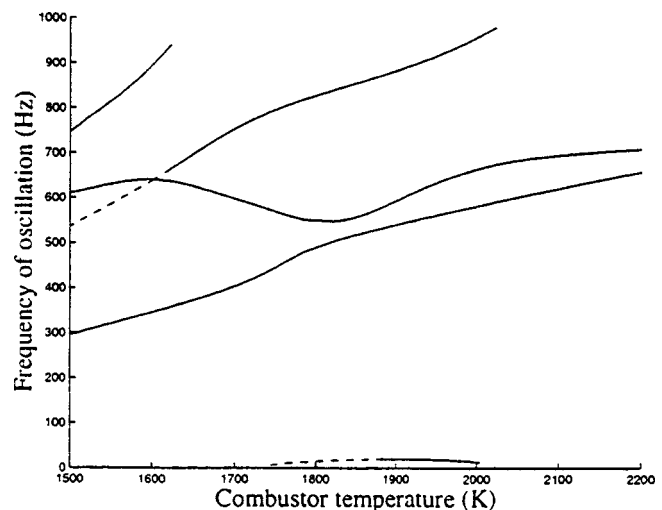


Fig. 7 Frequencies of oscillation with a Helmholtz resonator connected to the plenum, derived flame model with entropy diffusion — stable, - - - unstable

The Helmholtz resonator is not of course the only damping mechanism in the combustor. Other significant energy losses include a loss of kinetic energy at exit from the premix duct and the convection of acoustic energy out of the combustor. But when there is a self-excited oscillation, the energy gain from the unsteady combustion exceeds these losses. The introduction of a Helmholtz resonator provides additional tunable damping, which can be used to extend the stable operating range.

7 Fuel Flow Rate Variation

In this section we consider variations in fuel supply rate $\dot{m}_f$, and the effect that these have on the stability characteristics of the combustion system. For linear harmonic perturbations, fluctuations in equivalence ratio at a distance $l+x$ downstream of the fuel bars are now given by

$$\frac{\hat{\phi}}{\bar{\phi}} = \left(\frac{\hat{u}_F}{\bar{u}_F} - \frac{\hat{u}}{\bar{u}} \right) e^{-i\omega(l+x)/\bar{u}}. \quad (19)$$

We will now couple the fuel supply system to the plenum/combustor acoustics by determining the fuel supply velocity u_F driven by pressure perturbations p'_F at the location of the fuel bars.

We model the fuel supply by a duct of cross-sectional area S_f and length l_f , with mean flow velocity $\bar{u}_f$, speed of sound c_f , and of density ρ_f . The inlet is choked and the downstream end fuel is injected into the premix duct through a nozzle of cross-sectional area S_n .

Again disturbances with time-dependence $e^{i\omega t}$ are considered. Within the fuel line the perturbations in the flow quantities can be written as

$$\begin{aligned} p'_f(x,t) &= e^{i\omega t} (A_f e^{-i\omega x/(c_f+u_f)} + B_f e^{i\omega x/(c_f-\bar{u}_f)}), \\ u'_f(x,t) &= \frac{e^{i\omega t}}{\rho_f c_f} (A_f e^{-i\omega x/(c_f+\bar{u}_f)} - B_f e^{i\omega x/(c_f-\bar{u}_f)}), \\ \rho'_f(x,t) &= \frac{p'_f(x,t)}{c_f^2}, \end{aligned} \quad (20)$$

where A_f and B_f are the amplitudes of the travelling acoustic plane waves and $M_f = \bar{u}_f/c_f$.

In order to have a definite inlet condition we assume that the fuel supply is choked at $x=x_0$, a distance l_f upstream of the fuel line exit into the premix duct. The inlet condition is then taken as

$$\frac{p'_f}{\bar{p}_f} + \frac{u'_f}{\bar{u}_f} = 0, \quad (21)$$

which leads to a relation between A_f and B_f

$$B_f = \frac{1+M_f}{1-M_f} e^{-2i\omega x_0/c_f(1-M_f^2)} A_f. \quad (22)$$

Equations (20) and (22) can be combined to determine the impedance at the downstream end of the fuel duct $x=x_0+l_f$,

$$\begin{aligned} \frac{p'_f}{u'_f} &= \bar{\rho}_f c_f \frac{(1-M_f)e^{-iF} + (1+M_f)e^{iF}}{(1-M_f)e^{-iF} - (1+M_f)e^{iF}}, \\ &= -\bar{\rho}_f c_f \frac{1+iM_f \tan(F)}{i \tan(F) + M_f}, \end{aligned} \quad (23)$$

where

$$F = \frac{\omega l_f}{c_f(1-M_f^2)}. \quad (24)$$

The exit velocity of the fuel is given in terms of the pressure drop across the short nozzle in the fuel supply line

$$p_f + \frac{1}{2} \rho_f u_f^2 = p_F + \frac{1}{2} \rho_f u_F^2, \quad (25)$$

where p_F is the pressure in the premix duct at the exit of the fuel supply. The unsteady part of Eq. (25) can be rearranged, with use of continuity of mass flux $S_f \rho_f u_f = S_n \rho_f u_F$, to give

$$u'_F = \frac{p'_F}{p'_f/u_f + \bar{p}_f/\bar{u}_f(1-S_n^2/S_f^2)}. \quad (26)$$

By substituting from Eq. (23) into Eq. (26), an expression for $u'_F/\bar{u}_F$ can be derived in terms of the pressure perturbation p'_F within the combustion system,

$$u'_F/\bar{u}_F = u'_f/\bar{u}_f = K_f p'_F / \bar{\rho}_f c_f^2, \quad (27)$$

where the full form of K_f is given in the Appendix.

Finally we note that the inertial balance across the air in the premix duct gives p_F the pressure at the fuel bars as a function of the pressure at inlet to the premix duct p_2 and the air velocity u_3 in the duct,

$$p'_F = p'_2 + \frac{1}{2} (\rho_2 u_2^2 - \rho_3 u_3^2)' - i\omega \bar{\rho}_3 l_n u'_3, \quad (28)$$

where l_n is the distance from the entrance of the premix duct to the fuel injection nozzle. Substitution for p'_F in Eq. (27) leads to

$$u'_f/\bar{u}_f = K_f (p'_2 + \bar{\rho}_2 \bar{u}_2 u'_3 - \bar{\rho}_3 \bar{u}_3 u'_3 - i\omega l_n \bar{\rho}_3 u'_3) / \bar{\rho}_f c_f^2, \quad (29)$$

which relates the fractional changes in inlet fuel velocity to flow perturbations in the plenum/combustor.

Fluctuations $u'_F/\bar{u}_F$ in fuel velocity drive equivalence ratio changes as described by (19). When this is used in the flame model Eq. (14) and combined with the plenum/combustor equations, we again have a system of equations in matrix form, with $\mathbf{X}$ for this case again given in the Appendix.

It is a straightforward matter to determine the zeros of $\det \mathbf{X}$ numerically. Figure 8 shows the variation of frequencies of oscillation with the fuel line length, l_f . The frequencies can be divided into two families. There are four lines with frequency that varies inversely with fuel line length, these are associated with the fuel supply line, and are the fundamental frequency, and harmonics, of the fuel supply line, which has acoustically one open and one closed boundary. There are four other frequencies, these remain relatively constant, and are at the values of the frequencies ob-

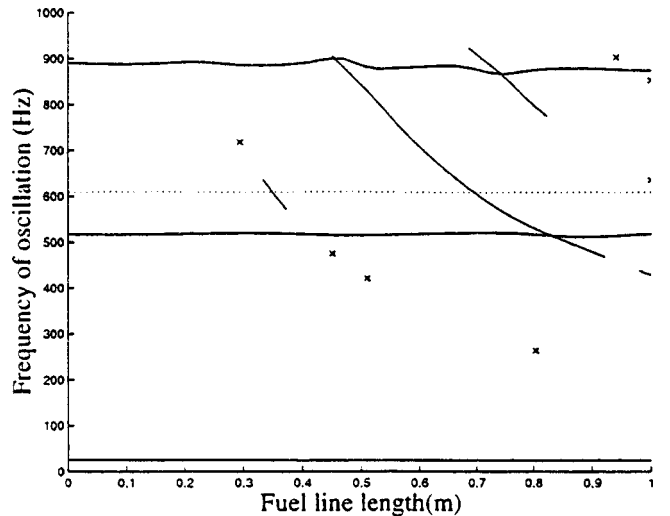


Fig. 8 Frequencies of oscillation, fluctuating fuel supply. Derived flame model with diffused entropy, at a combustor temperature of 1900 K.

Downloaded 02 Jun 2010 to 171.66.16.123. Redistribution subject to ASME license or copyright; see http://www.asme.org/terms/Terms_Use.cfm

with

$$\begin{aligned}
e_1^+ &= e^{i\omega l_1 / (c_1(1-M_1))}, \\
e_1^- &= e^{-i\omega l_1 / (c_1(1+M_1))}, \\
e_2^+ &= e^{i\omega l_1 / (c_2(1-M_2))}, \\
e_2^- &= e^{-i\omega l_1 / (c_2(1+M_2))}, \\
e_3^+ &= e^{i\omega(l_1+l_2)/(c_2(1-M_2))}, \\
e_3^- &= e^{-i\omega(l_1+l_2)/(c_2(1+M_2))}, \\
e_4^+ &= e^{i\omega(l_1+l_2+l_3)/(c_4(1-M_4))}, \\
e_4^- &= e^{-i\omega(l_1+l_2+l_3)/(c_4(1+M_4))}, \\
e_5^+ &= e^{i\omega(l_1+l_2+l_3+l_4)/c_4(1-M_4)}, \\
e_5^- &= e^{-i\omega(l_1+l_2+l_3+l_4)/(c_4(1+M_4))}, \\
\Omega_1 &= \omega \xi \left(\frac{1}{2} (a+b) \right) / \bar{u}_3, \\
\Omega_2 &= \omega l_7 S_7 / (S_3 \bar{u}_3).
\end{aligned}$$

When the entropy term is to be neglect in the application of the downstream boundary condition the term in the ninth row and seventh column of $\mathbf{X}$, in this case $-M_4 e^{-i\omega l_4 / u_4}$, is replaced by 0.

For the flame model shown in Sec. 5, the ninth row of the original matrix $\mathbf{X}$, given above, is replaced by

$$0 \quad 0 \quad 0 \quad 0 \quad 0 \quad 0 \quad 0 \quad 0 \quad +\bar{Q}(q_a+q_b+q_c)/\bar{u}_3 \quad 1,$$

with

$$\begin{aligned}
q_a &= (+0.05825/(1-0.05825\bar{\phi}) - 1/\bar{\phi} - B/\bar{\phi} + 2C(\bar{\phi}-D)) \\
&\times \frac{1-e^{-i\Omega_1}}{i\Omega_1} e^{-i\Omega_2} \\
q_b &= \frac{1-e^{-i\Omega_1}}{i\Omega_1} \\
q_c &= \phi(B/\bar{\phi} - 2C(\bar{\phi}-D)) e^{-i\Omega_2}.
\end{aligned}$$

For fluctuations in the fuel supply, shown in Sec. 9, the ninth row of the original matrix $\mathbf{X}$, given above, is replaced by

$$\begin{aligned}
&0 \quad 0 \quad \bar{Q}(q_a+q_c)K_f/(\bar{\rho}_f c_f^2) \cdot (1+M_2)e_2^+ \\
&\bar{Q}(q_a+q_c)K_f/(\bar{\rho}_f c_f^2) \cdot (1-M_2)e_2^- \quad 0 \quad 0 \\
&0 \quad -\bar{Q}(q_a+q_c)K_f/(\bar{\rho}_f c_f^2) \cdot \frac{1}{2} M_3/c_3 \\
&-\bar{Q} \frac{q_a q_b + q_c}{\bar{u}_3} - \bar{Q} \frac{q_a + q_c}{\bar{\rho}_f c_f^2} \cdot \rho_3(i\omega(l_3-l_7) + \bar{u}_3)K_f \quad 1,
\end{aligned}$$

where

$$K_f = \frac{-M_f - i \tan}{M_f(M_f^2(S_f^2/S_n^2 - 1) + 1) + i M_f^2 S_f^2/S_n^2 \tan},$$

and

$$\tan = \tan(\omega l_f / c_f (1 - M_f^2)).$$

It is noted that in these calculations the ratio S_f/S_n is 2.

References

- [1] Rayleigh, Lord, 1896, *The Theory of Sound*, Macmillan, London.
- [2] Hubbard S., and Dowling, A. P., 1998, "Acoustic Instabilities in Premix Burners," Paper No. AIAA 98-2272.
- [3] Dowling, A. P., and Hubbard, S., 2000, "Instability in Lean Premixed Combustors," *Proceedings Institution Mechanical Engineers, Part A, Journal of Power and Energy*, **214**, pp. 317-332.
- [4] Lieuwen, T., and Zinn, B., 1998, "The Role of Equivalence Ratio Oscillations in Driving Combustion Instabilities in Low Nox Gas Turbines," *Proc. 27th International Symposium on Combustion*.
- [5] Schuermans, B., Polifke, W., and Paschereit, C., 1999, "Modelling Transfer Matrices of Premixed Flames and Comparison With Experimental Result," ASME Paper No. 99-GT-132.
- [6] Dowling, A. P., and Ffowcs Williams, J. E., 1983, *Sound and Sources of Sound*, Ellis Horwood, London.
- [7] Marble, F. E., and Candel, S. M., 1977, "Acoustic Disturbances From Gas Non-Uniformities Convected Through a Nozzle," *J. Sound Vib.*, **55**, No. 2, pp. 225-243.
- [8] Abu-Off, G. M., and Cant, R. S., 1996, "Reaction Rate Modelling for Premixed Turbulent Methane-Air Flames," *Proceedings of the Joint Meeting of Spanish, Portuguese, Swedish and British Sections of the Combustion Institute, Madeira*.
- [9] Dowling, A. P., 1999, "A Kinematic Model of a Ducted Flame," *J. Fluid Mech.*, **394**, pp. 51-72.
- [10] Cummings, A., 1983, "Acoustic Nonlinearities and Power Losses at Orifices," *AIAA J.*, **22**, pp. 786-792.

Flamelet Model of NO_x in a Diffusion Flame Combustor

D. V. Volkov

A. A. Belokin

D. A. Lyubimov

V. M. Zakharov

R&D Center ECOLEN,
Moscow, Russia

G. Opdyke, Jr.

Power Tech Associates,
Media, PA

This paper describes a model used for the prediction of the formation of nitrogen oxides in modifications of an industrial diffusion flame, natural gas fueled can combustor. The flowfield inside the modified combustors is calculated using a Navier-Stokes solver. A fast chemistry assumption is used for modeling the heat release. Calculated turbulence parameters are then used for the calculation of the NO_x formation rate in the post-processing mode with the aid of a flamelet model. The flamelet model permits the use of detailed kinetics with only minimal computational expense. The dependence of the NO_x formation rate on the mixture fraction and scalar dissipation is calculated separately for each given condition. The validation of the model predictions is based on field test data taken earlier on several low NO_x modifications recently applied to an industrial, reverse flow can type combustor. The reduced level of NO_x emissions was achieved in these modifications by changes in the air distribution within the combustor liner. A comparison of the predicted and measured NO_x emission levels shows good potential of the flamelet model. [DOI: 10.1115/1.1374438]

Introduction

The number of works devoted to the numerical simulation of processes in gas turbine combustors has grown dramatically. Different combustion models are applied for the description of the turbulence-chemistry interaction beginning from fast chemistry without influence of turbulence, through the eddy break-up models with simplified kinetics, to the very complicated and expensive joint probability density function (PDF) approach. Different versions of the flamelet models are also used for combustion simulation. The advantages of flamelet models are the physically clear basis and moderate cost of implementation into multidimensional CFD codes. Within the framework of the flamelet approach, the only parameter which is responsible for turbulence-chemistry interaction is the scalar dissipation, N . That is why the model equations can be solved separately for a set of values of scalar dissipation and stored in a table usually called the flamelet library. This library can then be used by CFD codes for the estimation of flame properties. No kinetics calculations are done by the CFD procedure. That is, chemical calculations are detached from the hydrodynamics and no limitations exist on the size of the kinetic scheme.

One of the flamelet versions for nonpremixed flames was developed by Kuznetsov [1,2]. It was applied successfully to the simulation of laboratory jet flames (see [3]). An integral method for the prediction of NO_x emissions from gas turbine combustors was constructed on the basis of this model ([4]), see also [5,6]. In the present paper, the flamelet model is used together with a three-dimensional Navier-Stokes solver for the prediction of NO_x formation in modifications of an industrial diffusion flame, natural gas fueled can combustor. The joint use of CFD and the flamelet model allows one to take into account combustor geometry, plus a set of operating parameters, including pressure, composition, and temperature of the fuel and oxidizer, on NO_x emission. Experimental data on a particular combustor are not required to start the calculations because there is no calibration in the model.

Power Tech Associates (PTA) recently applied some modifications to lower NO_x emissions from the industrial W251AA combustor ([7]). The reduced level of NO_x emissions (compared to the

standard combustor) was achieved in these modifications by increasing air supply to the forward part of the liner. The tests were carried out at an installation in Colorado. The NO_x emissions data taken there are used for comparison with the joint CFD and flamelet model predictions. While the emission measurements were taken carefully, there is always some small variability in measurement to be expected (see [8]).

Filling in the Flamelet Library

The idea of the flamelet approach for the description of turbulent combustion was formulated at the end of the 1970s and the start of the 1980s (e.g., [9–11]). The flamelet approach based on the ideas of Kuznetsov [1,2] is used in this study. In diffusion flames the combustion process occurs mostly in thin layers in the vicinity of near-stoichiometric zones. Gas temperature and concentrations of radicals have their maximum values in these zones. Since the NO_x formation rate is very sensitive to temperature, it is clear that the nitric oxide is formed mainly in these regions. For nonpremixed flames with some additional assumptions the system of reacting species transport equations can be reduced to a system of ordinary differential equations with mixture fraction Z as an independent variable (e.g., see [12]).

$$N_s \frac{d^2 C_i}{dZ^2} + R_i = 0 \quad (1)$$

where $N_s = D(\nabla Z)^2$ is the scalar dissipation on the stoichiometric surface $Z = Z_s$, C_i is the mass fraction of the i th component, R_i is the corresponding chemical source term, and D is the molecular diffusivity.

Boundary conditions for Eqs. (1) can be posed at $Z=0$ (pure oxidizer) and $Z=1$ (pure fuel). It should be stressed that Eq. (1) is written for “instantaneous” parameters and averaging is needed for evaluation of the mean values in the turbulent case. The adopted averaging procedure is described later. In turbulent flows scalar dissipation characterizes the decay of the mean composition nonuniformity down to the molecular level and consequently the instantaneous values of species fluxes into the reaction zone.

The kinetics scheme by Miller and Bowman [13] was used in the present study to model chemical source terms in Eq. (1) with the reaction $\text{CH} + \text{N}_2 = \text{HCN} + \text{N}$ corrected according to Glarborg et al. [14].

Equation (1) can be solved numerically with various values of N_s as a parameter. The dependence of the NO_x formation rate (which is of interest in the present work) on the mixture fraction

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-099. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

and scalar dissipation, $R_{NO_x}(Z, N_s)$, can be calculated from Eq. (1) and stored in the flamelet library. Such a library must be created for every regime of the set of parameters considered (pressure, composition, and temperature of fuel and oxidizer).

Flowfield Calculation

The computer code developed for the calculation of incompressible variable density turbulent flows is used for the flowfield calculation in combustors. Transport equations for several turbulence characteristics (mixture fraction and its variance, turbulence kinetic energy) are solved together with Favre-averaged Navier-Stokes equations. These characteristics are used for the flamelet modeling of nitrogen oxides formation in the post-processing mode, i.e., after finishing the hydrodynamics calculations. A one-equation model for turbulent viscosity, proposed by A. N. Secundov, was applied for the turbulent stress approximation. Its description may be found in a review of Bradshaw, Launder, and Lumley [15]. A fast (equilibrium) chemistry assumption together with a bimodal probability density function (PDF) for the mixture fraction is used to model the heat release. Only intermittency on the lean side of the front is taken into account using an approximate relation. Such a model for heat release gives a density field close to that predicted by a full flamelet model with a more complicated PDF (which is described later) but requires less computation.

The pseudo-compressibility concept is employed for the numerical integration of the Navier-Stokes equations. The computer code is based on the algorithm developed by [12] Rogers, Kwak, and Kiris [16,17]. This method (originally used for incompressible flows) was extended for the calculation of variable density flows (Vasiliev et al., [18]). Convective terms of equations are approximated using a third order upwind scheme. A standard MINMOD operator is used for the approximation of the convective terms in scalar transport equations to avoid possible oscillations in the regions with large gradients of flow parameters. A detailed description of the algorithm may be found in a paper by Vasiliev et al. [18].

Calculation of NO_x Production in a Combustor

The mean NO_x production rate must be estimated at every computational cell using "instantaneous" values stored in the flamelet library and the probability density function (PDF):

$$\overline{\rho R_{NO_x}} = \int \rho R_{NO_x}(Z, N) p(Z, N) dZ dN \quad (2)$$

where ρ is the density, $p(Z, N)$ is the joint PDF of the mixture fraction and scalar dissipation. The procedure described here follows Buriko et al. [3].

The PDF can be subdivided into the turbulent and nonturbulent parts, and only the intermittency on the fuel lean side of the flame is taken into account

$$\rho p(Z, N) = (1 - \gamma) \rho(0) \delta(Z) \delta(N) + \gamma \rho p_t(Z, N)$$

where γ is the Favre intermittency factor, $p_t(Z, N)$ is the joint PDF of the mixture fraction and scalar dissipation in the turbulent fluid, $\rho(0)$ is the density of pure oxidizer (at $Z=0$), δ is the delta function. In turbulent flows the fluctuations of mixture fraction and scalar dissipation can be considered approximately independent, so

$$p_t(Z, N) \approx p_t(Z) p_t(N).$$

NO_x formation rate R_{NO_x} depends weakly on scalar dissipation because of the existence of two opposite effects: the temperature decrease and radical concentrations increase with the growth of scalar dissipation. These two effects are nearly counterbalanced. For this reason, fluctuations of scalar dissipation in the turbulent fluid are ignored in this study.

$$p_t(Z, N) \approx p_t(Z) \delta(N - N_t)$$

where $N_t = \overline{N}/\gamma$ is the mean value of the scalar dissipation in a turbulent fluid and $\overline{N}$ is the mean value of scalar dissipation. Taking into account that NO_x is not produced in pure oxidizer (at $Z=0$), Eq. (2) can be reduced to

$$\overline{\rho R_{NO_x}} = \gamma \int \rho R_{NO_x}(Z, N_t) p_t(Z) dZ.$$

At every computational cell the mean value of scalar dissipation is estimated using the usual modeling relation based on the calculated turbulence characteristics. Scalar dissipation at the stoichiometric surface N_s is believed to be equal to its value averaged in a turbulent fluid. The instantaneous distribution of $R_{NO_x}(Z, N_s)$ for a given value of N_s is obtained from the flamelet library using interpolation. This distribution in Z -space is then averaged using the probability density function $p_t(Z)$, which shape was proposed by Kuznetsov [19] (see also [12]) and depends on the average value of the mixture fraction and its variance, which are taken from hydrodynamic calculations. The shape of the PDF is based on asymptotic solutions of the PDF transport equation, namely the Airy function in the intermittent region and Gaussian in the nonintermittent region. The intermittency factor γ is calculated using a semi-empirical relation for jet flows.

An averaged NO_x source term is then integrated over the volume of the combustor. This gives the total NO_x flux, which can be used for the calculation of emission index or for the mean concentration of NO_x since the air and fuel fluxes are known.

Test Case Calculations

It had been demonstrated recently that it is possible to diminish the NO_x emission from traditional combustors by air redistribution within the combustor ([7]). The experiments had been carried out at the installation originally equipped with W251AA combustors (see Fig. 1). Several modifications to lower NO_x had been tested successfully. These modifications had different air distributions along the liner, generally to make the head-end leaner. These data were used for the validation of the numerical procedure developed.

Calculations were performed for a standard and three modified combustors using typical operational regime parameters (1.06 MPa pressure and inlet air temperature of 616 K). In Table 1 the air flow rate in the head end of the can (which includes flow rates through the combustor dome and Row 1 and Row 2 holes in the liner walls) is presented.

The flamelet library for the described conditions contained 203 solutions of Eqs. (1) from equilibrium ($N_s=0$) to the critical value of scalar dissipation which corresponds to flame extinction ($N_s=N_{cr}=188$), plus one more solution for the nonreacting case, which was used for $N_s > N_{cr}$. The number of solutions in the library was chosen to guarantee the accuracy of linear interpolation for NO_x production rate better than two percent.

Some simplifications of the combustor head end geometry were adopted in the calculations in order to diminish computational expenses. The main simplification was the changing of the number of natural gas injection holes from 8 in the real combustor to 6 in the numerical model. This allowed the consideration of a 60-deg sector of the liner as a computational domain. The computational grid consisted of 72, 47, and 17 cells in the streamwise, radial, and circumferential direction, respectively. The view of the grid is shown in Fig. 2.

Calculations for the standard and PTA4 variants were repeated using coarser ($65 \times 38 \times 14$ cells) and finer ($106 \times 61 \times 23$ cells) grids to estimate the computational dependence on grid spacing. The grid variation had some (not large) influence on the velocity field. The total NO_x production rate was reasonably insensitive to the computational grid spacing, however, resulting in a variation of less than five percent.

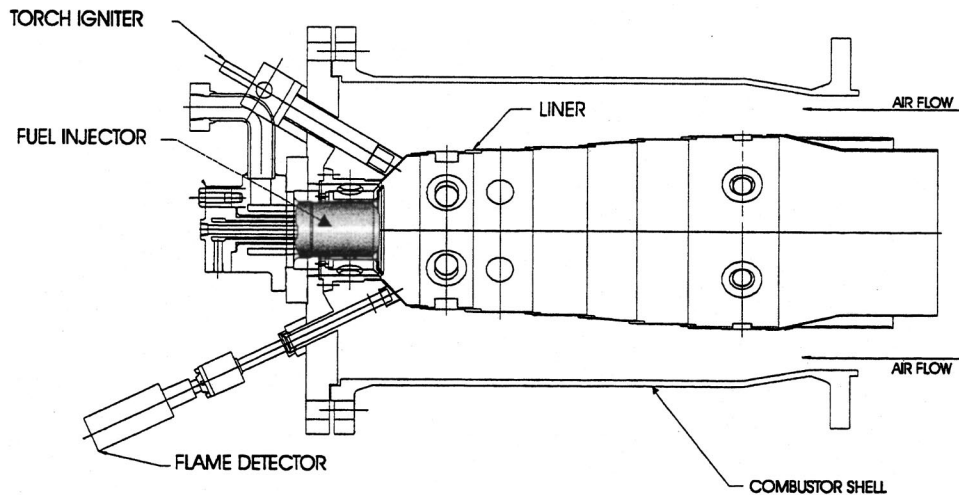


Fig. 1 View of W251 family combustor

Table 1 Results of NO_x calculations and comparison with experiment

	Head part air, % Eng	Measured NO _x reduction, %	Calculated NO _x reduction, %	Measured NO _x ppmvd	Calculated NO _x ppmvd
Standard	29.5	0	0	129	95.6
PTA2	53.7	22.5	31	100	66
PTA4	59.0	24.8	35.1	97	62
PTA5	62.0	36.4	39	82	58.3

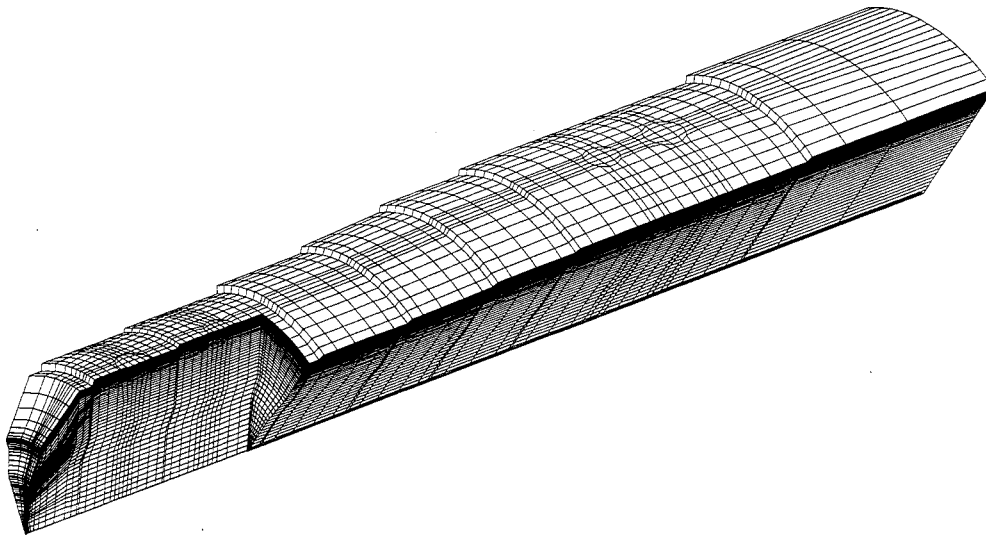


Fig. 2 Computational grid

Uniform velocity profiles were assumed in all inflow holes and slots. The flow rate was postulated based on hydraulic calculation data from PTA. Scoop holes were treated as plain holes with the discharge coefficient equal to 1.0. The effective area of the plain holes was decreased proportionally to their discharge coefficient.

Converged solution of the problem was obtained usually during one-night calculations on a personal computer with a 450 MHz Pentium-II processor.

Comparison of calculated and measured emission levels is presented in Table 1. It is seen that the absolute values of calculated NO_x concentrations are lower than the experimental data for all cases. However, the predicted direction of the change in NO_x is proper, so that the relative reduction of NO_x is reasonable. This is shown in Table 1 and also in Fig. 3. The calculated fields of

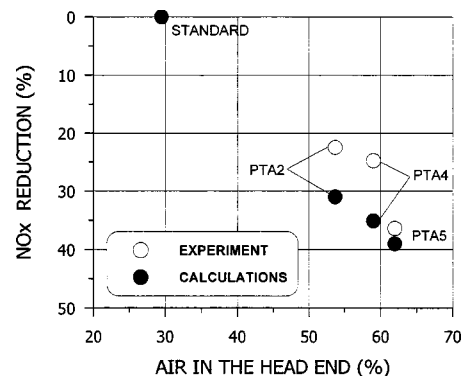


Fig. 3 Comparison of calculated NO_x reduction with experimental data

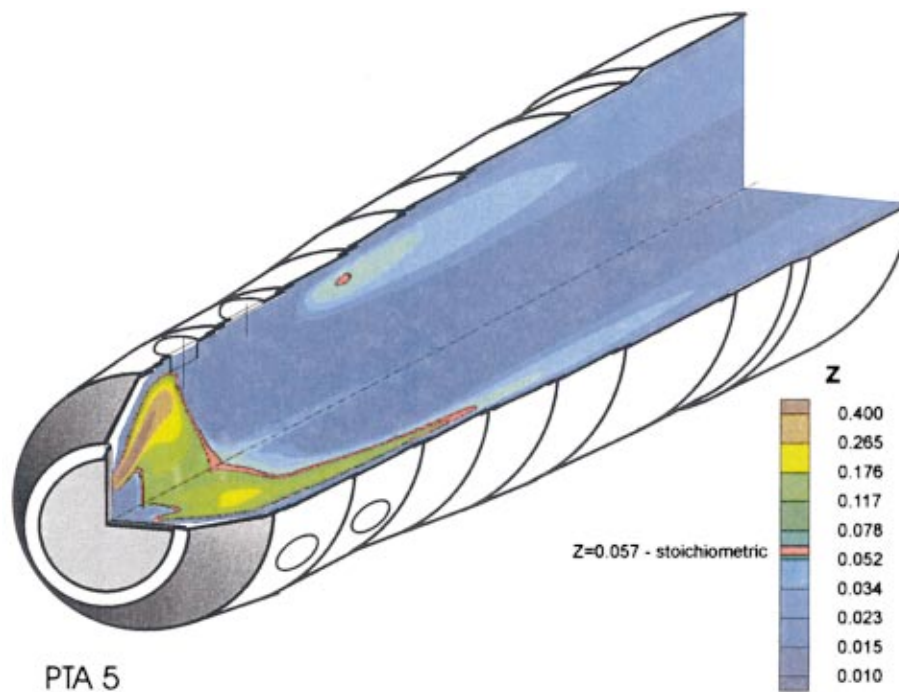


Fig. 4 Calculated mixture fraction field

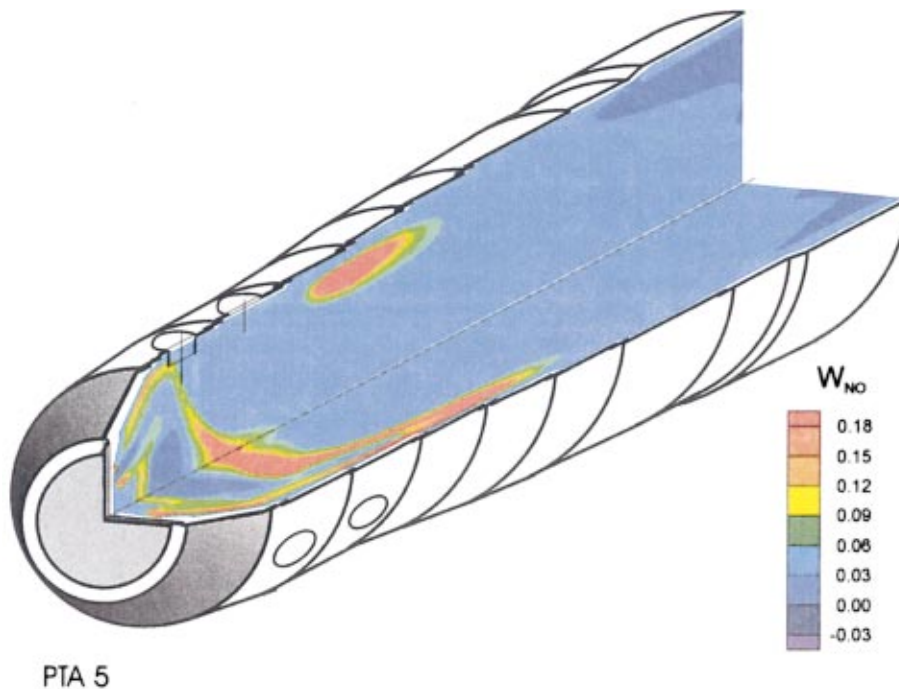


Fig. 5 Calculated NO formation rate

mean mixture fraction and nitrogen oxide formation rate are illustrated in Figs. 4 and 5, respectively, for one of the combustor modifications.

The inaccuracy of the NO_x prediction can be connected with limitations of the flamelet model. However, it could also be caused by the limitations of the detailed chemistry model, the limitations of the adopted turbulence model, and that of the current CFD procedure. The number of grid cells currently achievable is not sufficient for a detailed description of the flow near the walls of the liner including air holes and cooling films. The

accuracy of turbulence modeling in complex flow inside combustors is questionable and this will be the subject of future studies.

Conclusions

A numerical procedure for the prediction of nitrogen oxide formation in a diffusion type combustor is described. It is based on the joint use of the computational fluid dynamics method and a flamelet model for nonpremixed combustion. The field test engine

emission data for several modifications of an industrial natural gas fired combustor for lower NO_x are used for the validation of the prediction method. It is demonstrated that the described procedure captures the influence of the air distribution on the nitrogen oxide emission trends with acceptable accuracy.

Acknowledgments

The authors are grateful M. DeCorso, P. Holden, D. Moen, and G. Vermes from PTA for the provided data and for numerous fruitful discussions. The help of B. Brainin in the preparation of the paper is greatly appreciated. The activity of Russian authors in combustion studies is supported partially by the Russian Foundation for Basic Research (projects 98-01-00785, 98-02-16443).

References

- [1] Kuznetsov, V. R., 1977, "Mixing Up to a Molecular Level and the Development of a Chemical Reaction in a Turbulent Flow," *Fluid Dyn.*, **12**, pp. 369–377.
- [2] Kuznetsov, V. R., 1982, "Influence of Turbulence on High Nonequilibrium Concentrations of Atoms and Free Radicals in Diffusion Flames," *Fluid Dyn.*, **17**, pp. 815–820.
- [3] Buriko, Yu. Ya., Kuznetsov, V. R., Uryvsky, A. F., Volkov, D. V., and Zaitsev, S. A., 1994, "A Test of a Flamelet Model for Turbulent Non-premixed Combustion," *Combust. Flame*, **96**, pp. 104–120.
- [4] Kuznetsov, V. R., 1983, "Nitric Oxides Formation in Gas Turbine Combustors," *Proc. of Central Institute of Aviation Motors*, No. 1086, Central Institute of Aviation Motors, Moscow (in Russian).
- [5] Buriko, Yu. Ya., Gorbato, A. A., Tchepin, S. A., Zaitsev, S. A., and Volkov, D. V., 1995, "NO_x predictions for Aerocombustors and Gas Turbine Engines Based on Flamelet Model," 1995 *Yokohama International Gas Turbine Congress*, Vol. II, Gas Turbine Society of Japan, pp. 291–296.
- [6] Buriko, Yu., Zakharov, V., Belokon, A., and Opdyke, G., 1999, "Test Validation of Flamelet Model Predictions of NO_x," ASME Paper No. 99-GT-296.
- [7] Scheirer, S. T., and Nolan, J. P., 1994, "NO_x Reduction Program for CCP Brush Cogen Facility W251AA Gas Turbines," *Proceedings of the 1994 American Power Conference*, American Power Conference, Chicago, IL.
- [8] Hung, W. S. Y., and Cambell, A., 1998, "Uncertainty in Gas Turbine NO_x Emission Measurements," ASME Paper No. 98-GT-75.
- [9] Williams, F. A., 1975, *Turbulent Mixing in Non-reactive and Reactive Flows*, S. N. B. Murthy, ed., Plenum, New York.
- [10] Bilger, R. W., 1980, "Perturbation Analysis of Turbulent Non-premixed Combustion," *Combust. Sci. Technol.*, **22**, pp. 251–261.
- [11] Peters, N., 1984, "Laminar Diffusion Flamelet Models in Non-premixed Combustion," *Prog. Energy Combust. Sci.*, **10**, pp. 319–339.
- [12] Kuznetsov, V. R., and Sabel'nikov, V. A., 1990, *Turbulence and Combustion*, P. A. Libby ed., Hemisphere, Washington DC.
- [13] Miller, J. A., and Bowman, C. T., 1989, "Mechanism and Modeling of Nitrogen Chemistry in Combustion," *Prog. Energy Combust. Sci.*, **15**, pp. 287–338.
- [14] Glarborg, P., Miller, J. A., and Kee, R. J., 1986, "Kinetic Modeling and Sensitivity Analysis of Nitrogen Oxides Formation in Well-Stirred Reactors," *Combust. Flame*, **65**, p. 177.
- [15] Bradshaw, P., Launder, B., and Lumley, J., 1991, "Collaborative Testing of Turbulence Models," AIAA Paper No. 91-0215.
- [16] Rogers, S. E., and Kwak, D., 1990, "Upwind Differencing Scheme for the Time-Accurate Incompressible Navier-Stokes Equations," *AIAA J.*, **28**, pp. 253–262.
- [17] Rogers, S. E., Kwak, D., and Kiris, C., 1991, "Steady and Unsteady Solutions for the Incompressible Navier-Stokes Equations," *AIAA J.*, **29**, pp. 603–610.
- [18] Vasiliev, V. I., Volkov, D. V., Zaitsev, S. A., and Lyubimov, D. A., 1997, "Numerical Simulation of Channel Flows by a One-Equation Turbulence Model," *ASME J. Fluids Eng.*, **119**, pp. 885–892.
- [19] Kuznetsov, V. R., 1972, "Probability of Passive Scalar in Turbulent Shear Flows," *Fluid Dyn.*, **5**, p. 86.

Self-Excited Oscillations in Combustors With Spray Atomizers

M. Zhu

A. P. Dowling

K. N. C. Bray

Department of Engineering,
University of Cambridge,
Trumpington Street,
Cambridge CB2 1PZ, UK

Combustors with fuel-spray atomizers are susceptible to a low-frequency oscillation, particularly at idle and sub-idle conditions. For aero-engine combustors, the frequency of this oscillation is typically in the range 50–120 Hz and is commonly called “rumble.” In the current work, computational fluid dynamics (CFD) is used to simulate this self-excited oscillation. The combustion model uses Monte Carlo techniques to give simultaneous solutions of the Williams’ spray equation together with the equations of turbulent reactive flow. The unsteady combustion is calculated by the laminar flamelet presumed pdf method. A quasi-steady description of fuel atomizer behavior is used to couple the inlet flow in the combustor. A choking condition is employed at turbine inlet. The effects of the atomizer and the combustor geometry on the unsteady combustion are studied. The results show that, for some atomizers, with a strong dependence of mean droplet size on air velocity, the coupled system undergoes low-frequency oscillations. The numerical results are analyzed to provide insight into the rumble phenomena. Basically, pressure variations in the combustor alter the inlet air and fuel spray characteristics, thereby changing the rate of combustion. This in turn leads to local “hot spots,” which generate pressure fluctuations as they convect through the downstream nozzle. [DOI: 10.1115/1.1376717]

1 Introduction

Most types of combustion-driven devices experience combustion instabilities. These instabilities can cause serious problems, such as mechanical vibration, noise, enhanced corrosion, and unpredictable performance. Combustors with fuel-spray atomizers are particularly susceptible to a low-frequency oscillation at idle and sub-idle conditions. For aero-engine combustors, the frequency of this oscillation is typically in the range 50–120 Hz and is commonly called “rumble.” In the past, the rumble has caused an audible, and sometimes disturbing noise in the aircraft cabin. However, combustors are required to have ever wider operating ranges and reduced emissions, and this is leading to design changes which can greatly increase the susceptibility to rumble. Radiated sound levels due to rumble have become excessive, and in some cases the pressure fluctuations have been so intense that they stall the engine. In practice, it is known that certain atomizers or atomizer/combustor combinations are particularly susceptible to self-excited oscillation, and yet there is no systematic way of predicting such effects. Industry has a pressing need for methods of analysis giving them the capability to predict the effect of design changes.

In previous work ([1,2]), we have demonstrated that a numerical model can be used to investigate the forced unsteady combustion due to specified time-dependent variations in inlet air and fuel supplies. The influence of atomization has also been investigated. In this paper, we use the same numerical combustion code but couple it to realistic dynamic inlet and outlet conditions to investigate self-excited oscillations. The mechanism for instability can be described as follows: at idle and sub-idle conditions, the pressure drops across the air and fuel supply systems are low and the inlet flows alter in response to pressure oscillations in the combustor. In particular, since atomization is strongly dependent on mass flow rate, the droplet size distribution at inlet is altered. This changes the time taken for the fuel to evaporate and, at the low air temperature at idle, causes unsteadiness in the rate of combustion and its location. This in turn produces regions of nonuniform tem-

perature or “hot spots.” These “hot spots” then convect downstream. It is well accepted that the convection of gas containing nonuniform temperature regions or entropy spots through a nozzle is a significant source of noise ([3,4]). The convection of the density variations in the “hot spots” through the area contraction at combustor exit may generate pressure fluctuations, and the upstream propagating sound wave can modify the pressure differences across the air and fuel inlets. If the phase relationship is suitable, a self-excited combustion oscillation can be established.

In next section, the mathematical models used in the calculations are briefly described. The dynamic boundary conditions are developed in Section 3. With these models, test cases were devised and calculated to examine the rumble phenomenon. The results of numerical calculations are described in Section 4. The mechanism of instability is investigated in more detail in Section 5. Finally, general conclusions and further work are discussed.

2 Mathematical Models

The combustion and atomizer models are identical to those in our previous work on driven perturbations of a combustor at idle ([2]). However, we will summarize the main elements of the model for completeness.

The gaseous flow field is modeled though the equations of motion in Favre ensemble-averaged form. Source terms appear in the continuity and momentum equations due to evaporation and the force exerted by the liquid phase. The frequency of the rumble oscillation is far below typical turbulent eddy frequencies so quasi-steady flow conditions will prevail. The turbulence time scales, as well as the local turbulence intensity, are therefore obtained by the well-known $k-\epsilon$ turbulence model, which is known to give satisfactory predictions of a gas turbine combustor flow field under steady flow conditions.

The thermochemistry is described in terms of the mixture fraction in the gaseous phase and its variance, which are obtained by solving their transport equations together with momentum and $k-\epsilon$ equations. The mixture fraction is defined as the proportion by mass of material originating in fuel stream assuming all molecular diffusivities are equal. With the assumption that all relevant chemical time scales are smaller than the mixing time scales, the calculation of the complex chemical reaction can be performed independent of the solution of the turbulent flow. In the simulation, the coupling between the turbulence and the chemical

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-108. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

reaction is considered by the laminar flamelet presumed pdf method. The fuel used in the calculations is kerosene. The mixture fraction distribution is represented by a beta function, which has been shown to be a good approximation for turbulent jets. The laminar flamelets have been computed and stored in a library, from which values are “drawn” by interpolation during the computation.

The description of the spray is based on Williams’ spray equation ([5]). This equation can be written as

$$\frac{\partial f_s}{\partial t} + \frac{\partial}{\partial x_i} [U_i f_s] = - \frac{\partial}{\partial U_i} [F_i f_s] - \frac{\partial}{\partial R} [\dot{R} f_s] - \frac{\partial}{\partial e} [\dot{Q} f_s], \quad (1)$$

where f_s is the droplet number density function in the joint space spanned by x , U , R and e . Here U is the droplet velocity; F is the total external force on the droplet per unit mass; R denotes the instantaneous droplet radius, and $\dot{R}$ is its rate of change. $\dot{Q}$ is the rate of change of a droplet’s internal energy e . The solution to this spray equation is obtained by means of a Monte Carlo method. The Lagrangian equations for trajectory, velocity, size, and temperature are solved. The droplet dispersion in a turbulent flow is calculated according to the Gaussian distribution ([6]).

The characteristics of the spray to be described as part of the inlet boundary condition include mean droplet size, droplet size distribution, droplet number density, and cone angle. The atomizer and fuel spray cone are small and the rumble frequency is much lower than the spray cone break-up frequency. We therefore assume that the fuel spray responds in a quasi-steady way to the flow through the atomizer. One of the difficulties in the study is that the unsteady combustion is closely coupled with the fuel spray formation process. A number of potential mechanisms may be involved in the atomization process, depending on the type of fuel injector used. Unfortunately, these mechanisms are not well understood. From results obtained over a wide test range, Lefebvre drew the general conclusion that liquid viscosity has an effect that is quite separate and independent from that of surface tension ([7]). He suggested the following equation to describe the steady-state airblast atomization droplet diameter

$$\overline{D}_{32} = 10^{-3} \left(1 + \frac{\dot{m}_f}{\dot{m}_a} \right)^{0.5} \times \left[\frac{(\sigma \rho_f)^{0.5}}{\rho_a U_a} + 0.06 \left(\frac{\mu_f^2}{\sigma \rho_a} \right)^{0.425} \right] \quad (2)$$

where $\dot{m}$ is the mass flux rate, ρ the density, μ the dynamic viscosity, and σ the surface tension. The suffices a and f denote the air and liquid in the atomizer, respectively. Once the Sauter mean diameter (SMD) $\overline{D}_{32}$ is obtained, the droplet size distribution can be calculated from the non-dimensional expression

$$\frac{dV}{V} = 13.5 \left(\frac{D}{D_{32}} \right)^3 \exp \left(-3 \left(\frac{D}{D_{32}} \right) \right) d \left(\frac{D}{D_{32}} \right), \quad (3)$$

where dV/V is the volume fraction occupied by droplets within the size range D to $D+dD$. We use Eqs. (2) and (3) to relate the fuel droplet size to the instantaneous air and fuel flow for low-frequency fluctuations. The fuel spray cone angle is described by random numbers which are generated according to a gamma distribution with its mean specified according to atomizer geometry.

The algebraic finite volume approximation to the conservation equations for gas flow field are written in fully implicit forms. The

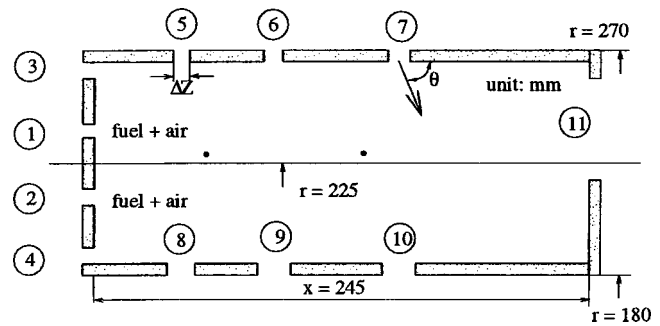


Fig. 1 Schematic diagram of the geometry

method of solution is based on the so-called pressure-implicit by splitting operator (PISO) algorithm, in which the coupling between the pressure and velocity is handled by an operator splitting technique and solved in a predictor-corrector fashion ([8,9]).

3 Boundary Conditions

The geometry used in the numerical simulations is the same as the previous work ([2]). The calculation has been conducted in an idealized two-dimensional axisymmetric annular combustor a section of which is shown in Fig. 1. The boundary conditions used in the numerical computation were specified to be representative of an aero-engine at idle.

Inlet Conditions. Fuel and air are injected through ports 1 and 2, and diffusion air is added through ports 3 to 10 as shown in Fig. 1. The inlet air stagnation temperature is 366 K. The mass flow rates are constant through ports 3–10, and chosen to be representative of these in an aero-engine combustor (see Table 1). The orifice width Δz of each port is chosen to give the same momentum flux in the two-dimensional model as in the engine. The air flow through ports 1 and 2 is unsteady and described by the discharge relationship, which may be expressed as

$$\dot{m}_a = c_d A \sqrt{2 \rho_a (p_0 - p_s)}, \quad (4)$$

where c_d is the discharge coefficient 0.710 here. A is the exit area of orifice of the atomizer. p_0 is the stagnation pressure of the supplied air 225 kPa and p_s the static pressure downstream of injector. The fuel mass flow rate is calculated by the flow number, which is defined as

$$\text{flow number} = \dot{m}_f / \sqrt{\rho_f \Delta p}, \quad (5)$$

where Δp is the pressure drop across the atomizer. Fuel is equally divided between inlets 1 and 2 and injected with the mean speed of 70 m/s and a mean mass flow rate of 0.03 kg/s corresponding to the flow number of 0.5818. The temperature of this liquid fuel is 288 K. We assume the supply volumes to be sufficiently large that the stagnation pressures of the inlet air and fuel are constant. The air and fuel flow rates therefore change according to Eqs. (4) and (5) as the combustor pressure p_s oscillates.

Outlet Condition. The guide vanes near turbine entry are nearly choked and hence the nondimensional mass flow,

Table 1 The inlet geometry used in the numerical simulations. *: mean value, perturbations described by Eq. (4).

Inlet No.	1	2	3	4	5	6	7	8	9	10
x mm	0.0	0.0	0.0	0.0	38.0	77.0	135.0	38.0	77.0	135.0
θ deg.	90.0	90.0	90.0	90.0	58.12	51.92	77.859	62.745	57.76	71.91
r mm	238.0	216.0	270.0	180.0	270.0	270.0	270.0	180.0	180.0	180.0
Δz	1.832	2.019	3.572	3.929	3.408	3.030	4.780	5.112	9.415	7.624
$\dot{m}_a$ kg/s	0.383*	0.383*	0.311	0.224	0.140	0.300	0.679	0.452	1.041	1.211

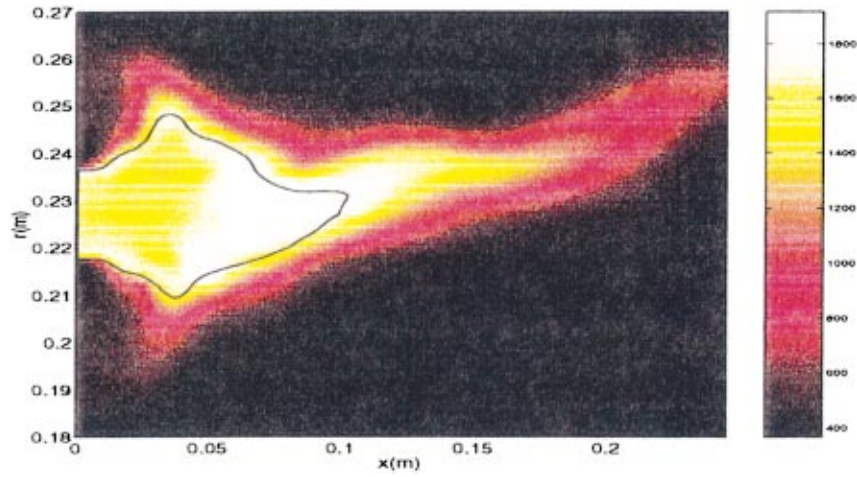


Fig. 2 Contour plot of the mean temperature distribution at idle conditions. The black line indicates the mean position of the stoichiometric curve.

$\dot{m}(t)\sqrt{c_p T_0(t)}/p_0(t)$, is specified and constant there according to the choking condition. We assume that the distance between the end of our computational domain (station 11 in Fig. 1) and turbine entry is compact, thus there is negligible storage capacity between the combustor and turbine. Therefore, the instantaneous mass flows at the combustor exit and turbine inlet are equal. Moreover, we assume that there is no further combustion between the combustor exit and the choking plane and that the accelerating flow in this short region is adiabatic and reversible. Hence, with the assumption of parallel flow at combustor outlet, the instantaneous stagnation pressures and temperatures are the same. The constraint of constant nondimensional mass flow rate at the choking plane can therefore be converted into a boundary condition at combustor

exit ([4]). The instantaneous stagnation pressure $p_0(t)$ at the end of our computational domain can be calculated by the relationship

$$\frac{\sqrt{c_p}}{p_0(t)} \int 2\pi r \rho(r,t) u(r,t) \sqrt{T_0(r,t)} dr = \text{const.} \quad (6)$$

where $T_0(r,t)$ is the stagnation temperature and c_p is the specific heat at outlet. The constant on the right-hand side of Eq. (6) can be obtained by a steady calculation in the same conditions.

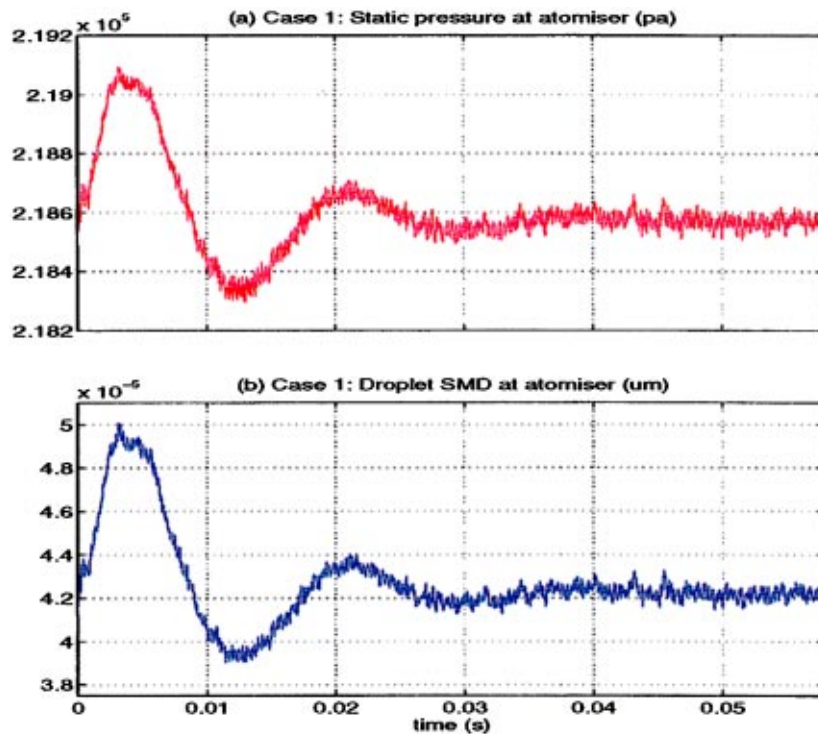


Fig. 3 The time traces of the static pressure and droplet Sauter mean diameter (SMD) at atomizer outlet during the transition period of computational fluid dynamics (CFD) calculations without the choking condition

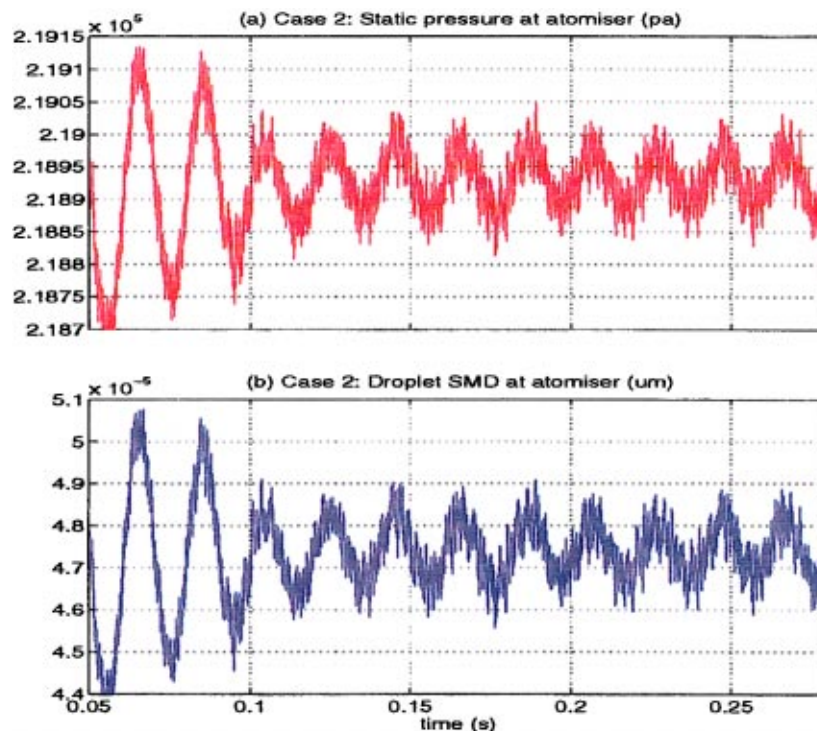


Fig. 4 The time traces of the static pressure and droplet Sauter mean diameter (SMD) at atomizer outlet during the transition period of computational fluid dynamics (CFD) calculations with the choking condition

4 Numerical Results

The calculation was started from a cold flow, and reached a limit-cycle oscillation with artificial relaxation. The time-step was selected to ensure that the acoustic waves travel less than one cell per time-step. A contour plot of a typical mean temperature distribution is shown in Fig. 2.

The computation involves coupling between the air and fuel supplies, unsteady combustion, and the downstream boundary condition. The unsteady combustion is strongly influenced by the fuel spray characteristics. In this study, Eq. (2) is used to couple the droplet SMD to the inlet flow. Moreover, in the calculation, the parameters in Eq. (2) can be adjusted to examine the effects of the atomiser in the combustion oscillation processes.

In order to examine the influence of the outlet boundary condition, the results for an open boundary, $p(t)$ constant, and for the choked turbine boundary condition, Eq. (6), are demonstrated here. For the open end, the time traces of the static pressure and droplet SMD at inlet boundary are shown in Fig. 3. It is seen that the amplitudes of the variations of the static pressure and SMD decay to the noise level after a few cycles of oscillation. In contrast in the results with the choked turbine condition shown in Fig. 4, it can be seen that an oscillation with constant amplitude was built up after a transient period. Here, it should be pointed out that the transient fluctuations are not true physical processes as the appropriate initial condition in the CFD calculation is generally unknown. Nevertheless the results still clearly demonstrate the influence of the outlet boundary condition on the self-excited oscillations.

It should be noted that the downstream boundary is reflective and so issues associated with the numerical implementation of nonreflective boundary conditions ([10]) do not arise. The open end reflects a downstream propagating acoustic wave with the same amplitude: We see from Fig. 3 that this is not sufficient to give self-excited oscillations. In addition to reflecting the acoustic wave, the boundary condition in Eq. (6) describes how an entropy

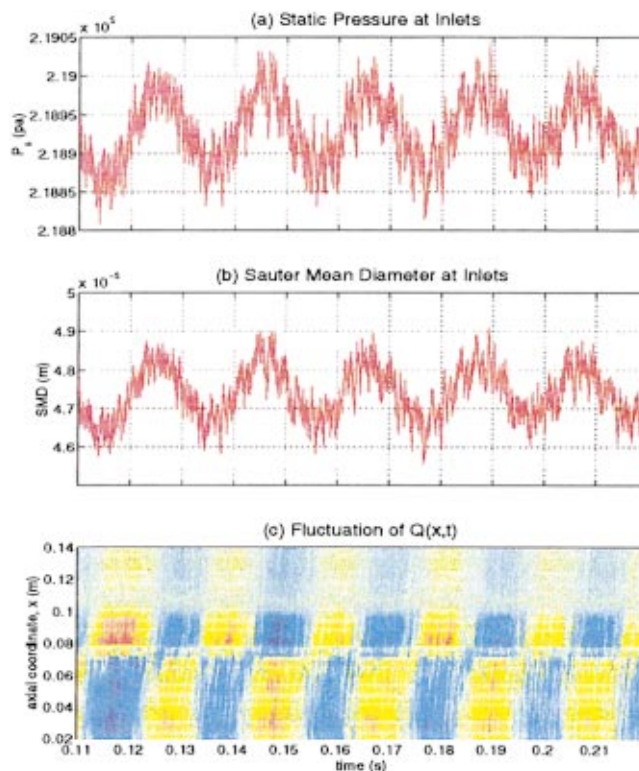


Fig. 5 The variation of the area-averaged rate of heat release/unit length due to the self-excited oscillations in the combustor. The changes of the static pressure and Sauter mean diameter (SMD) in the atomizer are shown in (a) and (b), respectively.

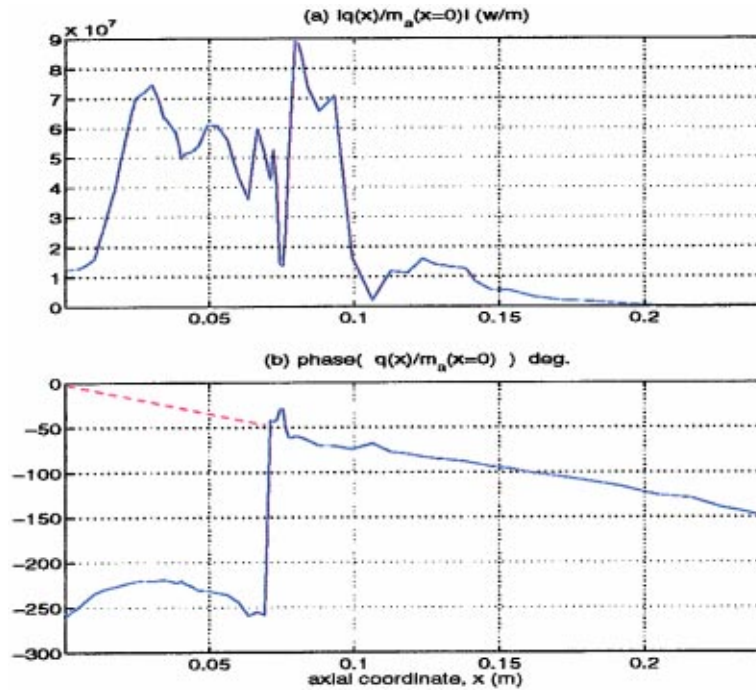


Fig. 6 The transfer function between the heat release rate per unit length and the inlet air mass flow rate in the frequency of the self-excited oscillation (50 Hz)

fluctuation or “hot spot” leads to an additional upstream propagating acoustic wave as it convects through a choked turbine. The conversion of energy from entropy into sound at the downstream boundary plays an important role in the self-excited oscillations.

Shown in Figs. 5(a) and (b) are time traces of the static pressure p_s and droplet SMD at the inlet boundary when the final steady-state oscillation is reached. The variations of p_s and SMD are in phase because an increase of p_s leads to a decrease of the pressure difference across the atomizer. Thus, the mass flow rates of air and fuel as well as their velocities are decreased, and the droplet SMD is increased according to Eq. (2). From Figs. 5(a) and (b), it can be seen that the frequency of oscillation is about 50 Hz. This frequency is close to the range of experienced on engines at similar design and flow conditions, typically 55–65 Hz. Although within a two-dimensional geometry, it is not possible to get the convection speed and hence the frequency exactly right, the modeling includes enough detail to get realistic self-excited oscillations: The frequencies are of the right order, and more importantly sufficient physics has been captured to highlight the main mechanisms of instability.

5 Mechanism of Instability

In order to understand the mechanisms that drive the rumble instability, we first investigate the form of the rate of heat release. Its essential features can be expressed in a concise way by integrating over the combustor cross section to determine $q(x,t)$, the rate of heat release per unit length of combustor. The fluctuation of $q(x,t)$ is shown in Fig. 5(c). Here the colors red and yellow indicate that $q'(x,t)$ is larger than the local mean value $\bar{q}(x)$, and blue and dark blue indicate when it is smaller. We can see that the local heat release rate also varies harmonically at the same frequency as the pressure and SMD in Figs. 5(a) and (b), respectively. The phase of the heat release fluctuation is strikingly different upstream and downstream of the location $x=0.075$ m: Perturbations of the rate of heat release in the dilution zone ($x > 0.08$ m) are effectively 180 deg out of phase with those in the primary zone. More quantitative information on the phase rela-

tionship can be obtained by evaluating the temporal Fourier transforms of $q(x,t)$ and the air mass flow rate at inlets. The results of the magnitude and phase of the transfer function $\hat{q}/\hat{m}_a$ at the self-excited frequency (50 Hz) are plotted in Fig. 6. Two regions with distinct and different forms of flame responses are evident. A similar transfer function was obtained in our previous work on forced oscillations (see [2] Fig. 16) and we can use the understanding of the combustion dynamics built up there to explain the main features of Fig. 5.

To aid in this interpretation, Figs. 7 and 8 show time traces of the static pressure, mixture fraction, heat release rate, and entropy in the recirculation zone and at the outlet of combustor, respectively. High inlet air velocity leads to a decrease in the SMD at the inlet (see Fig. 7(a)). These smaller droplets evaporate quickly, and increase the mixture fraction in the primary zone (Fig. 7(c)) with a recirculation time delay of about 3 ms. Since the fuel-air ratio in the primary zone is richer than stoichiometric, an increase in mixture fraction there decreases the heat release rate (Fig. 7(d)). In contrast, in the dilution zone, where the mixture fraction is weaker than stoichiometric, the local rate of heat release is in phase with the mixture fraction (compare Figs. 8(c) and (d)). This leads to heat release rates throughout the diffusion zone in phase with the inlet air velocity with a convection time delay (see Fig. 6(b)).

The coupling between the distributed heat release and the inlet air flow rate is just as we have seen for previous forced calculations. This reinforces the usefulness of investigating the response of a flame to forcing inlet oscillation to determining the “flame transfer function.” But an understanding of self-excited oscillations must also include consideration of the feedback processes.

First we note that, as one would expect in this example where stagnation pressure of the supply is constant, it is static pressure variations in the combustor that drive the changes in inlet air flow rate. A decrease of the static pressure in the combustor leads to an increase in the inlet air mass flow rate (see Figs. 5(a) and 7(a)).

The development of the entropy wave is shown in Fig. 9 by giving time traces of the entropy fluctuations at approximately equidistant sample points along the main stream line. Two cycles

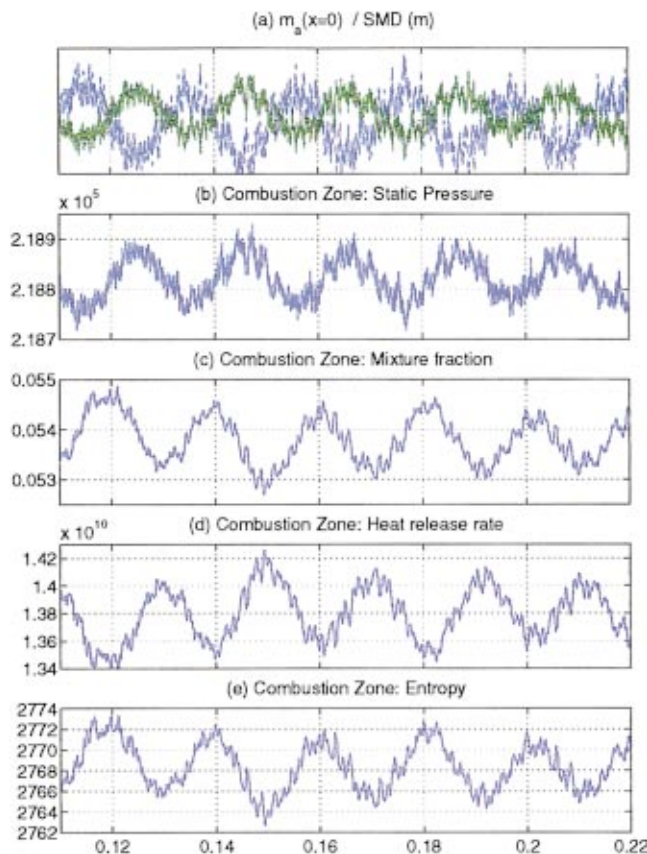


Fig. 7 The self-excited oscillations of the static pressure, mixture fraction, heat release rate, and entropy at the sample point in combustion zone. The variations of the air mass flow rate and Sauter mean diameter (SMD) in the atomizer are indicated in (a) where the green and blue lines indicate the SMD and the air mass flow rate, respectively.

of fluctuations are demonstrated here. Entropy fluctuations in the primary zone are small in amplitude, but become larger downstream of the stoichiometric curve. The dilution air provides some mixing but the amplitude of the entropy waves remains significant at the combustor exit. It is clear that the fluctuations of entropy are indeed small before the combustion zone and are increased afterwards. Moreover, like heat release rate, the phase is also shifted when crossing the stoichiometric line.

Entropy fluctuations in the dilution zone propagate downstream like a wave, travelling with the convection speed. In order to examine the interaction between the entropy and pressure waves, the pressure fluctuations at the same sample points are plotted in Fig. 10. We can see that the outlet boundary condition leads to the generation of a positive pressure wave, when the “hot spot” arrives at the exit. Moreover, since the pressure wave propagates with the speed of sound, the pressure is virtually in phase throughout the combustor.

The cycle of oscillation is now clear. A reduction in combustor pressure leads to an increase in inlet air velocity, and fuel droplets with small SMD. This increases the burning within the combustor and leads to a heat release rate and an entropy increase in the dilution zone, which lag the inlet air velocity by a convection time delay. As this entropy wave propagates through the downstream constriction, it generates a positive pressure pulse and completes half the cycle. It is clear from Figs. 9 and 10 that the time delay between the minimum of the static pressure at inlet and the peak of entropy at the exit is about 0.01 s. This half-cycle time corresponds to the frequency of 50 Hz. In the next half-cycle, the higher combustor pressure reduces the inlet air flow rate, and

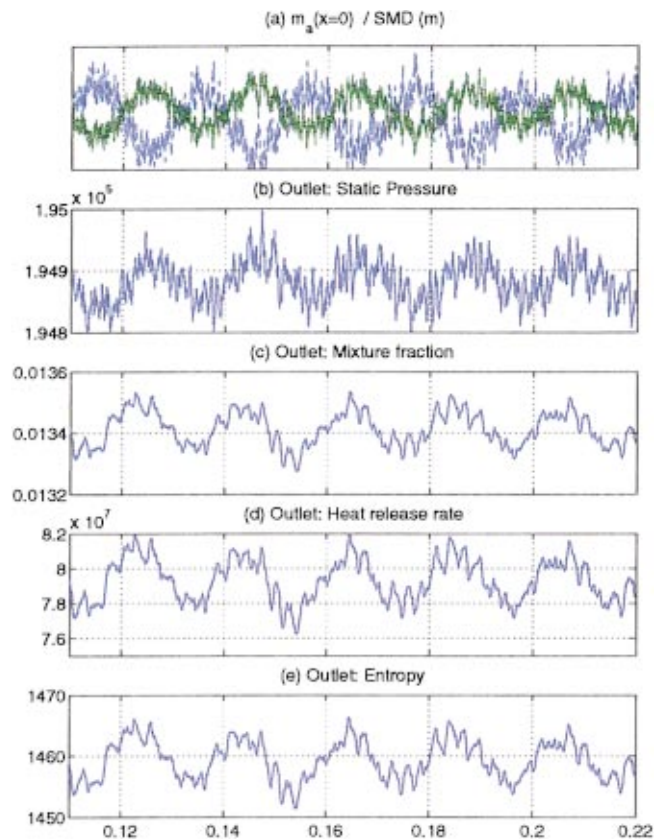


Fig. 8 The self-excited oscillations of the static pressure, mixture fraction, heat release rate, and entropy at the outlet. The variations of the air mass flow rate and Sauter mean diameter (SMD) in the atomizer are indicated in (a), where the green and blue lines indicate the SMD and the air mass flow rate, respectively.

leads to larger fuel droplets. Combustion is reduced and a “cold spot” convecting through the nozzle leads to a reduction in combustor static pressure. The magnitude of oscillation will greatly depend on the sensitivity of the SMD variation to the changes in atomizer air flow rate and the diffusion of the convected entropy wave before the exit nozzle.

In the examples shown in Figs. 4 and 5, the casing volume has been considered to be sufficiently large that its stagnation pressure remains constant irrespective of changes in its delivered mass flow rate. Preliminary calculations have been done to investigate the influence of size of the plenum formed by the casing. Its influence on the resonant frequency is small.

6 Conclusions

The self-excited oscillating behavior of spray combustion has been investigated through time-dependent calculations of the combustion process. The coupling between the inlet fuel and air, atomization and the downstream area constriction are fully considered. The calculated oscillation frequency is of right order as met on aeroengines operating at similar conditions.

From this study, it is clearly demonstrated that the low-frequency oscillation can be established by the interaction between unsteady combustion and the upstream and downstream boundaries. A decrease in combustor pressure causes an increase in the rate of heat release with a convection time delay. This is because the increased pressure difference across the atomizer leads to higher atomizer air velocities which break up the fuel droplets more effectively, thereby improving the combustion efficiency. These larger rates of heat release lead to “hot spots” or

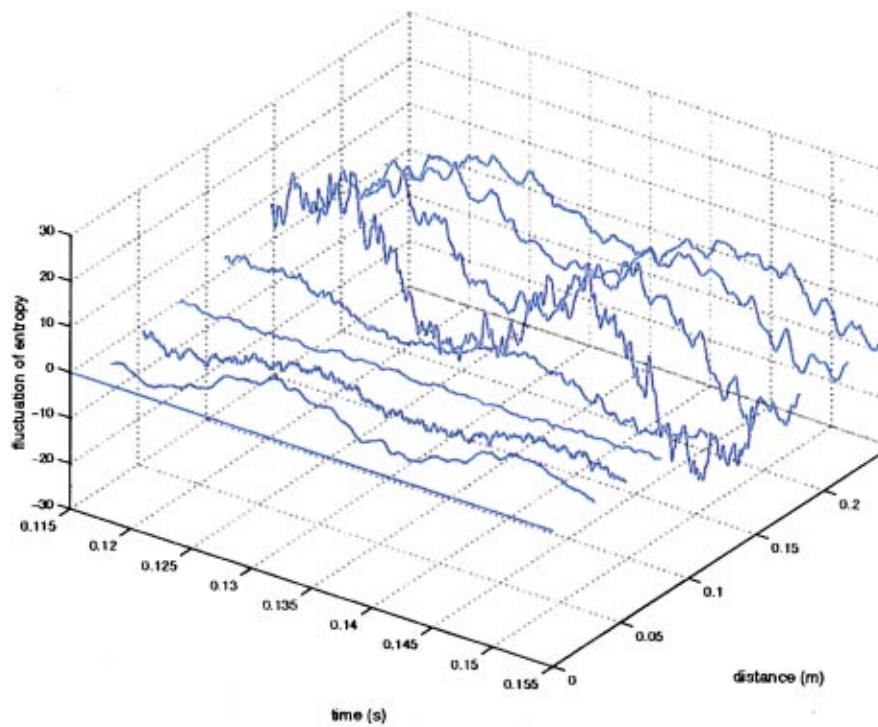


Fig. 9 The propagation of the entropy wave along the main stream flow

regions of low density. As these “hot spots” accelerate through the exit nozzle, a positive pressure wave is generated. This propagates upstream, reducing the atomizer air flow and leading to poorer atomization and less combustion. When the resulting “cold spot” accelerates through the nozzle, it cause a negative pressure wave, which again speeds up the atomizer air flow and enhances the atomization, thus completing the cycle. The period

of oscillation is approximately twice the convection time through the combustor. This corresponds to a frequency of oscillation of about 50 Hz for our combustor geometry.

The susceptibility to instability and the magnitude of oscillation when the system is unstable depend greatly on the sensitivity of the atomisation to change in air flow rate. Further work is needed

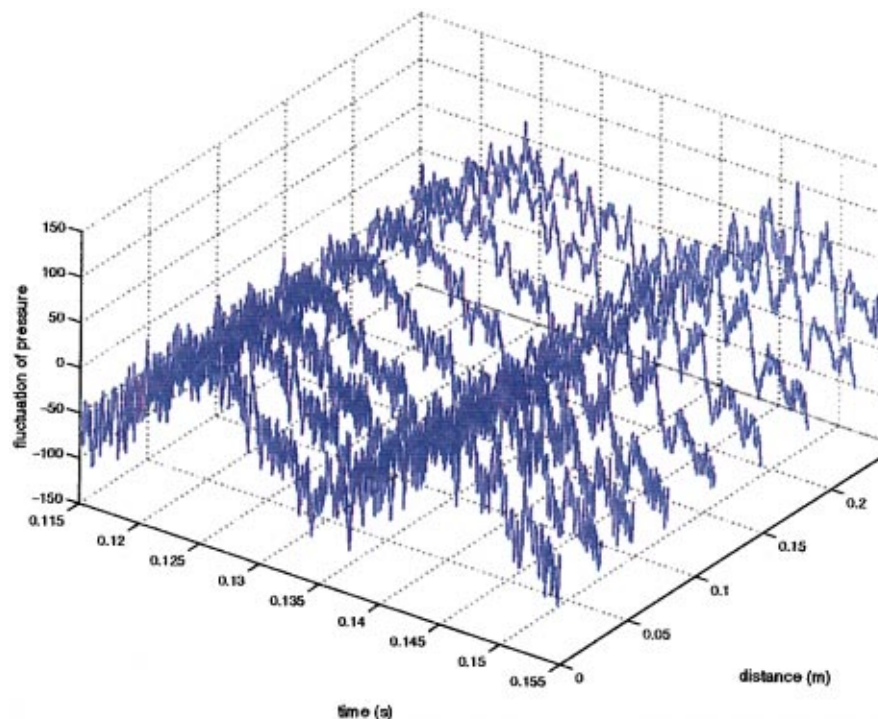


Fig. 10 The propagation of the pressure wave along the main stream flow

to obtain predictions for different atomisers and to investigate more fully the influences of the combustor/casing geometry.

Acknowledgment

This work was funded by the Engineering and Physical Sciences Research Council and Rolls-Royce whose support is gratefully acknowledged.

References

- [1] Zhu, M., Dowling, A., and Bray, K., 1999, "Combustion Oscillations in Burners With Fuel Spray Atomiser," ASME Paper No. 98-GT-302.
- [2] Zhu, M., Dowling, A., and Bray, K., 1999, "Forced Oscillations in Combustors With Spray Atomisers" ASME J. Eng. Gas Turbines Power, to appear.
- [3] Cumpsty, N., and Marble, F., 1977, "The Interaction of Entropy Fluctuations With Turbine Blade Rows: A Mechanism of Turbojet Engine Noise," Proc. R. Soc. London, Ser. A, **A357**, pp. 323–344.
- [4] Marble, F., and Candel, S., 1977, "Acoustics Disturbances From Gas Non-uniformities Convected Through a Nozzle" J. Sound Vib., **55**, No. 2, pp. 225–243.
- [5] Williams, F., 1985, *Combustion Theory*, 2nd Ed., Benjamin/Cummings, Menlo Park, NY.
- [6] Gosman, A., and Ioannides, E., 1983, "Aspects of Computer Simulation of Liquid-Fueled Combustors," J. Energy, **7**, No. 6, pp. 482–490.
- [7] Lefebvre, A., 1989, *Atomization and Sprays*, Hemisphere, New York.
- [8] Issa, R., 1986, "Solution of the Implicitly Discretised Fluid Flow Equations by Operator-Splitting," J. Comput. Phys., **62**, pp. 40–65.
- [9] Issa, R., Ahmadi-Befrui, B., Beshay, K., and Gosman, A., 1991, "Solution of the Implicitly Discretised Reacting Flow Equations by Operator-Splitting," J. Comput. Phys., **93**, pp. 388–410.
- [10] Poinso, T., and Lele, S., 1992, "Boundary-conditions for Direct Simulations of Compressible Viscous Flows," J. Comput. Phys., **101**, No. 1, pp. 104–129.

Research on a Methane-Fueled Low NO_x Combustor for a Mach 3 Supersonic Transporter Turbojet Engine

Y. Kinoshita

e-mail: kinoshita@ati.khi.co.jp

T. Oda

J. Kitajima

Akashi Technical Institute,
Kawasaki Heavy Industries, Ltd.,
1-1 Kawasaki-cho,
Akashi City, Hyogo, Japan

Methane-fueled low NO_x combustor research had been conducted under the Japanese supersonic/hypersonic propulsion research program. A unique form of premixture jet swirl combustor (PJSC) was proposed for the ultra low NO_x combustor of a Mach 3 turbojet engine. Fuel-air mixing tests and fundamental combustion tests were conducted to obtain the design data and combustion characteristics in the first phase of the research. A single can-type combustor was fabricated and high-temperature and high-pressure combustion tests were carried out for the evaluation on NO_x emission reduction capability of the PJSC concept in the second phase. In the final phase of research, a multisector combustor was fabricated and the performance demonstration test was conducted for the final evaluation of the pollutant exhaust emission goals and the combustor performance goals set in the HYPR project. The sequential three-phased program was completed successfully, and the project goals of NO_x emission, combustion efficiency, pressure loss and exit gas temperature pattern factor at the Mach 3 cruise condition, together with the ICAO regulatory levels for supersonic aircraft at LTO conditions, were all achieved in the performance demonstration test. [DOI: 10.1115/1.1377009]

Introduction

Environmental disruption of the ozone layer by NO_x emission exhausted from the high-speed airplane at high altitude has become a critical issue. ICAO, the international civil aviation organization, has been investigating a future emissions regulation that includes a strict regulatory level of NO_x emission in the stratosphere. Therefore it is very important to develop low-emission combustion technology, not only at landing and take-off conditions but also at high-altitude cruising conditions, for success in the development of the next generation high speed airplane.

The Super/Hypersonic Transport Propulsion System Research (HYPR) Project was funded by the Ministry of International Trade and Industry (MITI) during the last decade. The propulsion system investigated in the HYPR project was a combined cycle engine (CCE), which consists of a turbojet engine for the lower speed range from take-off to Mach 3 and a ramjet engine for the higher speed range from Mach 2.5 to 5. Liquid natural gas containing a high purity of methane was selected as fuel for its major advantage in dealing with the high temperatures created by high-speed flight, though its availability and handling are disadvantageous relative to kerosene. Another reason is that methane fuel is stable chemically and is appropriate for lean premixed combustion. The research on a methane-fueled low NO_x combustor for the Mach 3 turbojet engine started in 1991 and has just completed in March 1999 and was considered to be the most important component research program in the project.

Research and development of methane-fueled low NO_x combustors utilizing lean premix combustion technologies for land-based gas turbines have made much progress in recent years, but little for aircraft gas turbine application. There are many special performance requirements for aircraft gas turbine combustors

which differ from land-based ones, such as wide operating range, compactness, light weight, and totally reliable response to a rapid change in power demand.

Major species exhausted from aircraft engines are nitrogen oxides NO_x, unburned hydrocarbons UHC, and carbon mono-oxide CO for the gaseous products. CO and UHC are generated in a process of imperfect combustion, therefore they are considered to be critical at low-power conditions such as idle or taxiing. On the other hand, NO_x is generated by oxidization of nitrogen in air at high-temperature conditions in a combustion process, and NO_x emission shows a tendency to increase with increase in temperature and pressure of the engine cycle and by an increase in flight speed. Combustor inlet temperature and exit temperature become 1008 K and 1810 K, respectively, at Mach 3 cruise condition for the HYPR turbojet engine. These temperatures are extremely high compared to those of conventional subsonic combustors. An NO_x emission index of more than 50 g/kg fuel at Mach 3 cruise condition would be expected utilizing conventional combustion technology. This high level of NO_x emission will definitely not be accepted in view of the altitude environmental protection trend. NASA reported that the acceptable NO_x emission index was less than 5 g/kg fuel in the stratosphere [1]. The target of NO_x emission was decided to be below 5 g/kg fuel at Mach 3 cruise condition in the HYPR project, and this was equivalent to reducing NO_x emission to one tenth that of conventional technology. The HYPR's target on NO_x emission is illustrated in Fig. 1.

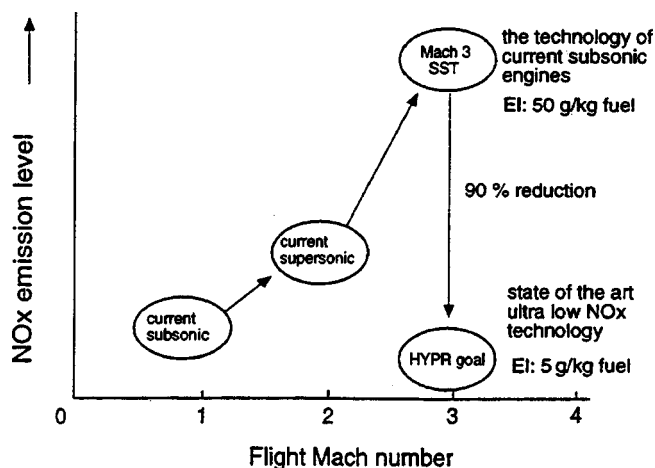
This paper describes the history of the low NO_x combustor research. The experimental efforts of the phase one and two programs are summarized briefly because these were already reported in detail in the previous paper [2]. This report focuses on the experimental efforts of the final phase program, especially on the results of the performance demonstration rig test study.

Program Plan and Goals

The methane-fueled low NO_x research was a multiyear effort, which was initiated in April 1991 and completed in March 1999.

- 1 To generate the combustion technology required to develop a

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-113. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.



Mach 3 supersonic transporter turbojet engine with significantly lower pollutant exhaust emission levels than those of current technology engines and

2 to demonstrate the low emission levels and other combustor performances in rig tests with a demonstration sector combustor.

The research was conducted in three sequential phases and the overall schedule is presented in Fig. 2.

Phase 1 Program. The objective was to identify the strategy for reducing NO_x emission to one tenth that of conventional technology. Phase 1 program efforts involved the proposal of a promising combustion methodology for obtaining the pollutant exhaust emission level reduction, and the fundamental combustion test for obtaining the basic design data and combustion characteristics of the low NO_x combustor proposed.

Phase 2 Program. The objective was to validate the possibility of the low NO_x methodology for achieving the target NO_x emission level. Phase 2 efforts involved the fabrication of a single can-type combustor and the high-temperature and high-pressure combustion test for evaluation of the capability of the low NO_x combustor.

Phase 3 Program. The objective of the final phase was to demonstrate the pollutant emission goals and the combustor performance goals. Phase 3 program efforts included the fabrication

JFY	1991	1992	1993	1994	1995	1996	1997	1998
	Phase 1							
	Fuel mixing test	Fundamental combustion test						
				Phase 2				
				High Temp. & press. combustion test				
						Phase 3		
						Demonstration combustion rig test		

Fig. 2 Low NO_y research overall schedules

Design point (Mach 3 cruise condition)		
NO _x emission index	g/kg fuel	≤5
Combustion efficiency		≥99.5
Total pressure loss ratio		≤5
Pattern factor		0.2
Off design point (LTO)		
ICAO Annex 16 volume II	Chapter 3 supersonic	

of a multisector combustor, the performance demonstration rig test then the final evaluation to the pollutant exhaust emission goals also combustor performance goals.

The program goals, which included the pollutant emission goals and combustor performance goals, are presented in Table 1. The most critical issue was NO_x emission at high Mach conditions, and the goal was below 5 EI at Mach 3 cruise. Another pollutant goal was to meet the ICAO regulatory levels for a supersonic aircraft.

Phase 1 Program and Major Results

HYPR's Low NO_x Methodology. There are two major approaches to realize ultra-low NO_x emission. One is the rich burn, quick quench, lean burn method, and the other is the lean premix combustion method. The rich burn, quick quench, lean burn combustor produces very low NO_x emissions ([3,4]). The major issues in this type of combustor are smoke production and heat damage of the primary zone liner wall ([5]). The lean premix combustor also can achieve very low NO_x levels by very lean and uniform mixtures ([6]). The issues for this type combustor are lean blow-out of the flame, auto-ignition or flashback in the premix chamber, and acoustic instability.

The lean premix combustion method was chosen for HYPR's low NO_x combustor for the following reasons. Methane is quite a stable fuel. In addition, the inlet air pressures are less than 1000 kPa, and they are lower than those of current efficient turbofan engines at climb and cruise conditions. Therefore, though the inlet temperatures are very high, auto-ignition is thought to be avoidable, if the mixing devices are well designed. Moreover, higher inlet air temperatures produce a wider burning range, and improve instability of lean flames. This makes it possible to burn at lower fuel-air ratio conditions and thus to generate less NO_x .

Premixture Jet Swirl Combustor Concept. Figure 3 presents a premixture jet swirl combustor (PJSC) concept based on the lean premix combustion method, which was proposed to realize ultra-low NO_x performance for the HYPR combustor. The combustor has a unique fuel preparation device, which is composed of six premixing chambers on the combustor dome, with each chamber connected to the dome surface at certain angles not

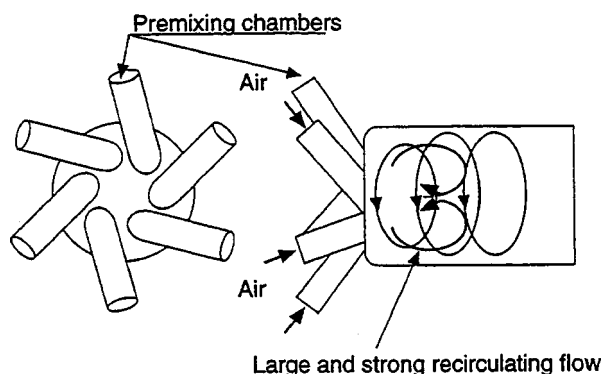


Fig. 3 Concept illustration of the premixture jet swirl combustor

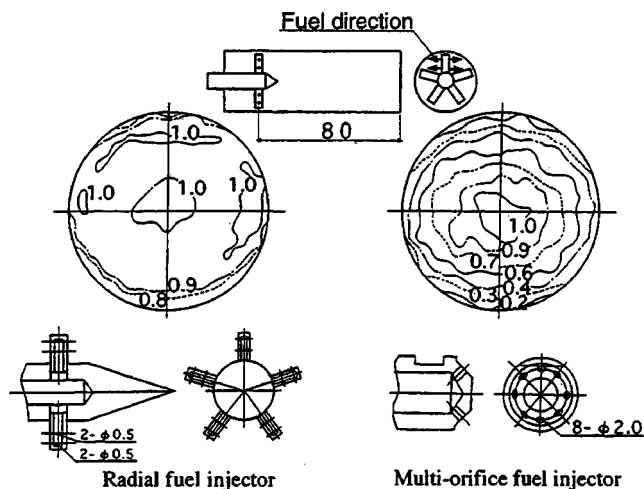


Fig. 4 Fuel concentration distributions of fuel injectors

only to the axis but also to the circumference of the combustor. This arrangement forms a large and strong recirculating flow by the impingement of six premixture jets.

The fuel preparation device does not include air swirlers and simply consists of fuel injectors and tubes, because the swirling flows from swirlers often cause flashback and auto-ignition problems. Main fuel injectors are inserted in the premixing chambers and fuel is injected into the main air stream through many small holes within the tube to mix with the air. United Technologies investigated the auto-ignition characteristics of methane-air mixtures to identify the exact auto-ignition delay times in the HYPR project ([7]). The mixing length of this combustor was defined utilizing the results to prevent auto-ignition so that the residence time should not exceed the auto-ignition delay time with some margin.

Fundamental Combustion Test. The fundamental combustion tests were planned to obtain combustor design data in the phase 1 program. Main design parameters were the relation between the configuration of the premixing chambers and mixing characteristics, such as mixing length, arrangement of premixing chambers, actual fuel air ratio within a premixing chamber, etc. Two experimental studies were investigated in the preliminary combustion test series. One was a fuel mixing study and the other was a fundamental combustion study with model PJSC combustors.

Fuel Mixing Study. Fuel distribution within the premixing chamber has a great influence on NO_x formation. Non uniform mixtures generate a large amount of NO_x and may easily cause auto-ignition. The fuel-air mixing test was carried out with the combination of a fuel injector and a premixing chamber as shown in Fig. 4. A sampling probe was located just at the exit of the premixing chamber and was moved to many measuring points automatically by an electric moving table. The mixture was sampled and introduced to a total hydrocarbon analyzer (FID system) at each point. Two major types of fuel injector were tested in this study, a radial fuel injector and a multi-orifice fuel injector. Fuel was compressed natural gas (CNG) supplied from high pressure bottles. MFR in this figure nearly coincided with that of the Mach 3 cruise condition, and the measured concentrations were normalized by a maximum concentration level.

Figure 4 shows the fuel concentration distributions for both the radial fuel injector and the multi-orifice fuel injector. The results show that the radial fuel injector exhibited a high degree of uniformity and no noticeably high-density region of fuel was observed. Other test results also showed that change in size of injection holes has little influence on the mixing and fuel distribution.

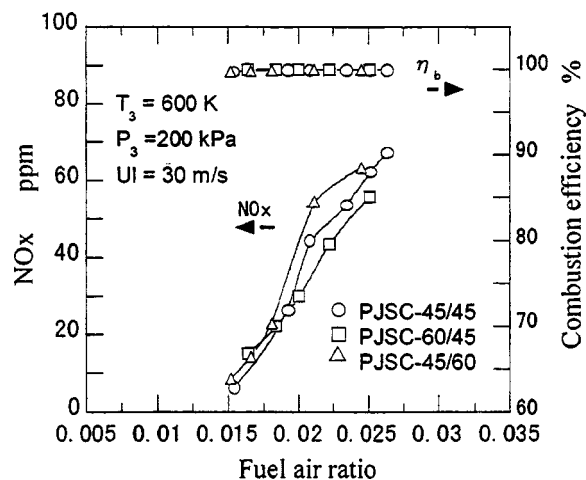


Fig. 5 Effect of premixing tube installation angles on NO_x and combustion efficiency

On the other hand, the multi-orifice fuel injector showed poor mixing characteristics which is not attractive for an ultra-low NO_x combustor.

The fuel mixing study demonstrated that the premixing fuel nozzle with a radial fuel injector achieved very uniform fuel concentration distribution desirable for an ultra-low NO_x combustor, and this type of fuel nozzle was selected as a primary candidate for the PJSC.

Fundamental Combustion Study. The combustor tested in this study was a can type PJSC as shown in Fig. 3. This study was focused on the design data acquisition for the premixing chamber arrangements and on obtaining their fundamental combustion characteristics.

The combustor was 150 mm in diameter and 300 mm in length, and it had a secondary air system of eight holes of 15 mm diameter and four dilution holes of 18 mm diameter. Six premixing chambers were mounted on the combustor dome at an angle α degrees to the vertical axis and β degrees to the circumferential direction. Three combustors were manufactured for the criteria above, which values of α/β of (45/45), (60/45) and (45/60). Fuel injectors were the radial type having five spokes and four holes of 0.8 mm diameter on each spoke, and the mixing length was 80 mm.

Combustion tests were carried out at a moderate inlet condition, with a temperature of 600 K and a pressure of 200 kPa. Figure 5 shows the effect of the premixing chamber installed angles on NO_x emissions and combustion efficiency. All three combustors, namely PJSC-45/45 A, PJSC-60/45 A, and PJSC-45/60 A, showed stable burning at a FAR over 0.015, and achieved a very high combustion efficiency of more than 99.5 percent. Lean blowout, however, occurred in the order of PJSC-60/45 A, PJSC-45/45 A, and PJSC-45/60 A at a FAR under 0.015. NO_x emissions increased rapidly with increasing fuel air ratio, and the effect of fuel air ratio on NO_x emissions was almost the same among the three combustors. The values of NO_x emissions, however, were different among the three combustors, and the lowest NO_x emissions were obtained with the PJSC-60/45 A combustor. The PJSC-45/45 combustor was selected as a primary candidate combustor from viewpoints of not only NO_x emissions but also flame stabilization.

Phase 2 Program and Major Results

HTHP Combustion Test Program. The high-temperature and high-pressure combustion test (HTHP) program was planned and conducted in the second phase of the low NO_x research. The objectives of the program were to prove the ultra-low NO_x con-

cept of PJSC and to obtain combustion characteristics of a model combustor at the high-temperature and high-pressure condition of the actual Mach 3 cruise.

HTHP Combustor Description. The HTHP combustor was a single can type PJSC with six premixing chambers mounted at angles of 45 degrees both α and β based on the preliminary combustion test results. The HTHP combustor was equipped with radial-type injectors for the main fuel nozzle and it had also a conventional multi-orifice primary fuel nozzle installed at the center of the dome for ignition and off design conditions. The combustor had no secondary air system and was designed to provide 85 percent of airflow for the fuel-air premixing, 10 percent for the liner cooling and 5 percent for the dilution. The combustor was 150 mm in diameter and 300 mm in length, just the same as the previous phase combustor. The combustor adopted a simple film cooling structure and its inner insulation liner was made of iridium alloy to resist high temperature and to protect the liner from heat damage. The HTHP combustor is shown in Fig. 6.

Test Results and Discussion. Figure 7 shows the combustion efficiency and NO_x emission index of the HTHP combustor at the Mach 3 cruise inlet condition. The combustor achieved very high combustion efficiency of about 99.9 percent over a wide range of FAR's tested. The levels of UHC and CO emissions were almost zero and less than 10 ppm, respectively. In addition, primary fuel injection extended stable and efficient burning to lower FAR, and combustion efficiencies of more than 99 percent were obtained at

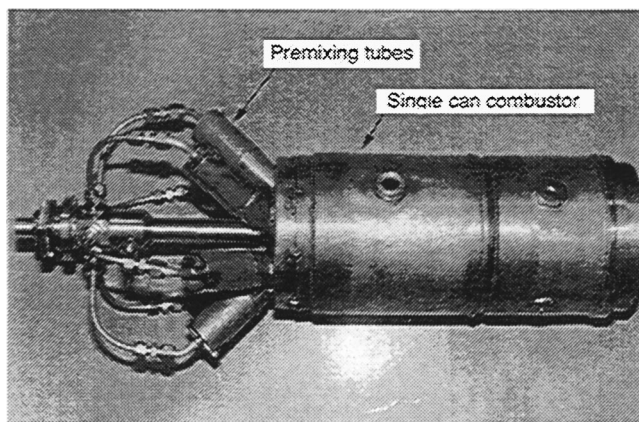


Fig. 6 Photograph of high-temperature and high-pressure (HTHP) combustor

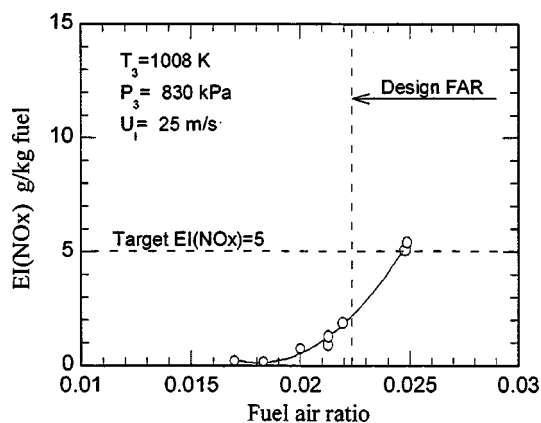


Fig. 7 NO_x emissions of high-temperature and high-pressure (HTHP) combustor at Mach 3 cruise

a FAR of 0.0025. There was no heat damage or significant combustion instability for the HTHP combustor during the test.

NO_x emission index increased with increase in fuel-air ratio, and the emission index of about 2 g/kg fuel was obtained at the design FAR of 0.0223. The HTHP combustor showed superior emission and combustor performance and met the NO_x emission and combustion efficiency goals, and the PJSC concept was proved to be the candidate for the demonstration sector combustor for the final phase of the low NO_x research.

Phase 3 Program and Details

Performance Demonstration Test Program. The Mach 3 turbojet combustor needs to have not only low pollutant emissions but also satisfactory combustor performance, such as combustion efficiency, pressure loss and pattern factor for the wide range of engine operations. Desirable combustor configurations were investigated in the early stage of this phase, to achieve the required performance characteristics listed in Table 1. Two candidate multisection combustors were designed and fabricated based on the investigation results, and a screening rig test was conducted to choose the last configuration. The combustor selected in the preliminary rig test was modified a little in its design utilizing the test data, and named the demonstration sector combustor. The demonstration rig test was conducted in the last stage of the final phase program. The objectives of the demonstration rig test were to demonstrate that the PJSC-type low NO_x combustor could achieve the target values and to obtain a useful NO_x emission database at high temperature and high pressure conditions for the lean premixed low NO_x combustor.

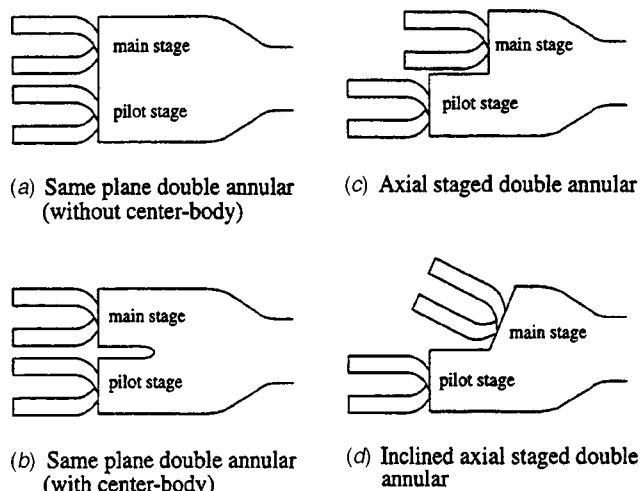
Candidate Low NO_x Combustor Configurations. There are two major approaches in the actual design of low NO_x combustors to accomplish the stringent target performance, namely, the staged combustor and the variable geometry combustor. Typical staged combustors are well known as double annular combustors (DAC) and some of them are actually in service now. Variable geometry combustors (VGC) were investigated in many development programs but difficulties in mechanical stability and reliability have prevented their use in service so far.

The double annular staged combustor concept was chosen as the basic configuration in the study, because firstly it has superior emission performance over a wide operating range, secondly we had prior experience of developing a DAC in another research project. The DAC features the use of multistage burning to provide proper combustion conditions not only at low engine power conditions to achieve low CO and UHC emission levels but also at high speed flight conditions to limit the NO_x emission levels. Four types of DAC configuration were investigated illustrated in Fig. 8. A same plane double annular combustor (SPDAC) is the basic configuration, and this can be subdivided into two further types: with or without a center body. The axial staged double annular combustor (ASDAC) is the other promising configuration, and this can also be subdivided into two types: simple ASDAC or inclined ASDAC. The advantages and disadvantages of each combustor were investigated, then each combustor was marked as shown in the figure. Simple ASDAC was considered to show satisfactory combustion performance at low-power conditions such as idle or descent, and achieved the best score. SPDAC without a center body got the second highest score for its simple construction, but it was thought that interference between the pilot stage and the main stage would be problematic and that combustion efficiency might be poor at low-power conditions.

These two combustors were chosen as the candidates in this study and were carried forward into the next study of the screening rig test.

Screening Rig Test Study

I The 60 deg sector combustors description. Two 60 deg sector combustors were designed based on the ASDAC and SP-



Feature of the each configuration candidates and evaluation			
	configuration	feature	evaluation
(a)	Same plane double annular (without center-body)	• short combustor • relatively simple configuration • interfere between the two stages	○
(b)	Same plane double annular (with center-body)	• short combustor • no interfere between the two stages • damages to the center-body	△
(c)	Axial staged double annular	• good in optimization in staging • no interfere between the two stages • long combustor • relatively large surface for cooling	○
(d)	Inclined axial staged double annular	• good in optimization in staging • no interfere between the two stages • long combustor • relatively large surface for cooling • increasing in combustor diameter	△

○ : good △ : behind in remark

Fig. 8 Staged combustor configurations and their features

DAC configurations selected. The general arrangement of the ASDAC design is shown in Fig. 9. The combustor consisted of two annular dome assemblies and the combustor cooling liners. The dome assembly of the outer annulus included three fuel preparation modules and was the main stage. The dome assembly of the inner annulus included two similar fuel preparation modules and was the pilot stage. These two dome assemblies were offset in the flow direction about 80 mm. The maximum dome height of the combustor was 162 mm, and maximum length was 255 mm. The

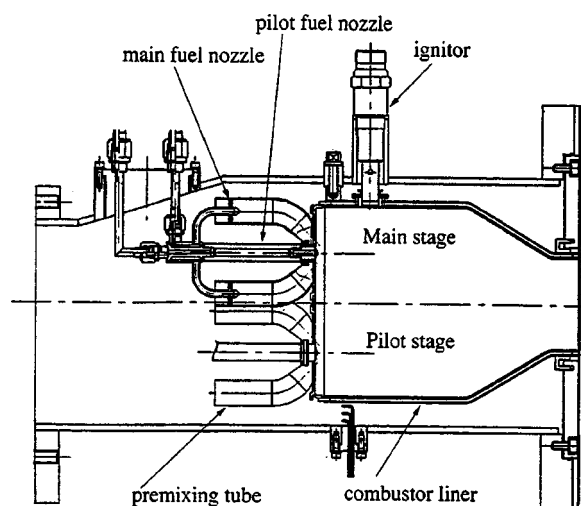


Fig. 9 Schematic of the same plane double annular combustor

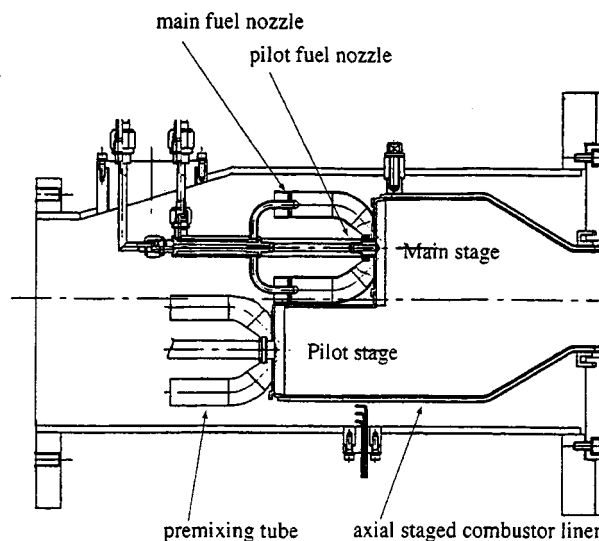


Fig. 10 Schematic of the axial staged double annular combustor

fuel preparation module was designed based on the PJSC concept but it was much smaller than that of the HTHP combustor. Each premixing chamber consisted of six stainless steel tubes but they were newly designed to bend near the dome for easy installation of the fuel nozzle. The fuel nozzle consisted of six main injectors arranged circularly with a pilot injector at the center. The main injector was the radial type, which showed a high degree of premixing performance, and it was inserted within the premixing tube. The pilot injector was of the multi-orifice type and was inserted into the combustion chamber directly. The combustor liners were of double wall construction to simulate transpiration cooling. This cooling structure showed superior cooling performance under high temperature operation. The dome and liners were made from a high temperature cobalt-based alloy, HA-188.

The general arrangement of the SPDAC design is illustrated in Fig. 10. The two dome assemblies of outer and inner annulus were set in the same plane. The fuel preparation module and the cooling structure of the combustor liners were the same as those of the ASDAC.

II Test objective and apparatus. The objective of the rig test was to screen the two candidates, ASDAC and SPDAC to define the configuration of the demonstration sector combustor. The test was conducted at the two most significant conditions simulating typical engine operation, Idle and Mach 3 cruise. Combustion efficiency and weak extinction are important issues at Idle because air temperature is very low at this condition, while NO_x emission and heat damage are primary concerns at Mach 3 cruise because air temperatures are high.

Figure 11 shows a schematic of the screening rig test apparatus. It consisted of an air inlet mouth section, a twin passage dump diffuser system, a low NO_x staged combustor system and an exit instrumentation section. The test apparatus was assembled in a pressure vessel. The exit instrumentation section has three ports for mounting temperature rakes and another three ports for mounting gas sample rakes in-line with and between the fuel preparation modules. The sample gas was dried and sent to a HORIBA gas analyzing system, which measured concentrations of CO , CO_2 , O_2 , NO_x , and UHC. Each temperature rake involved seven R-type thermocouples. Each gas sample rake involved four sampling tubes, and the rake was cooled by water to quench chemical reactions of exhaust gas sampled as soon as gas entered the rake. The combustor liners attached several thermocouples on the surface both the outer and inner liners for monitoring liner metal temperatures.

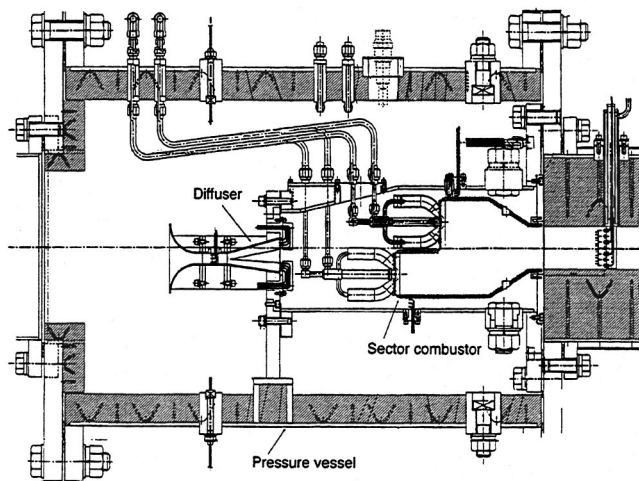


Fig. 11 Schematic of the screening rig test apparatus

Fuel was injected from the two pilot injectors of inner annular array and diffusion flame was formed in the pilot stage, at idle condition, while all the main fuel injectors were used both outer and inner annular array and lean premixed flame was formed in the main and pilot stage at Mach 3 cruise condition.

III Test results and discussion. As expected from combustor configuration investigations, combustion efficiency of the ASDAC was greatly superior to that of the SPDAC at Idle condition as shown in Fig. 12. ASDAC achieved a combustion efficiency of more than 99 percent at a fuel-air ratio of more than 0.015, and weak extinction occurred at a FAR of about 0.0133. Combustion condition of SPDAC, however, was poor and blow out was occurred at small FAR of 0.0182. Figure 13 shows the exhaust gastemperature distributions of the SPDAC measured at three circumferential positions. The temperatures were Hub biased and the distribution curves differed slightly and decreased toward the tip. SPDAC had no center body and this configuration was thought to cause poor combustion conditions, because a large quantity of cool main stage air flowed into the pilot stage flames, quenching the chemical reaction at the boundary between the pilot stage and main stage.

Figure 14 shows the NO_x emission index obtained at the high-temperature and high-pressure conditions of Mach 3 cruise for

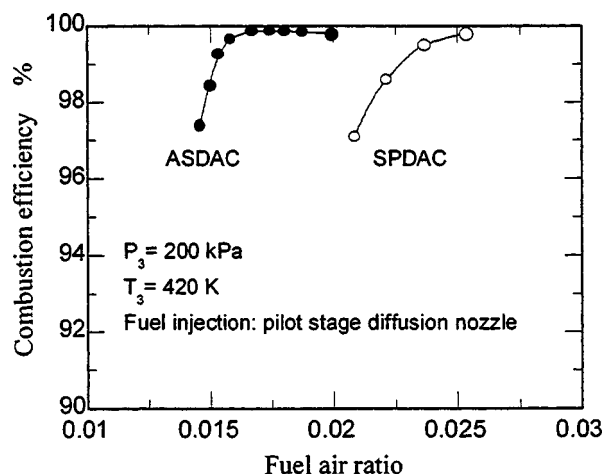


Fig. 12 Combustion efficiency of axial staged double annular combustor (ASDAC) and same plane double annular combustor (SPDAC) at idle

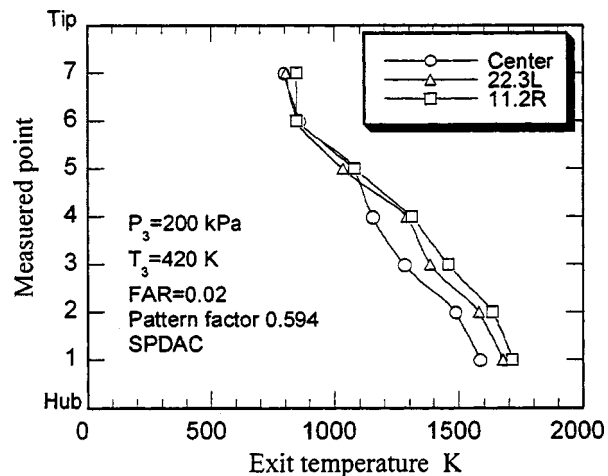


Fig. 13 Combustor exit temperature profiles of same plane double annular combustor (SPDAC) at idle

both combustors, but the inlet pressure for SPDAC was 700 kPa, just below the level of Mach 3 cruise. NO_x emission index for ASDAC and SPDAC were almost the same and about 2 g/kg fuel was obtained. Moreover, combustion efficiencies for the two were superior and showed almost 100 percent at this condition. This emission level was just the same as that of the HTHP combustor obtained in the Phase two program. This proved that the design method on the lean premix combustor based on the PJSC concept was established and met the requirement on NO_x emission sufficiently. Another concern was heat damage to the fuel preparation modules, combustor domes, and liners, but none occurred.

The screening rig test results identified ASDAC as the better configuration for the demonstration sector combustor, and encouraged us to hope to fully demonstrate the target performance.

Demonstration Rig Test Study

1 The demonstration sector combustor and test conditions.

The demonstration sector combustor (DSC) was also a 60 deg sector rig, and was designed by minor modifications to the airflow distribution between the pilot stage and main stage in the ASDAC. ASDAC showed better combustion than SPDAC at Idle, but not good enough to perform the wide range of stable burning and to meet the ICAO regulatory level targeted in this research.

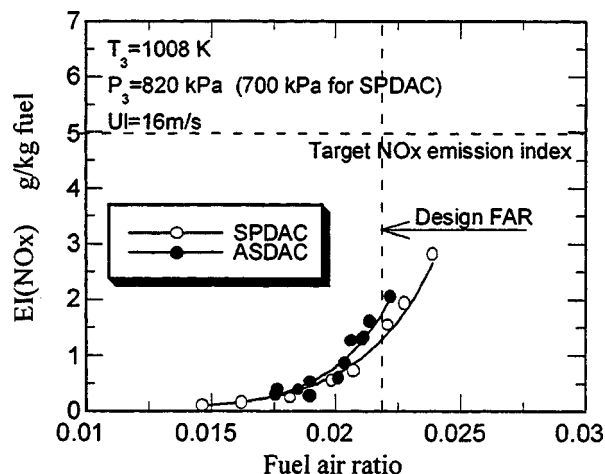


Fig. 14 NO_x emission index of axial staged double annular combustor (ASDAC) and same plane double annular combustor (SPDAC) at Mach 3 cruise

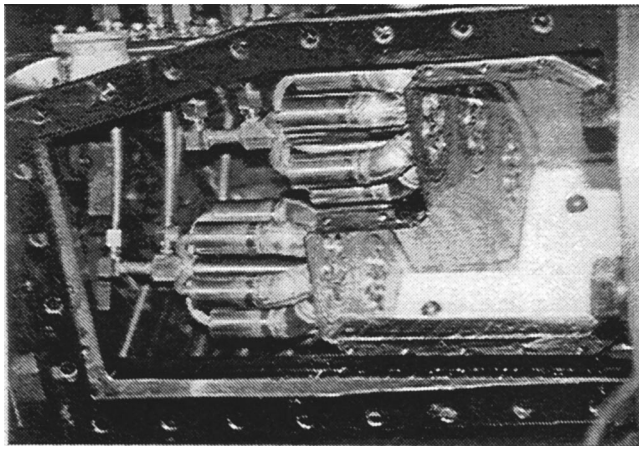


Fig. 15 Photograph of the demonstration sector combustor

The diameter of the premixing tubes was decreased to the inner fuel preparation modules, then airflow into the pilot stage was regulated to make the local fuel-air ratio richer. Figure 15 shows a side view of DSC.

The demonstration rig test conditions are listed in Table 2. Combustor operating conditions were the same as those of the Mach 3 turbojet engine except for the pressures at take off and subsonic climb conditions. The inlet pressures were almost half those of the actual take-off and subsonic climb conditions due to the limitation of the facility. The inlet temperatures, however, were able to reach the actual conditions with the high-temperature electrical heater.

II Staging mode description. Figure 16 illustrates the relation between actual fuel air ratios and combustion staging modes referenced to a typical engine operation. Staging mode 1 illustrates the pilot stage operation with conventional diffusion flame. This mode operates at low-power conditions, such as idle, descent, and approach. The primary stage operation with conventional diffusion flame achieves favorable weak extinction and maintains high combustion efficiency even at low fuel air ratios. Staging mode 2 illustrates dual stage operation, the pilot and main stage modes, with conventional diffusion flame. This mode operates at take-off and subsonic climb. Combustor inlet temperatures are relatively low even at take off and at subsonic climb in supersonic engines. Premix combustion has problems with flame stabilization and combustion instability at low inlet temperature conditions. Staging mode 3 illustrates dual-stage operation with the state of the art lean premixed flame. This mode operates at supersonic climb and cruise flight condition. Combustor inlet temperatures become extremely high in supersonic flight, in particular reaching 1008 K at Mach 3 cruise. This high temperature makes a lean premix flame stable, and the lean premix flame reduces NO_x emission greatly. HYPR's low NO_x combustor was thought to operate safely from idle to Mach 3 full climb and meet the required performance by changing the combustion mode at each of the above points in the flight envelope.

Table 2 Demonstration test conditions

reference engine operation	Inlet temperature K	Inlet pressure kPa	fuel air ratio
Idle	420	230	0.0125~0.0167
Descent	440	330	0.0125~0.0167
Approach	540	500	0.0125~0.0167
Climb/	630	640	0.0167~0.0286
Take off			
Mach 3 cruise	1008	820	0.018~0.025

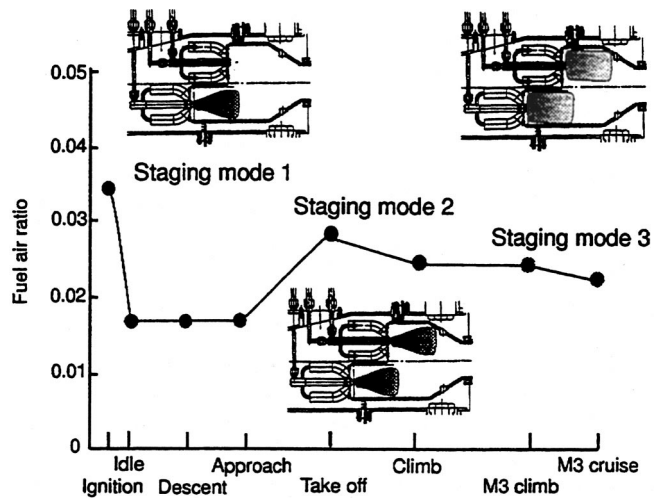


Fig. 16 Engine operations and combustion staging modes

Demonstration Rig Test Results and Discussion

I Emissions at Mach 3 cruise. Figure 17 shows the relation of fuel air ratio and emissions at $T_3 = 1008$ K and $P_3 = 820$ kPa obtained by the demonstration sector combustor. UHC was not emitted and CO levels were also extremely low for the wide range of FAR tested. NO_x emission levels increased rapidly with in fuel air ratio. However, the NO_x emission levels were very low owing to the success of lean premix combustion, though the inlet air and exit gas temperatures were both extremely high. NO_x emission of about 2.1 EI was obtained at the design fuel air ratio of 0.022, and this was almost equal to the level of HTHP combustor.

The result proved that DSC meets the NO_x emission goal.

II Combustor performance at Mach 3 cruise. Combustion efficiency was excellent for the wide FAR range tested, and nearly 100 percent was achieved at the design fuel air ratio of Mach 3 cruise. Total pressure loss of the combustor was 4.8 percent, i.e. 1.2 percent for the diffuser system, 2.8 percent for the wall loss of the combustion chamber and 0.8 percent for momentum loss, at Mach 3 cruise condition. Figure 18 presents the gas temperature distributions at combustor exit plane measured at the design fuel air ratio. Radial profiles were very uniform when operating both pilot and main stages in lean premixed mode. The circumferential

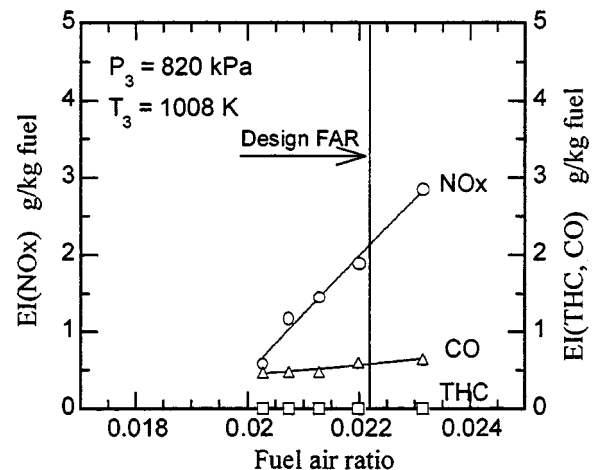


Fig. 17 Emission index of demonstration sector combustor (DSC) at Mach 3 cruise

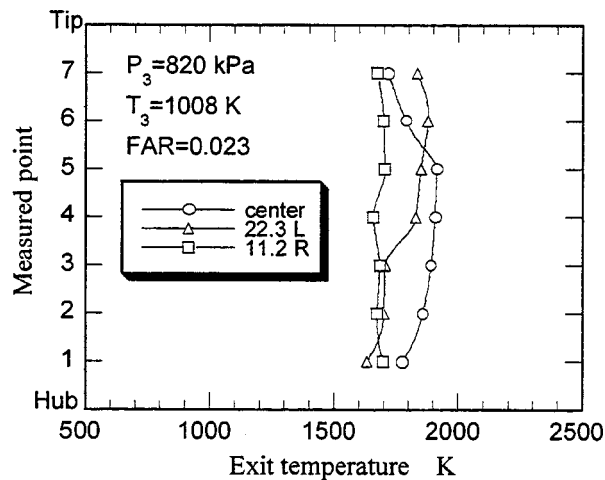


Fig. 18 Exit gas temperature profiles of demonstration sector combustor (DSC) at Mach 3 cruise

profiles were also uniform, so that an excellent pattern factor was obtained. PF was 0.15 at the design point and met the requirement.

Figure 19 shows the liner temperature distributions at the design point of Mach 3 cruise. Axial distributions were relatively flat and the maximum temperature was about 1200 K measured at exit part of both liners. This temperature level was just a little higher than conventional commercial combustor liners, but was quite acceptable for the developmental combustor. Neither deformation nor heat damage to the combustor were observed in an inspection after the demonstration rig test. Combustion instability and oscillation were also not experienced during the demonstration rig test.

Therefore, combustor performance goals included mechanical integrity at the design point of Mach 3 cruise were all achieved.

III Emissions at LTO conditions. The emissions exhausted from civil aircraft engines are regulated by many nations based on the standards prescribed by ICAO. The standards are designed to regulate the emission quantities generated and exhausted from the aircraft in and around airports during its landing and take off (LTO) cycles. Therefore, emissions at LTO are also important, and the goal, which meets the ICAO regulatory levels for a supersonic airplane, was also a high priority in HYPR project.

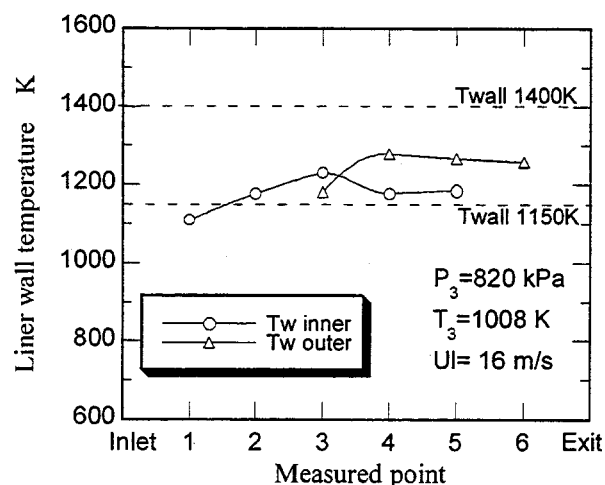


Fig. 19 Linear wall temperature profiles of demonstration sector combustor (DSC) at Mach 3 cruise

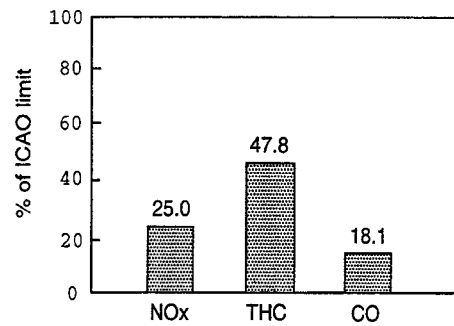


Fig. 20 Emissions at landing and takeoff (LTO) cycle to the ICAO regulatory limitations

Table 3 The demonstration rig test results

Parameters	Engine Operating Mode	Test Results	Project Goal
NO _x emission index g/kg fuel	M3 cruise	2.1	≤5.0
Combustion efficiency %	M3 cruise	100	≥99.5
Total pressure loss %	M3 cruise	4.8	≤5.0
Pattern factor	M3 cruise	0.15	≤0.20
ICAO regulatory levels	LTO	succeeded	Fig. 20

LTO emission levels of DSC to the ICAO regulatory limitations are shown in Fig. 20. Owing to successful operations of staging mode 1 and 2, excellent emissions characteristics were obtained at each LTO condition, such as idle, descent, approach, take-off, and subsonic climb. NO_x emissions at take-off and subsonic climb in the table were corrected by assuming the power of half for the pressure effect. Emissions at LTO conditions also met the requirement fully.

Conclusions

Methane-fueled low NO_x combustor research has been conducted in the HYPR project. The sequential three-phased program was planned and successfully completed and the project goals of pollutant emissions and combustor performance at Mach 3 cruise, with ICAO regulations for the LTO cycle were all achieved by the demonstration rig test in the final research phase.

The demonstration rig test results are summarized in Table 3 with each target.

Acknowledgments

This research was conducted under the entrustment contract with NEDO (New Energy and Industrial Technology Development Organization) as a part of the National Research and Development Program (Industrial Science and Technology Frontier Program) of Agency of Industrial Science and Technology, Ministry of International Trade and Industry.

Nomenclature

- A = area (m²)
- EI = emission index (g/kg fuel)
- FAR = fuel-air ratio
- MFR = momentum-flux ratio
- P = pressure (kPa)
- PF = pattern factor
- PLR = total pressure loss ratio (percent)
- T = total temperature (K)
- UI = combustion liner velocity (m/s) = $W_3 / (\rho_3 A_l)$
- W = airflow (kg/s)
- α = angle to the central axis (deg)

β = angle to the circumference axis (deg)
 η_b = combustion efficiency (percent)

Subscript

3 = combustor inlet
1 = liner

References

- [1] Shaw, R. J., Gilkey, S., and Hines, R., 1993, "Engine Technology Challenges for a 21st Century High-Speed Civil Transport," ISABE, **93**, pp. 673-683.
- [2] Kinoshita, Y., Kitajima, J., and Oda, T., 1997, "Low NO_x Combustor Research for a Mach 3 Turbojet—Concept Validation Test Results," ASME Paper 97-GT-153.
- [3] Rizk, N. K., and Mongia, H. C., 1990, "Ultra-Low NO_x Rich-Lean Combustion," ASME paper 90-GT-87.
- [4] Nguyen, H., Bittker, D., and Niedzwiecki, R., 1989, "Investigation of Low NO_x Staged Combustor Concept in High Speed Civil Transport Engine," Paper AIAA 89-2942.
- [5] Koff, B. L., 1993, "Aircraft Gas Turbine Emissions Challenge," ASME Paper 93-GT-422.
- [6] Mularz, E. J., 1979, "Lean, Premixed, Prevaporized Combustion for Aircraft Gas Turbine Engine," Paper AIAA 79-1318.
- [7] United Technologies Report, 1992, "JFY1991 Hypersonic Transport Ramjet Propulsion System Study Technical Results Report," United Technologies.

A. Mansour
M. Benjamin

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

D. L. Straub
G. A. Richards

U.S. Department of Energy,
National Energy Technology Lab,
Morgantown, WV 26505

Application of Macrolamination Technology to Lean, Premixed Combustion

Macrolamination, a novel manufacturing technique, is used to develop a dual-fuel premixer. A spatially distributed injection strategy is used to enhance fuel placement, distribution, and mixing inside the premixer. Parametric studies are conducted with different configurations of the premixer to determine the effects of residence time and nozzle configuration on pollutant emissions and flame stability. Diesel fuel (DF-2) and natural gas are used as fuels. Tests are conducted at a pressure of 400 kPa (5 atmospheres), and an inlet air temperature of 533 K. The pollutant emissions and RMS pressure levels are presented for a relatively wide range of nozzle velocities (50–80 m/s) and equivalence ratios (0.54–0.75). These results indicate very good pollutant emissions for a prototype design. These results also indicate that the time-lag model, previously associated with combustion oscillations for gaseous-fuel applications, also applies to liquid-fuel operation. [DOI: 10.1115/1.1378296]

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 combustors for both liquid and gaseous-fuel applications must meet a number of design constraints, including low pressure drop, minimum size, and resistance to flashback. 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.

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 auto-ignition, 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 Parker Hannifin. Experimental test data are presented that demonstrate the preliminary performance of this premixer in terms of both the measured pollutant emission levels and the RMS pressure levels.

Background

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. Using this macrolamination technology (U.S. Patent 5435884), multipoint 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 Hannifin Corporation has been tested at the National Energy Technology Laboratory as part of a Co-operative Research and Development Agreement (CRADA).

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 thermoacoustic 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.

One of the variables investigated in this study is the fuel-transport time, or residence time, in the premixer. 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 ([7–9]) show the value 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 [10] 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,

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-115. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

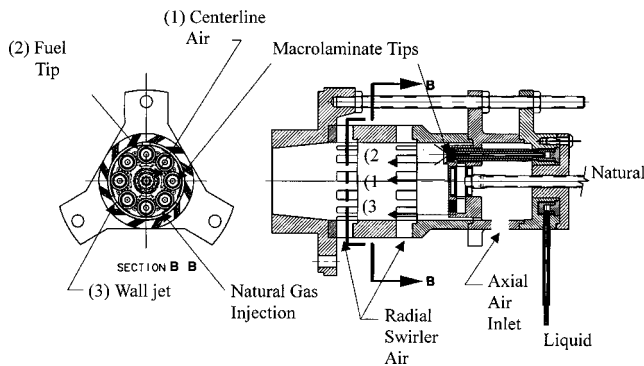


Fig. 1 Cross section of Parker-Hannifin premixer

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 τ^*f . This approach of correlating combustion oscillations for a given combustor is widely accepted for natural gas applications. However, for liquid-fuel applications, it is not clear whether this simple time-lag model applies for lean, premixed, prevaporized combustors.

In this paper, the prototype dual-fuel premixer is described, as well as the experimental combustor. In the results section, the qualitative effects of residence time, nozzle configuration, and equivalence ratio on the performance of the premixer are discussed. The effects of these variables on NO_x , and CO emissions are reported. The RMS pressure levels are also reported at each operating condition. In addition to conducting an evaluation of this prototype design, the validity of using a simple time-lag model to correlate combustion instabilities for liquid-fuel combustion is described.

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 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 fuel droplets on the premixer wall. The initial swirl strength, or swirl number, is maintained relatively low to minimize droplet impingement on the walls. The second swirler adds sufficient swirl to stabilize the flame. The droplet sizes reaching the second

swirler are significantly smaller because of vaporization. Therefore, staging the swirl-vane assemblies in this manner is expected to minimize fuel-droplet impingement on the premixer wall, while maintaining sufficient swirl to stabilize the flame.

Experimental Combustor. Many of the details of this combustor have been described previously ([7,10,11]). Therefore, only the changes and pertinent details are described here. A cross section of the experimental combustor is shown in Fig. 2. This combustor is designed specifically to study combustion dynamics, so the hard acoustic boundaries and lack of cooling air passages result in very strong acoustic feedback. It is important to note that this apparatus provides a rigorous test environment for premixers, and the observed RMS pressure levels may not be representative of premixer performance in a realistic gas turbine application.

A cylindrical plug forms the exhaust neck region (see Fig. 2). This plug is fabricated using refractory for the purposes of these tests. Only the material is different from the design described in Straub and Richards [11]. The dimensions and location of this plug are the same as that described in earlier work. An important difference between the results reported in this paper and those reported for previous work, is the location of the water spray that quenches the exhaust gases. In previous work, this water spray has been located immediately downstream of the exhaust neck. To minimize the potential for quenching unburned hydrocarbons or soot, the water quench has been moved approximately 3.8 meters downstream and the vessel length has been increased by 3.6 meters.

Roughly 10 cm downstream of the refractory insert, gas samples are extracted through a stainless steel water-cooled probe. The gas sample is extracted from three locations across the diameter of the exhaust neck region (see Fig. 2). Depending on the flow rate and the gas temperature, the bulk residence time from the inlet of the exhaust neck region to the stainless steel sample probe is 10–20 ms. The temperature of the gases exiting the sample probe is about 420 K (300°F). The gas sample then passes through a heated stainless steel line before being cooled in an ice bath 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 will be reported. However, it should be noted that the measured levels of unburned hydrocarbons are negligible for the range of operating conditions investigated.

A natural gas pilot is introduced into the corner region of the combustor through the upstream wall of the combustor liner (see Fig. 2). Pilot fuel is used for startup purposes only. None of the cases reported in this paper utilize pilot fuel to assist the main flame. The intent for operating in this mode is to eliminate the influence of the pilot fuel on the combustion oscillations.

The combustor pressure is maintained with a back-pressure 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 temperature is controlled using a nonvitiated air preheater, and is also independent of the combustor operating pressure. Therefore, the inlet-air temperature, air-flow rate, and combustor operating pressure can be selected arbitrarily, if necessary. The combustor operating pressure is set at 400 kPa (5 atm.), and allows the rig to operate over a wide range of air and fuel flows, while maintaining a low fuel consumption rate.

The Type 2 diesel fuel used in these tests is a research grade 0.05 percent sulfur fuel. This fuel is chosen to minimize the variability in fuel properties and any subsequent effects on the data. The properties of this fuel are listed in Table 1. The nitrogen content in the fuel is 80 ppm, 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 5 ppm at 15

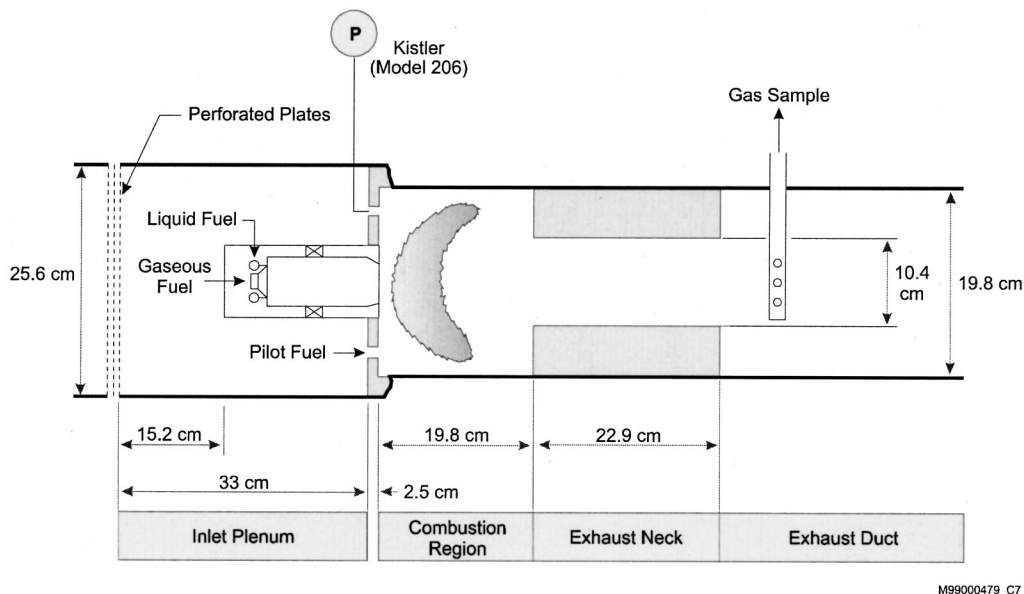


Fig. 2 Schematic of experimental NETL combustor

percent oxygen. It should be noted that this is approximately twenty percent of the measured NO_x levels for the best nozzle configuration tested.

Instrumentation. The oscillating combustor pressure is monitored using a fast response Kistler (Model 206) pressure transducer. The pressure transducer is installed in 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 Horiba (Model CLA-510SS) chemiluminescent analyzer. CO measurements are taken from a nondispersive infrared analyzer manufactured by Advanced Pollution Instrumentation Inc. (Model 300). All emissions are reported on a dry basis and have been corrected to 15 percent 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. Table 2 shows the nozzle configurations that are discussed in this paper. The overall length of the premixer is constant for the tests reported. The effect of the swirl-vane size and the effect of the downstream swirl-vane are investigated. The swirl-vane thickness (i.e., the number listed in Table 2) refers to the width of the swirl-vane at the outside diameter of the premixer. All of the swirl-vane assemblies have the same outside diameter, and the larger vane sizes correspond to smaller flow areas. Therefore, Case C has the largest flow area of the swirl-vanes tested.

Table 1 Diesel fuel properties

Carbon (wt%)	86.6
Hydrogen (wt%)	13.4%
Nitrogen	80 ppmw
Specific gravity (60/60)	.843
API gravity	36.28
50% boiling point	533°K

Both the operating pressure and the inlet-air temperature are fixed at 400 kPa (5 atm.) and 533 K, respectively. Although the test rig is capable of higher pressures and inlet-air temperatures, these operating conditions are selected to allow testing over a wide range of reference velocities and to conserve fuel.

Besides the geometric premixer configuration, the air flow and the equivalence ratio are also varied. Clearly, increasing the air flow decreases the residence time, but it should also be noted that the nozzle configuration also dramatically 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. This simplified approach is illustrated in Fig. 3. The fuel-transport time is estimated as

$$\tau = \tau_1 + \tau_2 + \tau_3. \quad (1)$$

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 second swirler; and τ_3 is the time required to advect the parcel of fuel-air mixture between the second swirler and the exit of the premixer (see Fig. 3). Therefore, the bulk fuel-transport time can be approximated using the nozzle configuration and approximate air splits as shown in Eq. (2).

Table 2 Premixer configurations tested. Swirler 2 designates the downstream swirl-vane assembly. Swirler 2 is replaced with a solid-spacer for Case B and Case C.

	Premixer length (cm)	Vane thickness swirler 1 (cm)	Vane thickness swirler 2 (cm)
Case A	14.7	0.51	0.51
Case B	14.7	0.51	N/A
Case C	14.7	0.38	N/A

Table 3 The effect of nozzle configuration on the 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	0.11	0.13	0.10	0.33	0.33
Case B	0.16	0.20	0.14	0.50	0.00
Case C	0.15	0.17	0.12	0.56	0.00

$$\tau \approx \left(\frac{L_1 - L_2}{y_1 * U} \right) + \left(\frac{L_2 - L_3}{[y_1 + y_2] * U} \right) + \left(\frac{L_3}{U} \right) \quad (2)$$

The reader should refer to Fig. 3 for the definition of the various parameters and Table 2 for the approximate air splits for each of the nozzle configurations tested.

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 air flows 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 4 shows the range of transport times obtained using the above equation for all three cases discussed in this paper as a function of average exit velocities. Note that Case A has the longest transport time, and Case B has the shortest transport time.

Results

The overall NO_x and CO emission data for DF-2 are shown in Fig. 5. It should be noted that each point shown in Fig. 5 represents a specific equivalence ratio and air-flow rate that is replicated for each premixer configuration, so an equal number of points are shown for each nozzle configuration. Although all of the configurations produced CO emissions below 15 ppm, Case B exhibited the lowest NO_x levels. Furthermore, Case B produced lower levels of NO_x over a wider range of operating conditions. Therefore, the configuration in Case B produces the best pollutant emission performance for the range of operating conditions investigated.

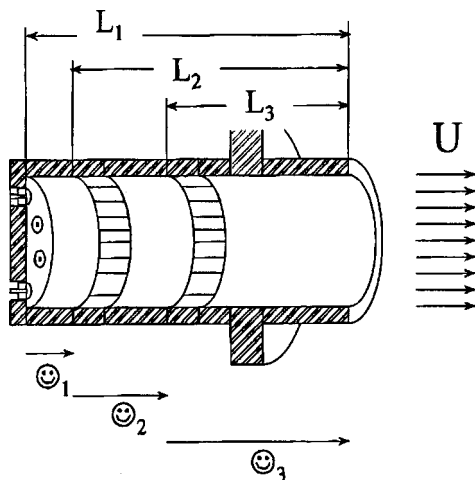


Fig. 3 Approach to calculate fuel residence time

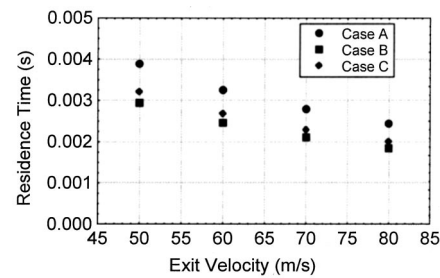


Fig. 4 Range of fuel transport times

Although Fig. 5 shows both the NO_x and CO levels, additional information is obtained by looking at the NO_x emissions as a function of the adiabatic flame temperature (see Fig. 6). The adiabatic flame temperature is calculated by the procedure outlined by Gulder [13]. These calculated temperatures agree to within one percent of values from chemical equilibrium codes. The data in Fig. 6 show that the NO_x levels vary significantly depending on the operating condition.

Consider the data from Case A on liquid fuel at a nozzle exit velocity of 80 m/s. As the equivalence ratio, or adiabatic flame temperature, is decreased, the NO_x emissions decrease until the adiabatic flame temperature approaches 1950 K. When the adiabatic flame temperature is decreased below 1950 K, the NO_x emissions actually increase. This behavior is repeatable to within about 3 ppm on the NO_x measurements. However, hysteresis is observed, so the actual flame temperature at which this transition occurs depends on whether this transition is approached from a lower flame temperature or a higher flame temperature. The data shown in this paper are collected as the flame temperature is decreased. As Fig. 6 shows, this behavior is not observed for all the premixer configurations.

The RMS pressure levels show the same "on-off" transition. Figure 7 shows both the NO_x emissions (thin line) and the RMS pressure levels (thick line) as a function of equivalence ratio for Case A (DF-2 fuel and a velocity of 80 m/s). As the NO_x emissions increase, the RMS pressure levels decrease sharply for values of equivalence ratio below 0.62. It is believed this behavior is caused by a change in the flame location. When the RMS pressure levels are low, the flame zone is believed to be located closer to the premixer tip. The change in flame location is confirmed by a thermocouple added to the wall of the premixer. This thermocouple registers temperatures above 533°K when the RMS pressure levels are low. When the RMS pressure levels are high (greater than 0.5 percent) and the NO_x levels are relatively low, the thermocouple reading is considerably lower. It is believed that the change in flame location produces a diffusion-type flame that results in higher NO_x emissions and lower RMS pressure levels.

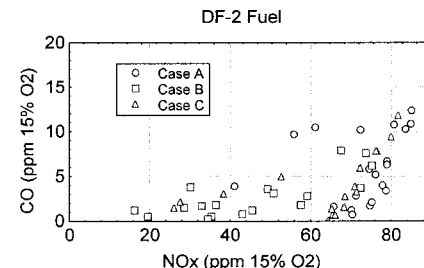


Fig. 5 Pollutant emissions for all nozzle configurations and all operating conditions ($50 < U < 80$ m/s, $0.55 < \phi < 0.75$, 5 atm, 533 K)

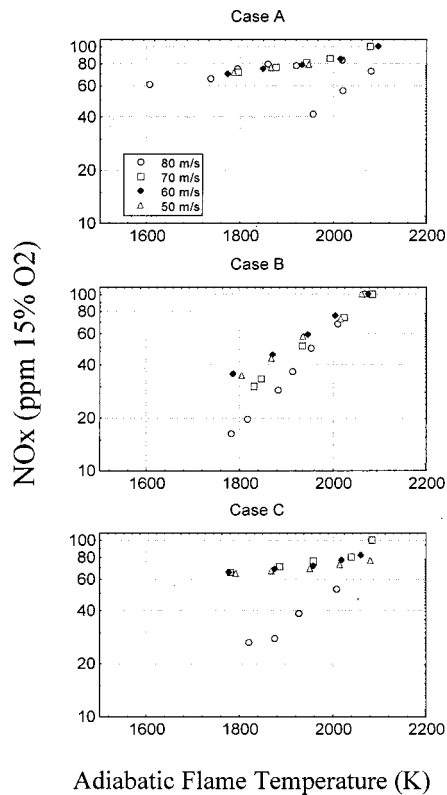


Fig. 6 NO_x emissions as a function of nozzle configuration and adiabatic flame temperature (DF-2 fuel). No pilot fuel is used, so leaner conditions are not investigated.

When the RMS pressure levels are high, the flame is likely burning in a premixed mode and the NO_x emission levels are reasonably lower.

Figure 8 shows the RMS pressure levels for all three cases studied (both natural gas and DF-2). Note that the RMS pressure levels for the liquid-fuel cases are extremely low (less than 0.5 percent) for many of the operating conditions investigated. At these low values of the RMS pressure, it is believed that the flame anchoring location has changed. Although the level of the combustion oscillations for Case B is higher, the measured NO_x concentrations are lower.

Figure 9 shows the RMS pressure levels for all of the operating conditions investigated. These RMS pressure levels are plotted against the product of the fuel time-lag, τ , and the dominant frequency, f in the pressure signal. The fuel time-lag is approximated

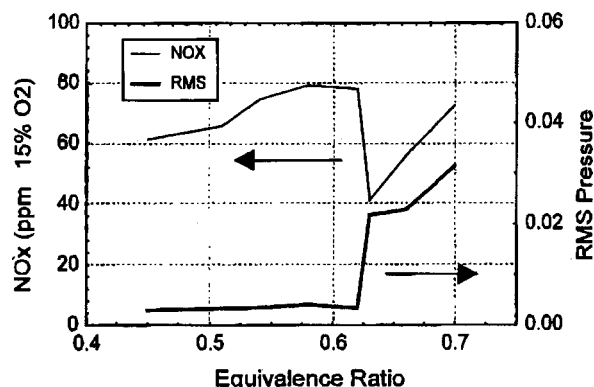


Fig. 7 NO_x /RMS pressure level (percent of ambient) relationship (Case A, 80 m/s, DF-2 fuel)

by Eq. (2). However, to account for the additional time lag between the tip of the premixer and the location of the flame zone, a constant flame-standoff distance of 2.54 cm is assumed for all operating conditions. Therefore, the time-lag values presented in Fig. 9 represent the sum of the fuel residence time in the nozzle (Eq. 2) and an approximate time lag between the nozzle exit and the flame front. This approach is identical to the approach used in previous work. Figure 9 shows that despite a factor-of-two change in the fuel-transport time, these results are correlated by a simple time-lag approach. In fact, the range of time-lag values over which large RMS pressure levels are observed is consistent with previous results from different premixers tested in this rig (Straub and Richards, [8]).

The liquid-fuel results also obey a time-lag model, as shown in Fig. 9. However, the bands appear to be considerably narrower than for natural gas, and a second band of data is observed around a τ^*f product of 0.8. These results differ from the natural gas operating experience that shows the region between τ^*f values of 0.7 to 1.1 are generally dynamically stable. Although these variations are not well understood, these differences may be explained by the inherent assumptions used in calculating the fuel-transport time. Perhaps small changes in the flame location, or shape, for the natural gas results can alter the actual fuel time-lag enough to cause the combustor to oscillate in a different τ^*f band. When calculating the fuel-transport time, assumptions are made about the flame location and shape. Perhaps a better measurement of fuel-transport time is warranted. In fact, the flame location and shape may play a much more critical role in determining the dynamic stability of the combustor.

To further investigate whether the DF-2 fuel data conforms to the behavior of a simple time-lag model, the frequency of the combustor oscillation is plotted as a function of the reference velocity (see Fig. 10). As explained in Straub and Richards [5] the oscillating frequency will cover a range of values near the natural frequencies of the combustor. According to the time-lag description, if the velocity decreases (fuel time-lag increases), the frequency of the oscillation will decrease to maintain a constant τ^*f product. If the frequency is sufficiently far from the natural frequency or the acoustic losses exceed the magnitude of the combustion driving term, the combustion will be stable. Figure 10 shows that the frequency decreases with velocity, as predicted by a simple time-lag model. Figure 10 shows how the air flow, or velocity, through the nozzle can be altered to control the τ^*f product.

Consider the data shown in Fig. 10. At 80 m/s, both operating conditions produced combustion oscillations (see Fig. 8, Case B, DF-2 Fuel), and the observed values of the τ^*f product are within the unstable range (0.37–0.43) shown in Fig. 9 for DF-2 fuel. By reducing the nozzle reference velocity from 80 m/s increases and the time-lag model predicts that the frequency must decrease to maintain a constant τ^*f product. At 70 m/s, Fig. 10 shows that the lean operating point is near the edge of the unstable range for the τ^*f product, and the frequency for the richer operating point has shifted to a lower value which puts it in the stable range. Indeed, the leaner condition is unstable and the richer condition is stable, as shown in Fig. 8. As the velocity is reduced further, the frequencies for both operating conditions transition into the 110–125 Hz frequency band. The acoustic feedback from this lower frequency range is not as strong as the 160–180 Hz band, so even though the time-lag model predicts an unstable phase relation at 50 m/s, the gain at this frequency is not large enough to make the system unstable.

Conclusions

A prototype lean premixed pre-vaporized fuel nozzle has been designed and successfully tested to demonstrate performance at

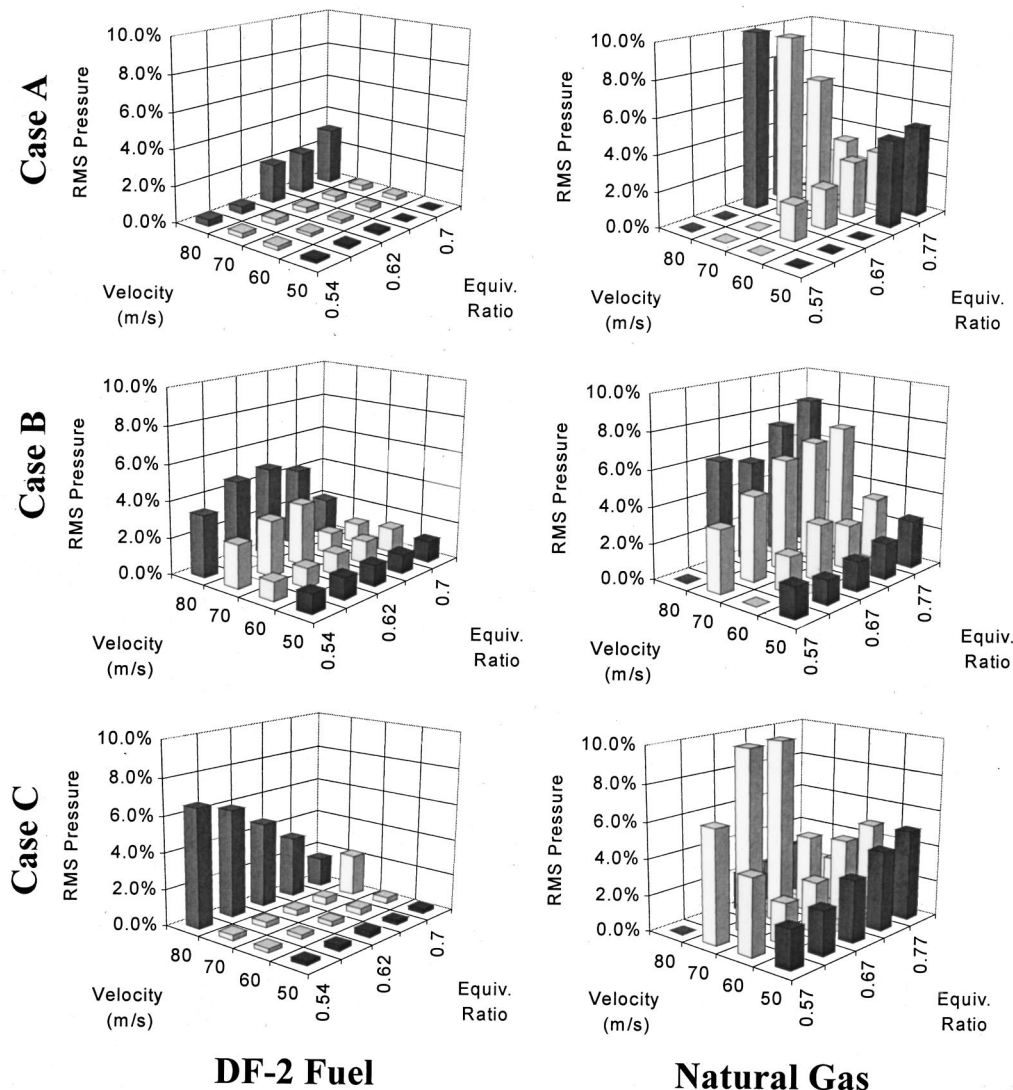


Fig. 8 Stability maps for each nozzle configuration and fuel type

533°K inlet-air temperature and 400 kPa (5 atmospheres). After testing a number of nozzle configurations, NO_x and CO emissions of less than 20 ppm are observed for both DF-2 and natural gas. 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.

The observed RMS pressure levels and NO_x emissions are very sensitive to changes in flame location. It has been observed that

when the RMS pressure levels are significant (i.e., greater than 0.5 percent), the observed NO_x levels are relatively low. However, sudden changes in the flame location, as indicated by a thermocouple installed along the wall of the premixer, can lead to significantly higher NO_x emissions and lower RMS pressure levels. The transition in the flame location is sensitive to the nozzle configuration and the operating conditions.

A time-lag model can be used to correlate and guide approaches

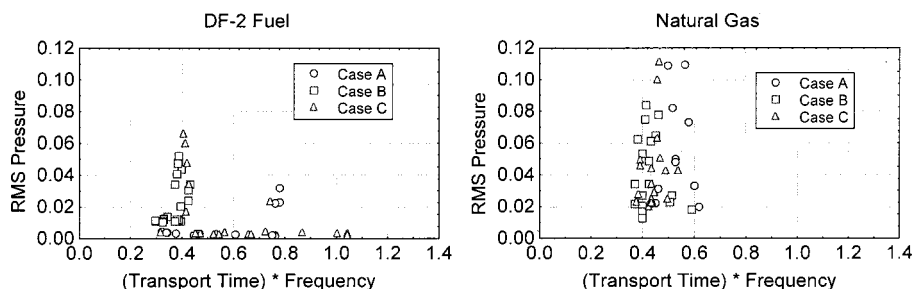


Fig. 9 Time lag model description of the experimental results. Both fuel types can be described by a simple time lag.

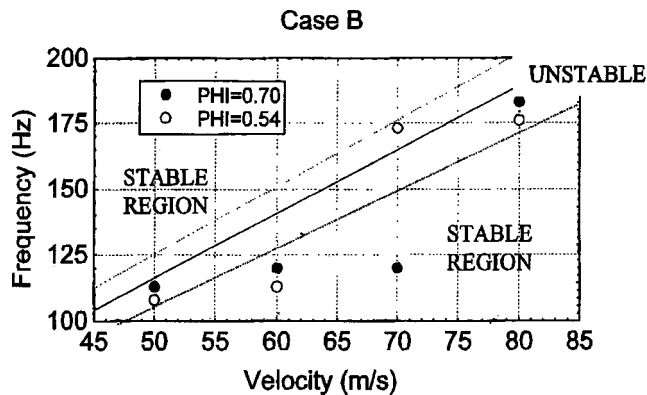


Fig. 10 Dominant frequencies observed as a function of velocity for two equivalence ratio conditions (Case B on DF-2 fuel). Constant values of (transport time)*(frequency) are also shown.

for solving combustion oscillations in liquid, and dual-fuel, applications. An approach to estimate the fuel-transport time in a pre-mixer with multiple air-injection locations has been described. In spite of a factor-of-two variation in the fuel-transport time, the observed combustion oscillations for lean premixed prevaporized liquid-fuel applications can be correlated using a time-lag description, similar to approaches that have been used for gaseous-fuel applications.

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 Symp. on Combustion, 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 96-GT-195.
- [4] Smith, K. O., and Cowell, L. H., 1989, "Experimental Evaluation of Liquid-Fueled, Lean-Premixed Gas Turbine Combustor," ASME Paper 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 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**, No. 2, pp. 294-302.
- [8] Straub, D. L., and Richards, G. A., 1998, "Effect of Fuel Nozzle Configuration on Premix Combustion Dynamics," ASME Paper 98-GT-492.
- [9] Lieuwen, T., and Zinn, B. T., 1998, "Theoretical Investigation of Combustion Instability mechanisms in Lean Premixed Gas Turbines," AIAA Paper 98-0641.
- [10] Putnam, A. A., 1971, *Combustion Driven Oscillations in Industry*, Elsevier, New York.
- [11] Straub, D. L., and Richards, G. A., 1999, "Effect of Axial Swirl-Vane Location On Combustion Dynamics," ASME Paper 99-GT-109.
- [12] Mahan, J. R., and Karchmer, A., 1991, "Combustion Core Noise," *Aeroacoustics of Flight Vehicles: Theory and Practice, Vol. 1: Noise Sources*, NASA Langley Research Center, available from AIAA Technical Library.
- [13] Gulder, O. L., 1986, "Flame Temperature Estimation of Conventional and Future Jet Fuels," ASME J. Eng. Gas Turbines Power, **108**, pp. 376-380.

Integrated Experimental and Numerical Approach for Fuel-Air Mixing Prediction in a Heavy-Duty Gas Turbine LP Burner

G. Mori

S. Razore

Ansaldo Ricerche s.r.l.,
Corso Perrone 25,
Genova I-16152, Italy

M. Ubaldi

P. Zunino

Università di Genova,
DIMSET, Via Montallegro 1
Genova I-16145, Italy

An integrated experimental-numerical procedure has been developed for fuel-air mixing prediction in a heavy-duty gas turbine burner. Optical measurements of the degree of mixing have been performed in a full-scale test rig operating with cold flow. Experimental data have been utilized to validate a CFD RANS numerical model. In fact, it is recognized that the turbulence behavior of jets in swirling air-flow stream is not accurately described by standard $k-\epsilon$ turbulence models; therefore advanced turbulence models have been assessed by means of experimental data. The degree of mixing between simulated fuel and air streams has been evaluated at the burner exit section by means of a planar Mie scattering technique. The experimental apparatus consists of a pulsed Nd:YAG laser and a high resolution CCD video camera connected to a frame grabber. The acquired instantaneous images have been processed through specific procedures that also take into account the laser beam spatial nonuniformity. A second-order discretization scheme with a RSM turbulence model gives the best accordance with the experimental data. Such CFD model will be part of a more general method addressed to numerical prediction of turbulent combustion flames in LP technology. [DOI: 10.1115/1.1378297]

Introduction

The achievement of ultra-low NO_x levels has become one of the most important factors of the engine competitiveness because of the more and more stringent legislation concerning environmental issues. It is well known that the degree of premixing in an LP burner is one of the most important parameter concerning NO_x emission, in particular in the pressurized lean combustion regime typical of gas turbine ([1]). Furthermore the degree of mixing has some considerable influences both on flame stabilization and combustion instabilities.

Ansaldo Ricerche has been working in gas turbine combustion projects through integration between experimental data and three-dimensional CFD codes. Both burner aerodynamics ([2]) and liquid fuel jet injection ([3]) have been assessed in previous works. Numerical activities have been validated with LDV cold flow measurements carried out in Ansaldo Ricerche laboratories. Such works contributed to the development of the enhanced version of the LPP hybrid heavy-duty gas turbine burner carried out by Siemens ([4]).

Lean premixed turbulent combustion has been studied through a reduced chemical mechanism developed by the University of Washington ([5]); at present an enhanced combustion model is under development for partially premixed conditions.

The present work is focused on mixing process between gaseous fuel and oxidant, which is recognized as one of the main aspects of mixture preparation and greatly affects flame characteristics ([6]). Mixing between air and fuel in LP burners is dominated by turbulence; numerical analysis demonstrates that the rate of diffusion driven by turbulent phenomena is almost three order of magnitude greater than that driven by molecular processes. Since simulation of turbulent swirling flows is far from a defini-

tive solution, the problem of mixing between fuel and oxidant has to be resolved before further considerations about flames.

It is well known that jet mixing is a typical application of LES/DNS methods because of the complexity of the turbulent structure; unfortunately these methods presently are not suitable for industrial applications, especially in design and development activities, which require a lot of parametric calculations. Therefore, the good accordance between RANS method results and experiments concerning velocity fields ([2]) and the continuous development of both turbulence models and discretization schemes encourage the attempts to predict mixing phenomena with RANS methods.

The main objective of the present paper is to outline a combined experimental-numerical approach for the assessment of fuel-air preparation in gas turbine LP burners. Experimental evaluation of the instantaneous concentration field allows investigating the fuel-air mixing process, which is strongly influenced by both small and large scale flow fluctuations. Furthermore, the comparison between experimental and numerical results is important for the assessment of turbulence models to be used for mixing process predictions.

Concentration measurements by means of sampling probes and subsequent gas analysis are not suitable because of single-point time-average intrusive characteristics of the procedure. Therefore a nonintrusive full-field planar Mie scattering technique has been developed and used to determine instantaneous distribution of fuel concentration in the exit plane of a LP burner.

The Fluent code has been adopted for the numerical simulations. Since the present study is focused mainly on the aerodynamic phenomena, the algorithmic structure of Fluent is maintained adding, for combustive applications, in house developed routines.

A systematic investigation of both different turbulence models and discretization schemes has been carried out in order to select the models better suited for mixing prediction and to assess their accuracy. To authors' knowledge such type of analysis has not yet been issued in the field of burner LP technology applications.

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-122. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

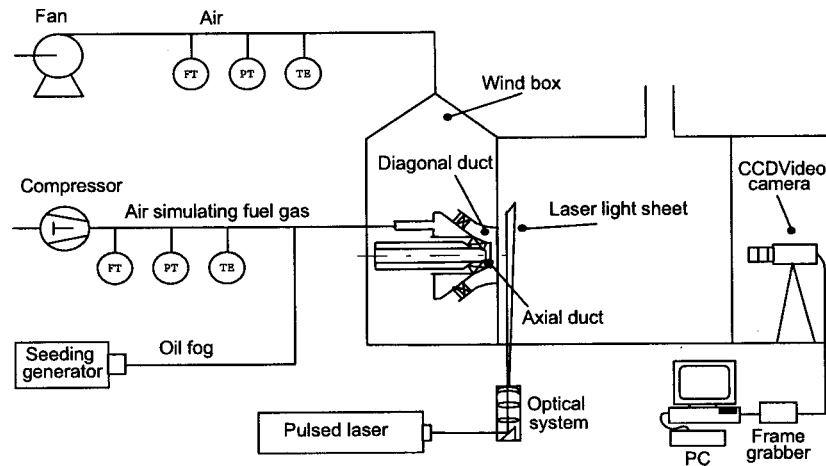


Fig. 1 Test rig scheme

Experimental Method

A lean premixed combustor, HR-type Siemens-Ansaldo hybrid burner for heavy-duty gas turbines has been employed in this study. The mixing phenomenon in the LP burner has been reproduced in absence of combustion, at atmospheric pressure and ambient temperature, simulating the natural gas by seeded air.

The seeding particles provide a method for tracking the flow and evaluating the instantaneous concentration field. A Nd:YAG pulsed laser has been utilized to illuminate the particles. Through an appropriate optical system the laser beam is expanded into a narrow light-sheet located at the burner outlet, normal to the burner axis, that provides a short duration illumination of the whole burner exit section. The laser light scattered by the crossing particles is imaged by a video camera and the instantaneous images of the turbulent mixing flow field are stored in a PC through a frame grabber.

Test Rig Description. The test rig (Fig. 1) consist of

- a feeding system for the air of 1.6 kg/s flow rate and $1 \cdot 10^4$ Pa head;
- a feeding system for the pressurized air simulating the natural gas fuel at $6 \cdot 10^5$ Pa pressure;
- a fog machine to produce small oil particles for seeding the air stream simulating fuel;
- a wind box $1.3 \times 0.7 \times 1.9$ m to house the burner; and
- the real lean premixed burner for heavy-duty gas turbine which is a HR-type Siemens-Ansaldo hybrid burner. The burner consists of two air ducts, one axial and one diagonal, each equipped with a swirler blade row. The flow simulating the fuel is injected inside the diagonal duct through the swirling vanes. A sketch of the burner is depicted in Fig. 2 ([4]).

Similitude Criteria. Due to the different working conditions between real turbine (in pressure and with combustion) and laboratory model (cold flow at the atmospheric pressure and room temperature) the following criteria, based on similitude theory, have been adopted:

- to simulate the air flow rate, the constant $\Delta p/p$ ratio criterion, where Δp is the pressure drop of the burner and p is the working pressure; and
- to simulate the natural gas flow rate, the constant momentum ratio criterion between the natural gas flow rate and the combustion air flow rate.

Diagnostics. The Mie scattering technique has been applied since the 1960s, as a local single point technique. Only in the last

decade, due to the advances in computer, imaging, and laser technology, it has been extended to full-field measurements in industrial scale combustors ([7,8]).

The planar Mie scattering technique is based on the property of particles of diameter of the order of the laser wavelength to scatter efficiently the light. In a flow seeded with particles of uniform size the intensity of the scattered light is proportional to the number of scattering particles contained in each control volume, which dimensions depend on the CCD camera pixel number, the extension of the investigated surface, and the light-sheet thickness. Therefore, considering two flow streams, one of which seeded, it is possible to deduce the mass concentration of the seeded fluid for each control volume from the value of the scattered light intensity.

If a short-duration pulsed laser is employed to generate the light-sheet, the turbulent flow field is frozen and the instantaneous full concentration field can be reconstructed from the flow picture imaged in the CCD camera.

To assure quantitative results, some assumptions need to be done:

- the particles are small enough to follow the air stream;
- the particle concentration is such that it does not alter the properties of the fluid;
- the seeding is monodispersed or, if it is polydispersed, the particle size distribution is uniform in space;
- the light extinction along the laser-sheet path is negligible; and
- the light intensity nonuniformity across the light-sheet is taken into account during data reduction.

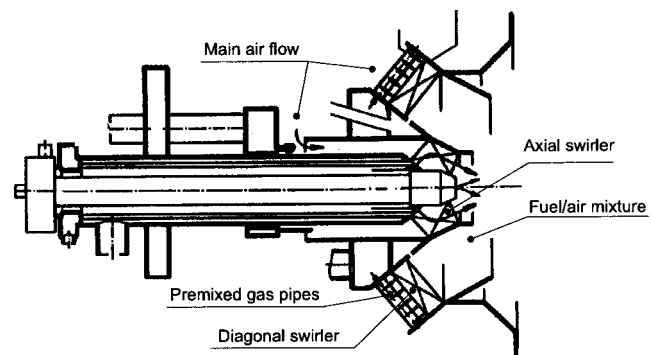


Fig. 2 Sketch of the HR burner (from Paper No. 98-GT-552)

All the above conditions have been carefully verified. The seeding particles generated by a Rosco oil fog generator are in the size range $0.5\text{--}2\text{ }\mu\text{m}$, which is considered adequate for flow fluctuations up to 10 kHz.

Extinction measurements were performed by uniformly seeding the air flow through the diagonal swirler with the same particle concentration adopted for the mixing analysis and imaging it at measuring plane. Negligible light attenuation resulted for the selected concentration of about 1000 particles per mm^3 . This value assured also low marker shot noise and good linearity of the photosensible elements without saturation. Spatial nonuniformity of the light sheet has been investigated and a correction has been applied, as explained in the data processing paragraph.

The technique has been applied to study the mixing process between the two air streams representing the air and the gas fuel at the exit section of the LP burner.

The air stream simulating the gas fuel was seeded and injected into the air stream flowing in the diagonal burner duct through ejection holes located in the diagonal swirling vanes. The oil particles flowing together with the air stream start to mix with the air inside the duct, and the mixing process is evaluated at a plane including the whole burner circular exit section (diameter 119 mm) placed 5 mm downstream of the burner exit. A scheme of the apparatus layout is depicted in Fig. 1.

To illuminate the flow at the burner exit section, a Nd:YAG Q-switched laser (Quanta System SYL200) with 300 mJ per pulse at 532 nm has been utilized. A laser light-sheet 1 mm thick and about 300 mm high was created by means of an optical system consisting of two spherical and one cylindrical lenses. The laser pulse duration was 8 ns and the pulse repetition rate was 12.5 Hz. The particle field was imaged with a high-resolution (1024×1024 pixels) 8-bit B/W CCD digital video camera (Pulnix TM-1001) equipped with a high-quality telephoto zoom. The frame rate of the camera was 15 Hz and the data were transferred to the PC by means of a frame grabber (Imaging Technology).

The cell size of the CCD sensor is $9\text{ }\mu\text{m} \times 9\text{ }\mu\text{m}$. With the adopted magnification factor of 0.07 the measuring volume size is $0.13 \times 0.13 \times 1\text{ mm}^3$. The correspondent smallest detectable flow structure, which is 2 pixels large, is 0.26 mm.

The 8-bit sensor resolution, considering that the maximum measured light intensity was about five times larger than the average intensity (corresponding to the nominal gas fuel concentration), leads to a quantization error for the mass concentration of about two percent of the nominal value.

Data Processing. During the experimental campaign several instantaneous images have been captured. Every image has been carefully checked in order to avoid pixel saturation.

Every selected image has been modified applying a calibration curve suitable to correct the nonuniform energy distribution across the light-sheet. The calibration curve was obtained with two different procedures. In the first one an image of an uniformly seeded flow at the burner exit plane was taken and a curve of the mean light intensity along the vertical axis was deduced. In the second procedure a filtered image of the cross section of the laser-sheet formed on a reflecting surface was taken and its light intensity distribution was deduced. The results of the two procedures were in good agreement.

Every corrected instantaneous image has been then normalized by means of its mean light intensity value which is representative of the completely mixed out flow concentration value. An example of corrected and normalized image is shown in Fig. 3. The figure shows the annulus, which corresponds to the burner diagonal duct exit, with superimposed the instantaneous normalized mass concentration gray scale plot.

Both the long time averaged values (Fig. 4) and the rms of the mass concentration (Fig. 5) have been deduced by ensemble averaging pixel by pixel the light intensity values of about 100 corrected and normalized images.

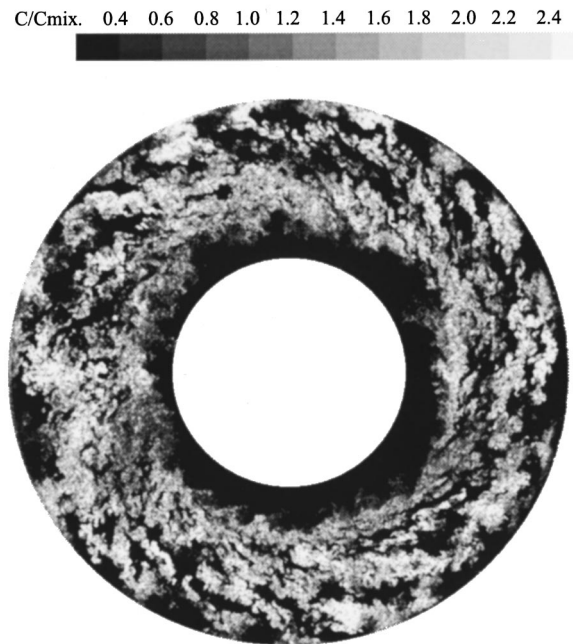


Fig. 3 Instantaneous normalized local fuel concentration

The circumferentially averaged fuel concentration and its root mean square have been obtained by circumferentially averaging the long time-averaged distributions.

Numerical Method

Calculations were performed with a customized version of the commercial code Fluent (version 5.1, 1999). The code is addressed to complex three-dimensional turbulent reactive flow analysis and can manage both structured and unstructured meshes.

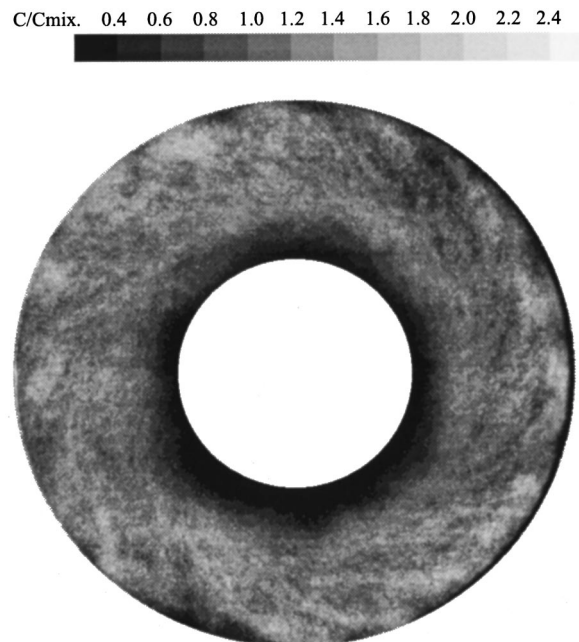


Fig. 4 Ensemble-averaged normalized local fuel concentration

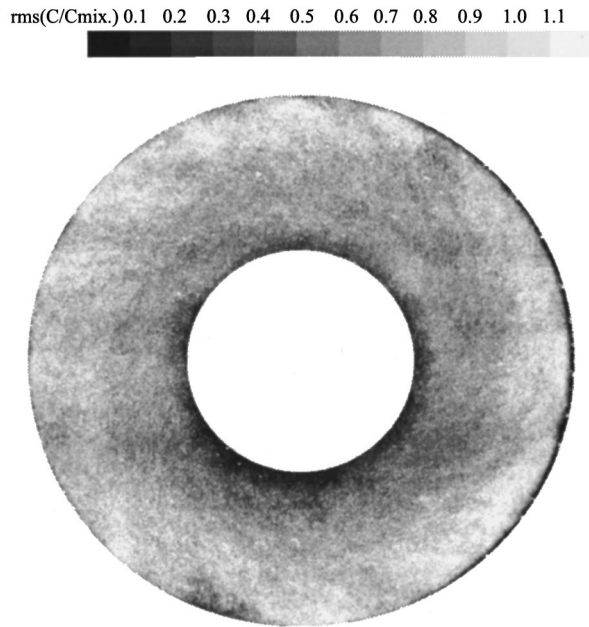


Fig. 5 Ensemble-averaged local rms of the fuel concentration fluctuation

Several advanced turbulent models ([9]) are available, including also a revised version of the six equations Reynolds Stress Model (RSM).

Both flow and chemical species transport equations have been solved by an implicit segregated scheme. Standard wall function is adopted for turbulent boundary layer conditions at walls.

Numerical Calculations. Three series of numerical simulations have been performed with the following objectives:

- 1 to compare the results at the atmospheric cold conditions with the real hot pressurized conditions. The following two main effects have been analyzed separately:
 - (i) the presence of flame in front of the burner and
 - (ii) the effect of pressure.

This investigation allows to verify the scaling between experimental and real conditions and assess the assumptions made for the experimental conditions setting

- 2 to define the best suited modeling for the numerical simulations.
- 3 to compare numerical results with experimental data in order to establish both the best calculation parameters and turbulence models.

Results Viewing. The fuel mass fraction is mainly analyzed on the two surfaces shown in Fig. 6, the conical combustion chamber inlet surface and the measurement plane, respectively. The contour plots of the fuel mass fraction, having a quite complex behavior, do not allow to compare the computed results and the experimental data quantitatively. Therefore, the local mass fractions belonging to different circumferential positions of a periodic sector of the above mentioned surfaces are shown in a scatter plot, as a function of the nondimensional radius $R/R_{\max}$ (Fig. 7).

To obtain overall information, the distributions are circumferentially averaged in order to plot mean and rms values as a function of the nondimensional radius. A conventional diagram of distributions is then depicted in Fig. 8. The three curves are: the circumferential averaged value, the average plus rms and the av-

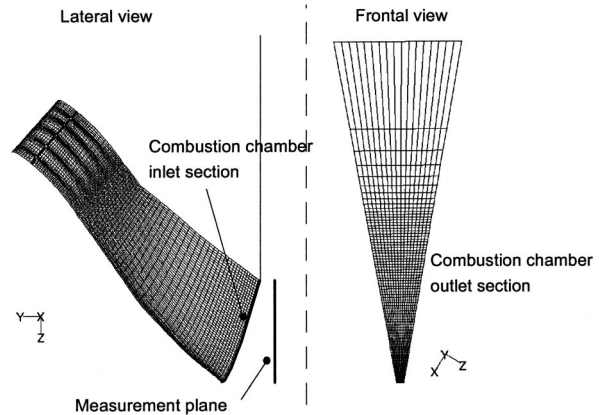


Fig. 6 Blade and mixing duct grid view

erage minus rms, respectively. These distributions allow to obtain an overall quantitative evaluation of both radial and circumferential unmixedness.

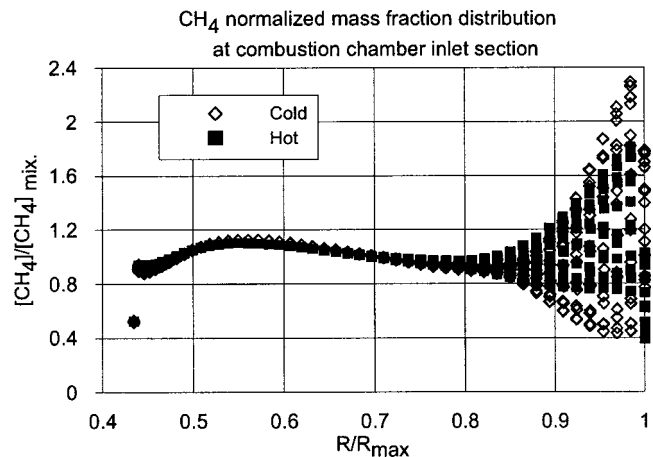


Fig. 7 Comparison between cold and reactive simulations

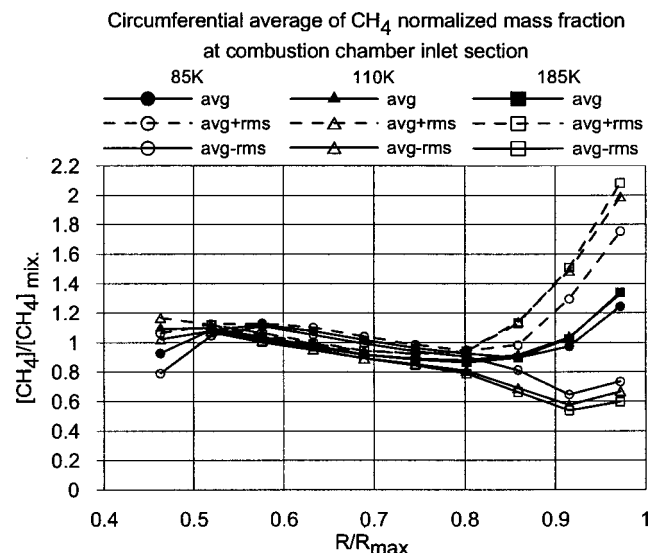


Fig. 8 Analysis of optimal grid dimension

Real Versus Experimental Conditions

(i) *Presence of Flame in Front of the Burner.* Combustion is simulated with a simplified EBU-one-step-reaction mechanism, because a good accuracy in flame simulation does not concern with this analysis (calculation is conservative because of the higher temperature due to the lack of dissociation effects). Radiative effect is included through a model that considers the emissivity of the gas.

The species distributions in the inlet combustion chamber plane for reactive and cold flows are shown in Fig. 7. The CH_4 normalized mass fraction is plotted in a two-dimensional diagram in function of the nondimensional radius. Each point represents a value of the CH_4 mass fraction in the relative cell center.

It should be noted that

- in both cases the distributions spread in the outer region of the surface analyzed, the dispersion is due to the concomitant higher turbulence levels and larger concentration nonuniformity occurring in the peripheral region of the burner exit section.
- the dispersion of data has the same behavior up to the nondimensional radius equal to 0.85.
- in the range of nondimensional radius between 0.85 and 1, the cold flow dispersion is greater than the combustive one; this result is not surprising because of the increasing of the turbulent intensity occurring before the flame front.

Therefore the cold flow conditions are more conservative than the real ones. So, due to remarkable reduction of computer effort, the successive calculations have been performed in nonreactive flow conditions.

(ii) *Effects of Pressure.* Two calculations have been performed, respectively, in pressurized and atmospheric conditions; the former at real operative full load conditions, the latter in the conditions selected for experiments. The differences between the two cases are negligible; that confirms the assumptions made on the basis of the similarity theory in the phase of selecting the experimental conditions. It was important to verify this aspect because not all the involved phenomena can be strictly treated with the adopted similarity theory.

Choice of the Grid. The three-dimensional domain includes one blade passage of the diagonal premixer burner duct. The region downstream of the blade row has been twisted on the basis of the blade trailing edge angle in order to have a good alignment between flow and cells surfaces. The overall domain includes also a sector of the axial swirler throat and a downstream region close to the burner outlet (not depicted on the figure for clearness reasons). The presence of these two parts is necessary because of their not negligible aerodynamic effects on the 10-mm downstream measurement section.

Fluid dynamic conditions at the axial swirler throat section have been taken from experimental LDV measurements carried out in the Ansaldo Ricerche laboratories.

A series of preliminary calculations has been performed in order to define the mesh size that guarantees the “grid independent” conditions for the simulations.

Computations were performed on both structured and unstructured grids with total cell numbers ranging between 85 K and 185 K. Results have been assessed through the species mass fraction dispersion at the combustion chamber inlet section.

Unstructured grids lead to a greater diffusion than structured ones. The same results obtained for structured topology have been reached only with greater cell overall number. This is due to a greater numerical diffusion occurring in the downstream blade row channel (where the most important phase of mixing process occurs).

Grid-independent conditions with structured topology are reached for the 110 K case, as shown by Fig. 8. Results do not change also with an adaptive grid refinement driven by the species

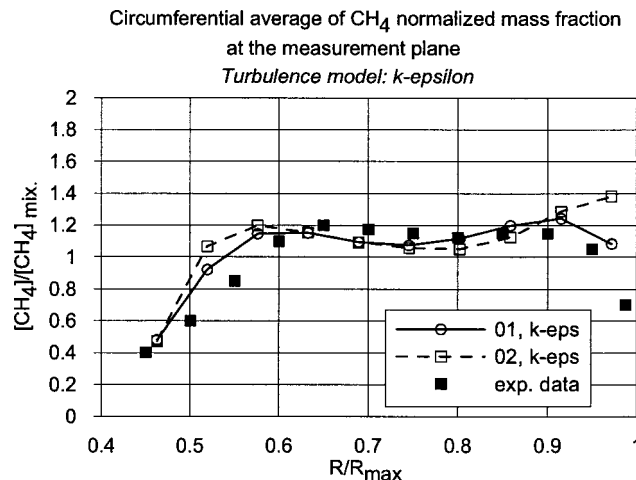


Fig. 9 Comparison between experimental and computed data. $k-\epsilon$ turbulence model: 01 first-order discretization scheme; 02 second-order discretization scheme

mass fraction gradient that allows a four times increase in cell number near the nozzles regions. Therefore a 110 K cells-structured topology has been adopted for the successive cases. In this conditions, convergence is reached in a few hours on a SGI Indigo 2 WS equipped by 350 MB RAM. Figure 6 shows a side view of the grid including burner blade and mixing duct.

Numerical Results: Comparison With Experimental Data

Comparisons between calculated and experimental data have been made in the measurement plane normal to the burner axis located at 5 mm downstream of the burner outer section.

The following parameters are taken into account:

Discretization Scheme

- first-order upwind
- second-order upwind

Turbulence Model

- standard, RNG and realizable $k-\epsilon$ models (Fluent user manual, 1998)
- RSM
- RSM—reduced Prandtl and Schmidt numbers

In the last case Prandtl and Schmidt numbers were set to 0.25 on the basis of the results of a three-dimensional simulation made for a 3 MW industrial gas turbine ([10]).

The RSM model is applied to the momentum equation only, being the scalar turbulent diffusion calculated by a correlation depending on Schmidt number.

Figures 9 and 10 show the diagrams of circumferentially averaged CH_4 mass fraction value against radius in the measurement plane, respectively for $k-\epsilon$ and RSM turbulence models. The experimental data are reported in both diagrams.

Only standard $k-\epsilon$ results are presented in Fig. 9, as both RNG and realizable methods give almost identical distribution only slightly more dispersed than the standard model.

It should be noted that

- first-order upwind $k-\epsilon$ model gives a good agreement with the experimental data.
- second-order upwind $k-\epsilon$ model gives a worse agreement than the first-order scheme, especially at the ends of the curve; it is difficult to interpret such results because the second-order scheme is theoretically less diffusive than the first-order scheme and therefore better suitable for jet spreading rate.

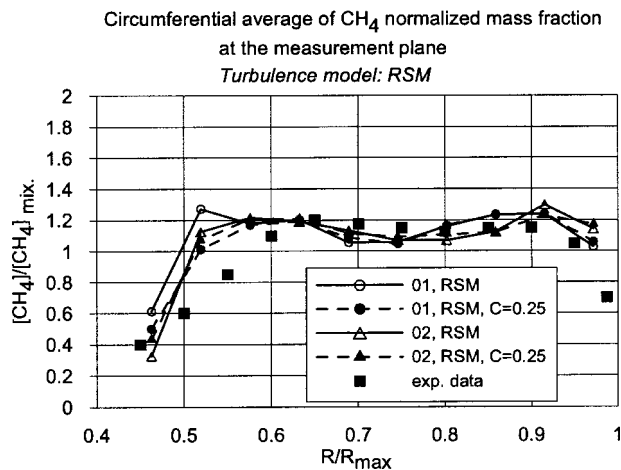


Fig. 10 Comparison between experimental and computed data. RSM turbulence model: 01 first-order discretization scheme; 02 second-order discretization scheme. $C=0.25$: case with Prandtl and Schmidt number reduced to 0.25

- RSM turbulence model calculations give similar results with the exception of first-order upwind scheme, which shows a maximum in the low radius region.
- every model overestimates the CH_4 mass fraction in the low radius region. This region is quite critical due to the presence of the flow coming from axial swirler, which interacts with the main flow from the diagonal swirler (in any case fuel mass flow rate evolving in the axial swirler is quite low).
- second-order schemes present a better agreement with the experimental data in the central region than the first-order schemes.
- all models give an overestimation in the outer region, close to nondimensional radius R equal to one. This behavior will be analyzed in more details, as such zone could be affected by a large experimental uncertainty (there is a lack of circumferential symmetry in the experimental concentration distributions).

The curves of the rms of mass fraction circumferential fluctuations are depicted in Fig. 11. In this case it should be noted that

- $k-\epsilon$ models underestimate the rms for all radial positions; only in the outer region the behavior is similar to the experimental one; the discrepancies are remarkable for $R < 0.8$ because the rms value computed is almost zero.

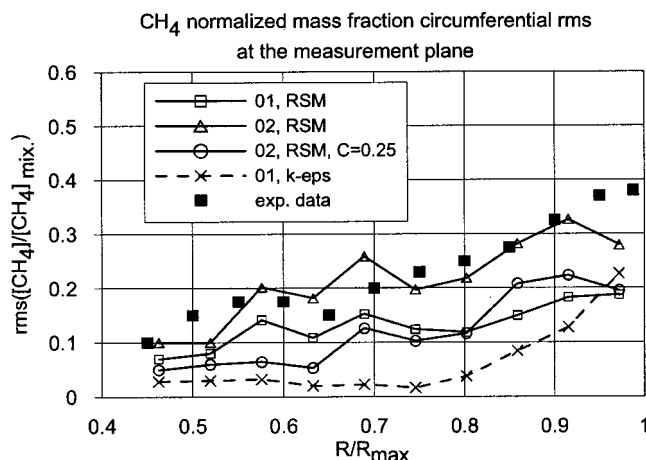


Fig. 11 Comparison between experimental and computed data; $k-\epsilon$ and RSM models

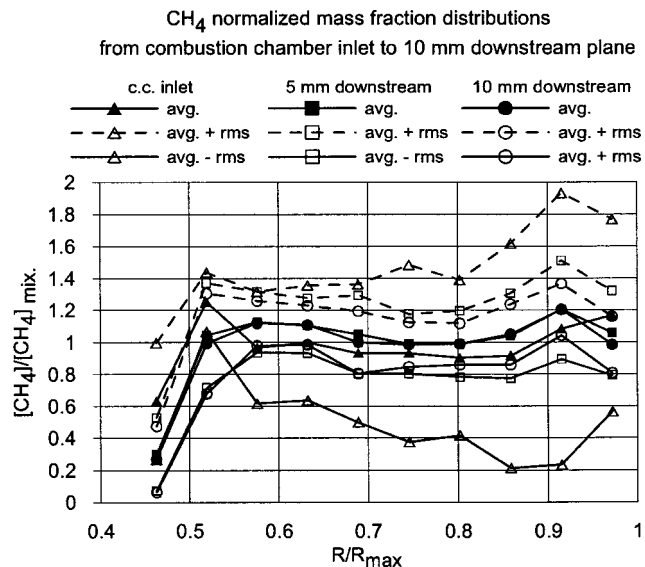


Fig. 12 Effect of combustion chamber sudden expansion of CH_4 distribution

- RNG and realizable $k-\epsilon$ models give results similar to standard $k-\epsilon$ methods.
- first-order upwind/RSM scheme underestimates the rms values. Curve behavior is better than $k-\epsilon$ one up to R equal to 0.9; the prediction in the outer region is poor because of the rms value decreasing.
- second-order upwind/RSM model gives the better rms prediction. Results are in good agreement up to $R=0.9$.

Overall, the second-order upwind/RSM model shows the best agreement with the experimental data. The $k-\epsilon$ model gives a good agreement with average values, but it overestimates the jet diffusion.

Assessment of Mixture Preparation. Experimental results show a quite good mixing behavior in radial direction as the circumferential averaged fuel mass fraction distribution is flat for a large range of the analyzed region (obviously fuel mass fraction decreases in correspondence of the axial swirler stream flow).

Concerning circumferential unmixedness (proportional to the rms value), it is interesting to analyze the mass fraction contours in successive sections along with the axial direction (Fig. 12). The unmixedness level is higher in the outlet section than in the 5-mm downstream section. This means that a strong circumferential mixing rate occurs in the burner downstream region; therefore the mixture homogeneity is expected to be fairly good near the flame front region (generally located more than 5 mm downstream of the throat burner section).

Conclusions

An integrated experimental and numerical approach has been developed in order to analyze the mixing process between fuel and air in a heavy-duty gas turbine burner. This work is part of a general Ansaldo Ricerche program about the simulation of advanced pressurized combustion systems.

Planar Mie scattering optical measurements have been performed in an atmospheric test facility that allows to test a real-scale gas turbine burner. Several instantaneous concentration distributions have been obtained at the burner exit allowing to evaluate the degree of mixing between gas fuel and oxidant streams. Knowledge of the instantaneous interfaces between the two streams helps understanding the phenomenon and can provide a criterion for optimizing mixture preparation.

CFD RANS model has been assessed through the analysis of several discretization schemes and turbulence models. A second-order/RSM model gives the best agreement with the experimental results. Great care was taken in mesh construction in order to avoid convergence problems and check grid independence.

The calculation of the species mass fraction distribution gives a remarkable aid for the design of novel gas turbine burners. Furthermore, the conditions at the combustion chamber inlet can be used as boundary condition in the combustive calculations; the effective fuel mass fraction distribution is fundamental for predicting both NO_x emissions and the presence of flame fluctuations.

Future activity (based also on LES simulations) is planned to better understand the limits shown by the present calculations. Most of the future efforts will be addressed to evaluate time-dependent aspects of the problem, because the reacting phenomena depend on the local instantaneous reactant concentration and the existence of locally rich and lean mixture pockets could influence also the dynamic characteristics of the combustion process.

Acknowledgments

The financial support of Ansaldo Energia for the experimental activities is gratefully acknowledged.

The authors are grateful to Siemens and Ansaldo Energia for the permission of paper publication.

The authors wish to thank Mr. Nino Elia for his invaluable contribution to the whole experimental activity.

References

- [1] Lefebvre, A. H., 1995, "The Role of Fuel Preparation in Low Emission Combustion," ASME Paper No. 95-GT-465.
- [2] Gazzani, F., and Mori, G., 1997, "CFD Modeling of Turbulent Premixed Combustion in a Heavy Duty Gas Turbine," Fluent Users' Seminar, Manchester, UK.
- [3] Mori, G., Hoffmann, S., and Di Meglio, A., 1997, "Numerical Procedure for Optimization of a Liquid Fuel Premix Burner Injection System," ILASS Int., Florence.
- [4] Hoffmann, S., Judith, H., and Holm, C., 1998, "Further Development of the Siemens LPP Hybrid Burner," ASME Paper No. 98-GT-552.
- [5] Nicol, D., Malte, P. C., Hamer, A. J., Roby, R. J., and Steele, R., 1998, "Development of a Five Step Global Methane Oxidation—NO Formation Mechanism for Lean Premix Gas Turbine Combustion," ASME Paper No. 98-GT-185.
- [6] Rutar, T., Martin, S., Nicol, D. G., Malte, P. C., and Pratt, D. T., 1997, "Effect of Incomplete Premixing on NO_x Formation at Temperature and Pressure Condition of LP Combustion Turbine," ASME Paper No. 97-GT-335.
- [7] Dugué, J., Mbiok, A., and Weber, E. R., 1994, "Mixing Characterization in Semi-industrial Natural Gas Flames Using Planar Mie-Scattering Visualization," Seventh Int. Symp. on Appl. of Laser Techniques to Fluid Mech., Lisbon.
- [8] Eaton, A. R., Frey, S. F., Cusano, D. M., Plesniak, M. W., and Sojka, P. E., 1996, "Development of a Full-Field Planar Mie Scattering Technique for Evaluating Swirling Mixers," *Exp. Fluids*, **21**, pp. 325–330.
- [9] Coklat, D., 1998, "Advanced Turbulence Modeling," lecture at "CFD in Combustion Engineering" course, Leeds University, Leeds, UK.
- [10] Liever, P. A., Myers, G. D., Hernandez, L., and Griffith, T., 1998, "CFD Assessment of a Wet, Low- NO_x Combustion System for a 3 MW-Class Industrial Gas Turbine," ASME Paper No. 98-GT-292.

M. Carl

T. Behrendt

C. Fleing

M. Frodermann

J. Heinze

C. Hassa

Institute of Propulsion Technology,
German Aerospace Center,
Linder Höhe, DLR
D-51147 Cologne, Germany

U. Meier

D. Wolff-Gassmann

Institute of Combustion Technology,
German Aerospace Center,
Pfaffenwaldring 38–40
D-70569 Stuttgart, Germany

S. Hohmann

N. Zarzalis

Daimler Chrysler Aerospace,
Dachauerstrasse 665,
D-80995 Munich, Germany

Experimental and Numerical Investigation of a Planar Combustor Sector at Realistic Operating Conditions

Results of an ongoing collaboration between the engine manufacturer MTU and the German aerospace research center DLR on the NO_x reduction potential of conventional combustors are reported. A program comprising optical sector combustor measurements at 1, 6 and 15 bars and CFD calculations is carried out. The aims are to gather information in the combustor at realistic operating conditions, to understand the differences between the sector flow field and data from tubular combustors, to verify the used CFD, and to discover the benefits and limitations of the applied optical diagnostics. Selected results of measurements and calculations of the isothermal flow and of measurements at 6 bars and 700 K at a rich-lean and overall lean AFR are reported. The used measurement techniques were LDA, PDA, Mie scattering on kerosene, quantitative light scattering, OH^* chemiluminescence, and LIF on OH. The measurements were able to confirm the intended quick and homogeneous mixing of the three staggered rows of secondary air jets. [DOI: 10.1115/1.1378298]

Introduction

In the framework of the Engine 3E-project of the German federal aeronautical research program, MTU and DLR cooperate on research and development of low emission aeroengines. The application in view is the turbofan engine with high bypass ratio. The cooperation's aim is a low-emission single annular combustor. It seeks to achieve the NO_x reduction with a better homogenization of the mixture in the combustor. To that end, DLR modified a test rig to accommodate rectangular combustor sectors at pressures up to 20 bars, undertook further development of measurement techniques for the application at higher pressure, and enhanced its capabilities in combustion CFD. MTU undertook the combustor development itself and also extended the capabilities of its numerical design tools and combustion CFD.

The choice of a rectangular combustor was motivated by the uncertainties that persist in design and scaling of combustors with CFD on the base of earlier designs and tubular combustor tests. Especially for the intended way of NO_x reduction by homogenization of the mixture, detailed and reliable information on reaction and flow inside the combustor is necessary. NO_x reduction in SAC combustors must rely on quick and homogeneous mixing of the secondary air after the primary zone, since it comprises the transition through stoichiometry from globally rich to lean AFR at full

power. Here the details of the jet mixing and its interaction with the primary zone are of interest, since the aerodynamics of the mixing zone can only be investigated in sector combustors.

A further aim of the study was to find the potential of optical diagnostics to describe the reacting flow field at pressures that reproduce the characteristics of fuel preparation in real engine operating conditions.

The workprogram was split into four parts:

- 1 atmospheric investigations of sprays from airblast nozzles,
- 2 isothermal investigations of the mixing inside the combustor,
- 3 adaptation of measurement techniques to higher pressure, and
- 4 investigations of the optically accessible rectangular combustor sector at 6 bars and 15 bars.

In this paper, results from the second and the fourth workpackage are reported. First, the test rig and the combustion chamber is briefly described, an overview of the used measurement techniques is given and selected experimental results without combustion and with combustion at 6 bars are presented before a short overview of the future work and a preliminary résumé are given.

Experimental Facility

The test rig and the combustion chamber have been specifically designed for optical investigations. The peculiarities of the research combustor can be discerned in Fig. 1, where a cross section through the combustor and the pressure casing is shown. For mea-

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-123. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

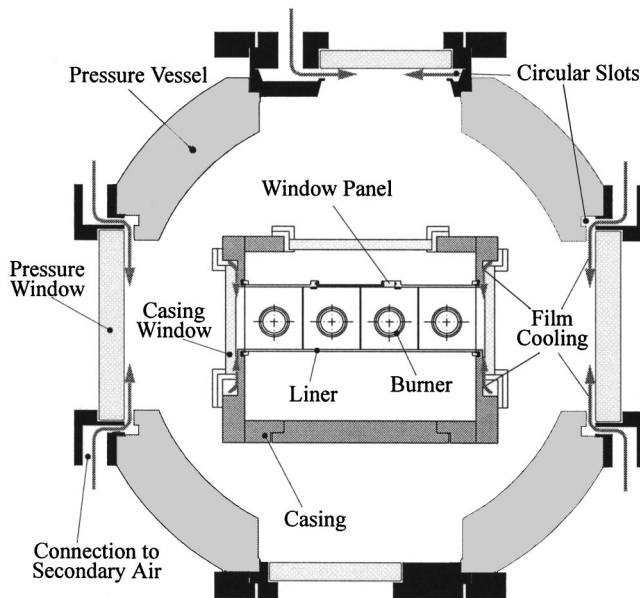


Fig. 1 Radial circumferential cross section of combustor and pressure vessel

surement techniques like LDA, that need optical access along a line, a curved sector was abandoned in favor of a rectangular sector with parallel side windows.

In circumferential direction the liner is subdivided into four sectors each including one burner. The number of the sectors was given by the distance of the combustor side walls, that was in turn decided by another project. In the direction of the main flow the liner is subdivided into the primary and the mixing zone. The mixing zone is equipped with three rows of holes on both sides of the liner. The holes of the first row are staggered with respect to the holes of the second and the third row and the upper holes are staggered with respect to the lower holes. The cross section through the burner axis intersects the hole of the first lower row and the holes of the second and third upper row. The walls of the inboard and outboard side of the liner are cooled by effusion cooling.

For the application of planar measuring techniques like planar laser-induced fluorescence, panels with quartz glass windows are placed on the outboard side of the combustor, which allowed a flexible introduction of high quality light sheets.

The pressure casing is designed for pressures up to 20 bars. The diameter of the side windows is 170 mm and 115 mm for the upper and lower window, that are mounted eccentrically in a rotatable metal plate to accommodate the different light sheet positions. Together with the side windows of the combustor and the combustor casing, the inner 60 percent of the sector width are completely accessible.

Figure 2 gives a schematic representation of the pressure vessel within the test plant. The pressure vessel is uncoupled from the bottom of the test stand by a supporting structure. The main air entering the combustor through the diffuser is preheated by a fuel oil heat exchanger for the tests at 15 bars. A maximum of 10 kg/s is possible. For the tests at 6 bars, an electrical heater is used to generate an inlet temperature of 700 K. The secondary air enters through three separately controllable lines, one for internal film cooling of the pressure windows, another for the film cooling of the casing windows and the third near the inlets as air purge for the pressure vessel. A maximum cooling air flow of 3 kg/s is available. A more detailed description of the test facility can be found in [1].

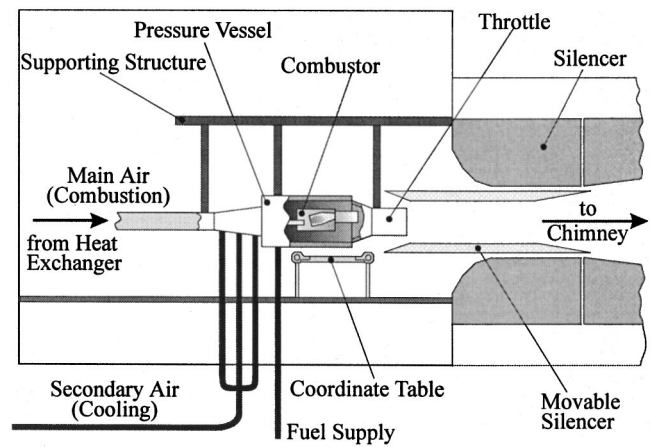


Fig. 2 Schematic view of test plant

Isothermal Investigations

In an aeroengine combustor, the combustion process is governed by the atomization of the liquid fuel and the mixing of primary and secondary combustion air. In the case of a rich quench lean (RQL) combustion system, the mixing process is of particular interest for two reasons. First, the recirculation of secondary air into the rich primary zone must be avoided in order to keep a homogeneous rich primary zone. Second, the mixing process (secondary air into primary air) must be very fast (<1 msec), to restrict the production of NO_x . The recirculation of the secondary air into the fuel rich primary zone would lead to near stoichiometric regions with locally high temperatures, in which NO_x formation would occur. The same is observed for long residence times in the quench module, where the stoichiometry of the fuel/air mixture changes from fuel rich to lean conditions.

Experimental Apparatus and Measuring Technique. The Mie scattering by quantitative light scattering (QLS) is used as planar point resolved technique to measure the dilution of one flow of air by a second. One stream of air is seeded with scattering particles and a laser light sheet is placed into the flow with the intensity of the scattering imaged onto a CCD camera signifying the local concentration of the seeded air stream in the mixing region. The technique is described in detail in [2].

The primary air through the combustor head and the secondary air through the mixing jets are separately controllable and each can be separately seeded. The optical access is through high quality quartz windows at the combustor side and through the combustor exit. The light sheet dimensions were 12×8 mm, giving a Pixel resolution of $31 \mu\text{m}$.

Numerical Solution Procedure. The mixing module can be optimized with regard to the residence time and the distribution of the quench air by means of computational fluid dynamics. The equations which are solved for the calculation are the three-dimensional Navier-Stokes equations. The turbulence is described by the well known $k-\epsilon$ model. The solution of a transport equation for a passive scalar gives the information about the quality of mixing. For a prediction of the quench air distribution, the boundary conditions are estimated with the help of a one-dimensional aerodynamic calculation, where the air split inside the combustor is determined. The three-dimensional calculations enable a reduction in the number of tests to run and therefore reduce costs and development time.

For the simulation only one single nozzle segment was considered and periodically boundary conditions were applied. The grid comprises from about 500,000 hexahedral cells. The commercial CFD package CFX-TASCFlow Version 2.73 was used to calcu-

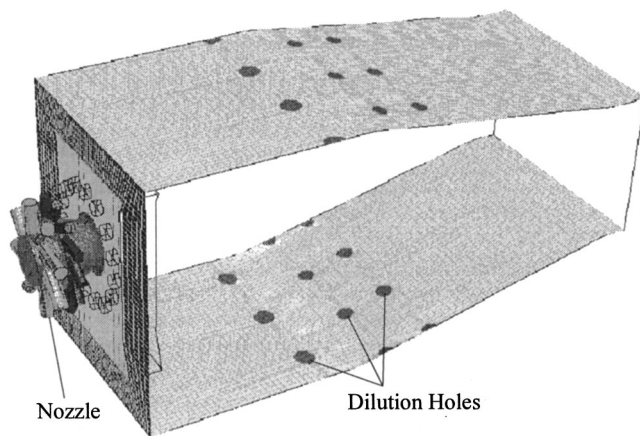


Fig. 3 Control volume and surface grid

late the isothermal flow field. The control volume and the surface grid are shown schematically in Fig. 3. A detailed drawing of the nozzle type used is shown in Fig. 4.

Results. For the prediction of the mixing of the quench air, a passive scalar was observed, whose value at the dilution holes was set to 1. The predicted mixing field in a longitudinal plane through the center of the combustion chamber is shown in Fig. 5.

The results of the simulation are compared to experimental results obtained by QLS measurements. The results of these measurements are directly comparable to the mixing field calculations by CFD. The experimentally determined mixing field of the quench air for the same plane as for the calculation is shown in Fig. 6.

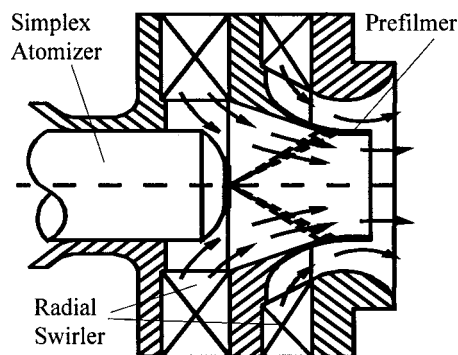


Fig. 4 Nozzle type

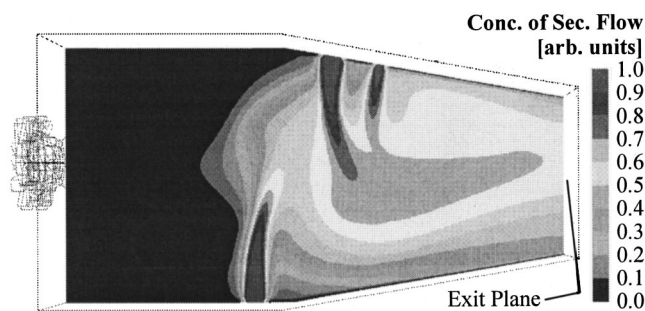


Fig. 5 Predicted mixing field in the plane through the center of the nozzle

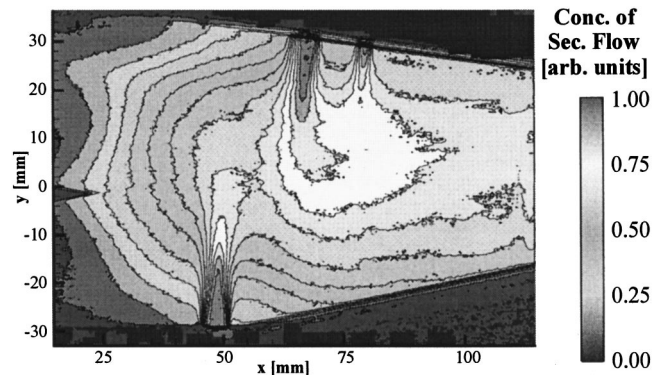


Fig. 6 Experimental mixing field in the plane through the center of the nozzle

Combustion Diagnostics

Laser Doppler Anemometry. The gas velocity was measured by Laser Doppler Anemometry (LDA). The light of a 3 W cw Ar⁺-Laser was passed through optical fibers to a two component optic. The resulting measuring volume was 0.32 mm × 3.0 mm. For the seeding of the air flow SiO₂ particles with a diameter of 0.5 μm were conveyed by a fluidized bed generator. The scattered laser light was detected by a receiving optic with an angle of 12° relative to the beam direction. The burst signals were analyzed via Fast Fourier Transformation (FFT). Details of the measuring arrangement are described in [1].

Phase Doppler Anemometry. The diameter of a spherical droplet and its velocity were measured with a Phase Doppler Anemometer (PDA). The specific instrument used was a dual PDA built by DANTEC [3]. The dimensions of the measuring volume were 0.32 mm in diameter and 1.5 mm in length. In Fig. 7 the optical set up of the PDA is shown. The Doppler frequencies and the phase shift were determined with a covariance processor. The signals were processed if the signal to noise ratio of every burst of a particle was -2 dB or better. For a more detailed description of the measurement technique see [4].

Mie Scattering of Kerosene. The distribution of the liquid kerosene in the near field of the nozzle was measured qualitatively by planar Mie scattering. The light of a pulsed Nd:YAG laser was formed to a planar light sheet with the dimensions of 50 × 0.2 mm by a multiple lens optic. The wavelength of the light was 355 nm and the mean pulse energy at the measuring plane was 15 mJ. The scattered light was detected by a UV-grade photo lens attached to an intensified CCD camera with an exposure time of typically 20 ns.

OH* Chemiluminescence. For the indication of the flame reaction zone the radiation of the electronically excited OH* radicals within the flame front was used, see [5]. The emission spectrum has a band structure with a maximum intensity at 310 nm.

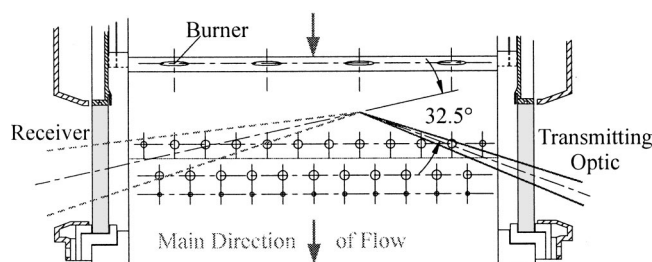


Fig. 7 Optical set up of PDA (top view onto combustor)

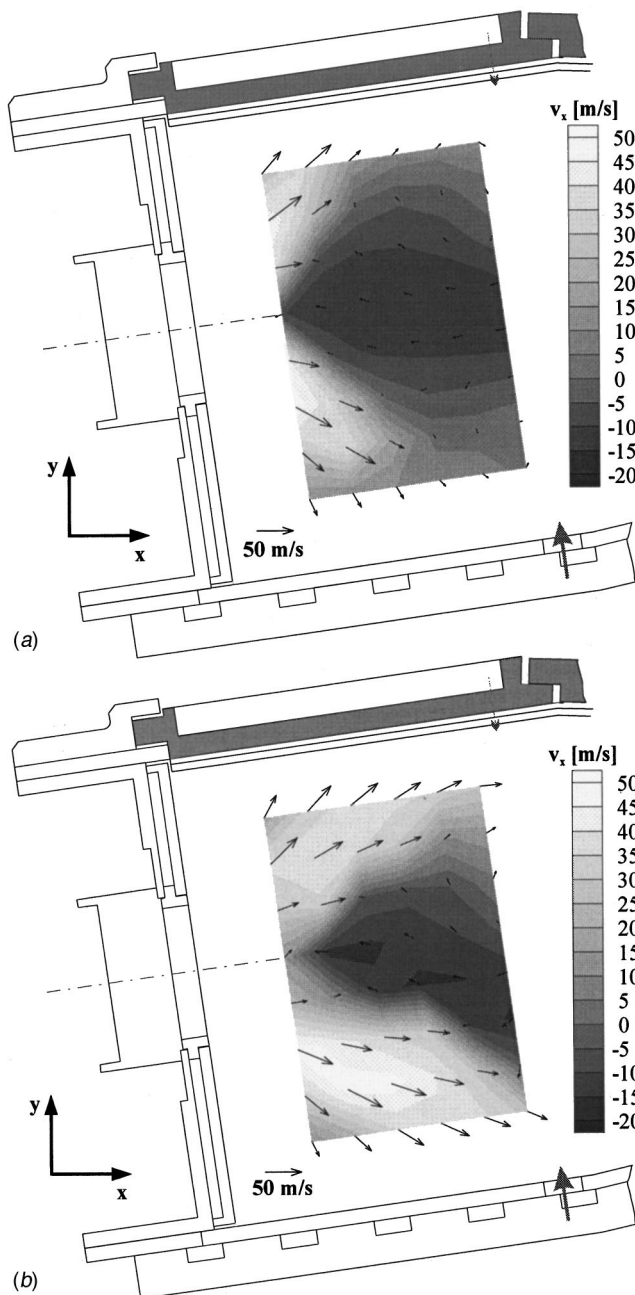


Fig. 8 Gas velocity in the primary zone ($P_{30}=6$ bar, $T_{30}=700$ K); (a) upper figure: AFR=90, (b) lower figure: AFR=40

The exposure time of the intensified CCD camera for a single image was increased from 20 ns to 10 μ s and an interference filter centered at 310 nm (FWHM: 10 nm) was placed in front of it. The OH* fluorescence was integrated over the entire width of the combustor.

Laser-Induced Fluorescence of OH. The planar detection of the OH radicals inside the regions of high temperatures was performed by laser-induced fluorescence (OH LIF). OH radicals were excited by ultraviolet radiation from a tunable dye laser at $\lambda \approx 283$ nm. The laser beam was shaped into a light sheet of about 0.3 mm thickness. In contrast to the integral measuring method of the OH* chemiluminescence, this is a spatially resolved laser induced process. The radiation of the excited OH radicals was de-

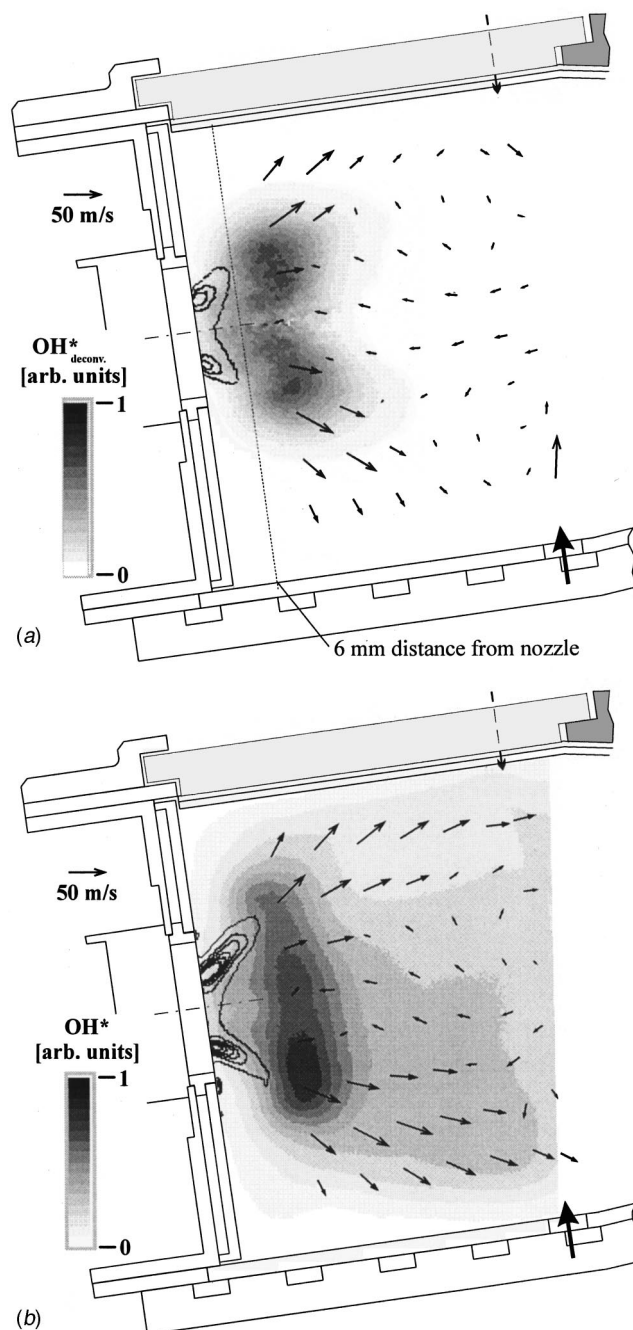


Fig. 9 OH* chemiluminescence, Mie scattering on kerosene and gas velocity in the primary zone ($P_{30}=6$ bar, $T_{30}=700$ K); (a) upper figure: AFR=90, (b) lower figure: AFR=40

tected by an intensified CCD camera at wavelengths around 312 nm. For the details of this measurement technique and the experimental arrangement, see [6].

Results on Combustion

Measurements with optical diagnostic systems were performed at pressures of 6 bars and 15 bars and at preheated air inlet temperatures of 700 K. The combustor air/fuel ratio (AFR) was varied from 40 to 90. Because of the ongoing analysis of the measurements at 15 bars, this paper is restricted to results at 6 bars.

The optical measurements focused on the primary zone and the mixing zone of the wall jets, since they are most important for the pollutant formation. The organization of the results will just fol-

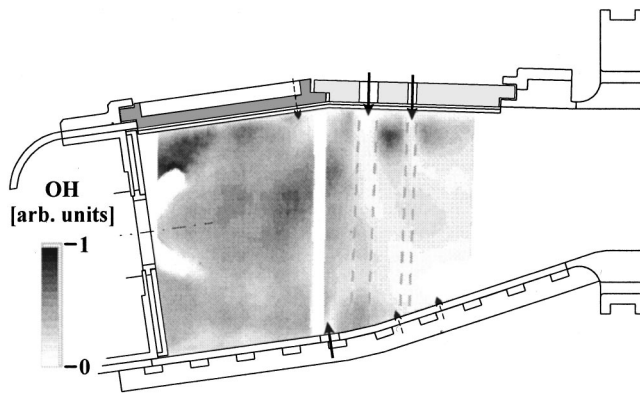


Fig. 10 OH distribution ($P_{30}=6$ bar, $T_{30}=700$ K, AFR=40)

low the direction of the flow, while pointing out the differences between the rich (AFR 40) and lean (AFR 90) operating condition relating to the primary zone on the way wherever possible. The measurements concentrated in the main direction of the flow on the left of the two middle sectors. All results reported here refer to an observation plane through the center of the nozzle in this sector.

Primary Zone. First a comparison of the gas velocity at both conditions is shown in Figs. 8(a) and (b). Here the horizontal velocity v_x and vertical velocity v_y are depicted by arrows, the grayscales additionally represent the horizontal component v_x only. The form of the recirculation and the different expansion angles of the high velocity region of the nozzle flow expose the asymmetry of the primary zone flow. This is caused by the asymmetric position of the nozzle in the combustor head, the center being positioned about ten percent above the middle of the combustor. In the considered plane, the recirculation zone is tilted downwards, although the first jet of the staggered first row issues from the lower side. This demonstrates the influence of the high confinement on the swirl flow. The asymmetry at the end of the primary zone can in turn result in variable penetration of the jets from the upper and lower side.

The other feature that catches the eye is the smaller expansion angle of the swirling nozzle flow and the narrower recirculation region at the rich condition. Results on the spray, the reaction and the temperature field are needed for an interpretation.

The superposition of the area taken up by the spray in a cross section through the nozzle center line and the contour of the reaction zone are shown in Fig. 9. The extent of the spray is measured by Mie scattering and the reaction by OH* imaging. Again the velocity measurements are represented with arrows to visualize the global flow structure. Figure 9(a) shows the lean condition: Here it was possible to deconvolute the OH* signal on the assumption of its axial symmetry [7].

The reaction stabilizes through the inner recirculation zone at the inner boundary of the spray. For the higher fuel load shown in Fig. 9(b), the OH* contour is much wider, almost reaching up to the combustor wall. Hence it is also influenced by the asymmetric nozzle position, which is the reason why the deconvolution did not work here. Consequently the original, not deconvoluted, line-averaged signal is shown in Fig. 9(b). The OH LIF image of Fig. 10 shows average OH number densities at both sides of the spray cone, signifying hot gases. The combined information of the OH* and the OH suggests, that combustion is stabilized at both the inner and outer recirculation zone. This furnishes an explanation for the above described differences in aerodynamics between the equivalence ratios, namely that the hot expanding gases in the corner squeeze the swirling flow, thus reducing its expansion angle and the width of the inner recirculation zone. Comparison of Figs. 9(a) and (b) also illustrates the different reaction zones for

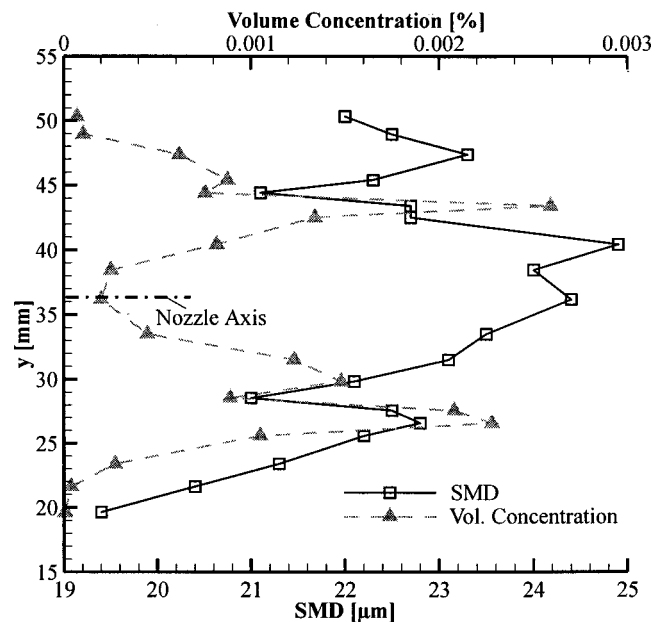


Fig. 11 SMD and volume concentration at an axial distance of 6 mm to the nozzle, cf. Fig. 9(a), ($P_{30}=6$ bar, $T_{30}=700$ K, AFR = 90)

the two operating points. At the lean condition, the OH* signal falls to zero 30 mm behind the combustor head, whereas for the rich condition two zones can be discriminated in the axial direction. The stabilization zone has the same axial extension as in the lean case, but the flame reaches up to the end of the primary zone.

The small overlap of the Mie scattering and the OH* for the AFR 90 condition in Fig. 9(a) points to a high degree of prevaporization. PDA measurements of the droplet size distribution and volumetric concentration of the spray support this observation. Figure 11 shows a profile of the Sauter mean diameter and the volume concentration measured at an axial distance of 6 mm to the nozzle. The volume concentration measured by PDA reflects the asymmetry of the gas flow seen in Fig. 8. The lower cone angle is larger than the upper one, cf. Fig. 9(a) and consequently the maximum concentration is 10 mm away from the nozzle centerline on the lower side and only 6 mm on the upper side. The increase of the droplet size inside of the spray cone from its low value at the maximum concentration is consistent with earlier observations of the influence of combustion on droplet size [8]. The quick disappearance of small droplets in the hot gas overcompensates the diameter diminution of the big droplets resulting in increasing droplet sizes. However, the effect is not as pronounced as at atmospheric pressure, because of the lower mean diameter and the rather low number of very big drops in the produced spray. At the maximum concentration positions, reaction has not yet begun. Hence, the size distribution in the spray cone is believed to be a good approximation of the starting conditions of the spray. The value of the SMD is about $23 \mu\text{m}$. This value correlates well with measurements made with similar nozzles and Kerosene at cold conditions and elevated pressures, ([9]). It can be concluded, that the airblast nozzle produces a rather fine spray at least at this atomizer loading. Unfortunately, the spray was too dense at AFR 40 to repeat the PDA measurements.

Mixing Zone. The penetration of the jets in the plane through the nozzle center as measured by LDA is shown with the velocity maps of Fig. 12 for AFR 40. The form of the presentation is the same as in Fig. 8, except that the gray scales represent the vertical velocity v_y .

There are three rows of holes, which are staggered in the circumferential and the axial direction, cf. Fig. 3. The first jet on the

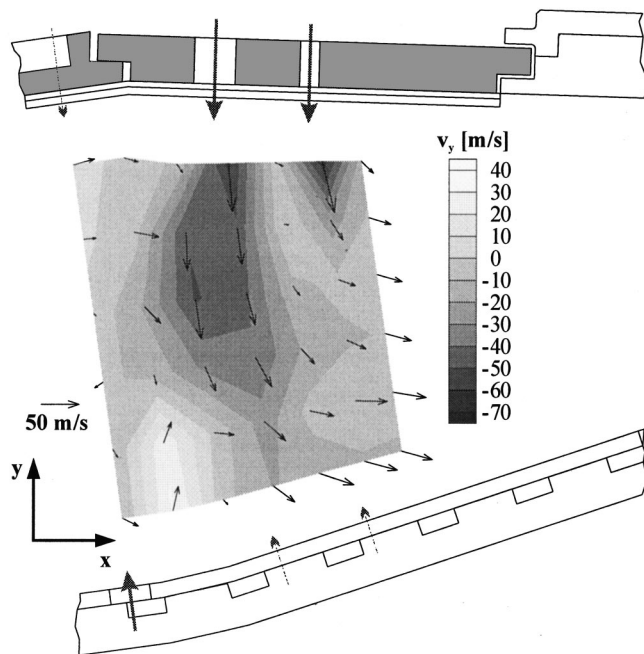


Fig. 12 Gas velocity in the mixing zone ($P_{30}=6$ bar, $T_{30}=700$ K, $AFR=40$)

lower side is quickly turned around by the primary zone flow, whereas the second jet on the upper side penetrates right through the measuring domain towards the lower wall. The OH LIF distribution of Fig. 10 shows the same structure, the penetration of the secondary air discernible by the white streaks of zero concentration. After the dissolution of the hot bubble behind the third jet, the combustion air is well mixed to low temperatures, signified by the low level of OH LIF. The plane between two nozzles exhibits the same low level of OH in the OH LIF measurement which is not shown here for brevity.

With this result, the combustion chamber holds the promise given by the isothermal secondary air distribution in Fig. 6, namely that the jet mixing is fast enough to suppress strong thermal NO_x formation in the secondary zone.

Some aspects of the turbulent mixing are highlighted with a statistical evaluation of the OH LIF and LDA measurements. Figure 13(a) shows a single shot OH picture. Comparison with Fig. 10 gives some information on the scale of the mixing at the end of the primary zone. Figure 13(b) shows the RMS value of OH extracted from 160 pictures. Areas of stable OH are the end of the primary zone and the area behind the second upper jet, where a recirculation of primary zone fluid is set up by the blockage of the jet and the lean burnout is intensified. Zones of high fluctuation are the collision region of the second upper and the first lower jet and the region between the last two lower jets.

This representation of the turbulent mixing can be supplemented by the distribution of the RMS value of the vertical velocity fluctuation measured by LDA in Fig. 14. The general level of the fluctuation in the mixing zone is twice as high as in the primary zone. Areas with very high fluctuation intensity are again the collision region between the first lower and the second upper jet as well as the penetration of the third upper jet and between the second and third lower jet. This picture stands in contrast to the fluctuations of the horizontal velocity, where the mixing zone exhibits a rather even distribution with values around 20 m/s RMS. This is also the level of fluctuation in the vertical direction, except for the high intensity regions. The observed degree of anisotropy of the velocity fluctuation may well be caused by precessing motions of the jets, as it is well known that the collision of jets is rarely stable. Unfortunately velocity measurements of the third

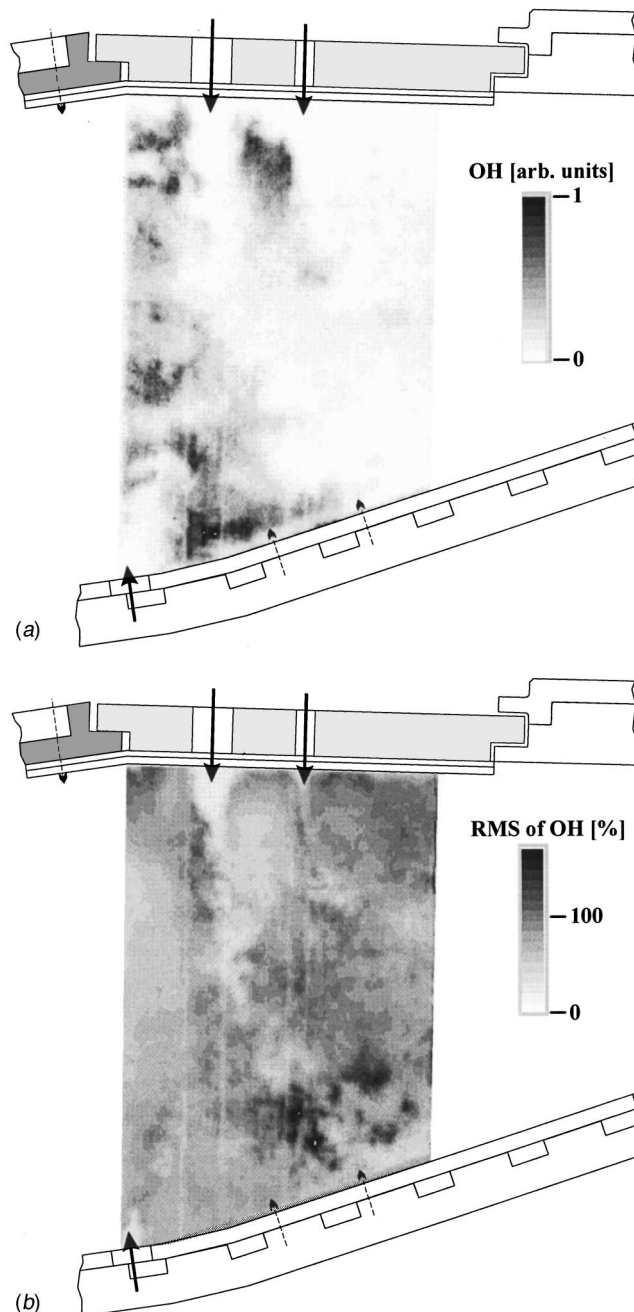


Fig. 13 OH LIF in the mixing zone ($P_{30}=6$ bar, $T_{30}=700$ K, $AFR=40$), (a) upper figure: single shot, (b) lower figure: RMS

component were not possible, because of the restrictions of the optical access by the windows of the pressure casing, such that the correlation of the radial and circumferential velocity could not be recorded. However, if the collision point of the first lower and second upper jet was unstable, the penetration of the third upper jet must also be unstable, as it sees a varying axial velocity at its root. In the same way the high fluctuation intensity region at the bottom between second and third jet results from the same coherent jet movement as on the upper side. These points have to be kept in mind when comparing measured results against Reynolds averaged calculations and probe measurements at the combustor exit and in the combustor, which might suggest better homogeneity than is actually reached in the combustor during the short residence time of the gases in the mixing and secondary zone.

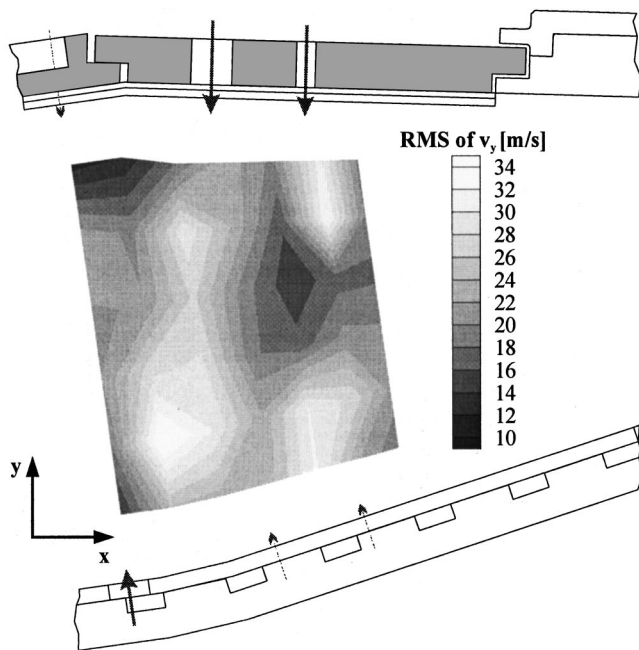


Fig. 14 RMS of vertical gas velocity in the mixing zone ($P_{30} = 6$ bar, $T_{30} = 700$ K, AFR=40)

Conclusion

The current status of the collaboration between MTU and DLR allows to draw conclusions on three aspects: The combustor, the CFD validation and the applicability of the combustion diagnostics. The results presented herein show, that the central aim of the mixing zone, the quick homogenization of the mixture is quickly achieved without hot streaks, such that the thermal NO_x production remains small. The differences observed in the primary zone combustion behavior between the rich and lean conditions highlight the feedback of the reaction on the primary zone flow field at realistic operating conditions and the influence of the rectangular confinement on the swirling flow from the nozzle.

Careful comparison of the results with numerical calculation has the potential to advance the use of CFD in combustor design with respect to two aspects: Since there is a considerable amount of local and global information, it can give advice on the degree of complexity and refinement that is needed in the physical models to reproduce the effects that are considered important for the goals of industrial CFD. Furthermore, a comparison with validated CFD on a tubular combustor would, at least for the used burner concept, generate rather accurate knowledge about the significance of tubular test results for the combustor development. For the comparison of CFD and measurement, further discussion on the importance of unsteady behavior in the combustor is needed.

The application of the combustion diagnostics was sometimes cumbersome but successful in the end, as the results of the different methods are consistent with each other and only their combined use creates the picture that is needed to satisfy the aforementioned goals.

The drawn conclusions point to the direction to take for the next phase of the program. It is necessary to complement the experiment with results for the comparison of equivalence ratio and pressure. The validation phase, that has begun with the isothermal CFD shown in this paper, will certainly need successive calculations with rising complexity to cover the questions of fuel preparation, turbulence-chemistry interaction, and pollutant formation that are addressed in the experimental part of the collaborative MTU-DLR program.

Acknowledgment

The project underlying this report was supported by means of the German Federal Minister for Research and Technology (BMBF) under supporting sign 20T9540.

Nomenclature

AFR	=	air fuel ratio
CCD	=	charge coupled device
CFD	=	computational fluid dynamics
DGV	=	Doppler global velocimetry
FFT	=	fast Fourier transformation
LDA	=	laser Doppler anemometry
LIF	=	laser-induced fluorescence
OH	=	hydroxide
PDA	=	phase Doppler anemometry
QLS	=	quantitative light scattering
RQL	=	rich quench lean
SAC	=	single annular combustor
SMD	=	Sauter mean diameter

References

- [1] Hassa, C., Carl, M., Frodermann, M., Behrendt, T., Heinze, J., Röhle, I., Brehm, N., Schilling, T., and Doerr, T., 1999, "Experimental Investigation of an Axially Staged Combustor Sector with Optical Diagnostics at Realistic Operating Conditions," RTO-MP-14, pp. 18-1-18-11.
- [2] Voigt, P., and Schodl, R., 1998, "Using the Laser Light Sheet Technique in Combustion Research," AGARD-CP-598, pp. 48-1-48-9.
- [3] *Dual PDA Manual*, 1996, DANTEC/invent measurement technology.
- [4] Behrendt, T., and Hassa, C., 1998, "Investigation of the Spray Dynamics of Aeroengine Fuel Injectors under Atmospheric and Simulated Pressure Conditions," AGARD-CP-598, pp. 5-1-5-12.
- [5] Dyer, M. J., and Crosley, D. R., 1982, "Two-Dimensional Imaging of OH Laser Induced Fluorescence in a Flame," Opt. Lett., 7, pp. 382-384.
- [6] Meier, U. E., Wolff-Gaßmann, D., Heinze, J., Frodermann, M., Magnusson, I., and Josefsson, G., 1999, "LIF Imaging of Species and Temperature in Technical Combustion at Elevated Pressure," Proc. 18th ICIASF'99, pp. 7.1-7.10.
- [7] Dasch, C. J., 1992, "One-Dimensional Tomography: A Comparison of Abel, Onion-Peeling, and Filtered Backprojection Methods," Appl. Opt., 31, No. 8, pp. 1146-1152.
- [8] Hassa, C., Deick, A., and Eickhoff, H., 1993, "Investigation of the Two-Phase Flow in a Research Combustor under Reacting and Non-Reacting Conditions," AGARD-CP-536, pp. 41-1-41-12.
- [9] Cao, M., Eickhoff, H., Joos, F., and Simon, B., 1988, "Influence of Operating Conditions on the Atomization and Distribution of Fuel by Air Blast Atomizers," AGARD-CP-422, pp. 8-1-8-8.

G. Klose¹

e-mail: goeran.klose@its.uni-karlsruhe.de

R. Schmehl

R. Meier

G. Maier

R. Koch

S. Wittig

Institut für Thermische Strömungsmaschinen,
Universität Karlsruhe (TH),
Karlsruhe 76128, Germany

M. Hettel

W. Leuckel

N. Zarzalis

Engler-Bunte-Institut,
Universität Karlsruhe (TH),
Karlsruhe 76128, Germany

Evaluation of Advanced Two-Phase Flow and Combustion Models for Predicting Low Emission Combustors

The development of low-emission aero-engine combustors strongly depends on the availability of accurate and efficient numerical models. The prediction of the interaction between two-phase flow and chemical combustion is one of the major objectives of the simulation of combustor flows. In this paper, predictions of a swirl stabilized model combustor are compared to experimental data. The computational method is based on an Eulerian two-phase model in conjunction with an eddy dissipation (ED) and a presumed-shape-PDF (JPDF) combustion model. The combination of an Eulerian two-phase model with a JPDF combustion model is a novelty. It was found to give good agreement to the experimental data. [DOI: 10.1115/1.1377010]

Introduction

Modern gas turbines are being designed with respect to high efficiency and a particular emphasis on low emissions according to strict limitations imposed for environmental reasons. In order to meet these requirements, advanced combustion concepts like lean-premix-prevaporize (LPP) combustion and rich-quench-lean (RQL) combustion are being introduced. These two technologies are based on the strategy of avoiding global as well as local stoichiometric conditions and, thus, keeping the thermal NO_x production on a low level. Due to the enormous increase in computing performance, computational fluid dynamics (CFD) becomes an important tool for the development of advanced combustor technologies.

Key aspects for correctly predicting thermal NO_x are the models representing turbulent swirling flows, spray propagation and evaporation as well as chemical reactions.

In the present study, a modified version of the standard $k-\epsilon$ (MKE) model was used to represent the turbulent swirling flow. The MKE model, developed at the Engler-Bunte-Institut, is known to improve the prediction of the circumferential velocity in swirling flows. It will be discussed subsequently.

For the prediction of the mixture formation governed by the evaporating fuel spray either Lagrangian or Eulerian methods are usually applied. Within both methods the gaseous phase is calculated by solving the Reynolds equations with a standard discretization method for instance with a finite volume method (FVM). In the framework of the Lagrangian method, the spray is represented by a superposition of a large number of representative droplet trajectories. The movement and evaporation of each droplet is described by a set of ordinary differential equations. These equa-

tions are solved by a numerical procedure different to that of the gaseous phase. To account for the interaction between the gaseous phase and the spray, several iterations of alternating solutions of the gaseous phase and the spray have to be conducted. Therefore, the computational effort for strongly interacting two-phase flows with the Lagrangian method is rather high. However, within the Eulerian method the spray is considered as a continuum which is represented analogously to single-phase flows by a set of transport equations. This description allows the gaseous phase and the spray to be discretized by the same method and, therefore, to be solved by the same numerical procedure. This approach promises to be more efficiently in terms of computational effort when predicting strongly interacting two-phase flows. In this study, a Eulerian model developed recently at the Institut für Thermische Strömungsmaschinen is being applied.

For the prediction of heat release, two different reaction models have been used. In the first stage, the eddy dissipation model which is valid for "mixed=burned" flame regimes was used. As swirling flames usually comprise highly turbulent flows, the interaction of turbulence and chemical reactions has to be considered carefully. Due to computational limitations, the chemistry model has to be restricted to very few reaction progress variables. Therefore, in the second stage a presumed-shape-PDF (JPDF) model, developed at the Engler-Bunte-Institut, was applied. This method relies on the local stoichiometry and one reaction progress variable (normalized temperature) for calculating the local heat release.

In the present study, predictions of a swirl stabilized model combustor operating with liquid fuel are presented. The MKE model is employed for the calculation of the turbulent swirling flow, the Eulerian model for the representation of the spray and either the ED or the JPDF model for the prediction of the heat release. All models were implemented into a commercial CFD code based on an algebraic multigrid FVM. The results will be discussed with respect to the effects imposed by the different models and compared to measurements conducted at the Institut für Thermische Strömungsmaschinen.

¹To whom all 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, Munich, Germany, May 8–11, 2000; Paper 00-GT-133. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

The MKE Turbulence Model

It is well known that the standard $k-\varepsilon$ (STKE) model fails to predict the velocity field of swirling flows in many cases, particularly in confined geometries. The main reason is, that the Boussinesq hypothesis is prone to overpredict the tangential shear stress component $\overline{u'v'}$, leading to circumferential profiles of the forced vortex type. In the calculations presented here, the MKE model developed by C. Hirsch [1] was used. He analyzed results of algebraic-stress model (ASM), Reynolds-stress model (RSM) and $k-\varepsilon$ model calculations in comparison with detailed turbulence measurements in strongly swirling flows and derived a correction for the $k-\varepsilon$ model, which was generalized subsequently for flows with significant streamline curvature. The MKE model has been applied at several occasions and its accuracy was found to be nearly as good as RSMs but with the advantage of greater numerical stability and requiring only slightly more computational time than the STKE model.

Eulerian Two-Phase Model

The Eulerian method is based on the assumption that the spray can be represented by an additional continuum penetrating the gaseous phase. For this continuum approach, equations for conservation of mass, momentum, and energy can be derived ([2]). These equations are coupled with the Reynolds equations of the gaseous phase by interfacial exchange terms. At the Institut für Thermische Strömungsmaschinen a Eulerian model for polydisperse, evaporating sprays was developed. A detailed description of this model is given in ([3]). In the frame of the Eulerian model, a polydisperse spray is represented by several droplet classes which are necessary to account for the different droplet starting conditions imposed by the atomization process. Each droplet class is described by the following set of transport equations:

$$\frac{\partial}{\partial t} \alpha_k \rho_k + \frac{\partial}{\partial x_j} \alpha_k \rho_k U_{k,j} = \frac{\partial}{\partial x_j} \left(\frac{\mu_{t,k}}{Sc_{t,d}} \frac{\alpha_k}{\partial x_j} \right) + \frac{6\alpha_k}{\pi D_k^3} \dot{m}_{\text{vap},k} \quad (1)$$

$$\begin{aligned} & \frac{\partial}{\partial t} \alpha_k \rho_k U_{k,i} + \frac{\partial}{\partial x_j} \alpha_k \rho_k U_{k,j} U_{k,i} \\ &= \frac{\partial}{\partial x_j} \left[\alpha_k \mu_{t,k} \left(\frac{\partial U_{k,i}}{\partial x_j} + \frac{\partial U_{k,j}}{\partial x_i} \right) \right] + \alpha_k \rho_k f_i - \alpha_k \frac{\partial}{\partial x_i} P_g \\ &+ \frac{3}{4} \rho_{g,\text{ref}} \frac{\alpha_k}{D_k} c_{w,k} |U_g - U_k| \cdot (U_{j,g} - U_{j,k}) \\ &+ \frac{6\alpha_k}{\pi D_k^3} \dot{m}_{\text{vap},k} U_{j,k} \end{aligned} \quad (2)$$

$$\begin{aligned} & \frac{\partial}{\partial t} \alpha_k \rho_k h_k + \frac{\partial}{\partial x_j} \alpha_k \rho_k U_{j,k} h_k \\ &= \frac{\partial}{\partial x_j} \left(\alpha_k \frac{\mu_{t,k}}{Pr_{t,d}} \frac{\partial h_k}{\partial x_j} \right) + \frac{3}{4} \frac{\alpha_k}{D_k} h t_k (T_{g,s} - T_{g,\infty}) \\ &+ \frac{6\alpha_k}{\pi D_k^3} \dot{m}_{\text{vap},k} h_{v,s} \end{aligned} \quad (3)$$

$$\frac{\partial}{\partial t} \alpha_k \rho_k D_k + \frac{\partial}{\partial x_j} \alpha_k \rho_k U_{j,k} D_k = \frac{\partial}{\partial x_j} \left(\alpha_k \frac{\mu_{t,k}}{Pr_{t,d}} \frac{\partial D_k}{\partial x_j} \right) + \frac{8\alpha_k}{\pi D_k^2} \dot{m}_{\text{vap},k} \quad (4)$$

The liquid volume concentration α_k , the velocity U_k , the enthalpy h_k , and the diameter D_k are transport quantities. The evaporation rate $\dot{m}_{\text{vap},k}$ and the heat transfer coefficient $h t_k$ of the droplets are calculated on the basis of the uniform temperature (UT) evaporation model ([4]). For a three-dimensional simulation $nc \cdot 6$ (nc =number of droplet classes) transport equations are necessary to represent the polydisperse spray. The whole set of transport equations can be recasted into a universal formulation

and discretized by a conservative FVM. The information exchange between the transport equations of the gaseous phase and the spray can be realized on the level of the nonlinear iteration being a part of the FVM. As a consequence, the convergence behavior of the Eulerian method is generally better compared to the Lagrangian method.

Turbulent Reaction Models

For the modeling of heat release two combustion models have been used.

Eddy Dissipation Model. According to Lilleheie et al. [5] the fuel consumption rate is calculated by

$$\overline{R}_{ED} = 24.3 \cdot \text{Re}_t^{-1/4} \cdot \overline{\rho} \cdot \frac{\varepsilon}{k} \cdot \min(\overline{\sigma}_{\text{fuel}}, \overline{\sigma}_{\text{oxygen}}) \quad (5)$$

where $\text{Re}_t = k^2/(\nu\varepsilon)$. This approach implies that the typical time scale of heat release t_c is much smaller than the turbulent time scale $t_t = k/\varepsilon$, leading to Damköhler numbers $\text{Da} = t_t/t_c \gg 1$.

Joint-PDF Model. Experimental investigations indicate, that in swirling flows t_c is of the same magnitude as t_t , leading to Damköhler numbers $\text{Da} \leq 1$. The numerical modeling of highly turbulent reacting flows requires detailed consideration of the statistical correlations between temperature and species concentrations since the time-averaged reaction rates are extremely sensitive to the level of the fluctuations of these quantities. Therefore, a *joint-probability density function (JPDF) model*, was used. It was developed by M. Philipp [6] and is based on presumed shape distributions (clipped Gaussian) for the PDFs. The JPDF is constructed by means of two statistical moments (mean value and variance) of two variables, one describing the local stoichiometry (mixture fraction f), the other representing the reaction progress (normalized temperature $\Theta = T/T_{\text{adiabatic}}$). For these statistical moments (f, f', Θ, Θ') four additional transport equations have to be solved. The calculation of the time-averaged reaction rate R_{JPDF} for each control volume of the calculation domain is described following. First, the two single PDFs for the variables f and Θ are constructed using the corresponding mean values and variances. Then the f -PDF is divided in i discrete intervals Δf_i , the Θ -PDF in j intervals $\Delta \Theta_j$ within the range of values $[0-1]$ for f and Θ . The probability of appearance of an arbitrary combination of f and Θ is represented by the $\text{JPDF}(f, \Theta)$. Assuming statistical independence of reaction progress from mixture fraction, the $\text{JPDF}(f, \Theta)$ can be yielded by simply multiplying the single PDFs.

$$\text{JPDF}(f, \Theta) = \text{PDF}(f) \cdot \text{PDF}(\Theta) \quad (6)$$

The chemical rate for every point of the JPDF is calculated with a global one-step mechanism following Westbrook and Dryer [7]. Integration of the JPDF weighted rates over the intervals yields the time-averaged reaction rates for each of the finite volumes.

$$\begin{aligned} \overline{R}_{\text{JPDF}} &= \sum_1^i \sum_1^j 2.12 \cdot 10^{11} \cdot \sigma_{\text{CH}_4}^{0.2}(f_i, \Theta_{i,j}) \cdot \sigma_{\text{O}_2}^{1.3}(f_i, \Theta_{i,j}) \\ &\cdot \rho_{i,j}^{1.5} \exp\left(\frac{-T_{\text{act}}}{T_{i,j}}\right) \cdot \text{JPDF}(f_i, \Theta_{i,j}) \cdot \Delta f_i \cdot \Delta \Theta_{i,j} \end{aligned} \quad (7)$$

The temperature $T_{i,j}$ is calculated from rescaling the normalized temperature Θ by $T = \Theta T_{\text{adiabatic}}$, where $T_{\text{adiabatic}}$ is the adiabatic temperature for stoichiometric mixture.

Simulation of a Low-NO_x Combustor

The performance and quality of different models described previously was studied by predicting the reacting flow field of a swirl stabilized combustor. The combustor chosen for this study was extensively investigated by means of optical measurements techniques at the Institut für Thermische Strömungsmaschinen ([8]) as a part of the German research program Engine3E. A detailed

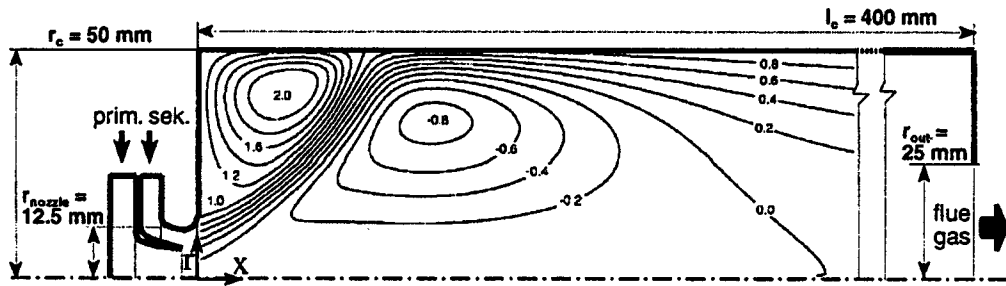


Fig. 1 Combustor geometry

description of the test rig is given in ([8,9]). The geometry of the combustor section of interest is presented together with calculated streamlines in Fig. 1. A blow up of the nozzle section is shown in Fig. 2. The duct consists of an radial swirl cup design dividing the primary air into two co-rotating air flows. A pressure swirl atomizer is mounted at the center of the swirl cup and supplies fuel to the atomizer lip (cf. Fig. 2). The swirl cup is attached to a cylindrical duct where the combustion takes place. The outlet of the duct is contracted by a ring with a inner diameter of $r_{out}=25$ mm confining the axial extension of the central recirculation zone. The combustor operates at atmospheric conditions and with a global equivalence ratio of $\Phi_{global}=0.5$. The parameters of the operating point investigated in this study are presented in Table 1. $S_o = \dot{D}/(lr_{nozzle})$ is the theoretical swirl number. The computational grid consists of $68 \times 74 \times 3$ cells and was found to be sufficiently fine.

Six different calculations were carried out to investigate the spray model as well as the combustion models. For the two-phase calculations dodecane was used to represent the fuel. For the fuel vapor the properties of CH_4 were used. To study the Eulerian spray model separately from the combustion models an isothermal (640 K) two-phase calculation was compared with a single-phase calculation where CH_4 was injected through a thin slot in the atomizer lip. In a second step the ED and the JPDF combustion model were used to predict the chemical reaction. These calculations were again carried out by using the Eulerian model as well as a direct CH_4 injection at the atomizer edge. The run-time for the Eulerian simulations was about four times larger compared to the single-phase calculations. For all calculations the MKE turbulence model was used to increase accuracy.

Discretization of the Spray. For the spray predictions, initial droplet conditions resulting from the atomization process have to be defined. Based on PDPA data recorded at the atomizer edge the following Rosin-Rammler distribution was determined:

$$\frac{\dot{V}_i}{\dot{V}} = 1 - e^{-0.6931(D/16.3 \mu m)^{3.0}} \quad (8)$$

Since this volume flux is confined to a rather narrow droplet diameter range, it was sufficient to discretize it into three diameter classes. In Fig. 3, a laser-light-sheet visualization of the spray behind the atomizer is shown. It can be seen that the dispersion of the spray is intensified due to a statistical motion imposed by the atomization process, affecting significantly the dispersion of the large droplets due to their inertia. Therefore, the largest droplet diameter class was split up into three direction classes each carrying one third of the volume flux. For all five droplet classes the same axial velocity of 50.0 m/s was chosen based on PDPA data.

Assessment of the Eulerian Model. In Fig. 3 the predicted equivalence ratio fields for the isothermal case are shown. The single-phase calculation refers to a virtual injection of CH_4 at the atomizer edge. It can be seen that the equivalence ratio predicted by the two approaches differs most in the area downstream of the atomizer edge. Contrary to the single-phase calculation, rather low equivalence ratios (Φ is everywhere below 0.52) are predicted by the Eulerian model. These differences are caused by the delayed onset of the evaporation of the droplets which is accounted for by the Eulerian model. Therefore, the more precise prediction of the equivalence ratio by the two phase model is required essentially for the simulation of the chemical reaction. Additionally, the mean trajectories of the five droplet classes are presented in Fig. 3. The widely spread trajectories of the large diameter classes reveal that the introduction of the directional droplet classes the fan-spreading effect ([10]) of the large droplets could be captured accurately by the Eulerian method.

In Fig. 4, the predicted liquid volume concentration field $\alpha = \sum_{k=1}^5 \alpha_k$ of the JPDF calculation is compared to a visualization of the spray. It can be seen that the zones where a high liquid fuel concentration is predicted correlate well with the high intensity

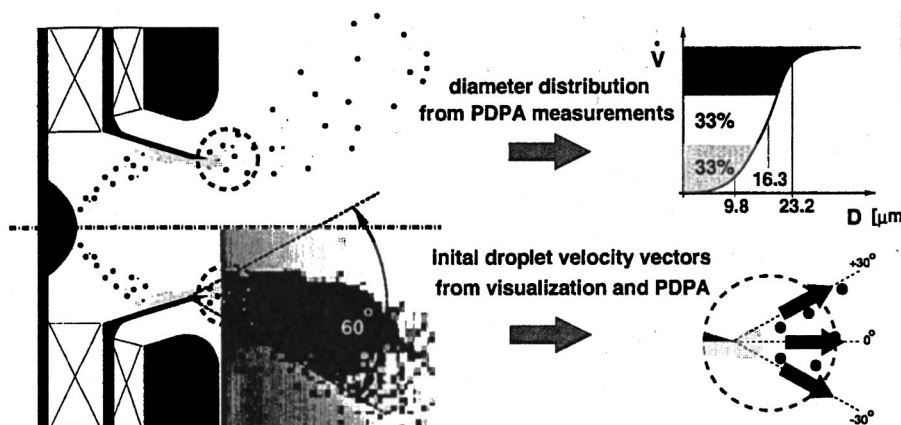


Fig. 2 Droplet initial conditions

Table 1 Parameters at the inlet

Gas Flow (Air)		Fuel (JetA4)	
$\dot{m}_0$	17.6 g/s	$\dot{m}_0$	0.599 g/s
T_0	640 K	T_0	380 K
p	1 bar		
Tu	10 percent		
$S_{o,prim}$	0.45		
$S_{o,sek}$	0.85		

(black) zones of the visualization. Therefore, the shape and the position of the spray prediction is in a good agreement with the measurements.

Assessment of the Combustion Models. Figure 5 shows the results of the calculations with gaseous fuel (one-phase flow), Fig. 6 the results with liquid fuel (two-phase flow), each showing a comparison of results, obtained when applying different combustion models. In both figures the equivalence ratio Φ , the fuel consumption rate $\dot{R}$ and the Temperature T are shown. In each plot $r=0$ corresponds to the symmetry axis and $x=0$ to the inlet of the combustion chamber. The end of the atomizer lip (ref. Fig. 1) is located at $x = -3$ mm and $r = 9$ mm.

The calculation with gaseous fuel and the ED model (Fig. 5) shows a reaction zone attached directly to the atomizer lip. This is in contradiction to the experimental results with gaseous fuel, where the reaction zone was found to be detached from the atomizer lip. The highest calculated fuel consumption rates in this case can be found in the shear layer between the fuel stream and the surrounding air. The temperature rises up to 2640 K which corresponds to the adiabatic temperature for stoichiometric combustion. According to this high temperature, the strong expansion

leads to large axial velocities. This results in a strong decrease of the swirl number S and subsequently to a smaller opening of the jet angle as well as to a delayed mixing in comparison to the isothermal calculation (ref. Fig. 3).

In contrast, the calculation with the JPDF model predicts a detached flame (Fig. 5) as can be seen in experiments. This is due to the fact that the fuel consumption rates are restricted by the limited kinetic time scale. The maximum temperature now is 1940 K, which corresponds to the adiabatic temperature for $\Phi_{global} = 0.5$. The opening of the jet angle is slightly smaller compared to the isothermal calculation and the mixture field is nearly unaffected by combustion (ref. Fig. 3). The calculated Damköhler number Da shows values below 0.1 in the whole reaction zone, indicating that a combustion model for calculating turbulent swirling flames has to take into account the limitation of the turbulent reaction rates due to chemical kinetics.

The calculation with liquid fuel injection using the ED model shows an attached reaction zone (Fig. 6), having a shape similar to the calculation with gaseous fuel and the ED model (ref. Fig. 5). In the attached reaction zone the temperature rises up to 2300 K, leading to large fuel evaporation rates and in addition with the smaller jet opening angle to a mixing field that is also very similar to the calculation with gaseous fuel and the ED model. In contrast the experimental results with liquid fuel, show a detached reaction zone (ref. visualization in Fig. 4).

According to the experiments, the calculation with liquid fuel injection applying the JPDF model (Fig. 6) yields a detached reaction zone. The shape and the axial and radial position of the turbulent flame front is in good agreement with the visualization (ref. Fig. 4). The distribution of Φ shows values up to 0.7, leading to a small region with high temperatures in the shear layer between the forward flow and the outer recirculation zone. The results indicate again the guidance of the used JPDF-model in com-

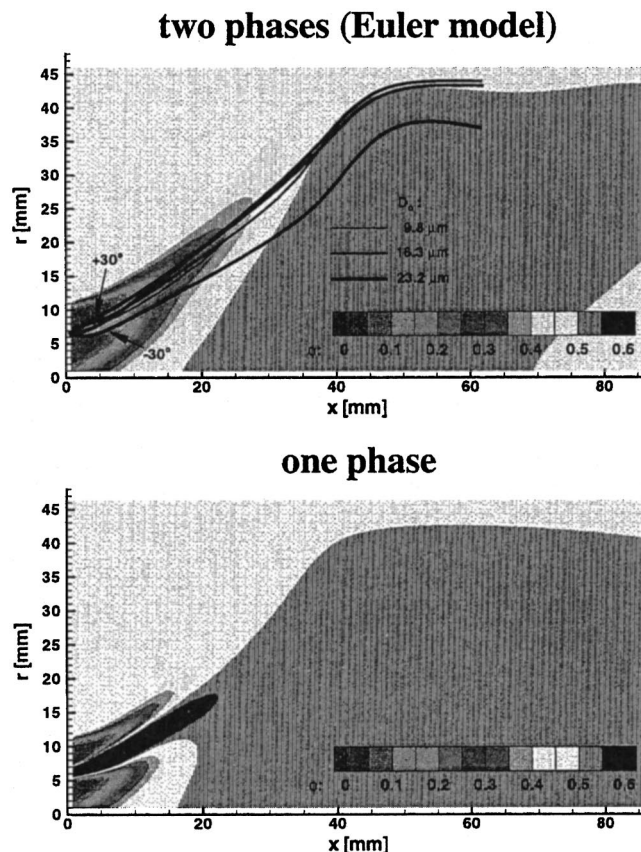


Fig. 3 Equivalence ratio fields (isothermal)

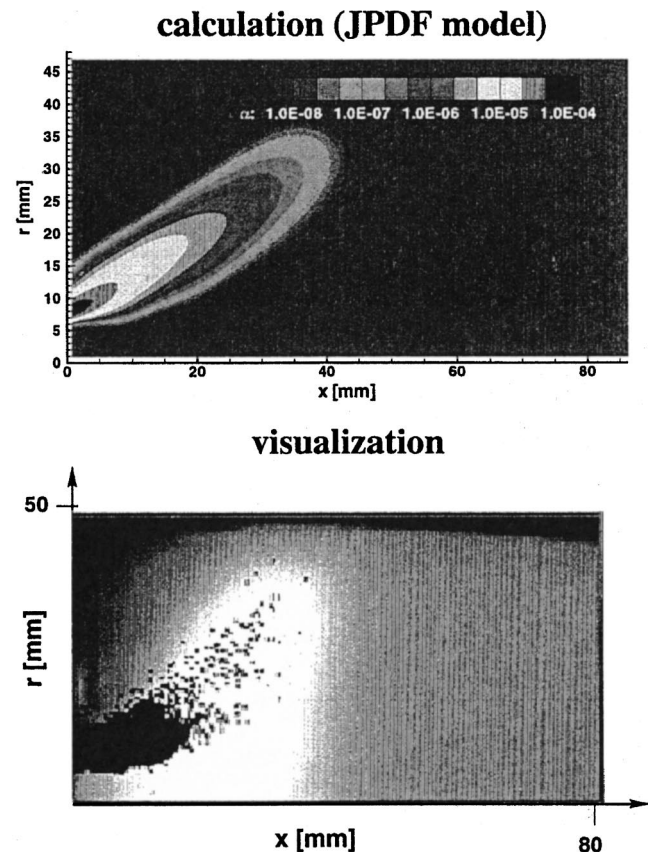
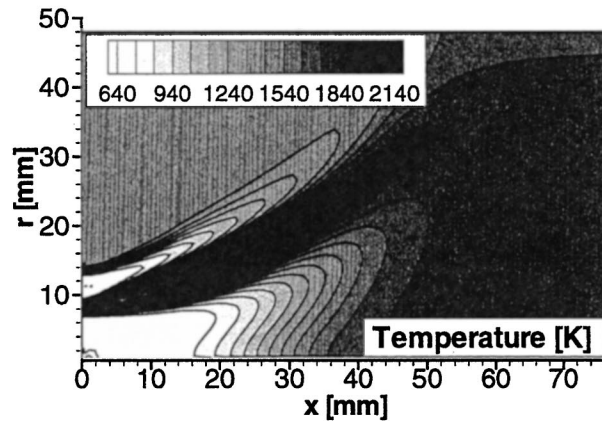
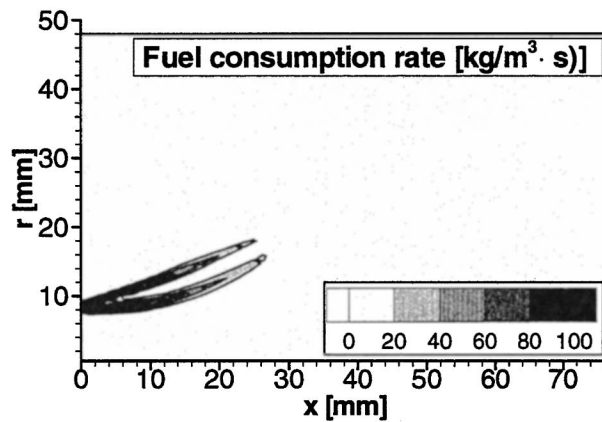
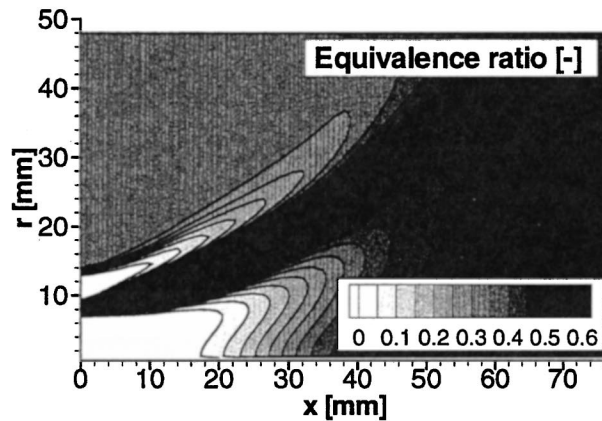


Fig. 4 Droplet concentration field

single phase

ED model



JPDF model

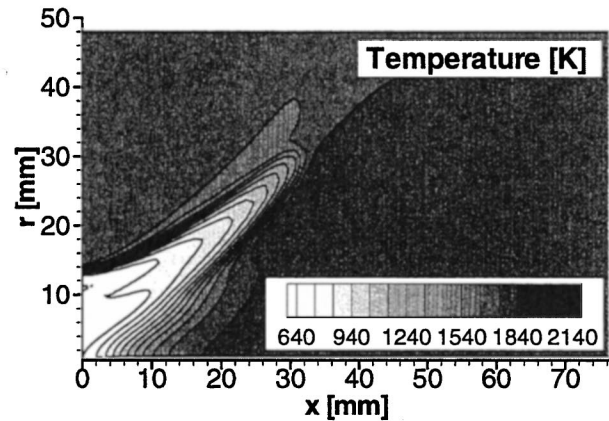
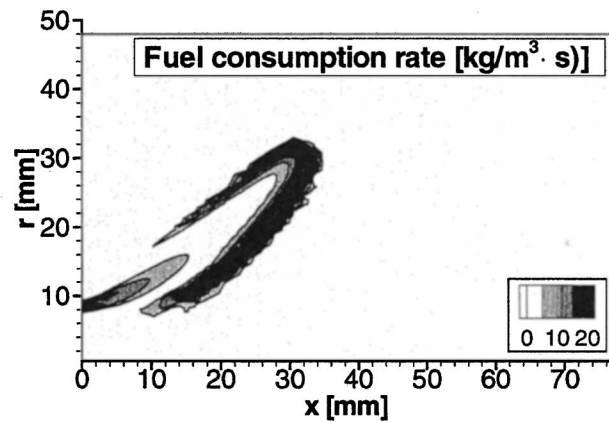
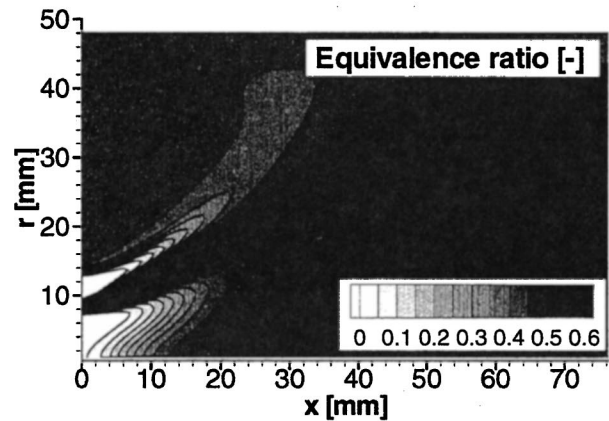


Fig. 5 Combustion results

parison to the the ED-model and the necessity to include to imply chemical kinetics in a combustion model which is used to calculate highly turbulent flames.

Conclusion

Liquid fuel spray flames are characterized by a strong interaction between the propagation and evaporation of the spray as well as the chemical reaction. Since the time scales of the turbulent

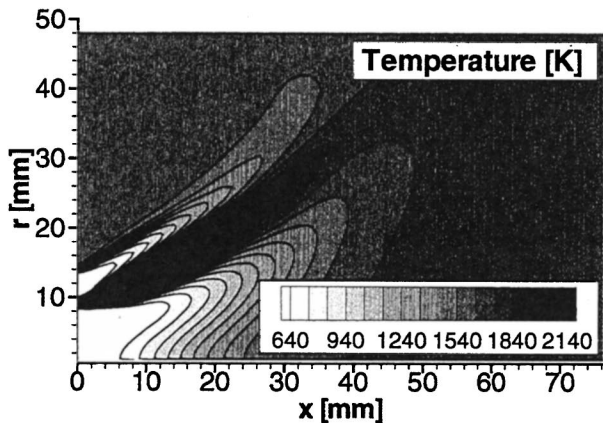
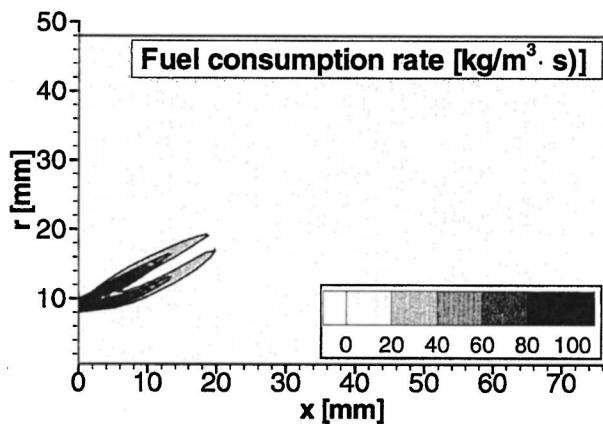
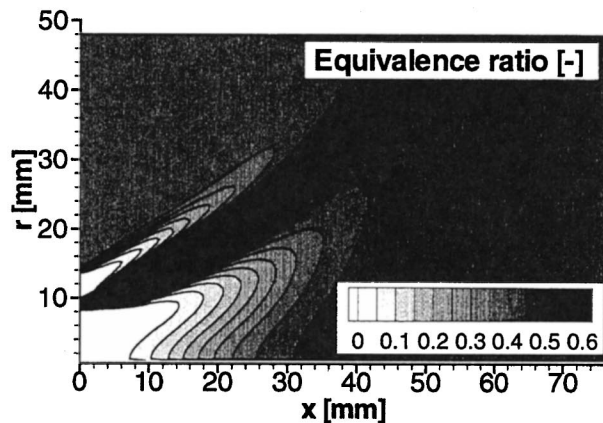
flow field, the mixture formation process and the chemical reaction are often of the same magnitude, all those physical effects must be accounted for by the numerical simulation.

In this study, the accuracy of an Eulerian two-phase model in combination with an eddy dissipation (ED) model and a joint-PDF (JPDF) combustion model was investigated.

In a first step, predictions of the evaporating spray without the reacting flow field were compared with single-phase calculations of the mixing process of CH_4 . These investigations revealed that

Euler model

ED model



JPDF model

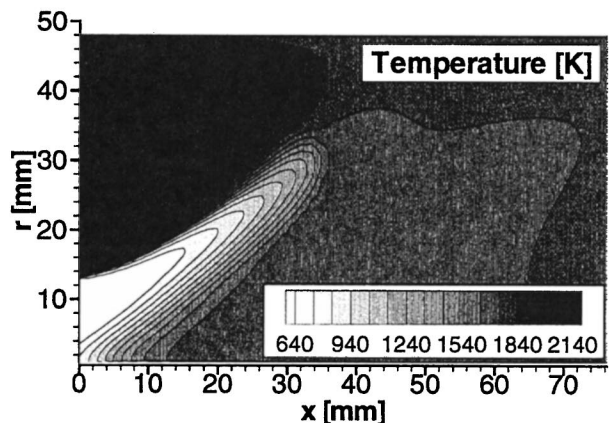
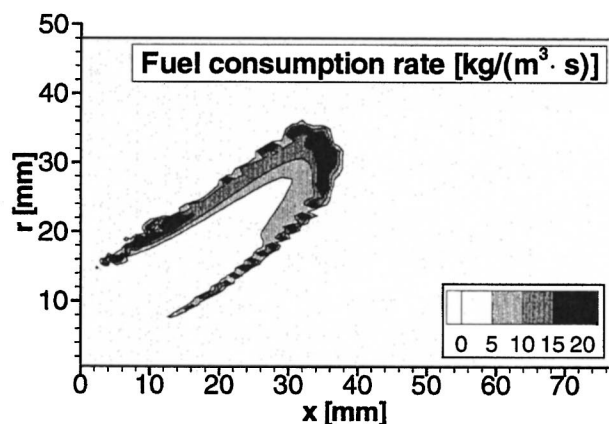
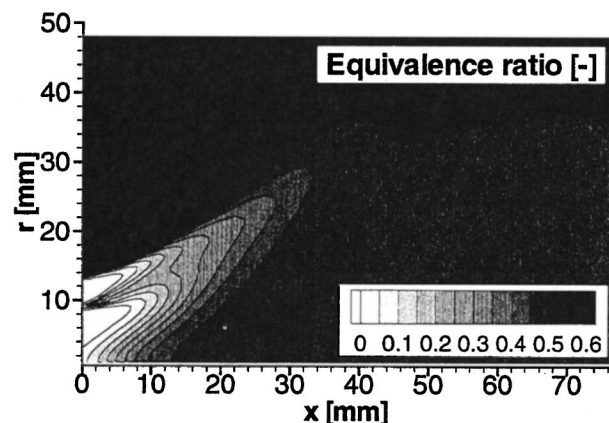


Fig. 6 Combustion results

droplet dispersion and evaporation is the key mechanism which determines the distribution of the equivalence ratio within the combustor properly.

In a second step, calculations of the reacting flow field were made for both gaseous and liquid fuel. For the liquid fuel flame the Eulerian two-phase model was used.

In each case, the deficit of the ED model becomes apparent, leading to an attached reaction zone with unrealistic high fuel consumption rates. This is in contradiction to the experiments where a detached reaction zone can be seen.

The calculations based on the JPDF model yield, in agreement to the experiments, detached reaction zones. The comparison of the calculation with liquid fuel with the visualization of the flame shows a good coincidence of the shape and the spatial position of the turbulent reaction zone. This is due to the fact that the limitation of the fuel consumption rate by the kinetic time scale is implied by the JPDF model. This is essential for calculating high turbulent swirl flames.

Finally it should be pointed out that the modified $k-\varepsilon$ model, the

Eulerian two-phase model, and the JPDF combustion model used in combination for the first time yielded good results and showed a numerical stable behavior.

Acknowledgment

This work has been founded by the Department of Education, Science, Research and Technology of Germany under contract No. 20T9540. The support is gratefully acknowledged.

Nomenclature

D	= droplet diameter
Da	= Damköhler number
$\dot{D}$	= tangential momentum flux
f	= body force
f	= mixture fraction
h	= enthalpy
$\dot{I}$	= axial momentum flux
ht	= heat transfer coefficient
k	= turbulent kinetic energy
L	= length scale
$\dot{m}$	= mass flux
P	= pressure
P	= probability density
Pr	= Prandtl number
R	= fuel consumption rate
r	= radius
S	= swirl number
Sc	= Schmidt number
T	= temperature
Tu	= degree of turbulence
U	= velocity component
x	= axial distance
Y	= mass fraction

Greek Symbols

α	= liquid volume fraction
ε	= dissipation rate of k
Γ	= diffusion coefficient
μ	= dynamic viscosity
Φ	= equivalence ratio
ρ	= density
σ	= species concentration
Θ	= normalized temperature

Subscripts

0	= initial state
o	= theoretical
act	= activation
c	= chemical
g, d	= gas, droplet
t	= turbulent
vap	= vapor
k	= droplet class

Superscripts

'	= fluctuation of a value
—	= time-mean value

References

- [1] Hirsch, C., and Leuckel, W., 1996, "Curvature for the k - ε Model for Engineering Applications," *Engineering, Turbulence Modeling and Experiments* 3, W. Rodi and G. Bargeles, eds., pp. 71–80.
- [2] Ishii, M., 1975, *Thermo-Fluid Dynamic Theory of Two Phase Flow*, Eyrolles, 1975.
- [3] Schmehl, R., Klose, G., Maier, G., and Wittig, S., 1998, "Efficient Numerical Calculation of Evaporating Sprays in Combustion Chamber Flows," *92nd Symp. on Gas Turbine Combustion, Emissions and Alternative Fuels, RTO Meeting Proceedings 14*.
- [4] Schmehl, R., Klose, G., Koch, R., and Wittig, S., 1998, "Droplet Evaporation and Transport," Technical Report, BRITE/EURAM Low NO_x Phase III, Institut für Thermische Strömungsmaschinen, Universität Karlsruhe (TH).
- [5] Lilleheie, N. I., Byggstøl, S., and Magnussen, B. F., 1989, "Calculations of a Methane Turbulent Diffusion Flame Using Detailed Chemical Kinetics in the Eddy Dissipation Concept," Technical Report, SINTEF/Norwegian Institute of Technology Division.
- [6] Philipp, M., Hoffmann, S., Habisreuther, P., Lenze, B., and Eickhoff, H., 1992, "Experimental and Numerical Study Concerning Stabilization of Strongly Swirling Premixed and Nonpremixed Flames," *24. Symp. (Int.) on Combustion*, The Combustion Institute, Pittsburgh, PA, pp. 361–368.
- [7] Westbrook, C. K., and Dryer, F. L., 1981, "Simplified Reaction Mechanisms for the Oxidation of Hydrocarbon Fuels in Flames," *Combust. Sci. Technol.*, 27.
- [8] Willmann, M., Meier, R., and Wittig, S., 1997, "Visualization of the Instationary Atomization and Spray Propagation Inside a Reacting Model-Gasturbine Combustor," *13th Annual Conference on Liquid Atomization and Spray Systems*, July 9–11, Florence.
- [9] Meier, R., Merkle, K., Maier, G., Zarzalís, N., Leuckel, W., and Wittig, S., 1999, "Development of an Improved Prefilming Airblast Atomizer for Gasturbine Application," *15th Annual Conference on Liquid Atomization and Spray Systems*, Toulouse, July 5–7.
- [10] Hardalupas, Y., Taylor, A. M. K. P., and Whitelaw, J. H., 1990, "Velocity and Size Characteristics of Liquid Fuelled Flames Stabilized by a Swirl Burner," *Proc. R. Soc. London, Ser. A*, **A428**.

Response of a Model Gas Turbine Combustor to Variation in Gaseous Fuel Composition

R. M. Flores

M. M. Miyasato

V. G. McDonell

G. S. Samuelsen¹

e-mail: gss@uci.edu

UCI Combustion Laboratory,
University of California,
Irvine, CA 92697-3550

The effect of fuel composition on performance is evaluated on a model gas turbine combustor designed to mimic key features of practical devices. A flexible fuel injection system is utilized to control the placement of the fuel in the device to allow exploration and evaluation of fuel distribution effects in addition to chemistry effects. Gas blends reflecting the extremes in compositions found in the U.S. are considered. The results illustrate that, for the conditions and configuration studied, both fuel chemistry and fuel air mixing play a role in the performance of the device. While chemistry appears to be the predominant factor in stability, a role is noted in emissions performance as well. It is also found that changes in fuel distribution associated with changes in fuel momentum for fixed firing rate also have an impact on emissions. For the system considered, a strategy for sustaining optimal performance while fuel composition changes is illustrated.

[DOI: 10.1115/1.1377011]

Motivation

Gas turbines have been developed to operate on a wide variety of fuels and the significant role of the fuel composition in both operability and emissions have been noted (e.g., [1]). Lean-premixed combustion is increasingly becoming the method of choice for reducing pollutant emissions from natural gas-fired gas turbine power plants. However, as these systems operate closer to the edge of combustion stability to reduce emissions of NO_x, modest upsets in operating conditions, including those due to variation in fuel composition, are becoming intolerable.

Typical constituent concentrations for natural gas within the United States average 93.9 percent methane, with ethane, propane, and higher hydrocarbons rounding out much of the remaining composition at 3.2, 0.7, and 0.1 percent, respectively. However, methane, ethane, and propane compositions can approach values of 74.5, 13.3, and 23.7 percent, respectively ([2]).

This wide range of compositional variability in the U.S. natural gas supply poses a significant challenge for the development of combustors that can maintain optimal performance despite a wide range of natural gas compositions. As a result, a need for the systematic study of the effects of fuel composition on the performance of combustion systems is required. In addition, delineation of the key phenomena occurring is required in order to develop combustor designs that are insensitive to variations in fuel composition.

Objectives

The objectives of the current study are to (1) delineate the effects of fuel composition on the overall performance of a model combustor and (2) relate the variation in performance to fuel injection effects (i.e., mixing) and/or reaction characteristics (i.e., chemical kinetics).

Approach

The approach taken is to develop and apply a model combustor with characteristics similar to those found in practice. A flexible

fuel injection system based on radial jets is employed in order to provide control over the fuel distribution entering the combustor. Performance maps based on CO, NO_x, and lean blow off (LBO) are generated as a function of fuel type and inlet fuel distribution. The fuel distribution is characterized at the inlet plane using an extractive probe to correlate the performance of the combustor.

Experiment

Test Facility. The test facility utilized provides a wide range of operating conditions and flow metering. The test stand is designed to operate at 1 atm with inlet temperatures up to 800 K. The model combustor test rig, shown in Fig. 1, is attached to a three-dimensional traverse system which allows the combustor to be moved as necessary to map out points both within and at the exit plane.

Model Combustor. The model combustor utilized is shown schematically in Fig. 2. Figure 3 shows the fuel injection options available. The fuel injection strategy involves a multipoint approach with the ability to control fuel flow splits between three independent fuel injection circuits. One available fuel injection option is to inject fuel radially from the centerbody into the swirl-

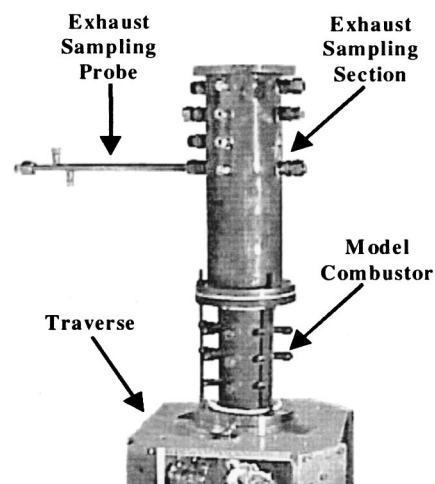


Fig. 1 Atmospheric test facility

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, Munich, Germany, May 8–11, 2000; Paper 00-GT-141. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

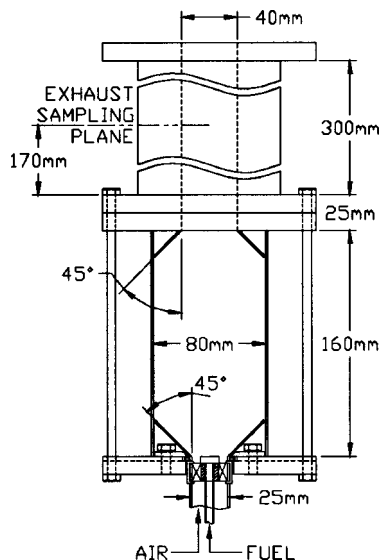


Fig. 2 Model combustor

ing air stream. This centerbody injection circuit, labeled “CB” in Fig. 3, consists of six equally spaced fuel holes located circumferentially along the side of the fuel manifold. The fuel manifold is mounted to the centerbody tip, and is positioned directly against the swirler hub. The second fuel injection option is to inject the fuel radially from the surrounding wall. This wall injection circuit, labeled “WJ” in Fig. 3, consists of six equally spaced fuel holes in positions that are staggered with respect to the centerbody injection holes. The final fuel injection option is to inject the fuel axially into the air stream. This axial injection circuit, labeled “PILOT” in Fig. 3, consists of a single fuel injection hole located at the tip of the centerbody.

The baseline configuration is a four vane axial swirler. The nominal firing rate for the system is 15 kW at 0.0093 kg/sec of air, though the fuel flow rates were varied to assess the system’s performance. An air inlet temperature of 700 K was utilized in all cases.

Diagnostics. Exhaust emissions of carbon monoxide (CO), carbon dioxide (CO₂), hydrocarbons (HC), oxygen (O₂), and nitrogen oxides (NO_x) were measured using Horiba Ltd. analyzers. These instruments are part of an integrated sampling and computer data acquisition system.

A 12.7-mm o.d. water-cooled, stainless steel bulk emissions probe is used to sample the exhaust emissions downstream of the exit plane of the combustor as shown in Fig. 1. This probe is designed to take an integrated average measurement of the emissions over the diameter of the sampling plane. The water in the

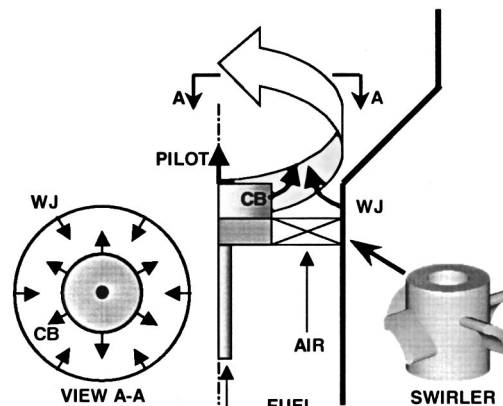


Fig. 3 Detail of fuel injection options

probe is heated to 325 K to protect the probe and quench the sample while avoiding condensation of water vapor in the sample. The emissions are pumped through a Teflon line heated to 408 K to prevent water condensation, and the sample is then split into two streams. The NO_x stream goes through a converter to reduce any NO₂ to NO prior to the water drop out.

All of the emissions measurements from the analyzers are recorded using a digital data acquisition system. The measurements from the analyzers are sampled over a 20 second period at a rate of 5 Hz. One hundred samples from each analyzer are averaged and recorded in a data file. The residence time associated with the gases flowing through the combustor and to the exhaust sample probe is 24.8 msec. The reference velocity (based on nonreacting, nonpreheated conditions) for the model combustor is 1.55 m/s.

The nonreacting fuel distribution at the swirler exit was mapped with a 12.7 mm o.d. stainless steel probe and a high range flame ionization detector (Horiba Model FIA-236-1).

Fuel Blending. For the current study, a subset of a comprehensive fuel blending system was utilized as illustrated in Fig. 4. Natural gas, ethane, and propane can be combined in any desired combination using this system. A series of Brooks mass flow controllers are utilized in conjunction with a LabView based control program which allows the user to set fuel composition and flow splits as desired. The overall accuracy of the blended fuel flow rate is ± 2 percent.

Noteworthy is the use of natural gas as a baseline fuel. In order to account for any compositional variations in the natural gas, a dedicated on-line gas composition analyzer is utilized to provide baseline concentrations.

Table 1 summarizes the typical natural gas composition utilized. Fortunately, methane comprises the vast majority of the natural gas. As a result, by studying the effect of adding 15 per-

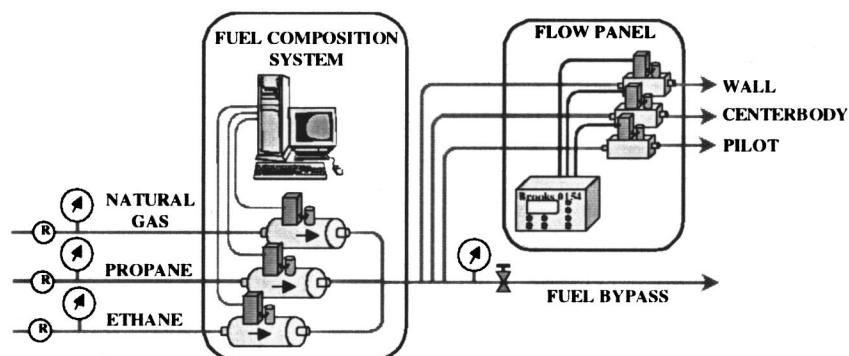


Fig. 4 Fuel blending system

Table 1 UCICL natural gas composition

n	Constituent		MF y(i) percent
1	Methane	CH ₄	96.975
2	Ethane	C ₂ H ₆	0.982
3	Propane	C ₃ H ₈	0.109
4	iso-Butane	C ₄ H ₁₀	0.014
5	n-Butane	C ₄ H ₁₀	0.015
6	iso-Pentane	C ₅ H ₁₂	0.004
7	n-Pentane	C ₅ H ₁₂	0.004
8	C6	C ₆ H ₁₄	0.001
9	C7	C ₇ H ₁₆	0.001
10	C8	C ₈ H ₁₈	0.000
11	CO ₂		1.574
12	O ₂		0.000
13	N ₂		0.322
Totals			100.00

Table 2 Gas compositions utilized

Blend	Wobbe Index* MJ/m ³	S.G. relative to air	LHV MJ/m ³
Natural Gas	44.3	0.576	33.6
85 percent Natural Gas/ 15 percent Ethane	46.8	0.645	37.6
80 percent Natural Gas/ 20 percent Propane	50.5	0.765	44.2
Propane	70.1	1.523	86.5

*Wobbe Index=LHV/[SG]^{1/2} ([1])

cent ethane or 20 percent propane by volume, the effects of the minor constituents present in the baseline natural gas are felt to carry a negligible effect relative to the results obtained.

For the present study, four fuel compositions were considered. Some of the associated properties are summarized in Table 2.

Results

Overall combustion performance is first summarized, followed by correlations with fuel distribution, and finally, fuel injection mechanisms. The combustor performance is based on the lean blow off (LBO) limits and the emissions of CO and NO_x, both of which are corrected to a fixed dilution of 15 percent O₂. In the current system, the LBO limit will depend upon both the mixing and reaction rates. As fuel composition changes, the reaction rate will change. An increased reaction rate will help extend the LBO limits. However, since a radial fuel injection strategy is utilized, and the fuel distribution depends upon the momentum ratio of the fuel jet to the swirling air, the fuel composition can also affect the jet behavior and the subsequent mixing. In addition, the NO_x levels will depend not only the local reaction temperature, which can be a function of fuel composition, but also upon differing NO_x formation mechanisms (e.g., [3,4]).

Combustor Performance

Emissions. The combustion performance for each fuel mixture was assessed via maps of NO_x and CO emissions, corrected to 15 percent O₂, as a function of injector fuel split (relative to the Pilot, Wall Jet, and/or Centerbody injectors). Figure 5 summarizes the results obtained for 100 percent natural gas with and without pilot fuel. The white regions on the lean side of each plot reflect regions beyond the LBO limits. Results are shown for NO_x emissions (Figs. 5(a) and 5(c)) and the corresponding CO emissions (Figs. 5(b) and 5(d)) as a function of overall equivalence ratio and percentage of fuel injected radially outwards from the centerbody (recall Fig. 3). Note that the fuel split between the centerbody and wall injectors is of the remaining fuel *after* the pilot fuel (if any)

is subtracted. The results illustrate several key features. First, for the cases without pilot fuel (Figs. 5(a) and 5(b)), the NO_x levels appear relatively insensitive to fuel split (percent Centerbody), while revealing typical dependency upon reaction temperature (Eq. Ratio). CO emissions reveal a “u-shaped” profile with an increase in CO under extremely lean conditions and at equivalence ratios greater than 0.47.

Figures 5(c) and 5(d) illustrate that the pilot fuel has a significant impact on the performance. Although the LBO limits are extended from $\phi=0.38$ to $\phi=0.32$, high amounts of CO and NO_x are emitted. Since the pilot fuel is injected axially into the central recirculation zone, a rich, high-temperature core with limited mixing must result to give rise to the higher NO_x and CO. As such, the extension of the LBO limit is handicapped by increased emissions.

As shown in Fig. 5, CO and NO_x emission levels often move in opposite directions. As a result, it is important to identify conditions and configurations that minimize both. This can be achieved by utilizing a “performance function.” J :

$$J = w_{\eta_c} \cdot J_{\eta_c} + w_{NO_x} \cdot J_{NO_x} \quad (1)$$

where η_c is the combustion efficiency, which is a function of CO and unburned hydrocarbon (HC) emissions, and the terms w_{η_c} and w_{NO_x} are weighting factors whose sum is equal to one. For this study, since HCs were zero for essentially all cases, the η_c represents a normalized function of CO emissions. The functions J_{η_c} and J_{NO_x} are defined as

$$J_{NO_x} = \begin{cases} 1 - 0.75 \cdot \left(\frac{[NO_x]}{5} \right)^4, & [NO_x] \leq 5 \text{ ppm} \\ (1 - 0.75) \cdot \frac{[NO_x]_{MAX} - [NO_x]}{[NO_x]_{MAX} - 5}, & [NO_x] > 5 \text{ ppm} \end{cases}$$

$$J_{\eta_c} = \begin{cases} 0, & \eta < 99.98 \text{ percent} \\ \left(\frac{\eta - \eta_{MIN}}{100 \text{ percent} - \eta_{MIN}} \right)^{1/2}, & \eta \geq 99.98 \text{ percent.} \end{cases}$$

The functions J_{η_c} and J_{NO_x} were selected to yield high values of J when the combustion efficiency is high (low CO) and the NO_x is low. From the definitions of J_{η_c} and J_{NO_x} , the values of J that may be obtained range from a minimum of zero, signifying poor performance, to a maximum of one, indicating ideal performance. For the results presented here, w_{η_c} and w_{NO_x} are equally weighted at 0.5.

With this combined CO and NO_x function, the areas on the map with optimal performance (or nonoptimal performance) can be identified and the extent to which fuel composition alters the regions of optimal performance noted.

Figure 6 shows the results from Fig. 5, presented in terms of the performance index J , with the results for three other fuel compositions. Figure 6 illustrates the utility of combining the NO_x and CO performance into a single quantity in order to evaluate operational trends. The pilot fuel has a major impact on performance for the conditions selected, and a distinct maximum in performance is observed for cases without pilot. The response appears as a ridge associated with a fairly broad range of equivalence ratios. However, the ridge has a peak which is a function of the fuel splits.

As a result, it can be seen that the fuel composition has an impact on the optimal fuel injection strategy for the conditions under considerations in this study.

To help provide additional insight into the behavior, some specific conditions were identified for further analysis. These cases are indicated on Fig. 6. One set reflects a fixed equivalence ratio

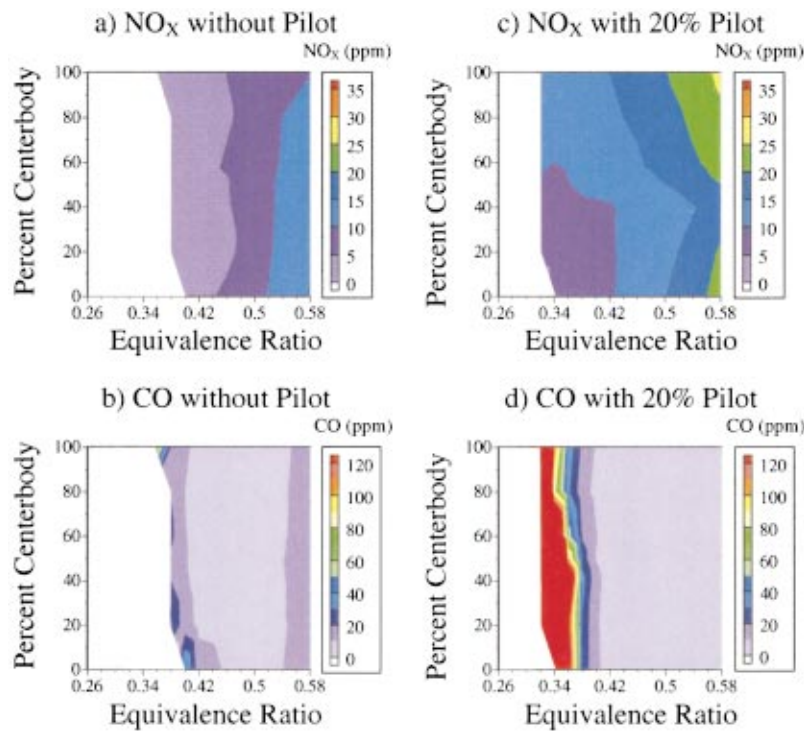


Fig. 5 Emission levels for 100 percent natural gas

and fixed fuel injection strategy to isolate the effect of fuel composition (cases 1–4). The other set reflects the optimal performance for each fuel composition (cases 5–7).

The isolated effect of fuel composition is summarized in Table 2 for an equivalence ratio of 0.42 and a fuel split of 80 percent centerbody—20 percent wall with no pilot fuel. These results show that, compared to 100 percent natural gas, CO levels de-

creased, while NO_x levels increased. Part of this behavior may be explained by the calculated reaction temperatures, which are also included in Table 3 for a fully premixed case.

Table 3 illustrates two key points. First, the average premixed temperatures are well below the threshold for NO_x formation (i.e., 1900 K). This means that, for the appreciable NO_x levels measured, local regions with higher temperatures (i.e., equivalence

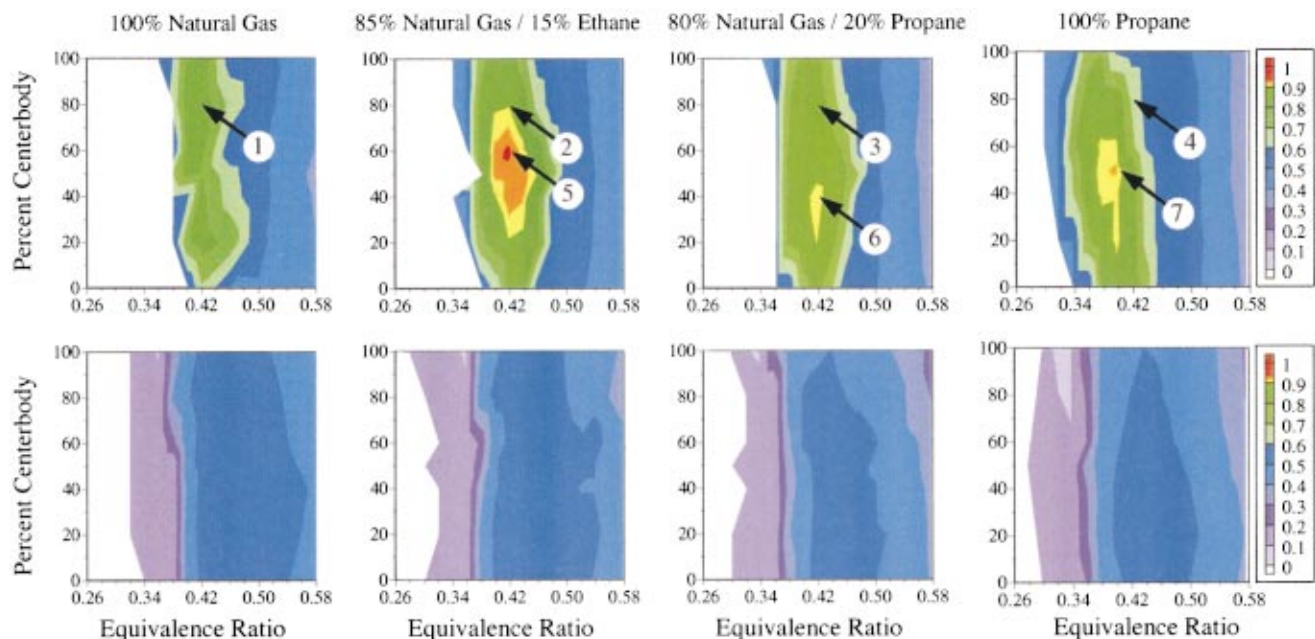


Fig. 6 Performance function for fuel composition (top: no pilot fuel; bottom: 20 percent pilot fuel)

Table 3 Relative emissions and calculated temperatures versus fuel composition for fixed equivalence ratio ($\phi=0.42$) and fuel split

Case	CO/CO _{nat gas}	NO _x /NO _{x nat gas}	T _{ad} *, K
85 percent nat gas 15 percent ethane (Case 2)	0.54	1.25	1616
80 percent nat gas 20 percent propane (Case 3)	0.86	1.25	1620
100 percent propane (Case 4)	0.55	1.87	1637

*Tadiabatic for Case 1: 1612 K

ratios) must be present if thermal NO_x is contributing. The results also illustrate that the peak temperatures will be higher for the fuels other than “pure” natural gas, thus providing one reason for the increased NO_x levels in these cases. The increased reaction temperatures may also lead to faster oxidation of CO. What is not revealed in Table 3 is the relative contribution of nonthermal NO_x mechanisms (e.g., N₂O pathway) which can also depend upon fuel type ([4]). For the conditions considered, it has been suggested that both mechanisms are important ([3]).

Lean Blow-Off Limits. The fuel composition and fuel injection strategy also have an impact on the LBO limits. “Pure” natural gas has the richest, LBO limit while the 100 percent propane case has the widest LBO range. Without pilot fuel, a slight trend of improved LBO limits with increased centerbody fuel injection is indicated.

With pilot fuel, the LBO range is broadened, but the overall performance is systematically worse compared to the cases with no pilot fuel. The reason for this degraded performance is attributed to higher emissions of both CO and NO_x, as illustrated in Fig. 5.

That the pilot fuel extends the LBO limits is not surprising and is, in fact, expected. The pilot fuel enriches the recirculation zone and thereby extends the limits. However, the improved LBO limits is accompanied by a degradation in performance (Fig. 6). Because the amount of fuel utilized may play a role, additional tests were conducted with 10 percent pilot fuel. These tests revealed similar trends. Consideration has not yet been given to the use of a more energetic fuel (e.g., H₂ addition—([5])) but remains the topic of future planned work.

In the case without pilot fuel, the fuel composition itself impacts the LBO limits in a manner consistent with reaction rates. Figure 7 illustrates this factor by comparing the calculated relative reaction rates of 100 percent methane, a mixture of 85 percent methane and 15 percent ethane, and a mixture of 80 percent methane and 20 percent propane. These amounts of ethane and propane lead to reaction rates that are a factor of ten or more faster. This, of course, is one reason why long premixing times can lead to problems with auto-ignition when fuel compositions vary. Recent work has illustrated a correlation of LBO limits with Peclet number, which is based in part on the kinetics of the process (e.g., [6]). This appears to be the case in the present study as well. With pilot fuel, less variation in LBO limits are noted, suggesting a different stabilizing mechanism in this case (e.g., since the fuel is injected directly into the reaction zone, a more “diffusion-type” reaction is present).

Fuel Distribution. Based on the emissions results, the chemistry of the process appears to be the primary reason for the varia-

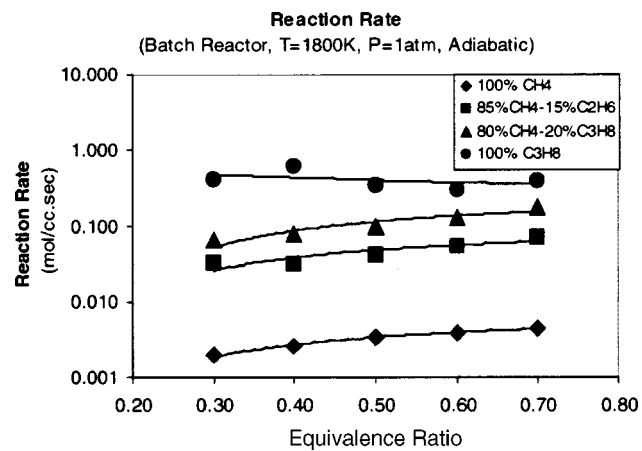


Fig. 7 Calculated reaction rates for methane—85 percent methane—15 percent ethane, and 80 percent methane—20 percent propane ([8])

tion in LBO. However, the relative contribution of fuel distribution must also be assessed to substantiate this conclusion.

The above measurements were taken at a fixed air flow (i.e., combustor pressure drop) while matching the firing rates for each fuel (equivalence ratios). Due to the difference in the heat content of the fuels, the gas volume flow rates also varied substantially. As a result, it might be expected that the fuel distributions will vary depending upon the gas blend. To start assessing this issue, calculations were done based on an empirical expression for a jet injected into a cross flow for the different fuels ([7])

$$x = D_{jet} \cdot q^{0.425} \cdot \left(\frac{y}{D_{jet}} \right)^{0.38} \quad (2)$$

where

$$q = \frac{\rho_{jet} \cdot V_{jet}^2}{\rho_{air} \cdot V_{air}^2}$$

Figure 8 shows the results for natural gas and propane for a fixed firing rate of 12.7 kW at a constant crossflow velocity. The results illustrate that the natural gas fuel jet penetration is limited to substantially less than half the annulus by the time it reaches the reaction zone; and accordingly, the propane jet penetration is 50 percent less (corresponding to a 2.5 flow rate ratio). These results illustrate the potential for fuel composition to play a role in the spatial distribution of the fuel.

To add to the understanding of the fuel distribution effects on performance, the spatial distribution of fuel entering the combus-

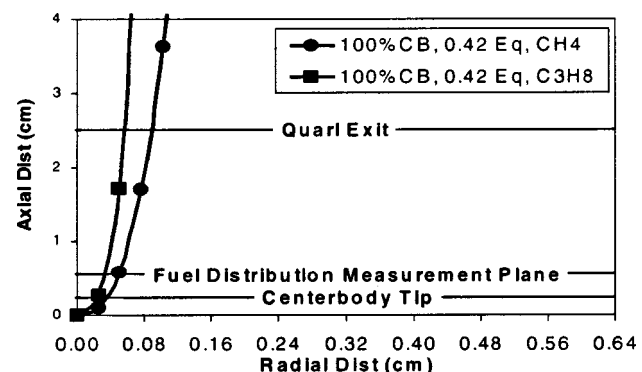


Fig. 8 Relative penetration of natural gas and propane at fixed firing rates for centerbody injection, wall jet injection, and a 50 percent split

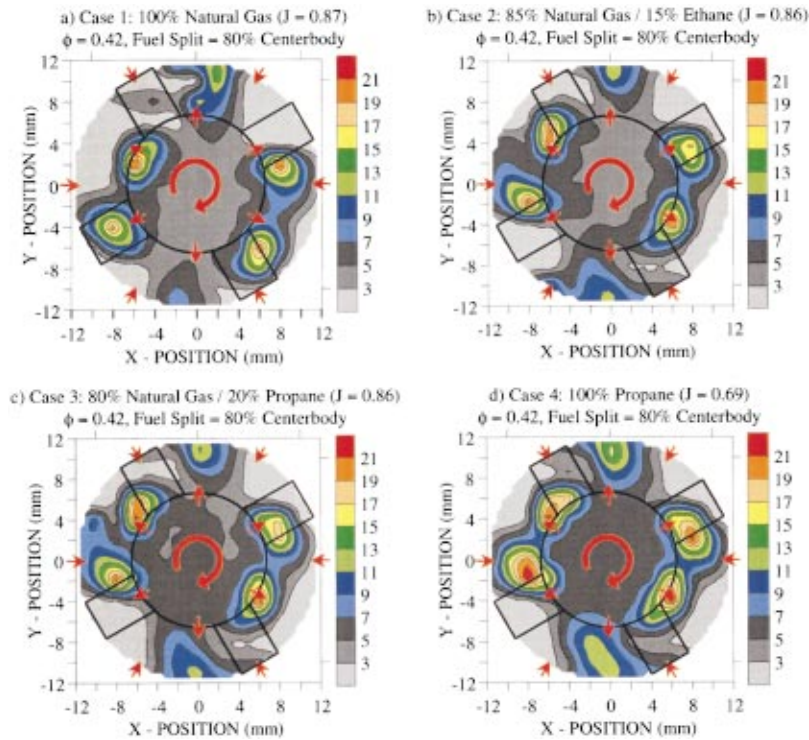


Fig. 9 Fuel distributions for cases 1–4 in Fig. 6 (concentrations presented in equivalent C atoms)

tor was measured for the seven cases which are numbered on Fig. 6. Nonreacting hydrocarbon measurements were conducted at a plane 3 mm downstream of the centerbody face. The results for the seven cases are shown in Figs. 9 and 10. The arrows indicate the direction and location of the fuel jets, with the swirl direction indicated by the counterclockwise arrow; the location of the four swirl vanes are also represented as the open boxes.

The results illustrate several key points. First, the distinct fuel jets are evidenced by the high fuel concentrations near the centerbody and wall jet injection locations. The six discrete centerbody injection points are easily distinguished in the results, suggesting that the fuel is not well mixed at the combustor inlet.

The results also reveal a distinct asymmetry in the jets, especially the jets at the “12 o’clock” and “6 o’clock” positions. Closer assessment of the geometry reveals the reason for the variation in the jet behavior. The alignment of the jets and vanes lead to three different patterns (with a symmetry plane consisting of a 180 deg sector). The position of the jet relative to vane plays a key role in the mixing. The 12 o’clock and 6 o’clock jets are oriented at the same location, as are the 1:30 and 7:30 and 4:30 and 10:30 pairs. Depending upon the location of these pairs relative to the position of the swirler vanes, the mixing varies considerably.

Considering the analysis from Fig. 8, it is anticipated that the propane would penetrate much less than the natural gas. However, for the results presented, the jet is heavily underpenetrating for either propane or natural gas. As a result, at the measurement plane shown in Fig. 8, differences may be difficult to resolve.

Indeed, based on the measured fuel distribution shown in Fig. 9, no clear differences are noticed as a function of fuel distribution for the same equivalence ratio (0.42), despite a large change in performance between cases 1 and 4. Since the distribution remains similar for the different fuel compositions, the changes in performance are attributed primarily to chemistry at the given equivalence ratio and fuel split.

To further explore the role of fuel distribution, Fig. 10 presents the distributions for cases 1 and 5–7. These cases represent the

optimal performance cases for each fuel blend. For these four cases, the NO_x and CO levels are comparable. When comparing the optimal cases, significant differences are exhibited compared to those in Fig. 9 (Figs. 9(a) and 10(a) are the same). The results displayed in Fig. 10 show a more uniform fuel distribution at the optimal conditions compared to the corresponding nonoptimal conditions shown in Fig. 9 (e.g., cases 4 versus 7, 3 versus 6, 2 versus 5). This suggests that fuel distribution does play a role in the performance in addition to the chemistry and that different fuel blends require different optimization conditions. To help quantify the differences, Table 4 shows the relative variation in fuel distribution for each case with the relative performance. Some trend with improved mixing (i.e., reduced normalized rms) is noted, for a given fuel composition (e.g., cases 4 versus 7, 3 versus 6, 2 versus 5).

Fully Premixed Studies. To further isolate the effects of chemistry from mixing, a study was conducted using fully premixed conditions. This was achieved by injecting the fuel well upstream of the inlet to the combustor and measuring the exhaust emissions at various equivalence ratios up to LBO limit. The fully premixed condition was verified using the setup used to measure the fuel distributions shown in Figs. 9 and 10.

The NO_x and CO emissions results from this study are shown in Figs. 11 and 12 in terms of calculated adiabatic flame temperature based on the equivalence ratio. A dependency of the lean blow off limit with fuel composition is illustrated (most clearly shown in Fig. 12). The trends are consistent with the observations based on the kinetics calculations (Fig. 7). The results also illustrate that the fuel composition plays a role in the emissions performance at temperatures above 1750 K. Propane generates more NO_x than natural gas at the same reaction temperature. This is attributed to the role of the nonthermal NO_x mechanisms that are impacted by fuel composition, including prompt NO_x and N_2O pathways. Finally, comparing the results from Fig. 11 to those in Fig. 5 indicates that the current system could achieve somewhat lower NO_x levels with improved mixing. However, the requirements of turn-

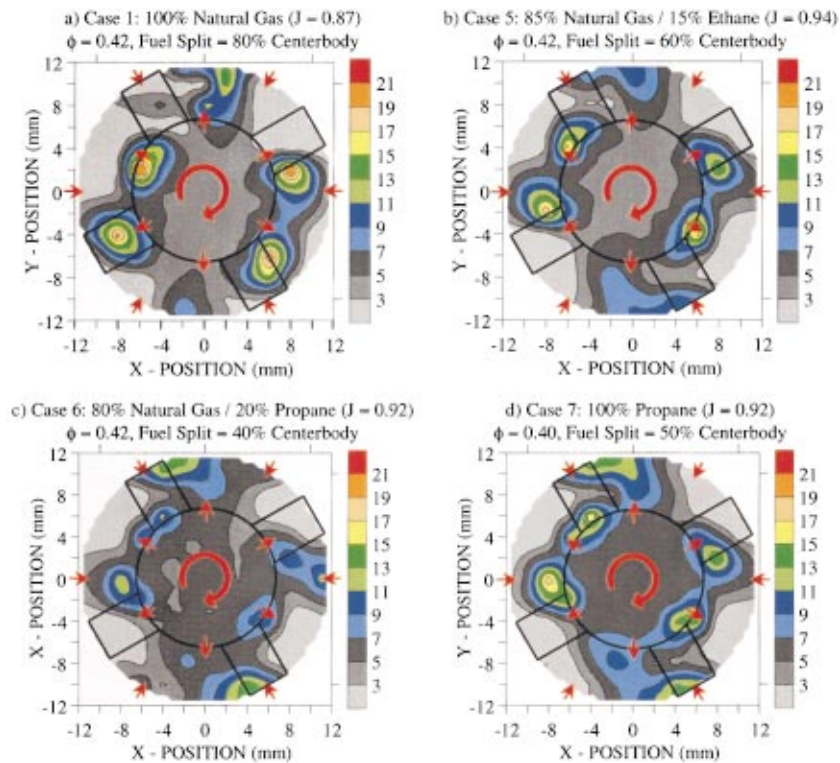


Fig. 10 Fuel distributions for optimal fuel injection strategies (cases 1, 5, 7) (coordinates are shown in millimeters and concentrations are presented in equivalent C atoms)

down, avoidance of flashback and autoignition, and robustness to fuel composition variability make a completely premixed system less attractive than the controlled lean rapid mix injection strategy utilized in the present study.

Summary and Conclusions

The results presented illustrate the role of fuel composition on the performance and fuel distribution in a model gas turbine combustor with controllable fuel injection strategies. The combustion performance in the present case is based on emissions of CO and NO_x, which are combined into a performance function, and the lean blow off limits.

The results indicate that:

- Changes in lean blow-off limits correlate with reaction rate, suggesting that a kinetic mechanism is responsible for stabilizing the reaction in the present case. This is reaffirmed by studies using fully premixed conditions.
- The measured fuel distributions for fixed operating conditions reveal little dependency upon fuel composition despite significantly varying momentum flux ratios. This is attributed to the

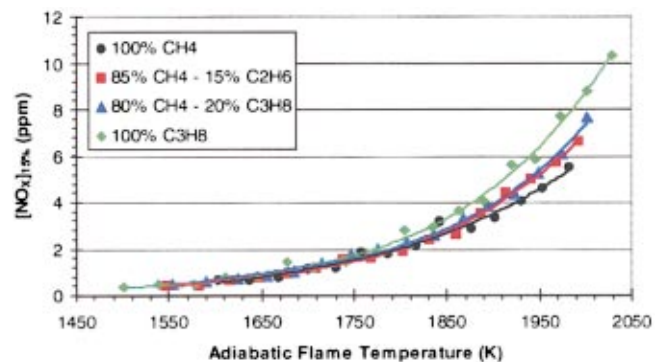


Fig. 11 NO_x concentrations for fully premixed cases

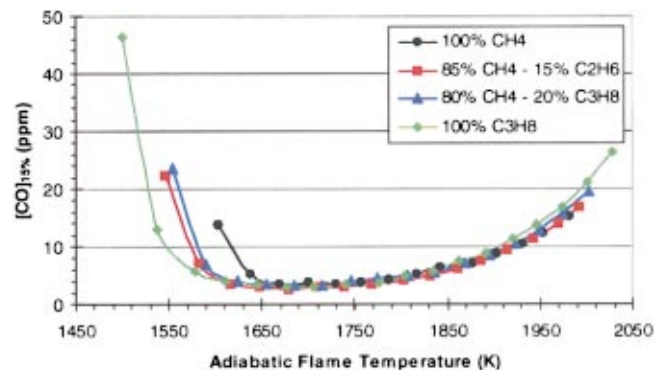


Fig. 12 CO concentrations for fully premixed cases

Table 4 Spatial variation versus fuel type and condition

Case	J , Performance	Rms Fuel Concentration/Mean Fuel Concentration
1	0.87	0.83
2	0.90	0.77
5	0.94	0.68
3	0.87	0.78
6	0.92	0.61
4	0.69	0.75
7	0.92	0.65

relatively weak penetration of the fuel jets in each case. This may be a strategy to minimize sensitivity of the mixing to fuel compositional changes.

- The fuel composition plays a role in NO_x emissions, with higher hydrocarbons generating more NO_x for a given firing temperature, independent of differences in mixing. This is attributed to the participation of nonthermal NO_x formation pathways that depend upon fuel composition.

- With the adaptive fuel injection strategy utilized in the present study, it is possible to compensate for fuel composition changes and maintain combustion performance at an optimal level. This is likely to have even more utility as load changes in addition to fuel composition.

Acknowledgments

The authors acknowledge the support of the US Department of Energy Advance Turbine Systems program and the South Carolina Institute for Energy Studies (SCIES). The contributions of Mr. Mark Vardakas, Mr. Junhua Chen, and Mr. Steve Hill are gratefully appreciated.

References

- [1] Meier, J. G., Hung, W. S. Y., and Sood, V. M., 1986, "Development and Application of Industrial Gas Turbines for Medium-BTU Gaseous Fuels," *ASME J. Eng. Gas Turbines Power*, **108**, pp. 182–190.
- [2] Liss, W. E., Thrasher, W. H., Steinmetz, G. F., Chowdiah, P., and Attari, A., 1992, "Variability of Natural Gas Composition in Select Major Metropolitan Areas of the United States," GRI-92/0123 Mar.
- [3] Nicol, D. G., Steele, R. C., Marinov, N. M., and Malte, P. C., 1995, "The Importance of the Nitrous Oxide Pathway to NO_x in Lean-Premixed Combustion," *ASME J. Eng. Gas Turbines Power*, **117**, pp. 100–111.
- [4] Steele, R. C., Jarrett, A. C., Malte, P. C., Tonouchi, J. H., and Nicol, D. G., 1997, "Variables Affecting NO_x Formation in Lean-Premixed Combustion," *ASME J. Eng. Gas Turbines Power*, **119**, pp. 102–107.
- [5] Nguyen, O. M., and Samuelsen, G. S., 1999, "The Effect of Discrete Pilot Hydrogen Dopant Injection on the Lean Blowout Performance of a Model Gas Turbine Combustor," *ASME Paper No. 99-GT-359*.
- [6] Hoffmann, S., Lenze, B., and Eickoff, H., 1998, "Results of Experiments and Models for Predicting Stability Limits of Turbulent Swirling Flames," *ASME J. Eng. Gas Turbines Power*, **120**, pp. 311–316.
- [7] Beer, J. M., and Chigier, N. A., 1972, *Combustion Aerodynamics*, Robert E. Krieger Publishing Co., Malabar, FL.
- [8] Kee, R. J., et al., 1999, *Chemkin Collection, Release 3.5*, Reaction Design, San Diego, CA.

Numerical Simulation of Swirl-Stabilized Premixed Flames With a Turbulent Combustion Model Based on a Systematically Reduced Six-Step Reaction Mechanism

D. E. Bohn

Professor,
Director of the Institute

J. Lepers¹

Research Engineer

Institute of Steam and Gas Turbines,
Aachen University of Technology,
Templergraben 55,
Aachen 52056, Germany

This paper presents the application of a detailed combustion model for turbulent premixed combustion to a swirl-stabilized premix burner. Computations are carried out for atmospheric pressure and elevated pressure of 9 atm. Results of computations for atmospheric pressure are compared to experimental data. The combustion model is of the joint-pdf type. The model is based on the characteristics of turbulent combustion under conditions typical for gas turbine burners. It incorporates a systematically reduced six-step reaction mechanism yielding direct computation of radical concentrations via transport equations or steady-state assumptions. The model is able to simulate combustion of fuel gases containing methane, carbon monoxide, hydrogen, carbon dioxide, and water. It is therefore applicable to both methane and low-BTU fuel gas combustion. Based on computed radical concentrations, a post-processor for NO_x formation is applied. This post-processor considers thermal formation of nitrogen oxides and NO formation via the nitrous oxide path. [DOI: 10.1115/1.1377597]

Introduction

Combustion models for lean premixed combustion have become a substantial aid in the design of dry low NO_x gas turbine burners operating without injection of water or steam into the combustion chamber.

Advanced combustion models have been developed for accurate prediction of heat release in the combustion chamber. Solving a transport equation for a chemical progress variable, they are highly efficient in computational time. A successful model of this kind is the flame-speed-closure model presented by Zimont et al. [1].

More detailed models for prediction of pollutant emissions are based on chemical kinetics. The kinetic mechanism applied can be either of an empirical or systematically reduced type.

The suitability of the former kind of reaction mechanisms for high pressure gas turbine combustion has made remarkable progress the last couple of years ([2,3]). Approaches to correlate NO_x production rate to CO concentrations seems to be a promising approach for simulation of pollutant formation in methane/air flames ([2,4,5]).

However, if studies of pollutant formation with varying fuel gas compositions including hydrogen and carbon monoxide are to be carried out, systematically reduced elementary reaction mechanisms show a greater flexibility, although being computationally more expensive.

The methodology of systematic reduction ([6]) yields a multi-step global reaction mechanism still including kinetic information of the underlying set of elementary reactions. Radical concentra-

tions are directly computed, either by solving transport equations or steady-state assumptions. An adequate systematically reduced six-step reaction mechanism for conditions of combustion encountered in gas turbine burners has been developed by Bohn et al. [7,8]. This mechanism is incorporated into a joint-PDF turbulent combustion model ([9]) which was validated for a test case of lean-premixed combustion.

This paper presents the application of this combustion model to a swirling lean-premixed flame at two different pressure levels (atmospheric and 9 atm). Experimental data was available for the atmospheric case so that a comparison of computations with measurements could be carried out.

Fluid Flow Model

The transient Navier-Stokes equations are used to model the fluid flow. They comprise equations for Favre-averaged transport of mass (1), momentum (2), and enthalpy (3)

$$\frac{\partial \bar{\rho}}{\partial t} + \nabla \cdot (\bar{\rho} \tilde{\mathbf{v}}) = 0 \quad (1)$$

$$\frac{\partial \bar{\rho} \tilde{\mathbf{v}}}{\partial t} + \nabla \cdot (\bar{\rho} \tilde{\mathbf{v}} \otimes \tilde{\mathbf{v}} - (\mu + \mu_t) \nabla \tilde{\mathbf{v}}) = -\nabla \bar{p} + \nabla \cdot ((\mu + \mu_t) (\nabla \tilde{\mathbf{v}})^T) \quad (2)$$

$$\frac{\partial \bar{\rho} \tilde{h}_{\text{tot}}}{\partial t} + \nabla \cdot \left(\bar{\rho} \tilde{\mathbf{v}} \tilde{h}_{\text{tot}} - \left(\frac{\lambda}{c_p} + \frac{\mu_t}{\sigma_t} \right) \nabla \tilde{h}_{\text{tot}} \right) = \frac{\partial \bar{p}}{\partial t} - \dot{q}_{\text{rad}}, \quad (3)$$

with σ_t being the turbulent Prandtl-Schmitt number according to Libby and Williams [10]. Radiative heat loss $\dot{q}_{\text{rad}}$ is modeled according to Müller [11], accounting for the gas phase radiation of water and carbon dioxide.

¹Joined B&B-Agema GmbH, Aachen, Germany in January 2001.

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-143. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

Table 1 Constants of the numerical scheme

C_1	C_2	C_μ	$\sigma_{t,k}$	$\sigma_{t,\varepsilon}$	C_{g1}	C_{g2}	σ_{gf}
1.44	1.92	0.09	1.0	$\frac{0.4187^2}{(C_2 - C_1)\sqrt{0.09}}$	2.52	2.0	0.7

Turbulence is considered with the standard $k-\varepsilon$ model solving transport equations for turbulent kinetic energy (4) and turbulent dissipation rate (5):

$$\frac{\partial \bar{\rho k}}{\partial t} + \nabla \cdot \left(\bar{\rho} \tilde{\mathbf{v}} k - \left(\mu + \frac{\mu_t}{\sigma_{t,k}} \right) \nabla k \right) = \bar{P} - \bar{\rho} \tilde{\varepsilon} \quad (4)$$

$$\frac{\partial \bar{\rho} \tilde{\varepsilon}}{\partial t} + \nabla \cdot \left(\bar{\rho} \tilde{\mathbf{v}} \tilde{\varepsilon} - \left(\mu + \frac{\mu_t}{\sigma_{t,\varepsilon}} \right) \nabla \tilde{\varepsilon} \right) = C_1 \cdot \frac{\tilde{\varepsilon}}{k} \bar{P} - C_2 \cdot \bar{\rho} \frac{\tilde{\varepsilon}^2}{k} \quad (5)$$

with P being the production term due to shear forces.

Turbulent viscosity is calculated as

$$\mu_t = C_\mu \bar{\rho} \frac{k^2}{\tilde{\varepsilon}} \quad (6)$$

For modeling of the presumed joint probability density function $\beta(T, f)$, further transport equations for mixture fraction (7) and variances of mixture fraction and temperature (8) (e.g. [12]) are solved:

$$\frac{\partial \bar{\rho} \tilde{f}}{\partial t} + \nabla \cdot \left(\bar{\rho} \tilde{\mathbf{v}} \tilde{f} - \left(\mu + \frac{\mu_t}{\sigma_t} \right) \nabla \tilde{f} \right) = 0 \quad (7)$$

$$\begin{aligned} \frac{\partial}{\partial t} \left(\bar{\rho} \left(\frac{\tilde{g}_f}{\tilde{g}_T} \right) \right) + \nabla \cdot \left(\bar{\rho} \tilde{\mathbf{v}} \frac{\tilde{g}_f}{\tilde{g}_T} \right) - \left(\frac{\mu}{\sigma_{gf}} + \frac{\mu_t}{\sigma_t} \right) \left(\frac{\nabla^2 \tilde{g}_f}{\nabla^2 \tilde{g}_T} \right) \\ = C_{g1} \frac{\mu_t}{\sigma_t} \left(\frac{(\nabla \tilde{f})^2}{(\nabla \tilde{T})^2} \right) - C_{g2} \bar{\rho} \frac{\tilde{\varepsilon}}{k} \left(\frac{\tilde{g}_f}{\tilde{g}_T} \right). \end{aligned} \quad (8)$$

Empirical constants appearing in the numerical scheme are listed in Table 1.

The set of equations is closed by the equation of state for ideal gases (9) and the constitutive equation of ideal gases (10).

$$\rho = \frac{p \cdot \sum_{i=1}^n \left(\frac{W_i}{Y_i} \right)}{R_m T} \quad (9)$$

$$\tilde{h}_{\text{tot}} = \int_{T_{\text{ref}}}^T c_p(T) \cdot dT + \frac{\tilde{\mathbf{v}}^2}{2} + \sum_{i=1}^n \tilde{Y}_i \cdot h_{i,\text{form}}^0 \quad (10)$$

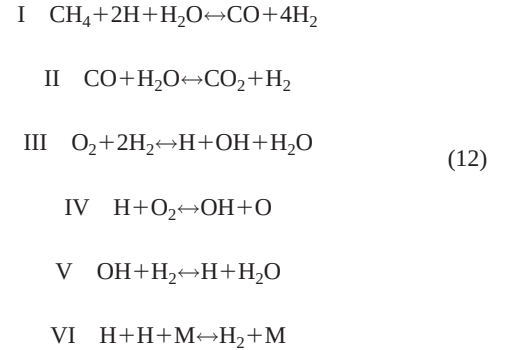
The temperature and species dependent values for the specific heats are taken from Kee et al. [13].

The Reynolds mean density is computed in accordance with the definition of the Favre-pdf:

$$\bar{\rho} = \frac{1}{\iint \frac{1}{\rho(T, f)} \cdot \beta(T, f) dT df} \quad (11)$$

Combustion Model

Main Species and Radicals. Chemical reaction is considered with a global 6-step reaction mechanism which was systematically reduced by Bohn et al. [7] from a set of 35 elementary reactions for lean to stoichiometric combustion of methane-air mixtures at pressures up to 50 bars ([14]), see Appendix). The global mechanism is:



Steady-state assumptions have been introduced for the radicals and intermediate species CH_3 , CH_3O , CH_2O , HCO , HO_2 , and H_2O_2 .

The global reaction rates I–VI include the complete kinetic information of the underlying set of 35 elementary reactions, where rate constants are computed by modified Arrhenius laws:

$$w_j = A_j T^{B_{jT}} e^{(-E_j/R_m T)} \quad (13)$$

Transport Equations for Chemical Species. Transport equations are solved for six chemical species, namely CH_4 , O_2 , CO_2 , H_2O , H , and O .

$$\frac{\partial \bar{\rho} \tilde{Y}_i}{\partial t} + \nabla \cdot \left(\bar{\rho} \tilde{\mathbf{v}} \tilde{Y}_i - \left(\rho D_i + \frac{\mu_t}{\sigma_t} \right) \nabla \tilde{Y}_i \right) = \bar{S}_i \quad (14)$$

The source term is computed from the reaction rates of elementary reactions as

$$\bar{S}_i = \bar{\rho} \int_{0.0}^{1.0} \int_{T_{11,(f)}}^{T_{u,1,(f)}} I_i \cdot \beta(T, f) \cdot dT df \quad (15)$$

with the integrand being an addition of elementary reaction rates j which contribute to depletion and formation of species i .

$$I_i = \sum_{j(i)} \left[v_j \cdot \frac{1}{\rho} \prod_{\kappa(j)} (c_\kappa(T, f)) \cdot w_j \right] \quad (16)$$

Computation of the Probability Density Function. The probability of mixture fraction and fluid temperature at any location in the flow field is given by a two-dimensional presumed probability density function (pdf). The pdf is based on mean values of mixture fraction and temperature, their respective variances (transport Eq. (8)) and their covariance.

The covariance is computed from gradient information of mean temperature and mean mixture fraction as a first-order approximation:

$$\beta(T, f) = \beta(T, f; \tilde{T}, \tilde{g}_T; \tilde{f}, \tilde{g}_f; \tilde{g}_{Tf}). \quad (17)$$

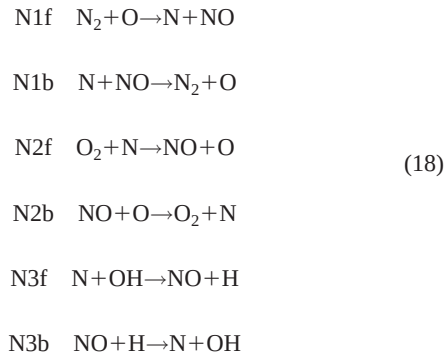
The presumed shape of the pdf is computed with a normalized β -function ([10]).

Formation of Nitrogen Oxides. Unburnt carbon monoxide and nitrogen oxides are the major pollutants relevant to gas turbine burner design. Further pollutants may result from specific fuel components like sulphur. However, they are not considered since ideally clean fuels are assumed.

The formation of nitrogen oxides in the flame and post-combustion regime is computed with a post-processor. The post-processing is based on the main species and radical distributions computed with the six-step reaction mechanism.

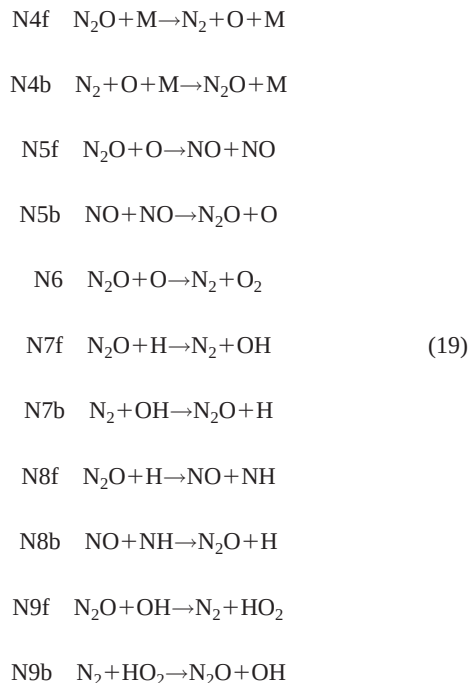
Formation of nitrogen oxides via the Zeldovich mechanism and via the nitrous oxide path is computed. The latter is of particular importance for high pressure combustion as it occurs in gas turbine combustion chambers ([15,16]).

The Zeldovich mechanism for NO_x formation includes six elementary reaction steps with the kinetic data being taken from Miller and Bowman [17]:



Since the N-radical is consumed sufficiently fast, it is assumed to be in steady state ([18]). O-, H- and OH- concentrations are predicted by the six-step reaction mechanism (12).

The nitrous oxide path is given by a total of 11 additional elementary reaction steps:



A further steady-state assumption is introduced for NH and an additional transport equation is solved for N_2O .

The initiating step for the nitrous oxide path is reaction N4b requiring a trimolecular reaction of nitrogen, an O-radical and a third body. This leads to a strong influence of pressure on the reaction rate.

Numerical Solution Method

A fully implicit finite volume scheme has been used for solving the coupled Eqs. (1)–(8) and (14). The SIMPLEC pressure-correction algorithm ([19]) is adapted on a nonstaggered grid, while avoiding the chequerboard oscillations by using the improved Rhie-Chow interpolation method ([20,21]). For diffusion terms a central differencing scheme is used, while advection terms are discretized by upwind differencing.

Application of the Flow and Combustion Model to a Swirling Flame at Atmospheric Pressure

Experimental Setup. The combustion model was applied for numerical simulation of premixed combustion in a strongly swirl-

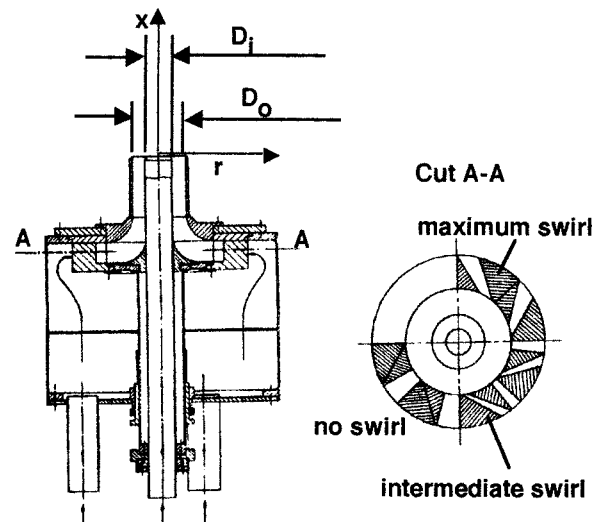


Fig. 1 Experimental setup ([11])

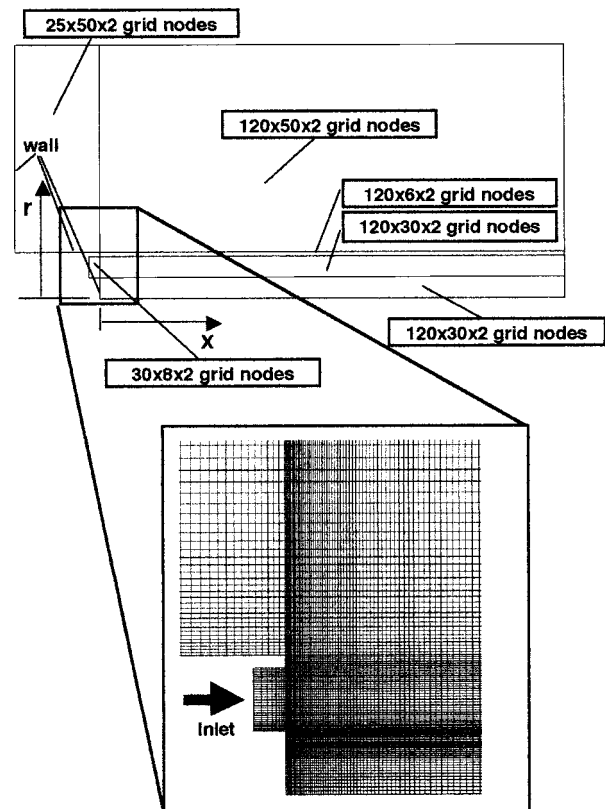
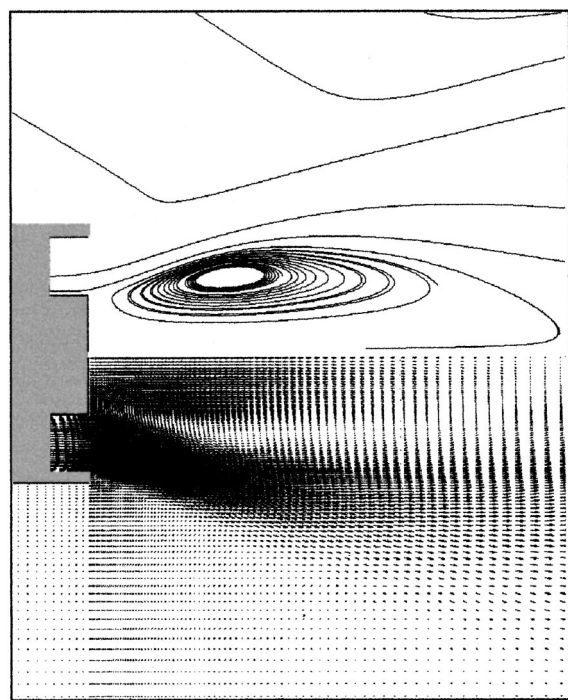


Fig. 2 Numerical discretization



Streak line plot and velocity vectors

Fig. 3 Numerical results for combustion at 1 atm and $T_{IN} = 300$ K

ing unconfined methane air-flame. Measurements of velocities and species concentrations were published by Philipp [12] and Leuckel et al. [22]. The experimental setup is shown in Fig. 1.

The methane air mixture is entering at atmospheric pressure and 300 Kelvin through adjustable swirlers (cut A-A). The outer tube of the burner has an inner diameter D_o of 90 mm.

For comparison with computations, operating conditions with an equivalence ratio of 0.91 and a swirl number of 1.5 were chosen. The swirl number represents the ratio of angular momentum to axial momentum. The quantity is nondimensionalized with a characteristic length scale ([12]).

Numerical Discretization and Boundary Conditions. The computational domain is shown in Fig. 2. Boundary conditions for axial velocities and turbulent quantities are set at the inlet so that computed velocities match measured data in the first measurement plane at $x = 0.05 D_o$. For building up of the radial pressure gradient of swirling flow at the inlet to the burner, a von Neumann boundary condition is applied.

Boundary conditions for chemical species and temperature at the inlet are set in accordance with the operating conditions of the burner given by Philipp [12]. The axial velocity profile set at the burner inlet according to the data provided by Philipp [12] can be seen in the bottom half of Fig. 3. Boundary conditions for entrainment flow are 300 K and a gas composition containing 21 Vol.-percent of oxygen and 79 Vol.-percent of nitrogen is provided.

The computational domain is discretized with a total of 30,820 nonequidistant grid points. For the atmospheric pressure case, a resolution of the radial flame front width with 10–15 grid points is achieved. At elevated pressure, a resolution of the radial flame front width with 5–7 grid points is obtained. The bottom part of Fig. 2 shows the grid close the burner inlet.

The computations are carried out three dimensionally with only one cell in circumferential direction due to rotational symmetry. The assumption of rotational symmetry is supported by gradients of all measured scalar quantities and axial velocity approaching zero close to the centerline.

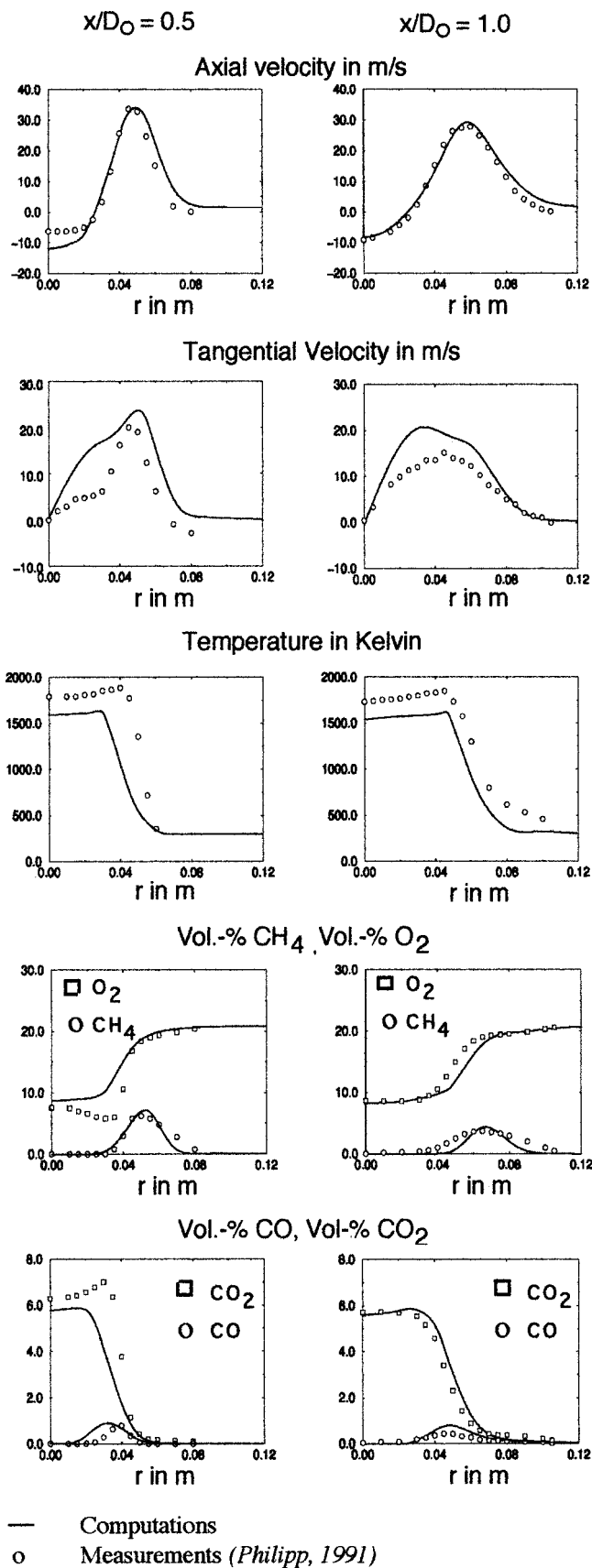


Fig. 4 Species distributions at axial positions $x/D_o = 0.5$ and $x/D_o = 1.0$ (1 atm, $T_{IN} = 300$ K)

Results and Discussion. Figure 3 shows velocity vectors and streak lines in the flow field for the validation case. The swirl-induced strong recirculation zone can clearly be detected.

Figure 4 shows radial distributions of velocity, temperature, and chemical species at axial positions $x=0.5 D_o$ and $x=1.0 D_o$. Computational and measured results are given. The radial distribution of axial velocity is in acceptable accordance with measurements, considering a standard $k-\epsilon$ model is used for turbulence. In particular, the location of maximum velocity is predicted correctly indicating an accurate computation of the radial size of the recirculation zone. This is achieved at the expense of over-predicted tangential velocities.

Maximum temperature is underpredicted by the computations presented. This is credited to overprediction of mixing with cold gases in the recirculation zone and over-prediction of radiative heat loss by the radiation model applied. The mean enthalpy defect of hot gases caused by the latter at axial position $x=0.5 D_o$ is 105 kJ/kg. A further part of the difference between temperatures measured and computed may be caused by higher hydrocarbons present in the fuel utilized experimentally.

The distribution of methane computed numerically appears to be less broad than measured. This indicates that turbulent diffusivity at the radial position where methane concentrations occur is somewhat underpredicted. The opposite is true for the inner part of the flow domain as can be seen from over-predicted diffusion of oxygen at $x=0.5 D_o$ and $r<0.04$ m. The damping effect of interacting pressure and density gradients on turbulence in this flow and combustion problem ([12]) could not be captured by the standard $k-\epsilon$ model applied.

The maximum concentration of CO computed numerically is in good accordance with measurements at position $x=0.5 D_o$. The decrease of computed CO concentrations between the two axial positions under consideration is found to be slightly slower than in measurements.

Numerical Computations for Gas Turbine Operating Conditions

The boundary conditions were subsequently altered to account for operating conditions typically found in gas turbine combustion chambers. Based on a polytropic efficiency of the compressor of 88 percent and a pressure ratio of 9, a combustor inlet temperature of 612 Kelvin is obtained. It is assumed that the compressor outlet temperature is approximately equal to the combustor inlet temperature as it holds for combustor design with the combustor liner being cooled in parallel flow (see Fig. 5).

The volumetric flow into the combustor and the velocity boundary conditions at the burner inlet are kept constant for comparability of results. To obtain these boundary conditions, the mass flow at the inlet had to be increased by a factor of 4.41 compared to computations at atmospheric pressure.

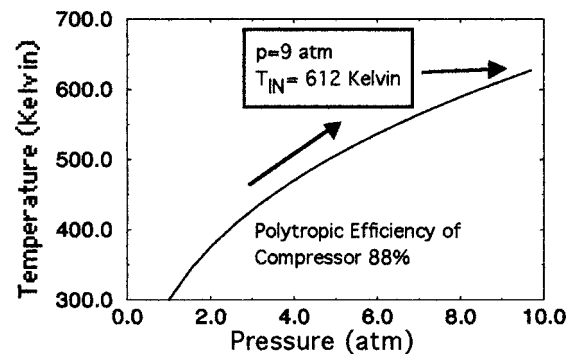


Fig. 5 Boundary conditions for computations under typical gas turbine combustor inlet conditions

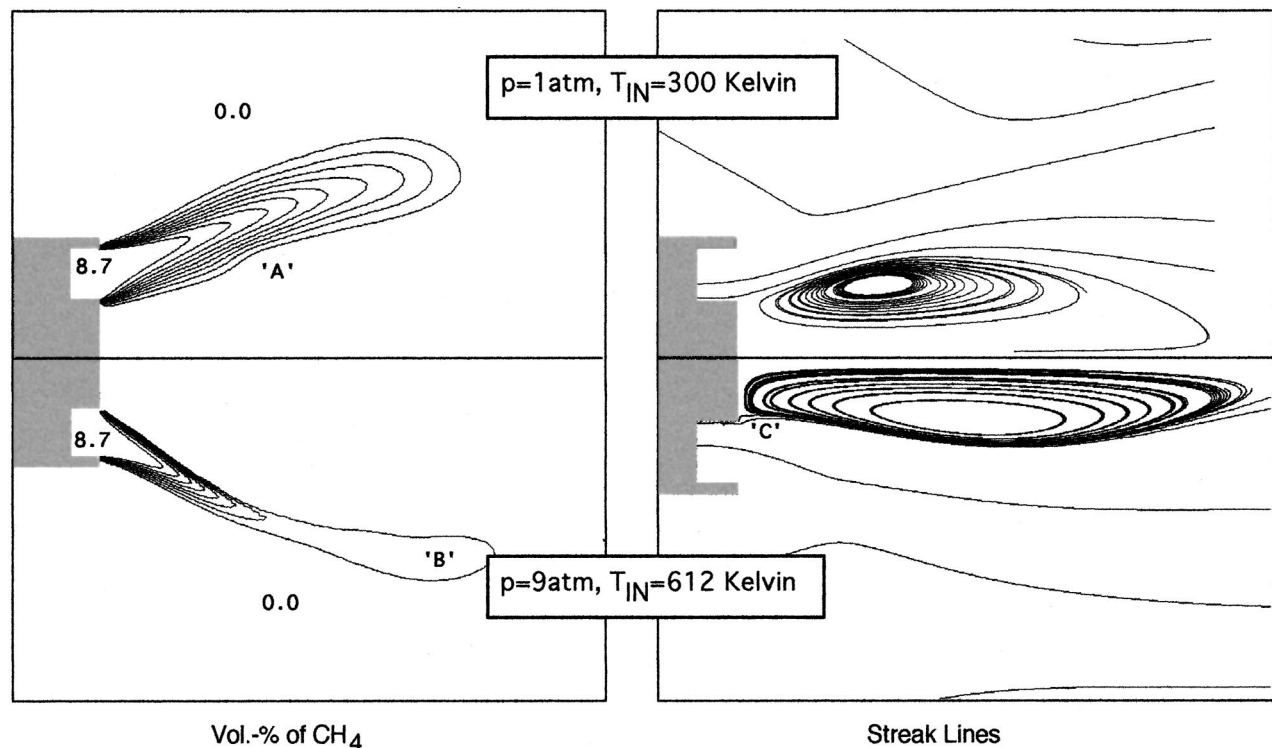


Fig. 6 Comparison of computations for atmospheric and elevated pressure

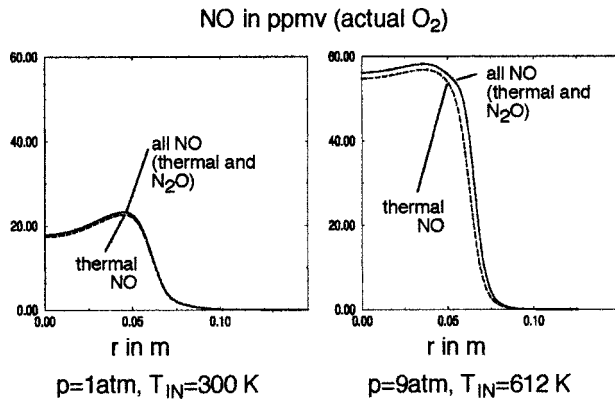


Fig. 7 Concentrations of nitrogen oxides at axial position $x/D_o = 1.0$

The left-hand side of Fig. 6 shows the distribution of CH_4 in the flow field at these elevated operating conditions in comparison to operation at atmospheric pressure. It can be seen that turbulent flame speed is increased and the flame front becomes thinner. At the outer regions of the flame front, ignition is delayed due to slight cooling effects of entrainment flow (position 'B'). The bend of isolines of methane for atmospheric pressure at position 'A' in Fig. 6 is caused by interaction of the flame front with the recirculation zone.

The recirculation zone for both operating conditions is shown on the right-hand side of Fig. 6. It can be seen that the axial length of the recirculation zone is increased whereas its radial width is reduced. This is credited to a higher pressure gradient at the inner radius of the burner inlet (position 'C') which is induced by a more rapid heat released compared to atmospheric combustion. This pressure gradient leads to a deflection of streak lines.

The limited radial width of the recirculation zone leads to a stronger interaction of the outer regions of the flame with the entrainment flow, leading to the delay of ignition mentioned above. To avoid quenching in the outer flame region, swirl of incoming flow would have to be increased.

Formation of Nitrogen Oxides at Atmospheric Operating Conditions and Elevated Pressure

In Fig. 7 the radial distribution of NO is given at position $x = 1.0 D_o$ for both operation at atmospheric pressure and operation at 9 atm.

For the atmospheric pressure case, post-processing for nitrogen oxide concentrations yields a maximum amount of 23.3 ppmv at the axial position considered in Fig. 7. Approximately two percent of NO are contributed by the nitrous oxide path. No experimental data was available for NO_x concentrations. Considering maximum temperatures have been underpredicted, NO_x concentrations in the test burner setup are expected to be higher than computed.

Most of the NO production occurs in the flame front region, where high radical concentrations are present. Besides that, NO concentrations are high in the recirculating flow region due to residence-time effects.

Switching to operating conditions with an inlet temperature of 612 K and a pressure of combustion of 9 atm yields maximum NO concentrations of 58.7 ppmv at the axial position under consideration. For these conditions, NO formed via the nitrous oxide path contributes 2.8 percent to the nitrogen oxide concentration found in the recirculation zone and approximately 16 percent to the nitrogen oxide concentration found in the flame front region.

Summary and Conclusions

Numerical results for computation of swirling premixed flames at two different pressure levels were presented. Computational results for atmospheric pressure were compared to measured data given by Philipp [12] and Leuckel et al. [22]. The model applied is based on a joint-pdf approach incorporating a systematically reduced six-step reaction mechanism. The model was developed to account for different fuel compositions including methane, hydrogen, carbon monoxide, carbon dioxide, nitrogen, and water. Radical concentrations are directly computed either by transport equations or steady-state assumptions. Post-processing for NO concentrations is based on these radical distributions.

Major results are:

- the combustion model presented is adequate for simulation of swirl-stabilized turbulent premixed combustion,
- direct computation of radical concentrations is an appropriate means for NO_x prediction with a post-processor, and
- further improvement is required for the turbulence and heat radiation model applied.

Acknowledgments

The authors would like to thank the "Deutsche Forschungsgemeinschaft" (DFG) for supporting this work as part of the "Graduiertenkolleg Turbulenz und Verbrennung."

Nomenclature

A	= frequency factor
B	= temperature exponent
C	= general constant
c_i	= species concentration
c_p	= specific heat
D	= binary diffusion coefficient
E	= activation energy
f	= mixture fraction
g	= variance
h	= enthalpy
k	= turbulent kinetic energy
p	= pressure
q	= heat
R_m	= universal gas constant
t	= time
T	= temperature
v	= velocity
W	= molar weight
w	= reaction rate
Y_i	= species mass fraction

Greek

β	= probability
ε	= turbulent dissipation
λ	= thermal conductivity
μ	= viscosity
ν	= stoichiometric coefficient
ρ	= density

Subscripts

form	= formation
rad	= radiation
t	= turbulent quantity
tot	= total quantity

Characteristic Numbers

Ka	= Karlovitz number
σ	= Prandtl-Schmitt number
Le	= Lewis number

Methane-air reaction mechanism ([14])

j	Reaction	A_j	B_j	E_j
1f	$H + O_2 \rightarrow O + OH$	2.0E14	0.0	70.338
1b	$O + OH \rightarrow O_2 + H$	1.575E13	0.0	2.89
2f	$O + H_2 \rightarrow OH + H$	1.8E10	1.0	36.95
2b	$OH + H \rightarrow O + H_2$	8.0E9	1.0	28.30
3f	$H_2 + OH \rightarrow H_2O + H$	1.17E9	1.3	15.18
3b	$H_2O + H \rightarrow H_2 + OH$	5.09E9	1.3	77.824
4f	$OH + OH \rightarrow H_2O + O$	6.0E8	1.3	0
4b	$H_2O + O \rightarrow OH + OH$	5.9E9	1.3	71.297
5	$H + O_2 + M^a \rightarrow HO_2 + M^a$	2.3E18	-0.8	0
6	$H + HO_2 \rightarrow OH + OH$	1.5E14	0.0	4.204
7	$H + HO_2 \rightarrow H_2 + O_2$	2.5E13	0.0	2.93
8	$OH + HO_2 \rightarrow H_2O + O_2$	2.0E13	0.0	4.1868
9f	$CO + OH \rightarrow CO_2 + H$	1.51E7	1.3	-3.17
9b	$CO_2 + H \rightarrow CO + OH$	1.57E9	1.3	93.521
10f	$CH_4(+M)^b \rightarrow CH_3 + H(+M)^b$	6.3E14	0.0	435.43
10b	$CH_3 + H(+M)^b \rightarrow CH_4(+M)^b$	5.20E12	0.0	-5.485
11f	$CH_4 + H \rightarrow CH_3 + H_2$	2.2E4	3.0	36.63
11b	$CH_3 + H_2 \rightarrow CH_4 + H$	9.57E2	3.0	36.63
12f	$CH_4 + OH \rightarrow CH_3 + H_2O$	1.6E6	2.1	10.30
12b	$CH_3 + H_2O \rightarrow CH_4 + OH$	3.02E5	2.1	72.942
13	$CH_3 + O \rightarrow CH_2O + H$	6.8E13	0.0	0
14	$CH_2O + H \rightarrow HCO + H_2$	2.5E13	0.0	16.71
15	$CH_2O + OH \rightarrow HCO + H_2O$	3.0E13	0.0	5.003
16	$HCO + H \rightarrow CO + H_2$	4.0E13	0.0	0
17	$HCO + M \rightarrow CO + H + M$	1.6E14	0.0	61.546
18	$CH_3 + O_2 \rightarrow CH_3O + O$	7.0E12	0.0	107.4
19	$CH_3O + H \rightarrow CH_2O + H_2$	2.0E13	0.0	0
20	$CH_3O + M \rightarrow CH_2O + H + M$	2.4E13	0.0	120.63
21	$HO_2 + HO_2 \rightarrow H_2O_2 + O_2$	2.0E12	0.0	0
22f	$H_2O_2 + M \rightarrow OH + OH + M$	1.3E17	0.0	190.5
22b	$OH + OH + M \rightarrow H_2O_2 + M$	9.86E14	0.0	-21.23
23f	$H_2O_2 + OH \rightarrow H_2O + HO_2$	1.0E13	0.0	7.536
23b	$H_2O + HO_2 \rightarrow H_2O_2 + OH$	2.86E13	0.0	137.29
24	$OH + H + M^a \rightarrow H_2O + M^a$	2.2E22	-2.0	0
25	$H + H + M^a \rightarrow H_2 + M^a$	1.8E18	-1.0	0

^athird-body efficiencies: $CH_4=6.5$, $H_2O=6.5$, $CO_2=1.5$, $H_2=1.0$, $CO=0.75$, $O_2=0.4$, $N_2=0.4$

^bLindemann form $k = k_\infty / (1 + k_{fall}/[M])$ where $k_{fall} = 0.0063 \exp(-18000/\mathcal{RT})$ rate coefficients are defined as: $k_j = AT^\beta \exp(-E/\mathcal{RT})$ units are: moles, cubic centimeters, seconds, Kelvins, kJ/moles

References

- [1] Zimont, V., Polifke, W., Bettelini, M., and Weisenstein, W., 1997, "An Efficient Computational Model for Premixed Turbulent Combustion at High Reynolds Numbers Based on a Turbulent Flame Speed Closure," ASME Paper 97-GT-395.
- [2] Nicol, D. G., Malte, P. C., Hamer, A. J., Roby, R. J., and Steele, R. C., 1998, "Development of a Five-Step Global Methane Oxidation—NO Formation Mechanism for Lean Premixed Gas Turbine Combustion," ASME Paper 98-GT-185.
- [3] Hamer, A. J., Roby, R. J., and Klassen, M. S., 1998, "Comparison of Reduced Chemical Kinetic Mechanisms for Pollutant Emissions Predictions in Gas Turbines," ASME IJPGC meeting, Baltimore, MD.
- [4] Polifke, W., Döbbling, K., Sattelmayer, T., Nicol, D. G., and Malte, P. C., 1995, "A NO_x Prediction Scheme for Lean-Premixed Gas Turbine Combustion Based on Detailed Chemical Kinetics," ASME Paper 95-GT-108.
- [5] Polifke, W., Döbbling, K., and Sattelmayer, T., 1995, "A Computational Model for Lean-Premixed, Highly Turbulent Combustion," CIMAC 21st International Symposium on Combustion Engines, Interlaken, Switzerland.
- [6] Peters, N., and Rogg, B., 1993, "Reduced Kinetic Mechanisms for Applications in Combustion Systems," (Lecture Notes in Physics), Springer-Verlag, Berlin.
- [7] Bohn, D., Krüger, U., and Matouschek, G., 1995, "Numerical Simulation of Premixed Flame Combustion using Systematically Reduced Reaction Mechanisms," International Gas Turbine Congress, Yokohama, JGTC-40, II, pp. 275–282.
- [8] Bohn, D., and Matouschek, G., 1997, "Numerical Combustion Simulations and Pollutant Prediction of Low- to Medium-Btu Gases in Premixed Flames using Systematically Reduced Multi-Step Reaction Mechanisms," 4th Int. Conf. on Technologies and Combustion for a Clean Environment, Lisbon, Portugal.
- [9] Bohn, D., and Lepers, J., 1999, "Numerical Simulation of Turbulent Premixed Combustion Using a Joint-PDF Approach Based on a Systematically Reduced Multi-Step Reaction Mechanism," ASME-Paper 99-GT-272.
- [10] Libby, P. A., and Williams, F. A., eds., 1994, *Turbulent Reacting Flows*, Academic Press, London.
- [11] Müller, U. C., 1989, "Der Einfluß von Strahlungsverlusten auf die thermische NO-Bildung in laminaren CH_4 -Diffusionsflammen," Diplomarbeit, Institut für Technische Mechanik, Aachen University of Technology.
- [12] Philipp, M., 1991, "Experimentelle und theoretische Untersuchungen zum Stabilitätsverhalten von Drallflammen mit zentraler Rückströmzone," Ph.D. thesis, University of Karlsruhe, Germany, (in German).
- [13] Kee, R. J., Rupley, F. M., and Miller, J. A., 1990, "The Chemkin Thermodynamic Data Base," Sandia Report SAND87-8215B.
- [14] Peters, N., 1992, "Fifteen Lectures on Laminar and Turbulent Combustion," ERCOFTAC Summer School, Aachen, Germany.
- [15] Nicol, D. G., Steele, R. C., Marinov, N. M., and Malte, P. C., 1995, "The Importance of the Nitrous-Oxide Pathway to NO_x in Lean-Premixed Combustion," Trans. ASME, **117**, pp. 100–111.
- [16] Steele, R. C., Jarrett, A. C., Malte, P. C., Tonouchi, J. H., and Nicol, D. G., 1997, "Variables Affecting NO_x -Formation in Lean-Premixed Combustion," ASME J. Eng. Gas Turbines Power, **119**.
- [17] Miller, J. A., and Bowman, C. T., 1989, "Mechanism and Modelling of Nitrogen Chemistry in Combustion," Prog. Energy Combust. Sci., **5**.
- [18] Göttgens, J., 1995, "Berechnung einer laminaren Diffusionsflamme mit reduzierter chemischer Kinetik" Ph.D. thesis, Aachen University of Technology, (in German).
- [19] van Doormal, J. P., and Raithby, G. D., 1984, "Enhancements of the SIMPLE Method for Predicting Incompressible Flows," Numer. Heat Transfer, **7**, pp. 147–163.
- [20] Rhie, C. M., 1981, "A Numerical Study of the Flow Past an Isolated Airfoil With Separation," Ph.D. thesis, University of Illinois at Urbana-Champaign.
- [21] Rhie, C. M., and Chow, W. L., 1983, "Numerical Study of the Turbulent Flow Past an Airfoil With Trailing Edge Separation," AIAA J., **21**, pp. 1527–1532.
- [22] Leuckel, W., Lauer, G., Hirsch, C., and Habisreuther, P., 1994, "Mathematische Modellierung der Wechselwirkung von Turbulenz und Reaktion unter den in Gasturbinenbrennkammern vorliegenden Bedingungen," AG Turbo (German Research Cooperation on High Temperature Gas Turbines), Turboflam Vorhaben 3.1.3.4, Schlußbericht, (in German).

The Nonlinear Response of Multimode Rotors Supported on Squeeze Film Dampers

R. Y. Yakoub

A. El-Shafei

e-mail: elshafei@ritec-eg.com

Department of Mechanical
Design and Production,
Faculty of Engineering
Cairo University,
Giza 12316, Egypt

This paper describes the development of an extremely fast method to obtain the unbalance response of multiple mode rotors supported on squeeze film dampers (SFDs). Planar modal analysis theory is used to model the rotor-SFD system. Undamped critical speed analysis is performed to obtain the rotor eigenvalues and eigenvectors. The SFD nonlinear forces are included in the modal force vector. The system differential equations are constructed and are uncoupled using the orthogonal properties of modal vectors. Assuming circular orbit, consistent with planar modes, the differential equations are converted into algebraic ones. A polynomial in speed is obtained through algebraic manipulations. This polynomial represents the steady-state behavior of the rotor-SFD system. The full unbalance response is directly obtained by finding the roots of the polynomial for each particular orbit. This method is extremely fast compared to numerical integration and to iterative methods. The developed method is useful in performing parametric studies and optimum design of SFDs. Twenty-five orders of magnitude computer time savings are reported. Parametric studies of an aircraft gas turbine fan rotor supported by an SFD are presented. The parametric studies show the possibility of appropriately locating the SFD, to dampen the rotor modes. In addition, parametric studies are also used to determine the effect of the SFD parameters on the AGTFR unbalance response.

[DOI: 10.1115/1.1370974]

Introduction

The nonlinear nature of squeeze film dampers (SFDs) does not allow parametric studies of the dynamic behavior of rotor systems to be readily performed. This leaves the analyst with the only option of performing parametric studies with simple models, and performing a final check with time-consuming numerical integration.

The objective of this paper is to provide SFD analysts and designers with a tool for performing parametric studies of SFD supported rotor dynamic behavior for multimode rotors. This is quite an ambitious objective given the current state of the art in nonlinear rotordynamic software.

Historically, only a few researchers attempted obtaining the nonlinear response of SFD supported multimode rotors. Although several researchers worked on obtaining the nonlinear response of simple rotor models incorporating SFDs (including Mohan and Hahn [1] and Taylor and Kumar [2,3] for rigid rotors; and El-Shafei [4,5] for flexible Jeffcott rotors including fluid inertia effects), yet most researchers avoided obtaining parametric nonlinear response of multi-mode rotors and resorted to numerical integration techniques (e.g., Becker and Steinhardt [6]).

Perhaps the first analysis of multimode rotors, incorporating SFDs is the work of Greenhill and Nelson [7] who used an iterative loop using a modified secant method to obtain the response of rotors executing circular centered orbits in SFDs. Later Mclean and Hahn [8] developed an algorithm that exploits the characteristics of a circular centered orbit to obtain the response of a multimode rotor on SFDs; however, their technique needs to be developed mathematically for each particular rotor, and the mathematics can be cumbersome for complicated rotors. Bayoumi [9] and El-Shafei et al. [10] extended the work of Greenhill and

Nelson [7] to examine elliptic orbits in multimode rotors using the technique of equivalent linearization developed by El-Shafei and Eranki [11] for rigid rotors.

The work presented in this paper is based on planar modal analysis ([12]), and is an extension of the work of Youssef and El-Shafei [13]. The technique is based on assuming a circular centered orbit in the SFD and analyzing the effect of SFD forces on the rotor with planar modes, which also imply by definition circular orbits ([12]). Circular orbits imply single frequency steady-state response and are applicable mainly to SFDs with retainer springs. The nonlinear forces from the SFD are transferred to the right-hand side of the modal equations, thus allowing the uncoupling of the modal equations. Gyroscopic effects are included when obtaining the planar modes, but their effect in the modal analysis was neglected, to allow the nonlinear analysis to be performed. Thus, complex modes are avoided. This technique was used by Youssef and El-Shafei [13] and Hathout et al. [14]. Through modal analysis and the assumption of circular centered orbits, an extremely fast algorithm is developed to obtain the full unbalance response of multimode rotors. The algorithm shows major computer time savings versus numerical integration and also the iterative algorithm developed by Greenhill and Nelson [7]. This allows the current algorithm to be used as a basis for an advanced optimization routine of SFDs, or in its current format can be used for parametric studies of SFD supported rotors.

In addition, parametric studies of an aircraft gas turbine fan rotor (AGTFR) are presented, thus exploiting the full capabilities of the developed algorithm. The utility of the fast algorithm and its capabilities are quite evident from the parametric studies.

Furthermore, parametric studies are conducted on the damper parameters, which can have major effects on the damper performance. These parameters are the damper radial clearance (c), the journal length (L), and the stiffness of the retainer spring (Kr). The study of the effect of these parameters on the unbalance response of the AGTFR rotor is also presented.

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, July 1998; final revision, Dec. 2000. Editor: H. D. Nelson.

Modeling Using Planar Modal Analysis Theory

Figure 1 shows a general system, which consists of a shaft, a number of disks, a number of bearings, and an SFD. Implementing the modal analysis technique to such a nonlinear rotor-SFD assembly was developed recently by Hathout et al. [14]. They performed the modal analysis method on the linear part of the rotor and transferred all nonlinear forces to the right-hand side as external forces acting on each damper in addition to the unbalance forces acting on the disks.

The equation of motion for the system shown in Fig. 1 is

$$[M]\{\ddot{z}\} + [G]\{\dot{z}\} + [K]\{z\} = \{f\}. \quad (1)$$

To apply modal analysis to the assembly shown above, an undamped critical speed analysis is performed on the rotor without including the SFDs but taking into consideration the bearing and support stiffness. Principal stiffness and damping properties of the bearings are used in the plane of the rotor, thus resulting in planar modes, and implying circular orbits ([12]). This undamped critical speed analysis is performed on the shaft, using available computer programs based either on transfer matrix methods or finite element methods, which include the rotor and disks gyroscopics in the analysis. The analysis would result in m undamped critical speeds of interest with their associated planar modes.

It should be noted that to perform true modal analysis, one should either use both the orthogonality properties of left and right eigenvectors or use complex modes. However, we wish to simplify the procedure, since our main objective is to perform the nonlinear analysis. But planar modes, used here, do not completely uncouple Eq. (1) because of the gyroscopic effect ([12]). However, for the class of rotor systems considered here the gyroscopic effect can be neglected (usually of order 2 percent or less). Note, however, that the main effect of the gyroscopics is on the critical speeds, which is still retained; however, their effect in the modal analysis is neglected in favor of performing a simplified nonlinear analysis.

Thus, using the modal transformation, in nondimensional form

$$\{z\}_{n \times 1} = [U]_{n \times m} \{\bar{q}\}_{m \times 1} \quad (2)$$

where $[U]$ is the modal matrix, the system differential Eq. (1) becomes

$$\{\bar{q}''\}_{m \times 1} + [\omega^{*2}]_{m \times m} \{\bar{q}\}_{m \times 1} = [U]_{m \times n}^T \{f^*\}_{n \times 1} \quad (3)$$

which represents the uncoupled system differential equation, in the modal coordinates, and consists of the linear part on the left-hand side, and the nonlinear part on the right-hand side, including the nonlinear SFD forces, and the unbalance forces. The force vector $\{f^*\}$ takes the form

$$\{f^*\} = \{F_{u1}^* \dots 0 \dots F_{u2}^* \dots 0 \dots F_d^* \dots\}^T$$

where the unbalance forces and the damper forces can be located anywhere on the rotor.

Equations (2) and (3) can be solved using numerical integration, but this process involves a huge computational time ([2]). In

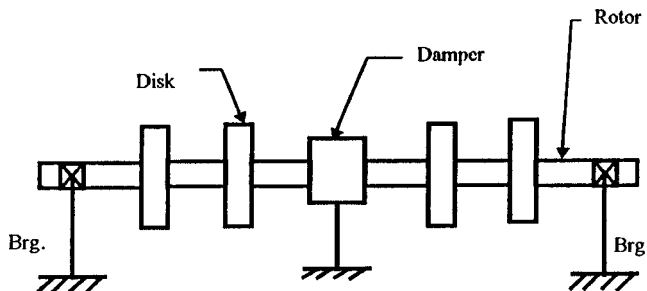


Fig. 1 Multidisk rotor supported by squeeze film damper (SFD)

the next paragraphs we describe an extremely fast algorithm to obtain the unbalance response of nonlinear multiple modes rotors, that would allow parametric studies to be performed.

Developing an Extremely Fast Method

Assuming that at steady state the rotor is executing a circular orbit with synchronous whirl, thus the whole rotor is whirling at the speed of the rotor rotation Ω^* . Then the motion of the center of the damper journal E , and S the geometric center of any of the disks can be expressed, in nondimensional form, as follows:

$$\bar{z}_E = \varepsilon e^{j\Omega^* \tau} \quad (4)$$

$$\bar{z}_S = R e^{j(\Omega^* \tau + \delta)}. \quad (5)$$

Consequently, since in (2) $[U]$ is real and $\{\bar{z}\}$ represents a circle, then $\{\bar{q}\}$ must also represent circular motion in the steady state. Thus

$$\{\bar{q}\} = \{\bar{Q} e^{j\Omega^* \tau}\} \quad (6)$$

where $\{\bar{Q}\}$ is the amplitude vector of $\{\bar{q}\}$.

By substituting for $\{\bar{q}\}$ from Eq. (6) into Eq. (3), one can obtain

$$[[\omega^{*2}] - [I]\Omega^{*2}]_{m \times m} \{\bar{Q}\}_{m \times 1} e^{j\Omega^* \tau} = [U]_{m \times n}^T \{f^*\}_{n \times 1} \quad (7)$$

where $[I]$ is the identity matrix.

The force vector $\{f^*\}$ can be expressed in a steady-state form by defining its elements (unbalance force and SFD forces). The unbalance force can be given in nondimensional form as follows:

$$F_u^* = \frac{m_d}{m_1} u^* \Omega^{*2} e^{j\beta} e^{j\Omega^* \tau}. \quad (8)$$

The SFD forces can be written in complex form as

$$F_d^* = (F_{dr}^* + jF_{dt}^*) e^{j\Omega^* \tau} \quad (9)$$

where F_{dr} and F_{dt} are the nonlinear SFD forces. These forces depend nonlinearly on the eccentricity ratio ε , and are given in details in [11].

Thus, the force vector (f^*) reduces to

$$\{f^*\} = \begin{Bmatrix} F_u^* e^{j\beta} \\ F_d^* \end{Bmatrix} e^{j\Omega^* \tau}. \quad (10)$$

By substituting for the force vector $\{f^*\}$ from Eq. (10) into Eq. (7), one can obtain

$$[[\omega^{*2}] - [I]\Omega^{*2}]_{m \times m} \{\bar{Q}\}_{m \times 1} = [U]_{m \times 2}^T \begin{Bmatrix} F_u^* e^{j\beta} \\ F_d^* \end{Bmatrix}_{2 \times 1}. \quad (11)$$

By defining a diagonal matrix $[A]$ which is the inverse of the receptance matrix to be

$$[A] = [[\omega^{*2}] - [I]\Omega^{*2}]. \quad (12)$$

Multiplying Eq. (11) by $[A]^{-1}$, one gets

$$\{\bar{Q}\}_{m \times 1} = [A]_{m \times m}^{-1} [U]_{m \times 2}^T \begin{Bmatrix} F_u^* e^{j\beta} \\ F_d^* \end{Bmatrix}_{2 \times m}. \quad (13)$$

It should be noted that the receptance matrix $[A]^{-1}$ may have a singular value at some critical speeds. Thus an artificial damping coefficient (ζ) is added to avoid the numerical overflow. This coefficient is chosen very small (about 1.E-10). In this case, the receptance equation is defined as follows:

$$[A]^{-1} = [[\omega^2] + j2\zeta\Omega^*[\omega] - [I]\Omega^{*2}].$$

Equation (13) is a set of nonlinear algebraic equations, that can be solved numerically by using Newton-Raphson method to obtain the modal vector $\{Q\}$. However, there are problems in using Newton-Raphson concerning speed, false convergence, trial and error in regions of multiple solutions and accuracy. To avoid this iterative method, a different approach, which has been used by Taylor and Kumar [3], and El-Shafei [4,5] for simple rotors, is suggested here to obtain the full unbalance response directly without the tedious process of trial and error. By substituting for the modal vector $\{Q\}$ from Eq. (13) into Eq. (6) and then into Eq. (2) considering the motion at the journal center (point E), one can get

$$\bar{z}_E = \left[\sum_{i=1}^m U_{i,\text{LSFD}} \bar{Q}_i \right] e^{j\Omega^* \tau} \quad (14)$$

where LSFD stands for the location of SFD.

By substituting from Eq. (4) into Eq. (14), one obtains

$$\varepsilon = \sum_{i=1}^m U_{i,\text{LSFD}} \bar{Q}_i. \quad (15)$$

Substituting for $\bar{Q}_i$ from (13) and after further algebraic manipulation of Eq. (15), the following form is obtained:

$$P_1 = P_2 e^{j\beta} \quad (16)$$

where P_1 and P_2 are polynomials in Ω^* with complex coefficients that are functions in ε .

Finally, by eliminating the angle β from Eq. (16) by dividing it into real and imaginary equations and using Cramer's rule, one can obtain

$$P_f = \sum_{i=0}^{8m} a_i \Omega^{*i} = 0. \quad (17)$$

P_f is the desired polynomial which describes the steady-state dynamics of a rotor with an unbalanced disk, which can be placed anywhere on the rotor, and is supported on a single SFD. The coefficients (a_i , $i=0, \dots, 8m$) of this polynomial, which is of order $8m$, are functions in the eccentricity ratio of the damper journal (ε). Once a value of ε is defined (in the range from zero to one), the polynomial coefficients (a_i) are calculated. The roots of the polynomial P_f , which represent the rotor rotational speed Ω^* at which this eccentricity ratio ε exists, are obtained by using the IMSL polynomial roots finding routines ([15]).

There are two IMSL routines to obtain the roots of a polynomial with real coefficients, DZPLRC and DZPORC. DZPLRC routine is based on Laguerre method, whereas DZPORC uses Jenkin-Traub method. DZPLRC runs much faster with order up to 35 times faster than DZPORC. Unfortunately, DZPLRC has problems with numerical overflow or underflow in spite of using double precision. To avoid these problems, the polynomial P_f must be scaled ([16]). The idea of this scaling is to make the value of the greatest positive root less than one. This can be done by the following substitution:

$$\Omega^* = \text{fac} \Omega^{**}.$$

fac is a selective quantity that would make the greatest root less than one. In our analysis it is taken somewhat higher than the nondimensional greatest critical speed. By this substitution, the polynomial is scaled and it is possible to use DZPLRC and the execution time can be reduced drastically ([16]).

The obtained roots ($8m$ roots) are complex, negative, and positive roots. The complex roots are ignored since they do not represent steady states. The negative roots are equal to the positive ones since the rotor can rotate in either positive (counter-clockwise) direction or negative (clockwise) direction. Thus, the negative roots are also ignored and only the positive roots are

Table 1 Comparison of computation time between the developed algorithm and other methods

Model Type	The Developed Algorithm	Iteration Method	FEM /EL	Numerical Integration Modal Analysis	FEM Numerical Integration
SU-Model	22 sec	5 hrs	1.5–2 hr.	4 hr.	days
LU-Model	22 sec	5 hrs	1.5–2 hr.	4 hr.	days
SC-Model	22 sec	5 hrs	1.5–2 hr.	8 hr.	days
LC-Model	22 sec	5 hrs	1.5–2 hr.	12 hr.	days

retained. This process is repeated to cover the range of the eccentricity ratio ε which is from $\varepsilon=0$ to $\varepsilon=1$, once this range is covered, the full unbalance response at the center of the damper journal is known.

By knowing ε and its corresponding speeds Ω^* , it is possible to evaluate the modal displacement vector $\{\bar{Q}\}$ from Eq. (14), after which all the elements of the vector $\{\bar{z}\}$ can be evaluated from Eq. (2).

At this point of modeling using this approach, one can obtain ε at the center of the damper journal (E), the vibration amplitude R at the center of the unbalanced disk (S), and the damper force F^* . All the above steps are organized in an algorithm to be executed by a digital computer. The algorithm is written in FORTRAN code, and it has the facility to constitute the coefficients of the desired polynomial (as discussed above) for m modes.

The solution thus obtained, represents the steady-state solution for the rotor system. To investigate the stability of the steady-state solution, a perturbation technique such as in the work of Rabinowitz and Hahn [17] is necessary to investigate the stability of the obtained solutions. This is particularly important in the regions of multiple steady states as in the case of the jump resonance phenomenon.

Algorithm Verification

To verify the algorithm described in this paper and to assess the validity of the assumptions made, a rotor of a test rig at Cairo University is modeled both by numerical integration and the proposed algorithm.

The results of the comparison are given in Yakoub and El-Shafei [18] for both short and long dampers, and show excellent agreement between the proposed algorithm and numerical integration.

Table 1 shows a comparison between the computation time of the various alternatives of obtaining the full nonlinear unbalance response of a particular rotor ([18]). The table compares the execution time, on a 486 DX4-100 computer, of the developed algorithm, the iteration method ([7]), the finite element/equivalent linearization method FEM/EL ([9]), and numerical integration both by modal analysis and by finite element method numerical integration. The advantage of the developed algorithm is clearly seen from Table 1, since the developed algorithm is 25 orders of magnitude faster than the closest algorithm. This is what makes the developed algorithm more suitable for parametric studies than comparable techniques.

The Advantages of the Developed Algorithm

The advantages of the developed algorithm can be summarized as follows:

1 The extremely fast execution time of 22 seconds on a 486 computer representing 25 orders of magnitude computer time savings represents the main advantage of the developed algorithm.

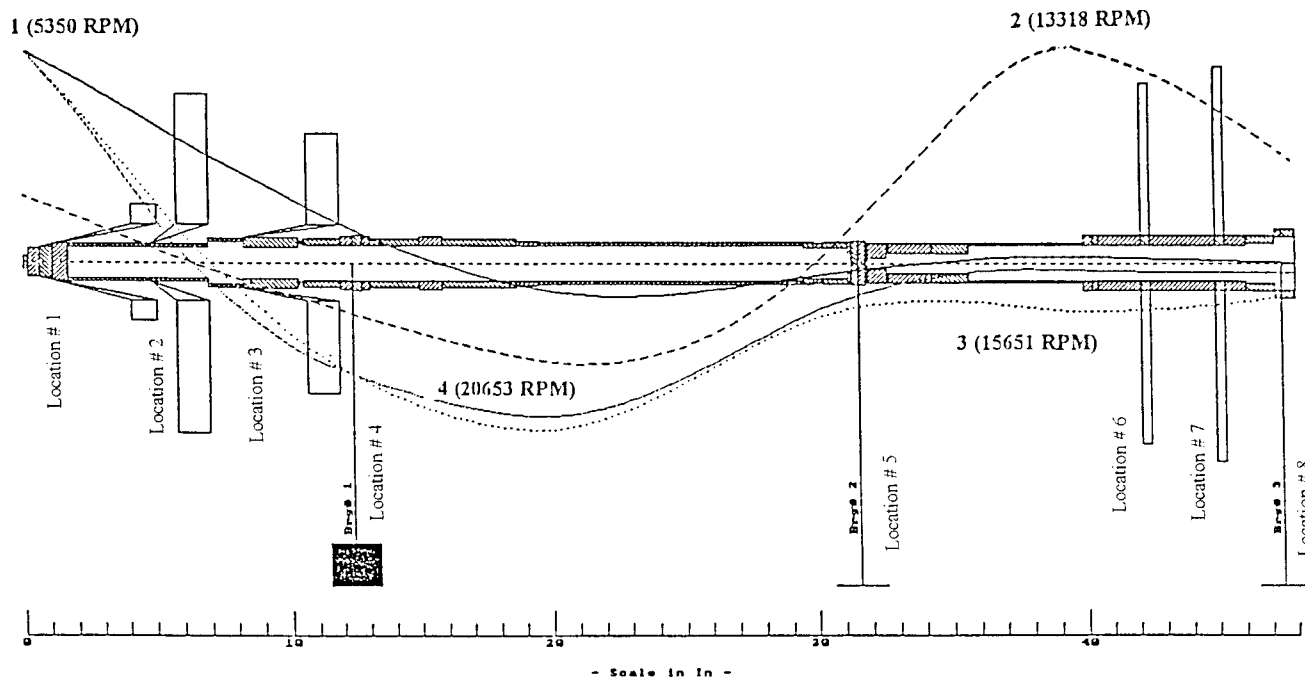


Fig. 2 Aircraft gas turbine fan rotor and its mode shapes

2 The developed algorithm depends on modal data thus no new mathematical development is required for new or modified rotors, and also makes it independent of the actual complexity of the rotor.

3 The developed algorithm can be considered as a post-processor to FEM or TMM methods thus enabling it to be a fast and independent algorithm.

4 The developed algorithm is able to obtain the full nonlinear response of the rotor, including jump resonance ([13]), since it is able to obtain all the solutions of the unbalance response. In this regard, it can be considered as a global numerical solver, without the need of tedious trial and error. This particular advantage is hardly matched by any other algorithm.

5 The accuracy of the developed algorithm is attested to by the examples of Yakoub and El-Shafei [18].

6 All those advantages are combined to form an elegant algorithm based on the previous work of Gunter et al. [12], Taylor and Kumar [3], and El-Shafei [4,5] to obtain an extremely fast algorithm to obtain the full nonlinear unbalance response of multi-mode rotors supported by SFDs. The algorithm is very effective in performing parametric studies and also can be used in design optimization of SFDs.

The next step in the development of this algorithm is to link it to an optimization routine to perform optimum design of SFDs for multimode rotors. The other planned direction for the development of the software is to use complex modes instead of planar modes in the analysis, thus uncoupling also the gyroscopic matrix.

Parametric Studies

Figure 2 shows a typical aircraft gas turbine fan rotor (AGTFR). This rotor is taken to undergo the parametric studies of locating SFDs and design of the dampers since it is a representative rotor for SFD application. The rotor is supported by three bearings. The first bearing is referred to as the fan bearing and the third bearing is the turbine end bearing. The second bearing is an intermediate differential bearing. The complete rotor data and its components are given in CRITSPD-PC, User Manual [19].

An undamped critical speed analysis in the speed range up to 25,000 rpm was performed. This analysis resulted in four critical speeds namely at 5335 rpm, 13007, 16375, and 22375 rpm. The resulting mode shapes are also shown in Fig. 2. The objective of the study is to locate the SFD at one of the three bearings to control the vibration of the rotor modes. In addition, the appropriate design parameters of the SFD are to be determined.

Appropriate Location of SFD

To ensure that the damper will contribute considerably in damping rotor vibrations, it should be located at a point, on the rotor, with a high modal activity, i.e., a high vibration amplitude. With mode shapes known, it is possible to assign where the damper should be located. Compromise between different modal behavior at different bearing locations will inadvertently be encountered. Because of the speed of the developed algorithm, all possible locations are investigated to decide which one is most appropriate. It is assumed that the damper will be mounted on one of the three rotor bearings.

In all the following parametric design studies the basic design of the damper is $c = 750 \mu\text{m}$, $L = 25.4 \text{ mm}$, $K_r = 350 \text{ N/mm}$, $\mu = 0.014 \text{ Ns/m}^2$ and $u = 0.15$. All parametric studies are made relevant to this basic design.

In performing the process of searching the appropriate location of SFD, the following steps were taken:

Table 2 AGTFR Components and their location

Component Name	Classification	Location #
Fan	Disk	1
1st stage fan	Disk	2
2nd stage fan	Disk	3
Fan bearing	Bearing	4
Differential bearing	Bearing	5
1st stage turbine	Disk	6
2nd stage turbine	Disk	7
Turbine end bearing	Bearing	8

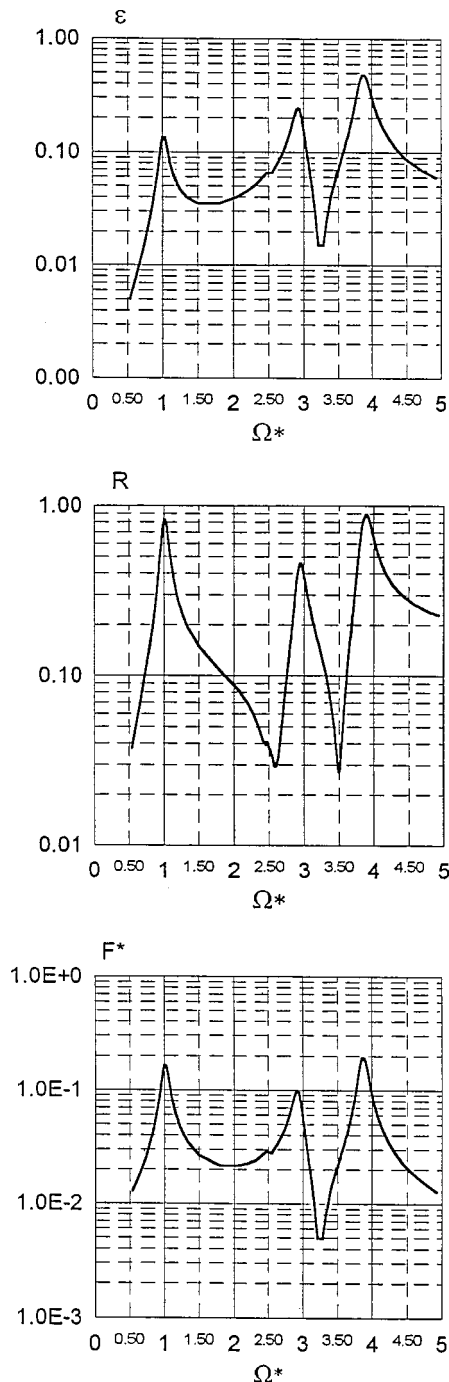


Fig. 3 The unbalance response of an aircraft gas turbine fan rotor (AGTFR). Case 41: Damper location: fan bearing (point #4), unbalanced disk: fan (point #1), short uncavitated damper (SU-model).

1 The rotor components are classified as either disks or bearings. Then they are numbered according to their order from the left end of the rotor. This classification and numbering are listed in Table 2 (see also Fig. 2).

2 The damper can be placed at the location # k ($k=4, 5$, or 8). While the unbalance can exist at the location of the disk # i ($i=1, 2, 3, 6$, or 7). Thus this case is called "Case ki ."

3 The contribution of the SFD in each case is investigated through the analysis of the unbalance response of AGTFR in each case ($3 \times 5 = 15$ cases).

4 The unbalance response of each case is obtained and plotted.

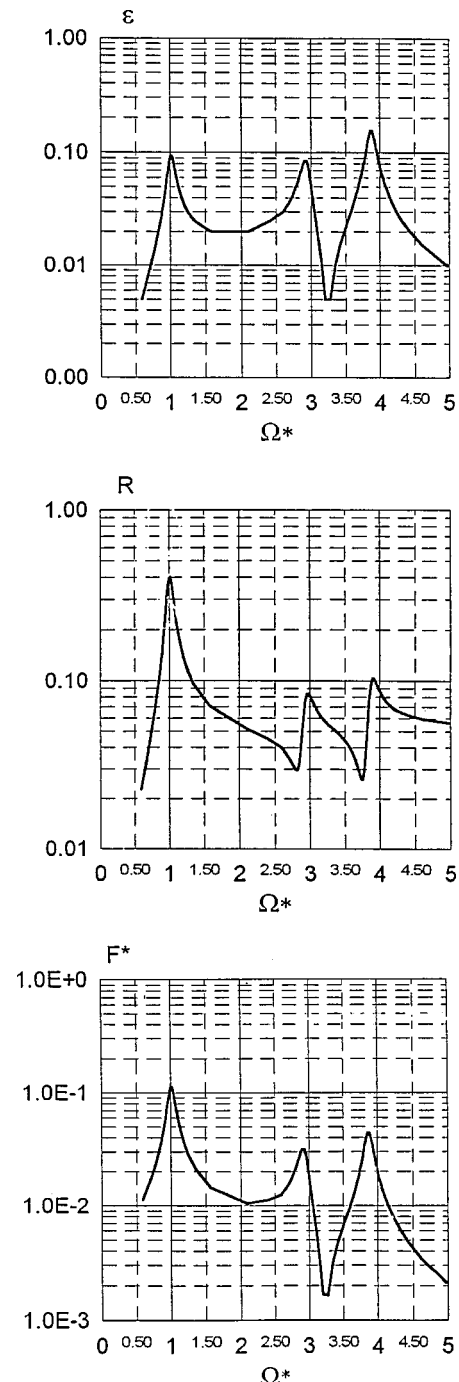


Fig. 4 The unbalance response of an aircraft gas turbine fan rotor (AFTFR). Case 42: Damper location: fan bearing (point #4), unbalanced disk: first stage fan (point #2), short uncavitated damper (SU-model).

The unbalance response of AGTFR covers three quantities, which are of interest. These quantities are: the vibration amplitude at the center of the damper journal (ε), the vibration amplitude of the center of the unbalanced disk (R), and the damper force (F^*).

Each of the three values ε , R , and F^* is plotted versus the nondimensional rotor rotational speed Ω^* . Ω^* is the rotor rotational speed divided by the first critical speed. Thus the rotor would exhibit four critical speeds at $\Omega^*=1.0, 2.489, 2.9$, and 3.86 . The scale of Ω^* axis is taken to be equal to 5 (i.e., 26675 rpm) to cover the whole range of the rotor rotational speeds.

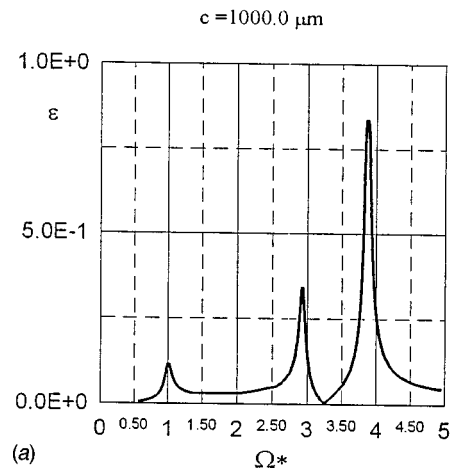
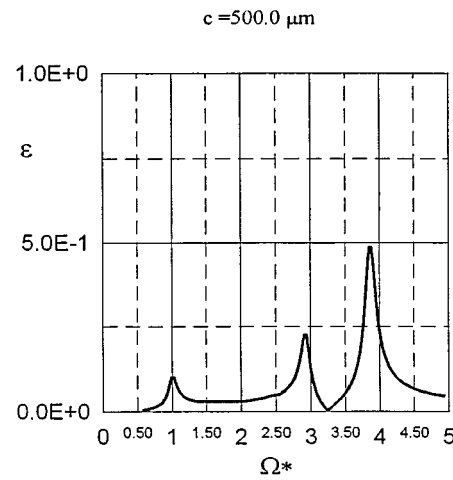
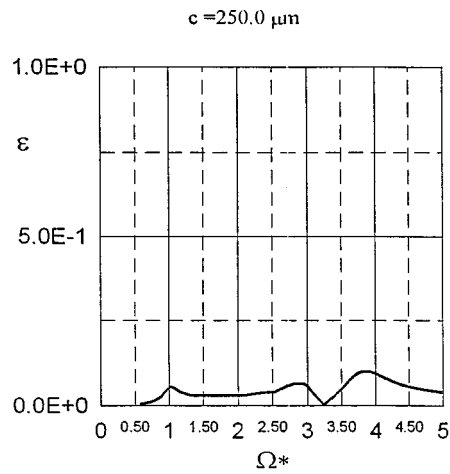


Fig. 5 (a) The effect of damper clearance on the vibration amplitude, short uncavitated damper (SU-model)

Figure 3 illustrates the unbalance response of AGTFR in case 41, which means that SFD is located at point 4 (fan bearing) and the unbalanced disk is located at point 1 (fan). It is observed that the rotor, in this case, exhibits only three modes which are at the first, third, and the fourth critical speeds, i.e., the second mode does not appear. This phenomenon can be generalized to all cases and is explained by the rotor mode shapes which are shown in Fig. 2. It can be seen in Fig. 2, that the value of the second mode at the unbalanced disk (at the fan) is relatively small as compared to the values of the other modes. Thus the contribution of the

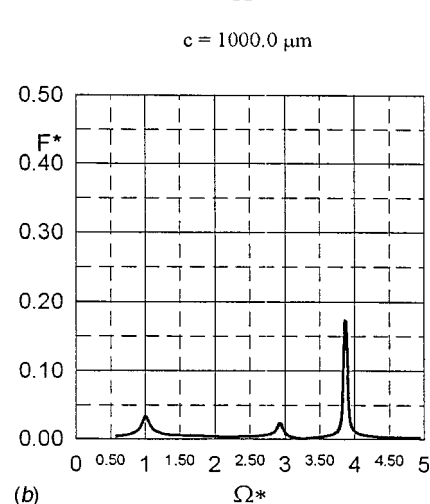
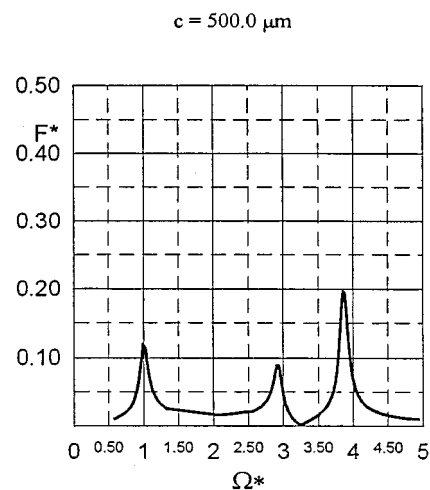
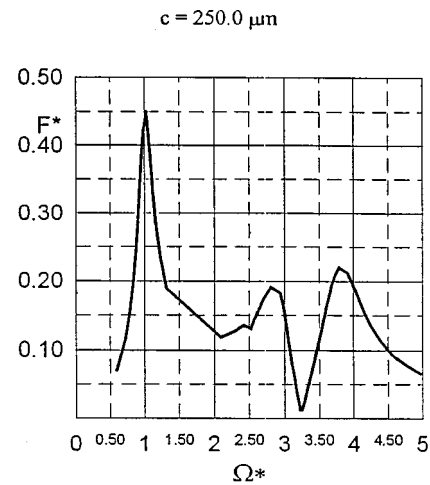
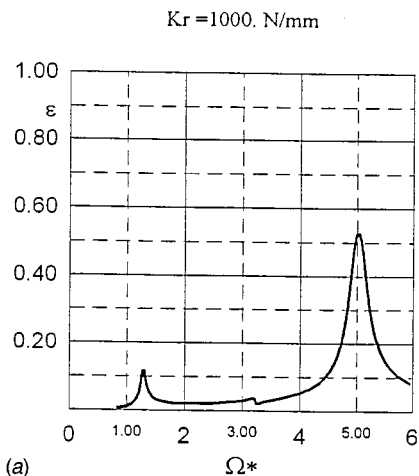
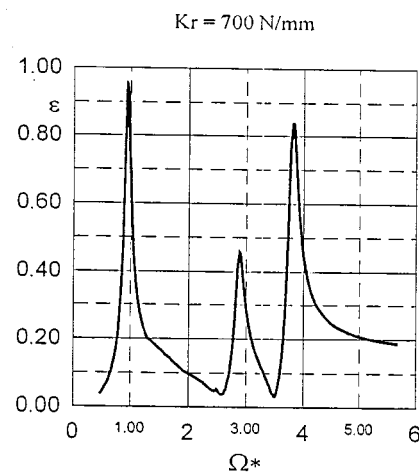
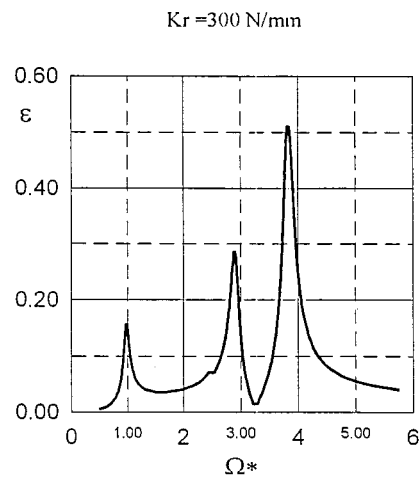


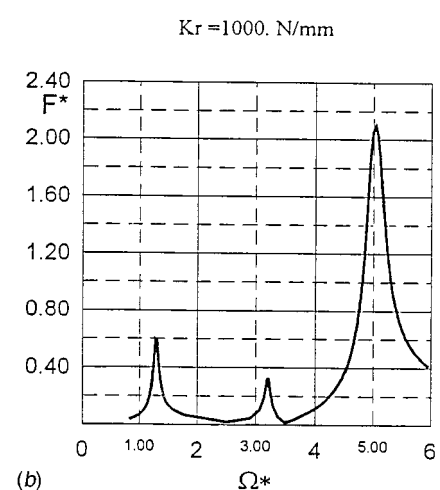
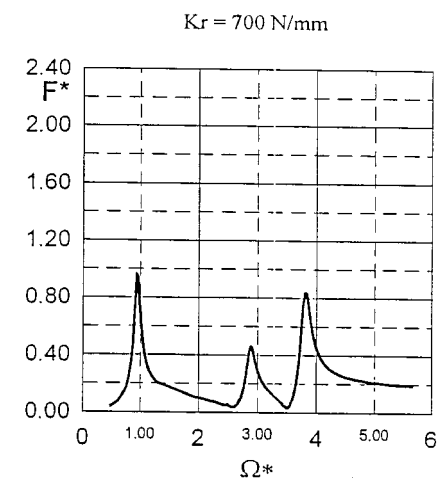
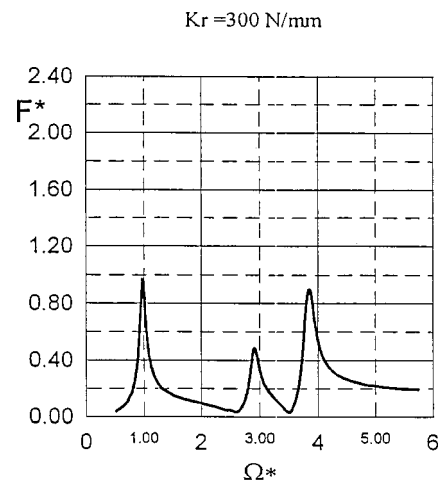
Fig. 5 (b) the effect of damper clearance on the damper forces, short uncavitated damper (SU-model)

unbalance force in exciting this mode is minor. Also Fig. 3 reveals that the vibration amplitudes and the damper force are generally high at the fourth mode.

The unbalance response of AGTFR, in case 42, is illustrated in Fig. 4. In this case the unbalanced disk is considered at the first stage fan rotor while the damper location remains fixed. It is noted that the vibration amplitudes and the damper force are decreased from those in case 41. This would agree with the basic knowledge



(a)



(b)

Fig. 6 (a) The effect of retainer spring stiffness on the vibration amplitude, short uncavitated damper (SU-model)

Fig. 6 (b) the effect of retainer spring stiffness on the damper forces, short uncavitated damper (SU-model)

of rotordynamics as well as the rotor mode shapes. Also this phenomenon is repeated in case 43. All the 15 cases were presented in ([18]), but the main results are obtained here.

For cases 46 and 47, where the unbalance is at the first stage turbine disk, it is observed ([18]) that the first mode disappears while the second mode appears. Also it is observed that the amplitudes of vibrations and damper force are getting lower.

It can be concluded from the cases where the damper is at the Fan bearing location 4, that the most dangerous place for the

unbalance is at the fan location (point 1). This conclusion can be simply obtained since the highest vibrations amplitudes are in case 41, which is shown in Fig. 3. Thus, in the design process of SFD, for AGTFR, it is advised to study the response due to the unbalance at the fan since it gives the most severe vibrations, and in practice the balancing of the fan rotor should be conducted with close tolerances.

The obtained conclusion can be also observed in cases 51 through 57, respectively ([18]). Moreover, the most important

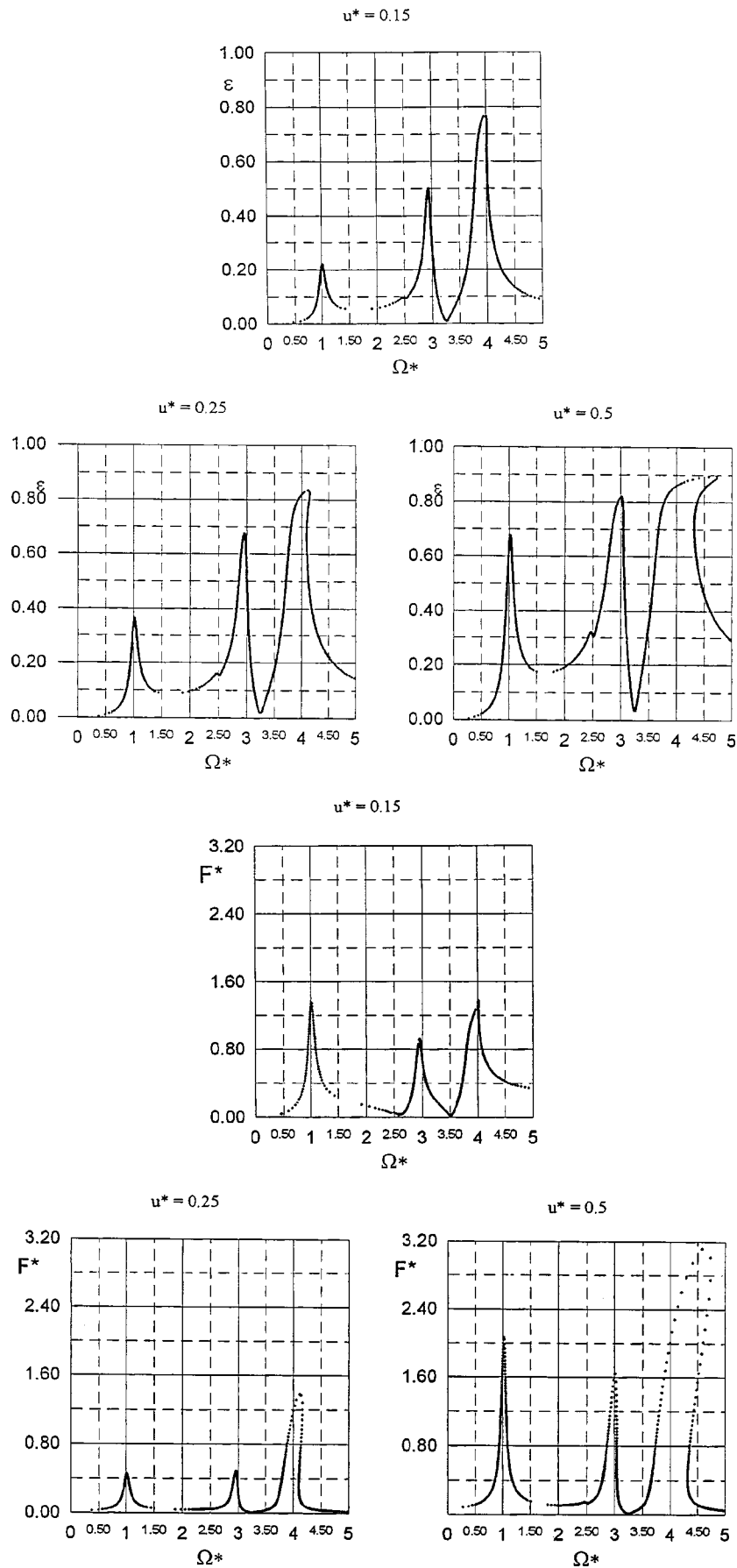


Fig. 7 (a) The effect of the unbalance on the vibration amplitude, short cavitated damper (SC-model); (b) the effect of the unbalance on the damper forces, short cavitated damper (SC-model)

conclusion, which can be obtained from this group of cases, is that the contribution of SFD in damping the rotor vibrations is minimal. It is observed from the analysis of the rotor unbalance response, which is illustrated for each case that the amplitudes of the rotor vibrations and damper force are higher than in the previous cases. This is because the damper is located near a common node of all the four modes, thus the damper will not be effective in controlling the vibrations of these modes.

This conclusion is confirmed by placing the damper at the turbine end bearing which constitutes cases 81 to 87 ([18]). Here it was concluded that the damper is effective in controlling the second mode effectively (at $\Omega^*=2.5$); however, it has a small effect on other modes since there is little deflection at the other three modes ([18]).

Through this parametric study of the vibrations of AGTFR, two points can be concluded: First the dangerous place of the unbalance is at the rotor fan, at which the rotor would exhibit severe vibrations, second the most appropriate location of SFD, at which it would successfully perform its task in damping vibrations, is at the fan end bearing, or at the turbine end bearing if it is desired to dampen the second mode. These two conclusions are very important in the design process of SFD. It should be emphasized that to generate all 15 cases and the conclusions in less than a day's work ([18]) can only be achieved by the developed algorithm, and confirms the usefulness and power of the developed algorithm.

The Effect of SFD Parameters on the Unbalance Response

The developed algorithm is also employed in this section to perform parametric studies for SFDs. Parametric studies is an important process to study the effect of SFD different parameters on the unbalances response. SFDs have three effective parameters which highly influence the performance of the damper. These parameters are (1) the damper radial clearance (c); (2) the journal length (L); and (3) the stiffness of the retainer spring (Kr). It is assumed that the viscosity and the damper radius are determined by other design considerations in the aircraft engine and are dictated on the SFD design.

Similar parametric studies were performed previously on simple models, by Taylor and Kumar [3] on a rigid rotor and by El-Shafei [4,5] on a flexible rotor exhibiting a single mode. In this section, parametric studies are performed on the aircraft gas turbine fan rotor (AGTFR), which exhibits multiple modes, case 41.

The damper radial clearance (c) is a significant factor that hugely affects the amount of damping provided by the SFD. Consequently, the vibration amplitude and the damper force are considerably changed by changing the damper clearance. Figure 5 illustrates the unbalance response at the center of the damper journal (ε) for different values of the damper clearance. It is obvious that as the damper clearance increases, the vibration amplitude (ε) increases. This is because the bearing parameter B is inversely proportional to the cubic of the damper clearance. Also shown in Fig. 5, the nondimensional damping force which changes considerably with c , but note, however, that F^* is defined to be $F_d^* = F_d / m_1 c \omega_{n1}^2$, thus the effect of c should be evaluated for dimensional quantities.

The effect of the journal length (L) on the unbalance response of AGTFR was studied in ([18]). It was shown that the vibration amplitude ε decreases with the increase in the journal length, while the damper force (F^*) increases with the journal length ([18]).

The retainer spring adds flexibility at the SFD location. The effect of the retainer spring is essentially to center the journal in the damper. However, the value of the retainer spring stiffness has to be well chosen, since increasing the value of the stiffness results in larger vibration amplitudes (El-Shafei [4,5]), up to the

point of changing the characteristics of the whole rotor system as shown in Fig. 6. Actually, the retainer spring is a very good tool for placing the critical speeds.

Nonlinear Response

All the previous parametric studies show well-behaved nonlinear response, because the damper parameters are well chosen, the unbalance values are relatively low, and the damper is not cavitated. For cavitated dampers at high unbalance values the jump resonance can occur. Figure 7 illustrates the nonlinear jump resonance phenomenon, i.e., multiple steady-state orbits at the same speed. It is very hard to obtain these multiple orbits with other numerical methods, because of the multiple steady states, which would require trial and error in other numerical methods. However, the developed algorithm does not require a trial and error procedure, and obtains the response directly. This is an additional advantage of the developed algorithm. It can be seen from Fig. 7 that the jump resonance occurs for multiple modes.

Conclusions

This paper described an extremely fast algorithm to obtain the nonlinear unbalance response of rotor-SFD systems. The system differential equations are constructed using the planar modal analysis theory. By assuming circular centered orbit, the differential equations reduced to algebraic ones. Extensive algebraic manipulations are exerted on these equations to constitute a polynomial in the rotor spin speed. By finding the roots of this polynomial, the full unbalance response is obtained for rotor-SFD systems. This method is extremely fast compared to numerical integration and iterative methods, showing 25 orders of magnitude computer time savings. This allows the developed method to be used in performing parametric studies and optimum design of SFDs.

Several important points can be concluded from the parametric studies:

- 1 The radial clearance is the most important parameter determining the amount of damping provided by the SFD.
- 2 The retainer spring should be as soft as possible, since at high retainer spring stiffness little motion occurs in the damper, thus reducing the damping capability.
- 3 The developed algorithm can easily determine multiple steady states represented by the jump resonance. Multiple modes can exhibit this phenomenon in SFDs.

Acknowledgment

The authors would like to acknowledge the fruitful discussions with Prof. M. Elaraby.

Nomenclature

- a_i = polynomial coefficients
- $[A]$ = inverse of receptance matrix
- B = bearing parameter = $\mu R_b L^3 / m_1 \omega_{n1} c^3$ for short damper, nondimensional = $\mu R_b^3 L / m_1 \omega_{n1} c^3$ for long damper
- c = clearance, m
- C_{rt}, C_{tt} = damping coefficients, Ns/m
- C_{rt}^*, C_{tt}^* = damping coefficients, nondimensional
- d = deflection at the location of the unbalanced disk, m
- e = journal eccentricity, m
- fac = scaling factor, nondimensional
- F_d = SFD damping force, N
- F_d^* = $F_d / m_1 c \omega_{n1}^2$ damping force, nondimensional
- F_{dr}, F_{dt} = damping force in both radial and tangential directions respectively, N
- F_u = unbalance force, N

$F_u^* = F_u / m_1 c \omega_{n1}^2$ = unbalance force, nondimensional
 $\{f\}$ = n by 1 complex force vector including the external forces, N
 $\{f^*\}$ = n by 1 force vector including the external forces, nondimensional
 $[G]$ = n by n gyroscopic matrix, Ns/m
 $[I]$ = identity matrix
 $j = (-1)^{1/2}$
 $[K]$ = n by n stiffness matrix, N/m
 m = number of modes
 m_1 = first modal mass, Kg
 m_{di} = mass of disk number i , Kg
 $[M]$ = n by n mass matrix, Kg
 P_1, P_2, P_f = polynomials
 $\{q\}$ = m by 1 complex vector containing modal displacements, m
 $\{\bar{q}\}$ = m by 1 complex vector containing modal displacements, nondimensional
 $\{\bar{Q}\}$ = m by 1 complex vector containing the modal displacements amplitudes, nondimensional
 $R = d/c$ = rotor deflection at the center of a disk, nondimensional
 R_b = damper inner radius, m
 t = time, s
 u = unbalance, m
 $u^* = u/c$ = unbalance, nondimensional
 $[U]$ = n by m modal matrix including system eigenvectors, nondimensional
 $\{U\}_{\text{LSFD}}$ = eigenvector at the location of SFD, nondimensional
 $\{z\}$ = n by 1 complex vector containing the global displacements, m
 $\{\bar{z}\} = \{z\}/c = n$ by 1 complex vector containing the global displacements, nondimensional
 $\varepsilon = e/c$ = eccentricity ratio, nondimensional
 μ = oil viscosity, Ns/m²
 $\pi = 3.141592654 \dots$
 τ = nondimensional time = $t \omega_{n1}$
 ω_{n1} = rotor first natural frequency, rad/s
 $[\omega^2]$ = m by m diagonal matrix containing square of system eigenvalues, rad/s²
 $[\omega^{*2}] = [\omega^2]/\omega_{n1}^2 = m$ by m diagonal matrix containing square of system eigenvalues, nondimensional
 Ω = rotor rotational speed, rad/s
 $\Omega^* = \Omega/\omega_{n1}$ = rotor rotational speed, nondimensional
 $\Omega^{**} = \Omega^*/fac$ = scaled rotor rotational speed, nondimensional
 $()^*$ = differentiating with respect to time

$()'$ = differentiating with respect to nondimensional time τ

References

- [1] Mohan, S., and Hahn, E. J., 1974, "Design of Squeeze Film Damper Supports for Rigid Rotors," ASME J. Eng. for Industry, **96**, pp. 976–982.
- [2] Taylor, D. L., and Kumar, B. R. K., 1980, "Nonlinear Response of Short Squeeze Film Dampers," ASME J. Lubr. Technol., **102**, pp. 51–58.
- [3] Taylor, D. L., and Kumar, B. R. K., 1983, "Closed Form Steady State Solution for the Unbalance Response of a Rigid Rotor in Squeeze Film Damper," ASME J. Eng. for Power, **105**, pp. 551–559.
- [4] El-Shafei, A., 1990, "Unbalance Response of a Jeffcott Rotor Incorporating Short Squeeze Film Dampers," ASME J. Eng. Gas Turbines Power, **112**, pp. 445–453.
- [5] El-Shafei, A., 1991, "Unbalance Response of Jeffcott Rotor Incorporating Long Squeeze Film Dampers," ASME J. Vibr. Acoust., **113**, pp. 85–94.
- [6] Becker, K. H., and Steinhardt, E., 1990, "A General Method For Rotordynamic Analysis," *Vibration and Wear in High Speed Rotating Machinery*, Kluwer Academic Publishers, Dordrecht, The Netherlands, p. 263.
- [7] Greenhill, L., and Nelson, H. D., 1982, "Iterative Determination of Squeeze Film Damper Eccentricity for Flexible Rotor Systems," ASME J. Mech. Des., **104**, pp. 334–338.
- [8] Mclean, L. J., and Hahn, E. J., 1983, "Unbalance Behavior of Squeeze Film Damped Multi-Mass Flexible Rotor Bearing Systems," ASME J. Lubr. Technol., **105**, p. 22.
- [9] Bayoumi, H. N., 1995, "Development of a Finite Element Program for Nonlinear Rotordynamics Analysis Using Equivalent Linearization," M.Sc. thesis, Cairo University.
- [10] El-Shafei, A., Bayoumi, H. N., and Mansour, A. M. A., 1998, "Nonlinear Rotor Dynamic Analysis Using Finite Elements and Equivalent Linearization," Fifth IFTOMM International Conference on Rotordynamics, Darmstadt, Germany, pp. 175–187.
- [11] El-Shafei, A., and Eranki, R. V., 1994, "Dynamic Analysis of Squeeze Film Damper Supported Rotors Using Equivalent Linearization," ASME J. Eng. Gas Turbines Power, **116**, pp. 682–691.
- [12] Gunter, E. J., Choy, K. C., and Allaire, P. E., 1978, "Modal Analysis of Turborotors Using Planar Modes-Theory," J. Franklin Inst., **305**, pp. 221–243.
- [13] Youssef, R., and El-Shafei, A., 1996, "The Unbalance Response of a Multiple-Mode Rotor Supported on Short Squeeze Film Dampers," presented at the International Gas Turbine and Aeroengine Congress and Exhibition, Birmingham, UK. Paper No. 96-GT-23.
- [14] Hathout, J. P., El-Shafei, A., and Youssef, R., 1997, "Active Control of Multi-Mode Rotor-Bearing System Using HSFDs," ASME J. Tribol., **119**, pp. 49–56.
- [15] IMSL MATH/LIBRARY-FORTRAN Subroutines for Mathematical Applications, Version 1.0, April 1987.
- [16] Murphy, B. T., 1993, "Improved Rotordynamics Unbalance Response Calculations Using the Polynomial Method," *Vibration of Rotating Systems*, ASME, DE-60, pp. 35–42.
- [17] Rabinowitz, M. D., and Hahn, E. J., 1977, "Stability of Squeeze Film Damper Supported Flexible Rotors," ASME J. Eng. for Power, **99**, pp. 545–551.
- [18] Yakoub, Y. R., and El-Shafei, A., 1998, "A Fast Method to Obtain the Nonlinear Response of Multi-Mode Rotors Supported on Squeeze Film Dampers Using Planar Modes, Part I and Part II," presented at the 1998 International Gas Turbine and Aeroengine Congress and Exhibition, Sweden, ASME papers 98-GT-412 and 98-GT-413.
- [19] CRITSPD-PC, Version 1.04, 1987, User's Manual, by RODYN-Vibration, Charlottesville, VA.

Reliable Rotor Dynamic Design of High-Pressure Compressors Based on Test Rig Data¹

N. G. Wagner

Development Mechanics,
DEMAG DELAVAL Turbomachinery,
Duisburg D-47015, Germany

The overall design of high-pressure centrifugal compressors is largely influenced by rotordynamic aspects. Rotor instability may restrict operating speed and/or maximum discharge pressure if the destabilizing effects have not been considered accurately during the design phase. A test rig for high pressures has been designed and operated successfully in order to achieve dynamic labyrinth seal coefficients through simulation of original conditions in every aspect. Details are given of the full-scale test rig, which uses active magnetic bearings as a key feature, as well as results from the comprehensive test program. Later on, these results are employed for the design of a compressor for very high pressures, demonstrating the complexity of this design task. Validation of the labyrinth test data and the rotor dynamic analysis is provided by the results from a PTC 10 class I test on a reinjection compressor. During shop testing, this machine has been run with and without swirl brakes and the test results agree very well with the predictions.

[DOI: 10.1115/1.1373396]

Introduction

The reliability of centrifugal compressors is by far the most important aspect for many users, for example, for those in the offshore industry. Above all in turbomachines with a very high power density, the aerodynamic forces which arise in the seals may endanger the operational reliability.

The literature indicates several examples in which high subsynchronous vibrations have made it necessary to perform subsequent alterations to the machine, e.g., Gelin et al. [1], Memmott [2], Sorokes et al. [3], Zeidan et al. [4]. It would be possible to avoid the costs incurred by this and, above all, the downtimes if precise knowledge of the excitation mechanisms were to be taken into account already at the project stage when performing the rotordynamic design.

What is needed is an analytical calculation of these forces, e.g., by using the CFD method of Athavale et al. [5], Weiser and Nordmann [6], or the bulk-flow method of Millsaps and Martinez-Sanchez [7], Scharrer [8], and others. The problems which have become known, some of them recently, clearly indicate that the calculation models available at present are not yet able to deal with the actual complexity of flow through the seals or are unsuitable for industrial application. Major deviations can be expected above all when performing extrapolations, cf. also Childs and Ramsey [9]. Experimental determination of the flow forces is therefore indispensable. The systematic measurement of the stiffness constants, performed for the first time by Benckert [10] is, however, insufficient. The damping forces likewise play an important part. These have been measured in particular at the Texas A&M University (e.g., Childs and Scharrer [11]) and are described in summarized form in Childs [12]. A uniform understanding of the main influencing variables, but also of the numerous contradictions, can be acquired through precise study of the literature, cf. Childs [12].

A need for comprehensive experimental investigation therefore continues to exist. The problems incurred by extrapolation can be avoided by using test specimens with original geometry and by simulating actual operating conditions. Such experiments have not hitherto been made owing to the very high expenditure involved in fulfilling these requirements.

Test Rig for Dynamic Labyrinth Coefficients

The basic requirements relating to the test stand are to produce significant gas forces and to take measurements which in every respect come as close as possible to the original conditions. The first of these requirements had led to the construction of a high-pressure test stand. Active magnetic bearings are suitable for simulating original operating conditions. They make it possible, independently of the rotor speed, to set the static eccentricity as well as a superimposed circulatory movement with freely adjustable frequency, amplitude and direction of precession. The force can be measured directly via the magnetic bearings. This permits direct calculation of transfer functions without any need for further transducers. From these transfer functions the dynamic seal coefficients are extracted as shown below.

Straight-through, stepped, and staggered labyrinth and honeycomb seals covering exactly the same geometries as utilized in high-pressure compressors were manufactured for the tests. Therefore, no extrapolations are necessary with regard to geometrical parameters.

The test stand is normally supplied with N₂ from a 250 bar accumulator station. The inlet pressure and the back-pressure are each set separately to the desired values using electronically controlled valves.

In addition to the pressures at the casing inlet and outlet, measurement is made of static and total pressures directly in front of the labyrinth inlet, cf. Fig. 1. This makes it possible to determine precisely the swirl at the labyrinth inlet, which is very important because the inlet swirl is a main influencing variable which is often only measured indirectly on other test rigs. More details of the test rig are given in Wagner and Steff [13].

Equations for Identification Purposes. The magnetic bearing controller design and the equations for identification shall be treated only briefly in this paper, while the full mathematical derivation can be found in Wagner and Steff [13] and Wagner and Pietruszka [14].

¹“Best Paper Award” received from the Structures & Dynamic Committee, 1999.

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-150. Manuscript received by IGTI, Mar. 1999; final revision received by the ASME Headquarters, Aug. 2000. Associate Editor: H. D. Nelson.

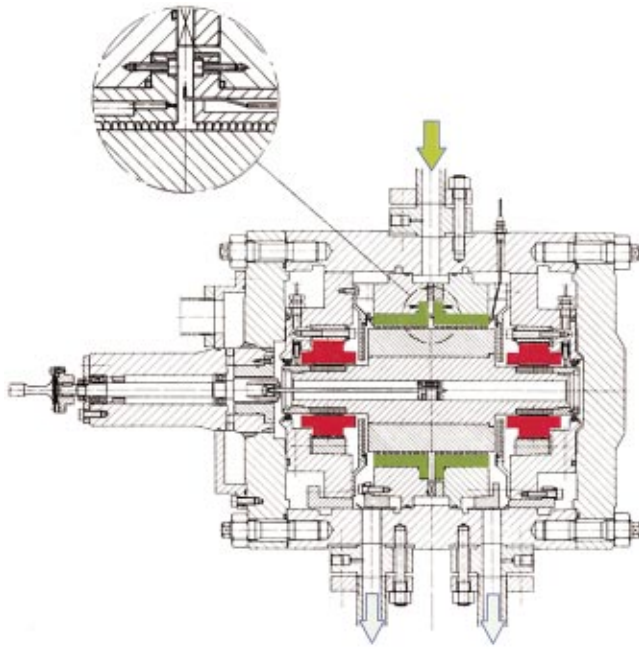


Fig. 1 Sectional view of the test stand

The labyrinth forces are applied as linear functions in displacement (F_k) and velocity (F_d):

$$\begin{aligned} -F_k &= -\begin{bmatrix} F_{kx} \\ F_{ky} \end{bmatrix} = \begin{bmatrix} k_{xx} & k_{xy} \\ k_{yx} & k_{yy} \end{bmatrix} \cdot \begin{bmatrix} x \\ y \end{bmatrix} \\ -F_d &= -\begin{bmatrix} F_{dx} \\ F_{dy} \end{bmatrix} = \begin{bmatrix} d_{xx} & d_{xy} \\ d_{yx} & d_{yy} \end{bmatrix} \cdot \begin{bmatrix} \dot{x} \\ \dot{y} \end{bmatrix} \end{aligned} \quad (1)$$

Only in those special cases where the rotor is centric are the two matrices in Eq. (1) skew-symmetrical:

$$\begin{aligned} d_R &= d_{xx} = d_{yy}, \quad d_Q = d_{xy} = -d_{yx}, \\ k_R &= k_{xx} = k_{yy}, \quad k_Q = k_{xy} = -k_{yx}. \end{aligned}$$

For identification of the stiffness and damping coefficients from Eq. (1) it is found to be practicable to represent the equation of motion and the equation of magnetic force in the sensor coordinates, marked by index "S." After several conversions of the equation of motion and transformation into the frequency domain we obtain finally the algebraic equation

$$+j\omega(d_R - jd_Q) + k_R - jk_Q - \frac{m}{2}\omega^2 + \frac{1}{2}k_a = \Lambda \quad (2)$$

where in

$$\begin{aligned} \Lambda &= q_{Rx} \cdot \frac{X_{SR}}{Z_S} + q_{Lx} \cdot \frac{X_{SL}}{Z_S} + k_{qLx} \cdot \frac{I_{Lx}}{Z_S} + k_{qRx} \cdot \frac{I_{Rx}}{Z_S} \\ &+ j \cdot \left(q_{Ry} \cdot \frac{Y_{SR}}{Z_S} + q_{Ly} \cdot \frac{Y_{SL}}{Z_S} + k_{qLy} \cdot \frac{I_{Ly}}{Z_S} + k_{qRy} \cdot \frac{I_{Ry}}{Z_S} \right) \end{aligned}$$

and

$$Z_S = X_{SL} + X_{SR} + j \cdot (Y_{SL} + Y_{SR})$$

with $q_{R,L,x,y}$, k_q magnetic parameters
 Z_S , $I_{R,L,x,y}$ complex combined vibration and bearings currents.

After separating the real and imaginary components, we obtain two real equations for determining the four labyrinth coefficients which are being sought

$$\omega d_Q + k_R = \frac{m}{2}\omega^2 - \frac{1}{2}k_a + \text{Re}(\Lambda) \quad (3)$$

$$\omega d_R - k_Q = \text{Im}(\Lambda).$$

The identification is thus reduced to a linear regression with ω as variable. The terms relating to damping correspond to the linear gradient and the terms relating to stiffness are obtained directly as an intersection or ordinates.

Prior to commencement of the labyrinth tests, extensive examinations were made to provide assurance about the operational reliability of the test stand and the quality of the measurements.

In order to measure and evaluate the overall transfer function Λ , the system has to be excited to vibrations with the aid of a test signal. The best results were achieved with a sweep sine excitation.

Static and Dynamic Calibration. The coefficients of the linear magnetic force equation depend on the bias current which serves to compensate static forces. Since the gas forces also have a static component, owing to unavoidable eccentricities, the bearing coefficients depend on the respective operating point and are therefore not constants in the usually accepted sense. These coefficients can be calculated specifically for each bearing from the condition of static equilibrium for each operating point.

At the beginning and end of each series of measurements, a dynamic calibration is made in addition to the above static calibration. For the dynamics of the PWM amplifiers, a corrective transfer function is inserted in Eq. (2).

This step can be expressed in an illustrative manner as a frequency-dependent evaluation of the force/current factor. The basis of this dynamic calibration is the equilibrium between the rotor inertia forces and the magnetic forces. This calibration is made upon every change in speed using the actual vibration spectra of the system. A typical result of this corrective transfer function has been provided in Wagner and Steff [15], where an uncertainty analysis can also be found.

Scope of Testing. Various inlet and back pressures are set. The pressure ratio is thus made to vary independently of the pressure level. In addition, by exchanging the swirl rings the circumferential velocity of inlet into the labyrinth is altered. Furthermore, for each setting of pressure the speed is increased progressively from zero up to maximum speed in order that the influence of all essential parameters can be clearly determined. The frequency range of the sweep sine is adapted continuously during the measurement in order to enable the coherence of all signal pairs in the whole frequency range to be as close as possible to the ideal value $\gamma^2 = 1$. This means that the quality of the result can be favorably influenced during the test and the need to repeat a test owing to a poor result of the off-line analysis can be avoided.

Each test evaluation commences with the static and dynamic calibration with the aid of the reference state measurement. This is followed by calculation of the averaged cross and auto spectra from which the overall frequency response Λ in Eq. (2) is finally determined.

The overall frequency response Λ is also filtered before making the linear regression according to Eq. (3), which provides the labyrinth coefficients being sought. When doing this, the frequency lines with a low coherence of the signal pairs are marked in such a manner that they are not taken into consideration for the linear regression.

Upon termination of the measurements with various swirl rings, the test seals are removed from the test stand and their geometry altered. The influence of the number of tips, the gap and the chamber height is measured step by step. Examination is made of straight-through labyrinths and staggered labyrinths in two different diameters as well as of stepped labyrinths and various honey-

combs. In addition to the difference between rotating and stationary sealing tips, the influence of various swirl brakes and of abradable rotor coatings is examined.

A total of 25 different seal designs were examined, with more than 1500 individual measurements being evaluated. This provides a comprehensive basis of data which covers the entire scope of practical applications.

Results From Seal Test Rig

When formulating the labyrinth forces in Eq. (1), a linear relationship was assumed with the rotor deflection and velocity. If this assumption is correct, then the measured overall frequency response must, according to Eq. (3), be a linear function in the angular frequency ω . In Wagner and Steff [15] this linear relationship was proved for labyrinths. This assumption is not valid unrestrictedly for honeycombs, however. In Fig. 2 the imaginary component, which contains the parameter k_Q , d_R relevant for stability, can be described by a straight line to a good degree of approximation. The real component, on the other hand, containing the parameter k_R , d_Q which influences the rotor elasticity, exhibits a clearly superlinear progression. In accordance with this, this curve must either be linearized in the frequency range in question or Eq. (1) must be extended by means of inertia terms. In labyrinth seals, on the other hand, the model indicated by Eq. (1) has been confirmed by measurements.

Usually, the labyrinth coefficients are employed as direct and cross-coupled variables k_R , k_Q , d_R , d_Q in which, with regard to the restoring force $F_R = k_R + \omega d_Q$, a good many more contra-

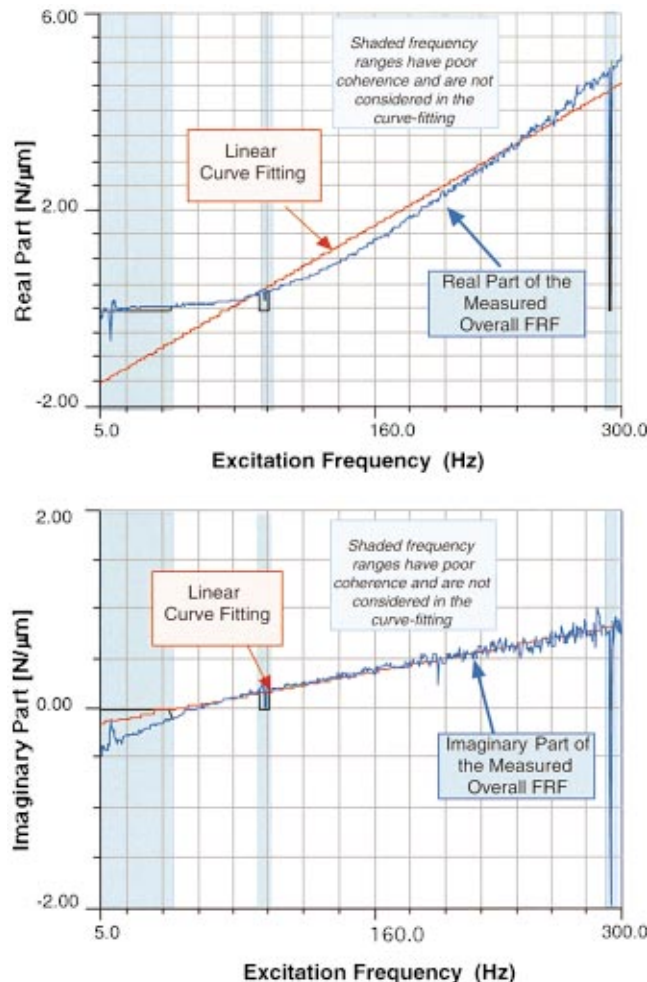


Fig. 2 Measured overall transfer function of a honeycomb seal

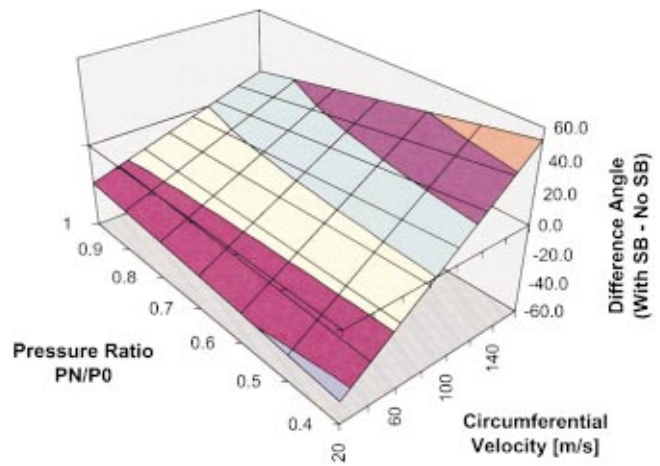


Fig. 3 Effect of curved swirl brake on stiffness angle

dictory statements are present than with regard to the tangential force $F_Q = k_Q - \omega d_R$. In particular, the changes of sign in these coefficients appear to be questionable. We therefore suggest that it would be best to change over from the cartesian components indicated above to a polar representation:

$$k = \sqrt{k_Q^2 + k_R^2}, \quad \tan \vartheta_k = k_Q / k_R \quad (4)$$

with the total stiffness k and the stiffness angle ϑ_k . The total damping d and the damping angle ϑ_d are calculated in a similar manner.

The measuring results relating to the test parameters are correlated in this polar representation. It is thus possible, for example, to represent the influence of the swirl brakes in a clearly illustrative manner. Swirl brakes reduce the destabilization by lowering the amount of total stiffness and total damping and by turning the vector through a given angle. Figure 3 shows as an example that the stiffness vector in a "staggered labyrinth" is rotated positively or negatively, depending on the peripheral speed, by the use of a swirl brake. In respect of a special case, Fig. 4 shows the stiffness and damping vectors with and without swirl brakes. This kind of representation has proved to be very beneficial for assessment of the seal forces because a change of sign in cartesian components due to a parameter variation may be easily explained being as just a change of the vector orientation from one quadrant to another.

After entering the orbit of the rotor, it is easy to designate the quadrants in which the stiffness and damping vectors stabilize or destabilize the rotor and increase or reduce the rotor elasticity.

As opposed to shunt holes, swirl brakes have the advantages of not requiring any additional bearing span and of their effect having been precisely quantified in the test rig presented in this paper,

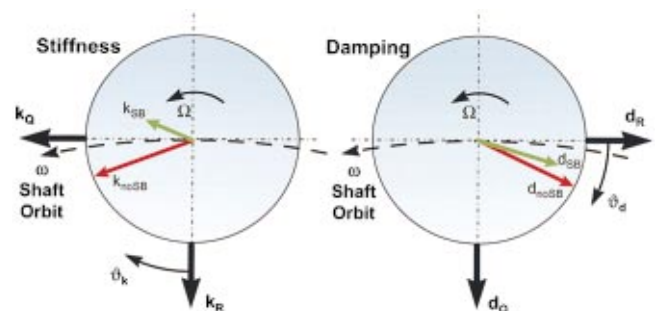


Fig. 4 Stiffness and damping vector with and without swirl brake

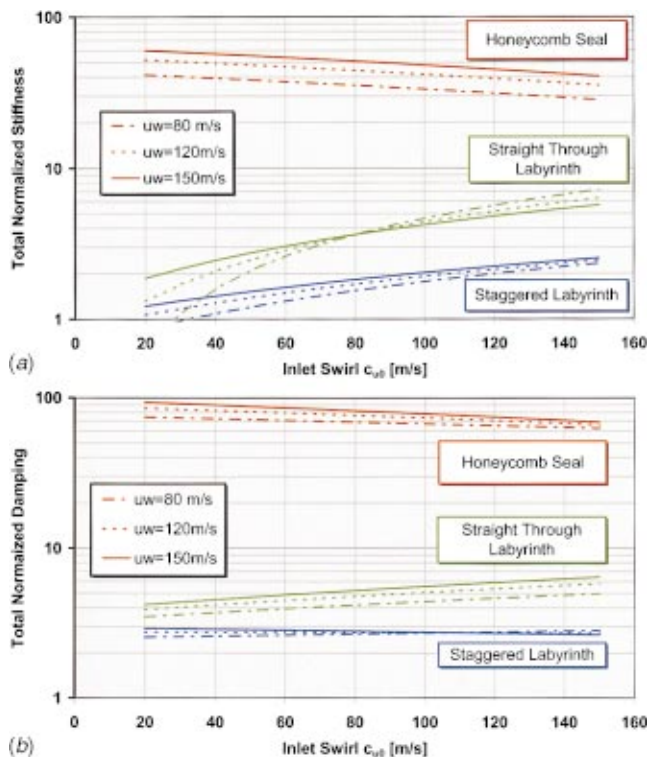


Fig. 5 Total stiffness and total damping for different types of seals

cf. also the data indicated in Wagner and Steff [13]. Moreover, cases are known in which shunt holes have proved to be ineffective, e.g., Gelin et al. [1] and Sorokes et al. [3].

A direct comparison of various seal types is possible using the available test data. Figure 5 shows the results of the established correlations for the total stiffness k and the total damping d in accordance with Eq. (4) for three different seal types in logarithmic representation. This calculation has been performed for a seal with a diameter of 200 mm, a length of 80 mm, a radial gap width of 0.2 mm at an inlet density of 200 kg/m^3 , and a pressure ratio of 0.45. The variations in shaft peripheral velocity and inlet swirl, which form the so-called “swirl ratio” are depicted in this figure. A staggered labyrinth produces the lowest stiffness but, at the same time, exhibits the lowest degree of damping. The forces which arise with honeycombs are greater than the labyrinth forces by one order of magnitude or more. Basically, the inlet swirl has a larger influence on the stiffness than on the damping.

The radial and tangential forces can also be calculated and compared with the aid of the stiffness and damping angles ϑ_k and ϑ_d . The characteristics of honeycomb seals differ considerably from

those of labyrinth seals. The restoring force of the former is several times higher than the tangential force and, owing to the very high cross-coupled damping, is highly dependent on the oscillation frequency. At low frequencies, the rotor stiffness is lowered, and at high frequencies it is increased. The tangential force always has a stabilizing effect in the honeycombs examined.

For rotordynamic calculation, honeycombs cause problems when they are employed at high densities. The spring and damping coefficients of the seal may then reach larger values than those of the journal bearings. Owing to unavoidable eccentricities, high static forces arise in the honeycomb, and these seriously affect the static bearing forces. A precise knowledge of the static bearing forces, however, is an important precondition for making rotordynamic calculation. Above all in light rotors, which are customary in the high-pressure range, critical speeds and log decrements may be heavily dependent on the installed seal eccentricity. If, furthermore, consideration is given to the extraordinarily strong influence of honeycomb geometry on the dynamic coefficients [16], then yet more uncertainties arise which may make it difficult for the rotordynamic analysis to agree with measured results. So the basic purpose of the rotordynamic analysis may not be met in such cases.

Contrary to this, the forces which arise in labyrinths are considerably smaller and therefore better able to be kept under control. The use of swirl brakes enables the destabilizing forces to be reduced to a sufficiently low level even in extreme cases.

Application of Test Results on a High-Pressure Compressor

The following example shows the application of the previously described test results on a reinjection compressor for a pressure rise from 400 bar to 600 bar. This compressor is a four-stage in-line machine with a power of 8.5 MW. The discharge density is 330 kg/m^3 . Shaft sealing can be provided by a dry gas seal or, alternatively, by a double-acting mechanical contact seal with a closed high-pressure loop (type “DEMPAC”). The rotor runs at 10,025 rpm and is supported in standardized four-segment tilting-pad bearings. Figure 6 shows the model used for the rotordynamic calculation.

A proven design concept is available through the design of the rotor with maximum stiffness. The flexibility ratio of this rotor is $N_{\text{max,cont}}/N_{\text{c,rigid}} = 1.06$ and the rotor deflection at midspan is only 1.6 times the bearing amplitude in the first mode. This results in a high level of logarithmic decrement, and the sensitivity to labyrinth excitation is lower than in an elastic rotor. The critical speed is in the vicinity of the operating speed as a result of such a design. In the majority of applications, the operation of a compressor close to resonance is avoided, though if an adequate reserve of damping is present, a rotor can definitely be operated close to resonance without risk. Stability under all operational conditions is, however, an essential requirement here since, if any

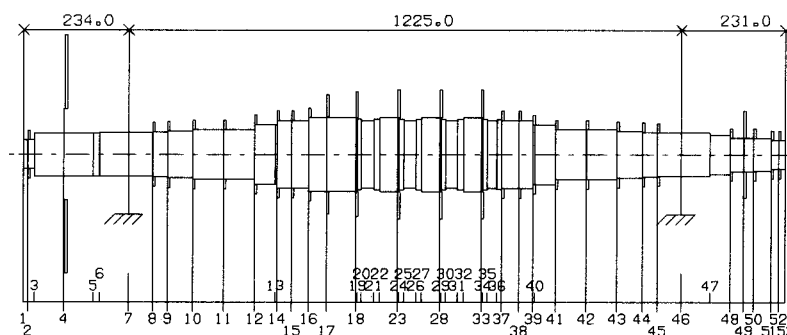


Fig. 6 Rotor model

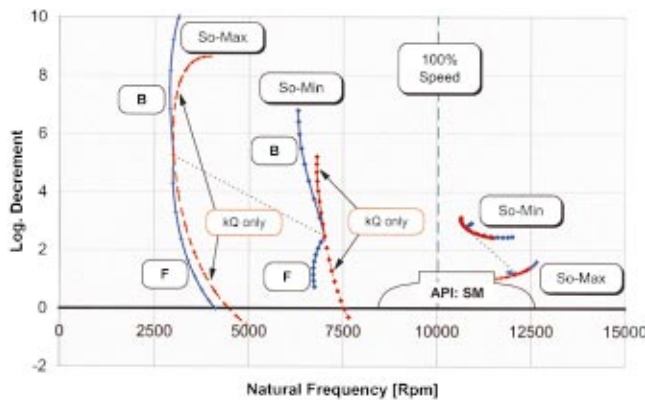


Fig. 7 Effect of increasing gas density from 0 percent to 200 percent on eigenvalues

instability should arise, this may cause the installation to shut down and the risk of damage to the machine cannot be excluded.

The stability analysis must therefore result in verification of adequate damping also for the most unfavorable operating situations. The manufacturing tolerances of the journal bearings play an essential part above all in the relatively light high-pressure compressors. Account must also be taken of the permissible variations in the oil temperature. Figure 7 shows the course of the eigenvalues in the complex plane when the density increases from zero to double the design value.

The calculations for the minimum ($So=0.05$) and maximum ($So=0.08$), Sommerfeld numbers indicate the influence of the bearing tolerances and of the oil temperature. The lower eigenvalue reacts much more sensitively to the Sommerfeld number than does the upper eigenvalue, and the same applies with regard to the influence of the gas forces. In the case of the upper eigenvalue, the gas forces indeed cause the position of the natural frequency to shift, yet the log decrement is reduced only slightly. The separation margin required under API 617 is observed even with double the design density.

For the lower eigenvalue, taking the case of a machine without load, the splitting of the double eigenvalue into the forward and backward modes is very clearly recognizable when the gas forces increase with increasing density. The greatest destabilization of the forward mode occurs at the maximum So number.

Figure 7 shows curve branches based on a calculation with a complete stiffness and damping matrix according to Eq. (1) or on a simplified calculation in which only the cross-coupled stiffness k_Q has been considered. As can be expected, the calculation with complete matrices results in a lower degree of destabilization and, in addition, it shows the influence of the gas forces on the natural

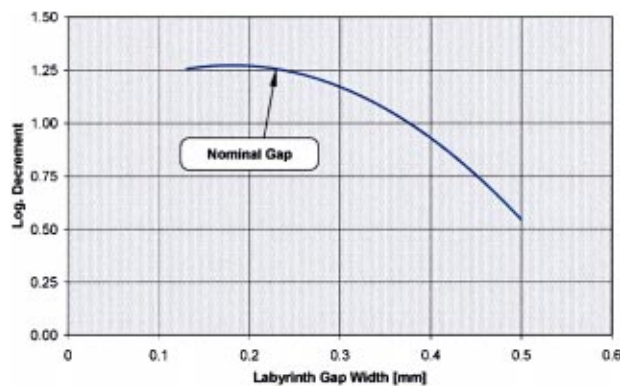


Fig. 8 Effect of labyrinth gap width on rotor stability

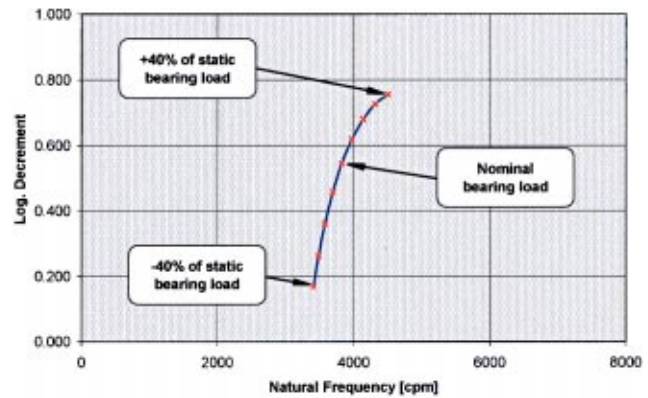


Fig. 9 Effect of side loads on rotor stability

frequencies. Owing to the very rigid rotor, this example only demonstrates a relatively small difference between these two calculations. In the case of a more elastic rotor, consideration of the complete matrices has a more serious effect than it does here. The same applies to the use of honeycombs at high gas densities, if the spring and damping coefficients extend into the same range of magnitude as the journal bearing coefficients.

For the impeller eye seals and the balance piston labyrinth, the spring and damping coefficients for various clearance widths are calculated on the basis of experimental data. The importance of examining the influence of clearance width has been described by Zeidan et al. [4]. They report that after the clearance had become enlarged through rubbing, a compressor exhibited high subsynchronous vibrations, and the theory failed when the labyrinth excitation was calculated for large clearance widths.

Figure 8 shows the course of the minimum logarithmic decrement over the clearance width for the most unfavorable combination of bearing geometry and oil supply temperature. An adequate damping reserve is still present even after the labyrinth clearance has become enlarged to 200 percent of the nominal value.



Fig. 10 Photograph of the reinjection compressor

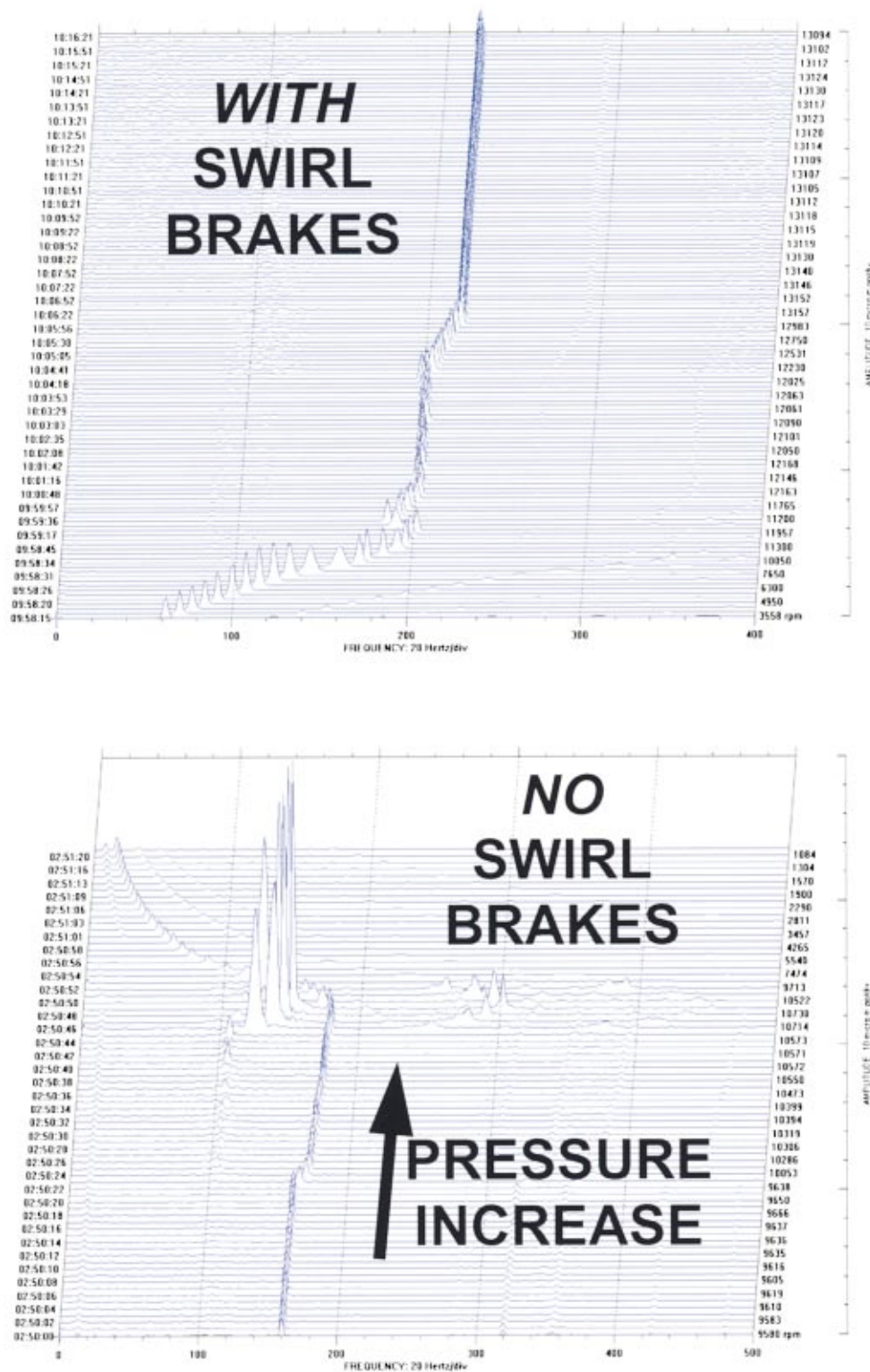


Fig. 11 Cascade plots for test run with and without swirl brakes

A further factor to be considered is that an asymmetrical distribution of pressure will arise in the volute. This may cause the static bearing force to alter, depending on the position of the outlet nozzle. Figure 9 shows, on the complex plane, the influence of such static gas forces on the eigenvalue with the smallest damping.

The basis of this representation is the assumption of the most unfavorable bearing geometry and oil temperature and of dam-

aged labyrinths. Even for this case of an assumed accumulation of worst-case conditions, an adequate damping reserve is still available.

Verification of Test Results on a FPSO Reinjection Compressor

In 1997 the company of the author received an order to supply a reinjection compressor with a design pressure of 418 bar. This

machine is envisaged for installation on an FPSO ship in the North Sea. The compressor in question is a five-stage in-line machine with a power requirement of 6.2 MW. The flexibility ratio of this rotor is $N_{\max, \text{cont}}/N_{c, \text{rigid}} = 1.78$, the amplitude ratio between midspan and bearing is 1.9 and the Sommerfeld number ranges from 0.06 to 0.11.

In view of the very short delivery period agreed upon, the designing of the compressor was commenced already during the negotiation stage. From a rotordynamic point of view, it was highly unfavorable that the customer should prefer the use of a triple dry-gas seal. During the stability analysis, the relatively large bearing span resulted in insufficient stability on assuming worst-case conditions.

The simplest solution to this problem was found to be the utilization of swirl brakes on shroud and balance-piston labyrinths. The swirl ratio present at the balance piston inlet had a relatively high value of $c_{u0}/u_w = 0.95$. Prediction of the swirl brake efficiency yielded a reduction of the cross-coupled stiffness for this operating condition by about 70 percent. This enabled adequate stability to be verified by calculation. The spring and damping coefficients of the labyrinths with and without swirl brakes are based on the results obtained from the test rig described above. The operational reliability of this machine thus depended decisively on whether or not the test data and, in particular, the effect of the swirl brakes coincided to an adequate degree with the conditions present in the machine.

Since a PTC 10 class 1 test with a hydrocarbon mixture was envisaged for the machine (Fig. 10), it was decided that the reliability of the stability analysis should be checked. For this purpose of verification, labyrinths without swirl brakes were manufactured in addition to those with swirl brakes.

For the first test run, the swirl brakes were installed and the machine was successfully tested. No noteworthy subsynchronous vibrations were ascertained at any of the operating points, cf. upper part, Fig. 11. The logarithmic decrement calculated for this condition was $\delta = 0.3$. For the second test run, the labyrinths without swirl brakes were installed. The stability analysis predicted the onset of the instability for the assembly without swirl brakes at 100 percent speed at discharge pressures between 240 bar and 320 bar, depending on supply oil temperature. The frequency of the mode, which becomes unstable, was predicted around 6000 cpm, also depending on supply oil temperature. By means of this configuration, and in harmony with the stability analysis, the stability limit was able to be approached in three different ways:

- 1 increasing of the speed at constant discharge pressure and constant oil temperature,
- 2 increasing of the discharge pressure at constant speed and constant oil temperature, and
- 3 increasing of the oil temperature at constant speed and constant discharge pressure.

A subsynchronous vibration in the magnitude of approximately $10 \mu\text{m}_{p-p}$ was considered to be the dimensional quantity for reaching the stability limit, in order to prevent the machine from being put unduly at risk. However, in one of the trials the step size of the pressure increase was too large and a high subsynchronous vibration arose quickly, resulting in a trip. From the lower part of Fig. 11 it can be ascertained that the frequency of the subsynchronous vibration increases after the onset of the instability. This can be explained by mechanical rub at the balance piston, which provides the effect of a bearing span reduction and thus raises the frequency of this vibration. Fortunately, only minor damage to the test seals without swirl brakes was observed.

Basically, all three methods of approaching the stability limit lead to the same result, only with slight differences in the frequency of the subsynchronous vibration. The experience gained from this and from other test runs shows that a very careful and slow parameter change enables observation of continuous growth

of the subsynchronous vibration up to a certain limit, at which the amplitudes grow suddenly to the excessive values of the corresponding limit cycle.

The validation of the test rig data on this machine is of great importance for the reliability and transferability of the test results. The disadvantages of having large safety reserves can be avoided through the application of these test results, without having to run the risk of instability occurring.

More details of the verification testing are provided in Wagner et al. [18].

Conclusion

The intention of this paper was to demonstrate reliable rotor dynamic design as a whole, from systematic R&D efforts up to verification in high-demand applications.

The key objective of this development project was to make available reliable labyrinth coefficients for the stability analysis. This aim was realized by developing and operating a test rig that enabled the original conditions to be simulated in every respect. The data basis generated on this test rig covers the entire range of application.

Use was made of an example to demonstrate the peculiarities relating to the rotordynamic design of a high-pressure compressor. The test data were validated by means of a full-load test on a reinjection compressor which was operated purposefully up to the stability limit.

Acknowledgments

The author wishes to thank the management of Demag Delaval Turbomachinery for authorizing the publication of this development project. The author acknowledges the contributions of Klaus Steff and Volker Hütten to this project. Results from the Class I test on the FPSO compressor have been provided by Delaval Stork V.o.f., Hengelo, NL.

Nomenclature

- d = damping coefficient
- F = labyrinth seal force
- k = stiffness coefficient
- m = mass of the test rotor
- q = parameter of the active magnetic bearing
- x, y = displacement of the rotor
- Λ = overall transfer function
- ϑ = angle of total stiffness and damping coefficient
- ω = angular frequency

Indices

- Q = cross-coupled coefficient
- R = direct coefficient
- SL, SR = sensor signal from left/right bearing

References

- [1] Gelin, A., Pugnet, J.-M. Bolusset, D., and Friez, P., 1996, "Experience in Full-Load Testing Natural Gas Centrifugal Compressors for Rotordynamic Improvements," ASME Paper 96-GT-378.
- [2] Memmott, E. A., 1992, "Stability of Centrifugal Compressors by Applications of Tilt Pad Seals, Damper Bearings, and Shunt Holes," C432/070 IMechE.
- [3] Sorokes, J. M., Kuzdzal, M. J., Sandberg, M. R., and Colby, G. M., 1994, "Recent Experiences in Full Load Full Pressure Shop Testing of a High Pressure Gas Injection Centrifugal Compressor," *Proc. of the 23rd Turbomachinery Symposium*, Texas A&M University, College Station, TX.
- [4] Zeidan, F. Y., Perez, R. X., and Stephenson, E. M., 1993, "The Use of Honeycomb Seals in Stabilizing Two Centrifugal Compressors," *Proc. of the 22nd Turbomachinery Symposium*, Texas A&M University, College Station, TX.
- [5] Athavale, M. M., Przekwas, A. J., and Hendricks, R. C., 1993, "A 3D-CFD Code for Accurate Prediction of Fluid Flows and Fluid Forces in Seals, Rotordynamic Instability Problems in High-Performance Turbomachinery," NASA Conf. Publ. 3239, pp. 137–148.
- [6] Weiser, P., and Nordmann, R., 1990, "Determination of Rotordynamic Coefficients for Labyrinth Seals and Application to Rotordynamic Design Calculations, Rotordynamic Instability Problems in High-Performance Turbomachinery," NASA Conf. Publ. 3122, pp. 203–222.

- [7] Millsaps, K. T., and Martinez-Sanchez, M., 1993, "Rotordynamic Forces in Labyrinth Seals: Theory and Experiment, Rotordynamic Instability Problems in High-Performance Turbomachinery," NASA Conf. Publ. 3239, pp. 179–208.
- [8] Scharer, J. K., 1987, A "Comparison of Experimental and Theoretical Results for Labyrinth Gas Seals," doctoral dissertation, Texas A&M University.
- [9] Childs, D. W., and Ramsey, C., 1990, "Seal-Rotordynamic-Coefficient Test Results for a Model SSME ATD-HPFTP Turbine Interstage Seal With and Without a Swirl Brake, Rotordynamic Instability Problems in High-Performance Turbomachinery," NASA Conf. Publ. 3122,, pp. 179–190.
- [10] Benckert, H., 1980, "Strömungsbedingte Federkennwerte in Labyrinthdichtungen," doctoral dissertation, University of Stuttgart, Stuttgart, Germany.
- [11] Childs, D. W., and Scharer, J. K., 1986, "Experimental Rotordynamic Coefficient Results for Teeth-on-Rotor and Teeth-on-Stator Labyrinth Gas Seals," ASME Paper 86-GT-12.
- [12] Childs, D. W., 1993, *Turbomachinery Rotordynamics, Phenomena, Modeling and Analysis*, John Wiley and Sons.
- [13] Wagner, N. G., Steff, K., 1996, "Dynamic Labyrinth Coefficients From a High-Pressure Full-Scale Test Rig Using Magnetic Bearings, Rotordynamic Instability Problems in High-Performance Turbomachinery," NASA Conf. Publ. 3344, pp. 95–112.
- [14] Wagner, N. G., and Pietruszka, W. D., 1998, "Identification of Rotor-Dynamic Parameters on a Test Stand With Active Magnetic Bearings," *Proceedings of the First International Symposium on Magnetic Bearings*, Zurich, Springer-Verlag, New York, pp. 289–299.
- [15] Wagner, N. G., and Steff, K., 1997, "Bestimmung der dynamischen Labyrinthkoeffizienten unter realistischen Betriebsbedingungen mit Hilfe aktiver Magnetlager, Schwingungen in rotierenden Maschinen," *Proceedings of Conference SIRM'97*, Irretier, Nordmann, Springer, Vieweg-Verlag Braunschweig/Wiesbaden, Germany.
- [16] Childs, D. W., Elrod, D., Hale, K., 1988, "Annular Honeycomb Seals: Test Results for Leakage and Rotordynamic Coefficients: Comparisons to Labyrinth and Smooth Configurations, Rotordynamic Instability Problems in High-Performance Turbomachinery," NASA Conf. Publ. 3026, pp. 143–159.
- [17] Childs, D. W., Baskharone, E., and Ramsey, C., 1990, "Test Results for Rotordynamic Coefficients of the SSME HPOTP Turbine Interstage Seal With Two Swirl Brakes, Rotordynamic Instability Problems in High-Performance Turbomachinery," NASA Conf. Publ. 3122, pp. 165–178.
- [18] Wagner, N. G., de Jongh, F. M., and Moffatt, R., 2000, "Design Testing and Field Experience of a High-Pressure Natural Gas Reinjection Compressor," *Proc. of the 29th Turbomachinery Symposium*, Texas A&M University, College Station, TX.

Comparison of the Dynamic Characteristics of Smooth Annular Seals and Damping Seals

J. M. Darden

E. M. Earhart

NASA Marshall Space Flight Center,
TD55, Building 4203,
MSFC, AL 35812

G. T. Flowers

Assoc. Professor,
Department of Mechanical Engineering,
Auburn University,
Auburn University, AL 36849-5341

Annular seals are known to enhance rotordynamic stability margins and minimize vibration response levels in high-speed rotating machinery. Theoretical predictions for the rotordynamic characteristics of annular seals exist but additional experimental data is needed to properly anchor these results. NASA's Marshall Space Flight Center (MSFC) has developed an annular seal test rig and facility to experimentally characterize axially fed annular seals. Annular seals with deliberately roughened stators (i.e., damping seals) have been shown analytically to increase stability margins of rocket engine turbomachinery by reducing the seal's whirl frequency ratio. The capabilities of MSFC's annular seal test rig have been enhanced to allow high fluid inlet preswirl testing that is more representative of actual turbopump seal boundary conditions. The purpose of this paper is to describe the effect of this realistic preswirl on the stabilizing capability of both damping and smooth seals. Centered seal results are presented for both a smooth annular seal and a damping seal. These results were obtained for a range of seal pressure differentials, shaft rotational speeds, and two levels of inlet fluid preswirl. [DOI: 10.1115/1.1383256]

Introduction

Vibration problems on the space shuttle main engine (SSME) turbomachinery have led engineers to seek ways to enhance rotordynamic stability margins and minimize vibration response levels. In response to these problems, engineers have employed annular seals in the SSME turbomachinery. One of the primary functions of the annular seal is to enhance rotordynamic stability margins. To that end, MSFC's annular seal test rig was modified to more realistically simulate the actual operating environment of a turbopump. The reconfigured rig has the capability to provide highly prerotated fluid at the entrance to a test seal. This condition is encountered frequently in turbomachinery applications. The fluid preswirl ratio is defined as the tangential velocity component of the fluid divided by the rotor surface speed at the entrance to a seal. Typically, the fluid in a turbopump has a significant tangential velocity component (in contrast with the axial velocity down the length of the seal) prior to entering the annular seal. A common example of a high preswirl condition is where fluid entering a seal is supplied by the discharge from an impeller.

High fluid preswirl is a significant source of destabilizing forces in a turbopump. In rotordynamic terms, this condition tends to reduce stability margins resulting in undesirable vibration response levels and could lead to dynamic instabilities. Experimental data is needed to assess the capacity of candidate annular seals to minimize whirl frequency ratios in the presence of a highly prerotated fluid inlet condition. Experimental results from annular seal testing is scarce in the technical literature, in general, but is particularly lacking high fluid preswirl test results. The objective of the present investigation, therefore, is to gain insight into the ability of annular seals to stabilize. Both a round-hole pattern damping seal and a smooth annular seal were tested under different (high and low) fluid preswirl conditions. Experimental results from the smooth seal served as a baseline from which the round-hole pattern damping seal could be assessed.

Background

Marquette, Childs, and San Andres [1] presented dynamic coefficients for a smooth annular seal over a wide range of speeds, pressures, and eccentricity ratios. Their work indicated that the rotordynamic coefficients for smooth annular seals are strongly dependent upon eccentricity. The experimental results showed a stronger dependence on eccentricity than the corresponding theoretical predictions. Childs and Fayolle [2] presented experimental results on two damping seals with a round-hole stator roughness treatment. The difference between the two seals was the depth of the flat bottom holes. Testing revealed that both seals had direct stiffness values much lower than the theoretical predictions. Moreover, the shallow depth hole pattern damping seal had a direct stiffness on the order of zero. The results also showed a general lack of dependence upon seal eccentricity on the experimentally obtained rotordynamic coefficients. Based upon the previously described work, additional experimental results are needed to anchor seal prediction codes.

Test Facility

MSFC's annular seal test rig is shown in Fig. 1. The rig supports testing of seal pressure differentials (ΔP) of over 138 bars (2000 psi) in de-ionized water. The test rig's shaft is powered by a 186 kilowatt (250 horsepower) steam turbine that provides a maximum operational speed range of just over 20,000 rpm. High-pressure de-ionized water with temperatures typically between 21.1°C to 32.2°C (70°F–90°F) is supplied to the test rig via a 11356 liter (3000 gallon) main water reservoir. The main water reservoir is pressurized by gaseous nitrogen. The water flows from the main reservoir through a bank of filters and then to the test rig. The water is routed to the upper and lower slave bearings as well as the test bearing.

The test bearing is mounted in the seal carrier assembly that is located approximately midspan of the shaft. The seal carrier assembly is supported axially by upper and lower orifice-compensated thrust bearings. This arrangement ensures that the seal carrier assembly remains parallel to the shaft during testing. It also allows the test bearing to translate in response to an externally applied load without having to account for additional load paths. The hydrostatically supported seal carrier assembly allows for a straightforward inertial correction to the excitation force.

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-177. Manuscript received by IGTI, October 1998; final revision received by the ASME Headquarters, March 1999. Associate Editor: D. Wisler.

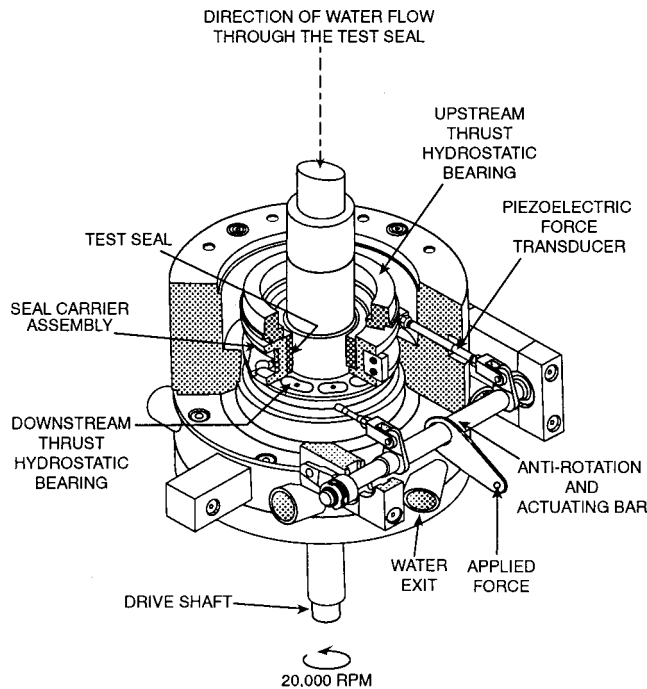


Fig. 1 Cut-away view of test rig

The highly prerotated fluid state was achieved by replacing the rig's shaft with a new shaft that incorporates ten radial fins located just upstream of the seal entrance. The radial fins are equally spaced around the circumference of the shaft. The symmetric fins have a height of 0.84 cm (0.33 in.) in the radial direction and a length of 5.08 cm (2.0 in.) down the long axis of the shaft. There is a 2.54 cm (1.0 in.) long cavity defined by the exit plane of the fins and the entrance plane of the seal to allow fluid mixing prior to its entry into the seal. The radial clearance between the fin tips and their mating housing is on the order of 0.13 cm (0.050 in.). Blade passing frequencies are well outside the frequency range of interest (20–200 Hz) so that the quality of the frequency response functions are not degraded.

Test Procedure

Candidate test seals were characterized at twelve operating conditions. The operating conditions consisted of a range of shaft rotational speeds (3978 rpm to 15,764 rpm), seal pressure differentials ranging from 92.9 bars (1347 psi) to 153.6 bars (2228 psi), and two different inlet preswirl conditions. The applied force is a band-limited random excitation between 20–200 Hz. High-frequency data, comprised of displacements, forces and pressures, is recorded on an analog tape recorder. The duration of each test is approximately 30 seconds. Following a test series, the measured data is digitized from the analog tape. The recorded measurements are subsequently downloaded to a computer for data reduction.

The relative displacement of the test seal is measured by four inductive probes mounted in 90-deg increments around the periphery of the seal carrier assembly as shown in Fig. 2. These internally mounted proximity probes allow direct measurement of the relative displacement of the fluid-film between the test seal and the shaft. In addition to the internally mounted proximity probes, there are four externally mounted proximity probes. The measurements obtained from these probes are used to inertially correct the excitation force applied to the seal carrier assembly by measuring the seal carrier assembly's absolute displacement as illustrated in Fig. 2. The seal carrier assembly's displacements are differentiated in the frequency domain to obtain acceleration. Before the acceleration levels are used to inertially correct the applied force, the acceleration of the test rig housing must be taken

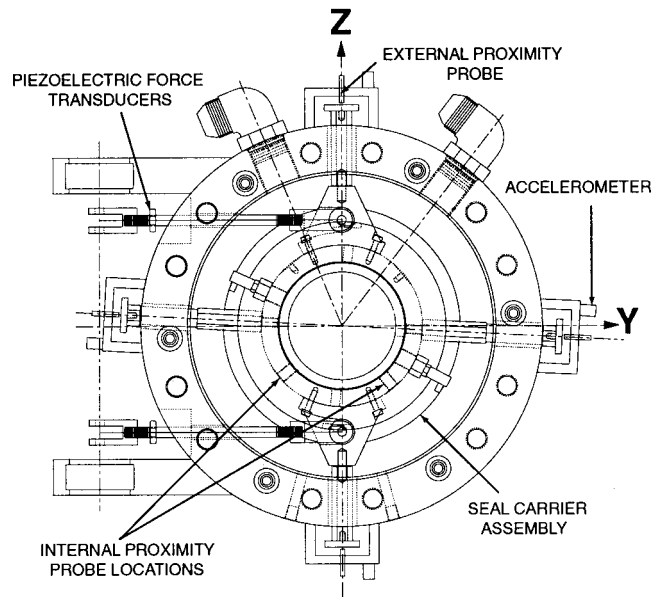


Fig. 2 Top view of the test section

into account. Test rig housing acceleration levels are measured using accelerometers mounted on each external proximity probe bracket. These measurements are used in conjunction with the acceleration levels obtained from the proximity probes to ensure that inertial frame acceleration levels are used to correct the applied force.

The equations are written using the coordinate system defined in Fig. 2. Seal reaction forces are modeled as

$$\begin{aligned} - \begin{Bmatrix} f_y \\ f_z \end{Bmatrix} &= \begin{bmatrix} K_{yy} & K_{yz} \\ -K_{yz} & K_{yy} \end{bmatrix} \begin{Bmatrix} \delta y \\ \delta z \end{Bmatrix} + \begin{bmatrix} C_{yy} & C_{yz} \\ -C_{yz} & C_{yy} \end{bmatrix} \begin{Bmatrix} \delta \dot{y} \\ \delta \dot{z} \end{Bmatrix} \\ &+ \begin{bmatrix} M_{yy} & M_{yz} \\ -M_{yz} & M_{yy} \end{bmatrix} \begin{Bmatrix} \delta \ddot{y} \\ \delta \ddot{z} \end{Bmatrix} \end{aligned} \quad (1)$$

where f_y , f_z are the y and z-components of the fluid film reaction forces, and δ_y , δ_z are the y and z-components of the relative displacement between the seal center and shaft. The rotordynamic coefficients are assumed to be skew-symmetric ($-yz=zy$ and $yy=zz$). The assumption of skew-symmetry is based upon the symmetry of the seal (and flow conditions) as well as the dynamic testing being performed in the centered seal state. Test data presented in Marquette et al. [1] indicate that the rotordynamic coefficients are skew symmetric for centered seals. After measuring displacements, forces, and acceleration levels, the individual time histories are transformed to the frequency domain. Spectral density methods are employed to obtain complex frequency response functions. The frequency response functions, in turn, are curve fit to yield rotordynamic coefficients. A complete description of the test rig and data reduction technique is provided in Darden, Earhart, and Flowers [3].

The test matrix for this investigation was significantly influenced by the high preswirl condition. The stability of the seal carrier assembly was predicted to degrade with increasing shaft speed. Since the stability of the seal carrier assembly is solely governed by the characteristics of the test seal, the maximum shaft speeds were limited to approximately half the speeds of the low preswirl condition tests. Moreover, earlier low preswirl testing of the damping seal revealed that the measured direct stiffness was lower than its predicted values. This result exacerbated the seal carrier's stability problem for damping seals at the high fluid preswirl condition. Consequently, tests involving damping seals in the high preswirl condition were limited to speeds lower than its

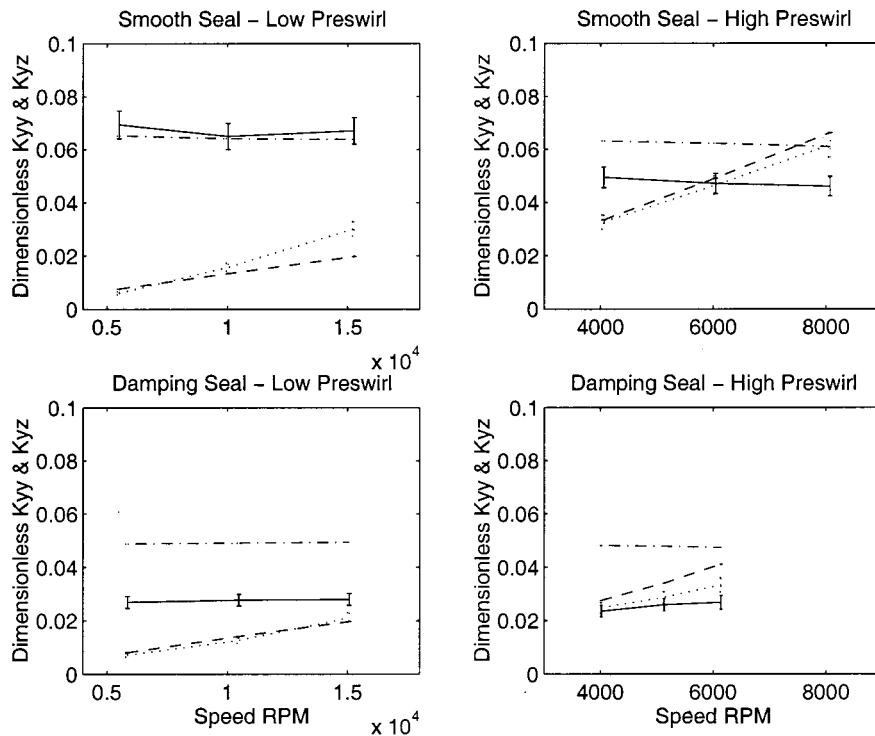


Fig. 3 Stiffness comparison for the low seal pressure differential case: solid=experimental direct stiffness, dash-dot=theoretical direct stiffness, dot=experimental cross-coupled stiffness, dash=theoretical cross-coupled stiffness

smooth seal counterpart. In hindsight, however, the damping seal tests could have been conducted at the same rotational speeds because of their whirl frequency ratios were significantly lower than predicted. This finding will be discussed further in the results section.

A smooth annular seal and a round-hole pattern damping seal were used in this investigation. The radial clearance between the shaft and stator for both seals is 0.053 cm (0.021 in.). Both seals have an approximate $L/D=0.5$ with a length of 4.57 cm (1.80 in.). The stator surface pattern for the damping seal was comprised of round-holes with flat bottoms. An individual round hole had a diameter of 0.37 cm (0.145 in.) and a depth between 0.033 cm (0.013 in.) and 0.043 cm (0.017 in.). The holes are distributed along the seal surface with a hole density of 0.35. The hole density is defined as the hole area divided by the total area.

The mechanical dimensions of MSFC's annular seal test rig are based roughly on the space shuttle main engine's (SSME) high pressure fuel turbopump (HPFTP) interstage damping seal configuration. The Reynolds number of the HPFTP interstage damping seals is on the order of 500,000. MSFC's annular seal test rig, in contrast, achieves a Reynolds number on the order of 100,000. The preswirl ratio of the fluid entering the HPFTP's interstage damping seal has been calculated to be 0.75. The test rig can produce a preswirl ratio of over 1.0.

Theoretical Predictions

Theoretical predictions for the performance of both seals under the various operating conditions were generated using a code developed by Padavala and Palazzolo [4]. The bulk flow model uses a Hir's friction factor for both seals. The value of the constants used in the smooth seal predictions were $M_s=M_r=-0.217$ and $N_s=N_r=0.0674$ (Childs and Kim [5]). The constants used in the damping seal predictions were obtained from Childs et al. [6] ($M_s=-0.0481$, $N_s=0.0322$, $M_r=-0.217$, and $N_r=0.0674$). Inlet preswirl values of 0.19 and 1.1 were used in the predictions for the low and the high preswirl tests, respectively. Values for both

preswirl conditions were based upon experimentally obtained whirl frequency ratios since these parameters were not measured directly. As a check, both preswirl test conditions were modeled using computational fluid dynamics (CFD) to determine the resultant fluid preswirl ratio of the fluid entering the seal. The CFD analyses yielded fluid preswirl ratios of 0.1 and 1.23 for the low and high preswirl conditions, respectively. An inlet loss factor of 0.1 was used in all of the predictions.

Experimental Results

The results of the investigation are given in Figs. 3–6. Dimensionless direct stiffness, cross-coupled stiffness, direct damping, and the resulting whirl frequency ratios are shown for low (average seal pressure differential=108.5 bars (1573 psi)) and high (average seal pressure differential=134.5 bars (1951 psi)) pressure cases. The experimental values depicted in the figures were nondimensionalized with the following relationships:

$$K_{yy}^* = \frac{c}{\pi d l \Delta P} K_{yy}$$

$$K_{yz}^* = \frac{c}{\pi d l \Delta P} K_{yz} \quad (2)$$

$$C_{yy}^* = \frac{c \omega_s}{\pi d l \Delta P} C_{yy}$$

This format acts to normalize differences in seal pressure differentials that exist in the larger categories of low and high seal pressures.

Figure 3 presents the dimensionless direct and cross-coupled stiffness curves for both the smooth seal and damping seal as a function of shaft rotational speed and preswirl condition at the lower seal pressure differential. Figure 5 provides the same infor-

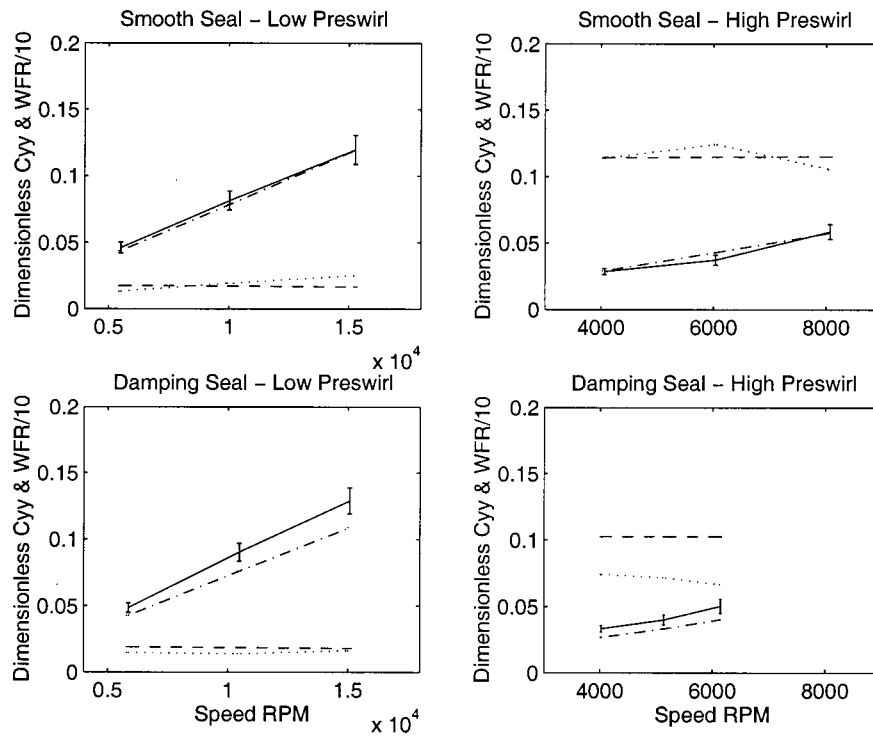


Fig. 4 Damping and whirl frequency ratio comparison for the low seal pressure differential case: solid=experimental direct damping, dash-dot=theoretical direct damping, dot=experimental whirl frequency ratio, dash=theoretical whirl frequency ratio

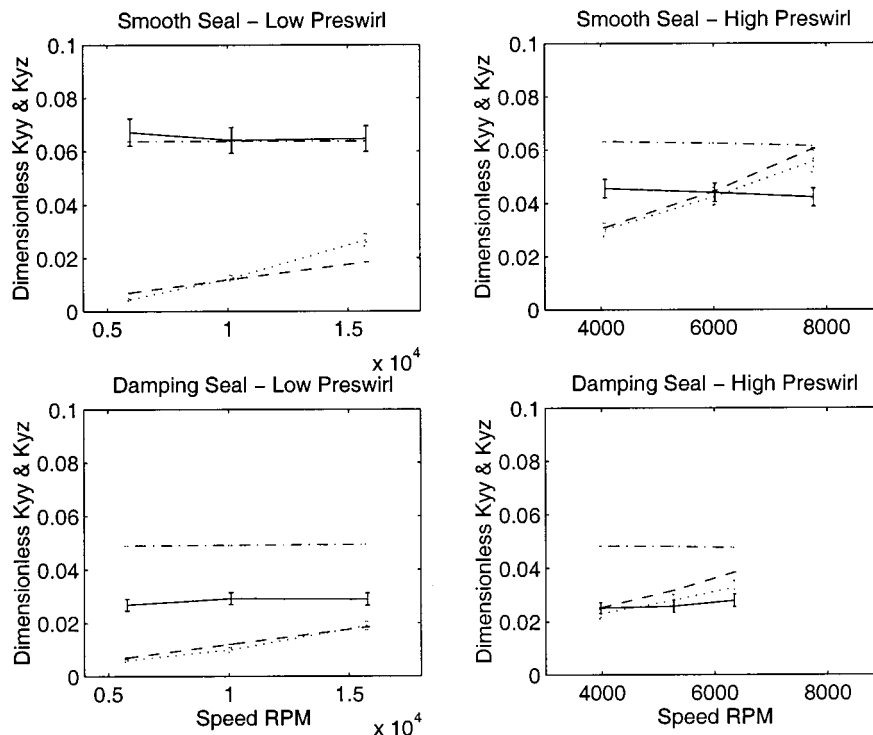


Fig. 5 Stiffness comparison for the high seal pressure differential case: solid=experimental direct stiffness, dash-dot=theoretical direct stiffness, dot=experimental cross-coupled stiffness, dash=theoretical cross-coupled stiffness

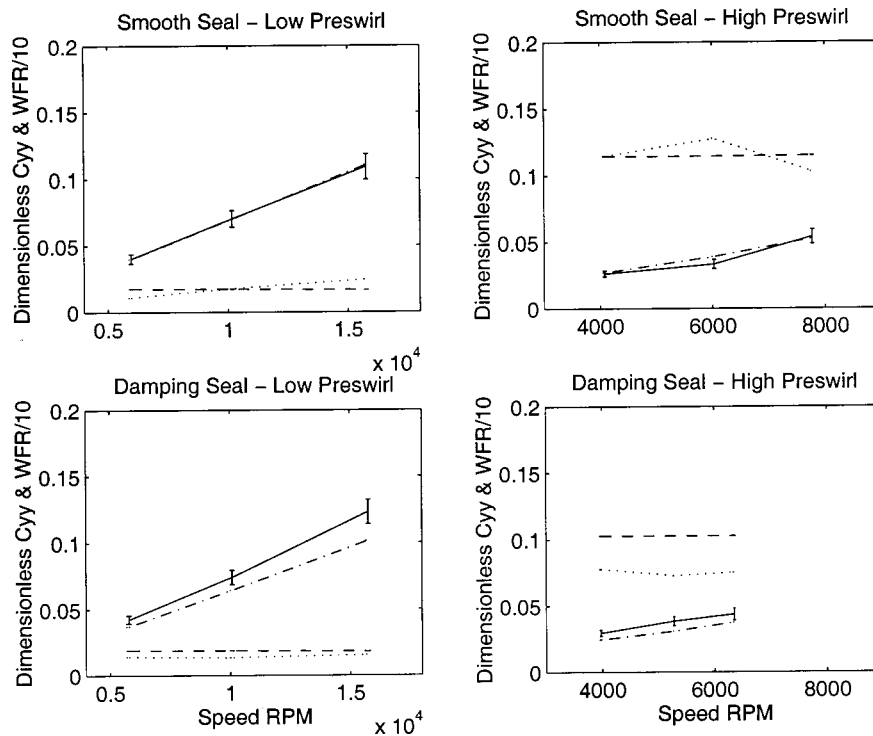


Fig. 6 Damping and whirl frequency ratio comparison for the high seal pressure differential case: solid=experimental direct damping, dash-dot=theoretical direct damping, dot=experimental whirl frequency ratio, dash=theoretical whirl frequency ratio

mation at higher seal pressure differentials. Figure 4 presents the dimensionless direct damping and whirl frequency ratio curves for both the smooth seal and damping seal as a function of shaft rotational speed and preswirl condition at the lower seal pressure differential. Figure 6 provides the same information at higher seal pressure differentials.

Direct Stiffness. Low Preswirl Testing:—Excellent agreement is obtained for direct stiffness of the smooth seal for both cases of high and low pressure. The direct stiffness results for the damping seal, on the other hand, indicates that the theory over predicts the stiffness coefficients by approximately 50 percent.

High Preswirl Testing: In contrast to the low preswirl results, the direct stiffness for the smooth seal is over predicted by approximately 20–30 percent. The direct stiffness for the damping seal is over predicted again by 50 percent.

The direct stiffness estimates agree with the theoretical trend to increase with increasing seal pressure differentials.

Cross-Coupled Stiffness. Low Preswirl Testing:—The dimensionless cross-coupled stiffness curves for the smooth seal indicate that this parameter is underpredicted at higher speeds. The damping seal cross-coupled stiffness curves indicate that theory over predicts by as much as 15 percent.

High Preswirl Testing: The smooth seal curves indicate that the theory over predicts the cross-coupled stiffness by roughly ten percent. Damping seal results show that the code over predicts this parameter by 10–20 percent.

The cross-coupled stiffness estimates did agree with the theoretical trend to increase with increasing shaft rotational speed.

Confirmation of Stiffness and Cross-Coupled Stiffness Results. Since the experimental estimates for the damping seal direct stiffness were considerably lower than their corresponding theoretical predictions, a series of static tests were performed to verify the dynamic testing results. Static test results from both high and low preswirl conditions are presented in Table 1. The

tests were performed by applying a series of static loads and recording the resulting displacements. The agreement between static and dynamic test results for the round-hole pattern damping seal provided validation of the lower than expected stiffness values.

Direct Damping. Low Preswirl Testing:—Excellent agreement between experimental results and theoretical predictions was obtained for the smooth seal direct damping for both high and low seal pressure differential levels. The dimensionless curves for the damping seal indicate that the theory under predicts the direct damping by 10–20 percent.

High Preswirl Testing: The theory does a reasonable job of predicting the damping coefficients for the smooth seal. The direct damping predictions and experimental estimates are within ten percent of each other. The dimensionless results for the damping seal show that the theory under predicts the damping value by roughly 25 percent.

Whirl Frequency Ratio. The whirl frequency ratio is given by

$$\text{WFR} = \frac{K_{yz}}{C_{yy}\omega_s} \quad (3)$$

where ω_s is the shaft rotational speed (rad/sec).

Since the fluid preswirl levels are not directly measured, the smooth seal test results were used to obtain the low and high test rig fluid preswirl values. The whirl frequency ratios of the smooth seal were averaged to establish a baseline from which fluid preswirl values could be determined from theoretical predictions. The close agreement between the experimental and theoretical whirl frequency ratios for the smooth seal reflect this effort.

Low Preswirl Damping Seal Testing: The average whirl frequency ratio for the damping seal was over predicted by approximately 20 percent.

Table 1 Comparison of the direct stiffness and cross-coupled stiffness for static and dynamic testing of the round-hole pattern damping seal

Test Description	RPM	ΔP (kPa)	K_{yy} (N/m)	K_{yz} (N/m)
Low Preswirl; Static Load	10144	1.16E+04	8.07E+06	3.34E+06
Low Preswirl; Dynamic Load	10275	1.20E+04	8.64E+06	3.19E+06
High Preswirl; Static Load	4909	1.31E+04	8.45E+06	9.22E+06
High Preswirl; Dynamic Load	5133	1.31E+04	8.32E+06	9.23E+06

Table 2 Test conditions and experimentally obtained rotordynamic coefficients for both the smooth seal and the round-hole pattern damping seal

Description	Pressure kPa	RPM	Stiffness N/m	Cross Coupled Stiffness N/m	Damping N*s/m	Cross- Coupled Damping N*s/m	Inertia Kg	Cross- Coupled Inertia Kg	Leakage l/s
Smooth Seal with Low Preswirl									
Run 30-96	9.29E+03	5.50E+03	1.59E+07	1.39E+06	1.82E+04	8.58E+02	1.45E+00	8.76E-01	1.55E+01
Run 31-96	9.26E+03	1.00E+04	1.48E+07	3.60E+06	1.77E+04	1.28E+03	8.23E-01	1.56E+00	1.52E+01
Run 32-96	9.14E+03	1.53E+04	1.51E+07	6.79E+06	1.68E+04	3.01E+03	1.09E+00	1.80E+00	1.52E+01
Run 34-96	1.28E+04	5.95E+03	2.12E+07	1.42E+06	2.04E+04	1.09E+03	7.01E-01	1.12E+00	1.68E+01
Run 35-96	1.21E+04	1.02E+04	1.92E+07	3.73E+06	1.98E+04	9.98E+02	7.36E-01	1.68E+00	1.67E+01
Run 36-96	1.15E+04	1.58E+04	1.83E+07	7.57E+06	1.87E+04	2.89E+03	8.93E-01	2.70E+00	1.63E+01
Smooth Seal with High Preswirl									
Run 5-98	1.13E+04	6.04E+03	1.31E+07	1.29E+07	1.64E+04	-6.30E+02	1.02E+00	3.64E+00	1.34E+01
Run 6-98	1.10E+04	8.07E+03	1.25E+07	1.67E+07	1.88E+04	-4.54E+03	-3.70E+00	1.16E-02	1.33E+01
Run 8-98	1.11E+04	4.06E+03	1.34E+07	8.87E+06	1.84E+04	1.28E+02	2.57E+00	2.75E+00	1.34E+01
Run 1-98	1.38E+04	6.02E+03	1.49E+07	1.44E+07	1.79E+04	-8.84E+02	9.81E-01	3.56E+00	1.51E+01
Run 3-98	1.31E+04	4.08E+03	1.47E+07	9.70E+06	1.98E+04	4.54E+02	3.10E+00	2.33E+00	1.47E+01
Run 4-98	1.24E+04	7.77E+03	1.29E+07	1.70E+07	2.02E+04	-3.09E+03	-2.99E+00	2.77E-01	1.43E+01
Damping Seal with Low Preswirl									
Run 2-97	1.03E+04	5.85E+03	6.82E+06	1.81E+06	2.01E+04	7.55E+02	2.19E+00	-5.99E-01	1.48E+01
Run 3-97	1.03E+04	1.05E+04	7.00E+06	3.19E+06	2.09E+04	8.90E+02	2.87E+00	-7.58E-01	1.47E+01
Run 4-97	1.03E+04	1.51E+04	7.07E+06	5.33E+06	2.06E+04	2.11E+03	1.94E+00	-3.20E-01	1.47E+01
Run 9-97	1.34E+04	5.78E+03	8.85E+06	1.98E+06	2.31E+04	1.02E+03	2.61E+00	-8.13E-01	1.72E+01
Run 10-97	1.33E+04	1.01E+04	9.58E+06	3.28E+06	2.31E+04	1.23E+03	4.03E+00	-9.12E-01	1.71E+01
Run 11-97	1.29E+04	1.58E+04	9.18E+06	6.04E+06	2.37E+04	1.72E+03	2.38E+00	-6.01E-01	1.68E+01
Damping Seal with High Preswirl									
Run 16-97	1.31E+04	5.13E+03	8.32E+06	9.23E+06	2.39E+04	5.41E+03	7.44E+00	-3.84E+00	1.67E+01
Run 17-97	1.27E+04	6.14E+03	8.37E+06	1.04E+07	2.44E+04	6.54E+03	7.85E+00	-4.17E+00	1.66E+01
Run 18-97	1.23E+04	4.02E+03	7.11E+06	7.48E+06	2.40E+04	2.07E+03	4.82E+00	-2.24E+00	1.63E+01
Run 19-97	1.60E+04	5.28E+03	1.01E+07	1.10E+07	2.73E+04	3.57E+03	5.62E+00	-3.40E+00	1.74E+01
Run 20-97	1.57E+04	6.35E+03	1.07E+07	1.26E+07	2.53E+04	7.02E+03	8.32E+00	-3.08E+00	1.71E+01
Run 21-97	1.44E+04	3.98E+03	8.89E+06	8.15E+06	2.52E+04	2.17E+03	5.20E+00	-1.87E+00	1.67E+01

High Preswirl Damping Seal Testing: The predicted whirl frequency ratios for the high preswirl case were calculated to be over 1.0. Experimental data, however, revealed that the average whirl frequency ratio was actually 0.73. This represents a significant increase in the damping seal's stabilizing capacity relative to its theoretical predictions.

Measurement Uncertainty

An uncertainty analysis that accounts for the bias errors was performed on the experimental data in accordance with the procedure outlined in the ANSI/ASME [7]. Normalized error bars are provided for each rotordynamic coefficient in Figs. 3–6. The average uncertainty for direct stiffness, cross-coupled stiffness, and

damping was 8.1 percent, 8.4 percent, and 8.7 percent, respectively. The maximum uncertainty value over the entire test matrix for direct stiffness was 9.6 percent. Likewise, the maximum uncertainty values for cross-coupled stiffness and direct damping were 9.2 percent and 10.5 percent, respectively. Cross-coupled damping, direct and cross-coupled inertia coefficients proved more difficult to measure. The average uncertainty for these parameters was 22.5 percent for cross-coupled damping, 52.9 percent for direct inertia, and 38.2 percent for cross-coupled inertia. Unfortunately, the maximum uncertainty values found in the test matrix for these measurements were on the order of 100 percent. Accordingly, these results are provided in the appendix but are not included in graphical form. (Table 2.)

Conclusions

The results from a theoretical and experimental comparison of the dynamic characteristics of a smooth annular seal and a round-hole stator pattern damping seal have been presented. Testing on the seals was performed over a wide range of pressure drops, running speeds, and two fluid inlet preswirl conditions. Both seals had an $L/D=0.5$ and the same radial clearance. Dimensionless curves for direct stiffness, direct damping, cross-coupled stiffness, and whirl frequency ratios have been provided. The trends of the dimensionless curves are remarkably consistent when comparing results from both seal pressure differential levels.

Comparison of the results from both seals brought several points to light. First, the theory does a better job, in general, of predicting the dynamic characteristics of the smooth seal. A direct comparison of the experimental data revealed that the resulting whirl frequency ratio of the damping seal was significantly lower than the smooth seal. The average whirl frequency ratio for the damping seal was 0.73 as compared with the smooth seal's 1.15. Both seals were tested in the same high fluid preswirl conditions. Moreover, the experimental direct stiffness values of the damping seal were notably lower than its corresponding predicted values. Both the damping seal's improved whirl frequency ratio and its lower than predicted stiffness needs consideration in the design process. One needs to be mindful, however, that these results are for a specific damping seal stator pattern and should not be generalized to all forms of the damping seal.

Nomenclature

c = seal radial clearance (L)
 C_{yy}, C_{yz} = damping coefficients (FT/L)

C_{yy}^* = dimensionless damping
 d = seal diameter (L)
 f_y, f_z = seal reaction forces (F)
 K_{yy}, K_{yz} = stiffness coefficients (F/L)
 K_{yy}^*, K_{yz}^* = dimensionless stiffness
 l = seal length (L)
 M_{yy}, M_{yz} = mass coefficients (M)
 ΔP = seal pressure differential (F/LL)
WFR = whirl frequency ratio
 ω_s = shaft rotational speed (RAD/T)

References

- [1] Marquette, O. R., Childs, D. W., and San Andres, L., 1997, "Eccentricity Effects on the Rotordynamic Coefficients of Plain Annular Seals: Theory Versus Experiment," *ASME J. Tribol.*, **119**, pp. 443–448.
- [2] Childs, D. W., and Fayolle, P., 1997, "Test Results for Liquid "Damper" Seals Using a Round-Hole Roughness Pattern for the Stators," unpublished report, Sept.
- [3] Darden, J. M., Earhart, E. M., and Flowers, G. T., 1999, "Experimental Rotordynamic Characterization of Annular Seals: Facility and Methodology," *ASME J. Eng. Gas Turbines Power*, **121**, pp. 349–354.
- [4] Padavala, S., Palazzolo, A. B., Vallely, D. P., Ryan, S. G., 1993, "Application of an Improved Nelson-Nguyen Analysis to Eccentric, Arbitrary Profile Liquid Annular Seals," Workshop on Rotordynamic Instability Problems in High Performance Turbomachinery, Texas A&M University, pp. 113–135.
- [5] Childs, D. W., and Kim, C. H., 1985, "Analysis and Testing for Rotordynamic Coefficients of Turbulent Annular Seals With Different, Directionally-Homogenous Surface-Roughness Treatment for Rotor and Stator Elements," *ASME J. Tribol.*, **107**, pp. 296–306.
- [6] Childs, D. W., Nolan, S. A., and Kilgore, J. J., 1990, "Additional Test Results for Round-Hole-Pattern Damper Seals: Leakage, Friction Factors, and Rotordynamic Force Coefficients," *ASME J. Tribol.*, **112**, pp. 365–371.
- [7] *Measurement Uncertainty*, 1986, ANSI/ASME PTC 19.1-1985 Part 1, 1986.

Self-Sensing in Fault Tolerant Magnetic Bearings

D. Montie

E. Maslen

Department of Mechanical, Aerospace, And
Nuclear Engineering,
University of Virginia,
Thornton Hall, McCormick road,
Charlottesville, VA 22903-2442

Several schemes have recently been proposed for achieving either fault tolerance or self-sensing in magnetic bearings. The present work describes the fundamental connection between ability to actuate and ability to sense in a partially failed magnetic bearing system. This relationship is then exploited to construct a self-sensing scheme which operates in the presence of detectable actuator or amplifier faults. Such an approach is advantageous in fault tolerant systems because it reduces or eliminates the need to address potential independent failure mechanisms in sensors and actuators. Based on a model reference parameter estimation mechanism, the self-sensing scheme is shown to provide acceptable position measurement accuracy and bandwidth under various actuator/amplifier faults which are actuator tolerable. Estimates of increase in noise floor and loss of bandwidth under fault conditions are provided. The issue of estimator convergence under fault conditions is examined in detail with comments on implementation complexity arising from scheduling convergence control on fault state.
[DOI: 10.1115/1.1383257]

Introduction

Two important issues in the design of magnetic bearings are cost and reliability. In addition to the purchase cost of a magnetic bearing system, it is important to consider the cost of maintenance. The expense of time and resources in repairing a failure (such as bearing coil, amplifier, controller, or sensor failure) may be substantial. Therefore, the ability of a magnetic bearing to be fault tolerant (with respect to failures of one or more of the above) is critical in many applications.

With a self-sensing system, position estimates are obtained by examining the current in the bearing coils, eliminating the need for discrete position sensors. The ability to eliminate the sensors on a magnetic bearing affects cost in both forms, as well as reliability. Self-sensing reduces the initial costs and size of the system. It also increases reliability by removing a failure susceptible part. While many self-sensing schemes have been demonstrated ([1–4]), this technology has not been incorporated into a fault tolerant system.

This combination of fault tolerance and self-sensing is crucial to successful application of magnetic bearings to machinery which is sensitive to cost and down-time, such as most turbomachinery. The potential cost advantages are significant, but more importantly, better reliability will enormously improve acceptance of this technology into common use in production turbomachinery. This is a primary motivation underlying the present work.

In a typical magnetic bearing, such as depicted in Figure 1, the work done by the magnetic field can be described ([5]) as

$$w = \frac{1}{2} I^T L I. \quad (1)$$

The force (along the x -axis, for instance) can be derived as

$$f_x = \frac{\partial w}{\partial x} = \frac{1}{2} I^T \frac{\partial L}{\partial x} I \quad (2)$$

in which I is assumed to be constant in evaluating virtual work.

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-178. Manuscript received by IGTI, Oct. 1998; final revision received by the ASME Headquarters, Mar. 1999. Associate Editor: D. Wisler.

As demonstrated in [6], the actuator can generate arbitrary forces in the x -direction if the matrix $\partial L / \partial x$ has both positive and negative eigenvalues. In the present work, we demonstrate that the key sensitivity matrix controlling the ability to sense in the x -direction is $\partial L^{-1} / \partial x$. The required property is that, again, this matrix should have both positive and negative eigenvalues.

Hence, while a formal proof is not yet developed, there is a strong suggestion that the ability to generate force implies the ability to sense and the ability to sense implies the ability to generate force. On this basis, it is posited that, in any stator axis along which an arbitrary magnetic force can be applied, position sensing can also be achieved. This means that the need for independent fault tolerance mechanisms in the actuators and sensors can be eliminated; for a bearing pole, position sensing is always possible if actuation can be achieved. Whatever scheme is used to preserve the ability to actuate under fault conditions will suffice to preserve the ability to sense.

This paper demonstrates the feasibility of a self-sensing scheme under actuator/coil failures as are typically addressed in a fault tolerant system ([6]). In outline, the first section of this paper develops the magnetics model which is used both in a simulation for performance evaluation of the parameter estimator and for derivation of an estimator controller. The parameter estimation scheme is then presented, followed by the techniques used for estimator controller design and construction. Simulation results are then discussed, with emphasis on estimation error, noise, and bandwidth under various fault states. Finally, conclusions and likely avenues of future research are provided.

Magnetics Model

The magnetic bearing model developed in the paper is based on a typical eight-pole system (see Fig. 1), with each coil wound independently and controlled with independent bi-state switching amplifiers. For the purposes of illustration and explanation, some fundamental calculations are examined using a sample three-pole bearing, with each pole independently wound and spaced 120 deg apart as indicated in Fig. 2.

Begin with the discrete form of Ampere's loop law for the coil fluxes:

$$R\Phi = N_l I. \quad (3)$$

To simplify, first assume all of the reluctance is due to the air gap, with no reluctance in the stator or journal, since the relative permeability for many magnetic materials is typically several thou-

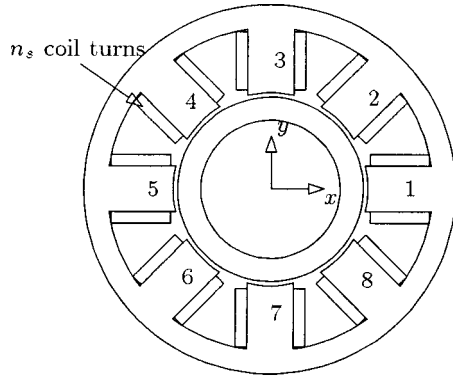


Fig. 1 Symmetric eight-pole magnetic actuator

sand or greater. Also, assume there are no magnetic nonidealities, such as hysteresis, saturation, or eddy current effects. These assumptions are valid in the linear region of the magnetic material's B-H curve and when the lamination thickness is small relative to reasonable skin depth (order of 0.2 mm).

For the three-pole bearing example, this yields

$$\begin{bmatrix} r_1 & -r_2 & 0 \\ 0 & r_2 & -r_3 \\ 1 & 1 & 1 \end{bmatrix} \begin{Bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \end{Bmatrix} = \begin{bmatrix} n_s & -n_s & 0 \\ 0 & n_s & -n_s \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} i_1 \\ i_2 \\ i_3 \end{Bmatrix}$$

where $r_i = 1/\mu_0 A_g (g_0 - x \cos \theta_i - y \sin \theta_i)$ and A_g is taken to be constant over the entire flux path.

Initially, the N_I matrix (and consequently the inductance matrix) has a nullity of 1 since the fluxes in poles one and two control the flux in the third pole (due to flux conservation). Thus, there is one redundant coil current, from which no flux in the circuit is affected. While this is beneficial in that one coil could fail with negligible loss in performance, the rank deficient inductance matrix means that there is one potentially unstable current state.

Following the method developed in [6], an independent decoupling choke (see Fig. 2) is added to the circuit to yield an inductance matrix which is both full rank (of size 3×3 in the example) and decoupled. By attaching each bearing coil in series with windings (of equal turns) around the ring, current vectors orthogonal to the nullity of the original N_I matrix will produce no flux in the ring. In this way, the positive mutual inductances produced by the ring exactly cancel the negative mutual inductances produced in the bearing circuit. The physical parameters of the ring are chosen to produce an inductance matrix that is diagonal under nominal conditions. For the three-pole example,

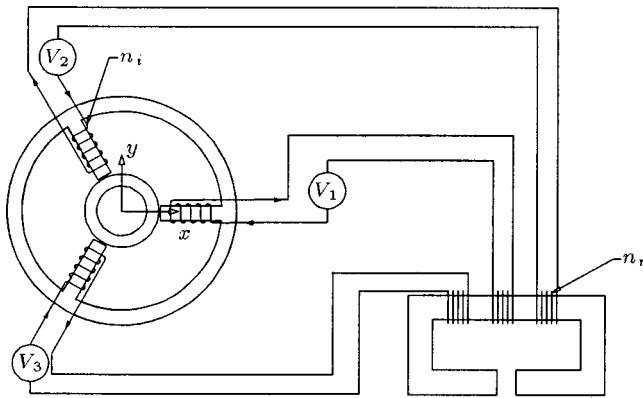


Fig. 2 Three-pole stator with a decoupling choke

$$\begin{bmatrix} r_1 & -r_2 & 0 & 0 \\ 0 & r_2 & -r_3 & 0 \\ 1 & 1 & 1 & 0 \\ 0 & 0 & 0 & r_r \end{bmatrix} \begin{Bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \\ \phi_r \end{Bmatrix} = \begin{bmatrix} n_s & -n_s & 0 \\ 0 & n_s & -n_s \\ 0 & 0 & 0 \\ n_r & n_r & n_r \end{bmatrix} \begin{Bmatrix} i_1 \\ i_2 \\ i_3 \\ i_r \end{Bmatrix}$$

Applying Faraday's Law and Ohm's Law to the magnetic circuit yields

$$V = N_V \dot{\Phi} + \Omega I, \quad (4)$$

or, in the example

$$\begin{Bmatrix} v_1 \\ v_2 \\ v_3 \end{Bmatrix} = \begin{bmatrix} n_s & 0 & 0 & n_r \\ 0 & n_s & 0 & n_r \\ 0 & 0 & n_s & n_r \end{bmatrix} \frac{d}{dt} \begin{Bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \\ \phi_r \end{Bmatrix} + \begin{bmatrix} \Omega_i & 0 & 0 \\ 0 & \Omega_i & 0 \\ 0 & 0 & \Omega_i \end{bmatrix} \begin{Bmatrix} i_1 \\ i_2 \\ i_3 \end{Bmatrix}$$

Assuming linear magnetics and no eddy currents, the fluxes are computed from (3):

$$\Phi = R^{-1} N_I I, \quad (5)$$

from which the example follows

$$\begin{Bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \\ \phi_r \end{Bmatrix} = \begin{bmatrix} r_1 & -r_2 & 0 & 0 \\ 0 & r_2 & -r_3 & 0 \\ 1 & 1 & 1 & 0 \\ 0 & 0 & 0 & r_r \end{bmatrix}^{-1} \begin{bmatrix} n_s & -n_s & 0 \\ 0 & n_s & -n_s \\ 0 & 0 & 0 \\ n_r & n_r & n_r \end{bmatrix} \begin{Bmatrix} i_1 \\ i_2 \\ i_3 \\ i_r \end{Bmatrix}$$

When this bearing is centered, $r_1 = r_2 = r_3 = r$:

$$\begin{Bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \\ \phi_r \end{Bmatrix} = \begin{bmatrix} 2a & -a & -a \\ -a & 2a & -a \\ -a & -a & 2a \\ b & b & b \end{bmatrix} \begin{Bmatrix} i_1 \\ i_2 \\ i_3 \\ i_r \end{Bmatrix} \quad \begin{matrix} a = \frac{n_s}{3r} \\ b = \frac{n_r}{r_r} \end{matrix}$$

The inductance matrix can then be computed from (5):

$$L \doteq N_V \frac{\partial \Phi}{\partial I} = N_V R^{-1} N_I. \quad (6)$$

From the previous derivation, L for the three-pole bearing example becomes

$$L = \begin{bmatrix} n_s & 0 & 0 & n_r \\ 0 & n_s & 0 & n_r \\ 0 & 0 & n_s & n_r \end{bmatrix} \begin{bmatrix} 2a & -a & -a \\ -a & 2a & -a \\ -a & -a & 2a \\ b & b & b \end{bmatrix} = \begin{bmatrix} 2an_s + bn_r & -an_s + bn_r & -an_s + bn_r \\ -an_s + bn_r & 2an_s + bn_r & -an_s + bn_r \\ -an_s + bn_r & -an_s + bn_r & 2an_s + bn_r \end{bmatrix}$$

Each off-diagonal term, when equated to zero (to yield a diagonal inductance matrix), produces the relationship

$$bn_r = an_s \Rightarrow \frac{n_r^2}{r_r} = \frac{n_s^2}{3r} \quad (7)$$

which determines the necessary geometry of the decoupling choke to make (7) valid. For a general symmetric p -pole stator, this relationship is $n_r^2/r_r = n_s^2/pr$. Using this design constraint for the decoupling choke produces a diagonal inductance matrix for $x = y = 0$: $L = \text{diag}(n_s/r)$.

Combining Eq. (4) and (5) yields

$$V = N_v \frac{d}{dt} (R^{-1} N_I I) + \Omega I \quad (8)$$

$$= L \dot{I} + \dot{L} I + \Omega I.$$

The dynamic behavior of the actuator is governed by

$$\dot{I} = L^{-1} (V - [\dot{L} + \Omega] I) \quad (9)$$

in which the flux and current are nonlinear functions of x, y, g , and V . L is invertible because of the addition of the decoupling choke.

Amplifier Simulation

Magnetic bearings typically use switching power amplifiers in a transconductance mode. These amplifiers compare a requested current to their actual output current and select the output voltage accordingly. In a bi-state amplifier, this output voltage is either $+v_{ps}$ or $-v_{ps}$; choosing $+v_{ps}$ will increase the coil current. In a tristate amplifier, a third output condition of 0 volts (coil shorted) is added. A specific limitation of the present work is that it assumes bi-state operation even though commercial practice has very broadly adopted tristate amplifiers because of their lower power losses and acoustic noise emission. The self-sensing scheme presented here does not fundamentally require bi-state operation, but the interrogation signal is much stronger and easier to process in bi-state than in tristate: Experimental results with tristate so far have been poor.

The actual amplifier scheme used in the present simulation is a time-delay algorithm ([7]). A comparator determines a desired output state based on the difference between the reference and measured currents. This output state is then implemented a short time later (13 microseconds in the simulation). The result is stable current control with a switching limit cycle with overall period of 52 microseconds: about 20 kHz. Similar behavior would be obtained with any of a number of other switching algorithms.

The target currents determine the force produced by the actuator. Meeker [6] provides a thorough coverage of this in both the nominal case and when the actuator has a fault. For the simulations here, the target currents were held at a nominal bias condition, so the effect of target current dynamic behavior was not examined. For the single magnet case, Noh [3] provides an extensive discussion of the effect of dynamic current reference.

Fault Conditions

The fault mechanism addressed in this work is an open circuit at any point in the series connection of amplifier, wiring, and coil. Such a fault includes fuse burnout, amplifier device burnout (in the open condition), coil wire burnout, and connector failure. It does not consider the shorted coil scenario or other short failures as in the power amplifier. In open circuit failures, the coil current goes to zero; for short failures, the coil voltage goes to zero and the current is driven by stator flux variation. While either mechanism is possible, industrial experience suggests that the open coil scenario is more commonly encountered.

Modeling an open coil failure is simple: The current in the failed coil is forced to zero. The actual coil voltage has no effect on the dynamics of the simulation so it can be assumed zero. In (9), this means that failure of coil i is equivalent to removing the i th row and column of each of the matrices.

Parameter Estimation Scheme

Information about the gap reluctances, and hence the rotor position, is most richly embedded in the time rate of change of the coil currents. Following [3], a set of filters is used to approximate the magnitudes of the slopes of each current. This is achieved with a high-pass filter (to remove the low-frequency components of the current), followed by a rectifier and a low-pass filter (to demodulate the amplitude of the signal). A simplified model of the action

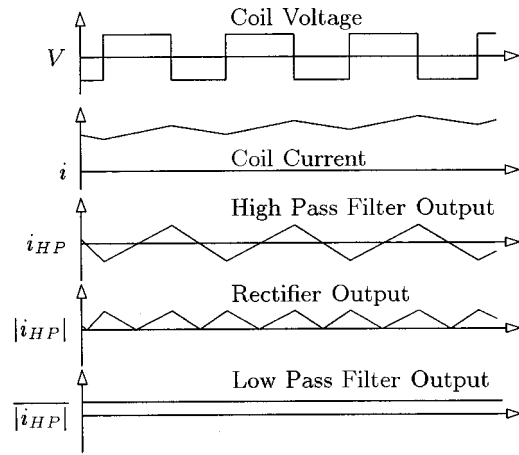


Fig. 3 Sequence of signals in envelope filter

of the envelope filter is a low-pass filter acting on $|I|$, with a cutoff frequency of about 2 kHz: Figure 3 illustrates the sequence of signal operations.

The output approximates the magnitude of the current slope up to the bandwidth of the filter. The filter output is, however, also a function of the amplifier duty cycle (directly related to actuator force slew rate). Consequently, it is undesirable to use this signal directly as a measure of gap length.

Instead, a parameter estimation scheme uses this information (which is monotonic in gap length ([3])) to estimate the actual gap length independently of the amplifier duty cycle. An analog circuit simulation of the magnetics is driven by the actual coil voltages. The air gaps (x, y, g) of the simulation are to be determined and form the output of the parameter estimator.

The simulation produces a set of coil currents which, if the simulation air gaps are correct, should match the actual currents. The simulation currents are filtered in the same manner as the actual currents and the error between the two filtered sets of currents is applied to a controller which attempts to determine the estimated position, as depicted in Fig. 4. If the loop closed by this controller is stable, then the error should converge to zero which is equivalent to the estimated gaps converging to the actual gaps. Noh [3] provides a very extensive discussion of this approach for the problem of estimating a single air gap.

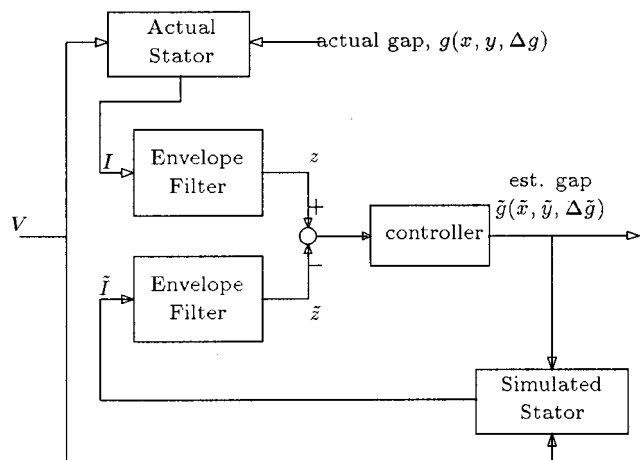


Fig. 4 Parameter estimation scheme

Plant Linearization

The first step in developing this controller is to extract a linearized model for the dynamics of the error between the actual filtered currents and the output of the simulation. The plant magnetics are described by (9):

$$\begin{aligned}\dot{I} &= A(x, y, g)I + B((x, y, g)V \\ A(x, y, g) &\doteq -L^{-1} \left[\frac{dL}{dt} + \Omega \right] \\ B(x, y, g) &\doteq L^{-1}\end{aligned}$$

and the simulation dynamics are described by

$$\dot{I} = A(\tilde{x}, \tilde{y}, \tilde{g})\tilde{I} + B(\tilde{x}, \tilde{y}, \tilde{g})V. \quad (10)$$

Following [3], the envelope filters are modeled as

$$\begin{aligned}\dot{a} &= -\alpha a + \alpha |\dot{I}| \\ z &= 4\Delta(I - \Delta)a.\end{aligned} \quad (11)$$

The nominal duty cycle of the amplifiers in two-state mode is 0.5. Thus, at the nominal duty cycle, z is a low pass version of $|\dot{I}|$. A similar equation can be written for the output filters acting on the simulation.

To eliminate the nonlinearity of the output equation in (11), expand in a Taylor's series as

$$z = 4\Delta_0(I - \Delta_0)a_0 + 4(\Delta - \Delta_0)(I - 2\Delta_0)a_0 - 4(\Delta - \Delta_0)^2a_0.$$

Noting that $\Delta_0 = \text{diag}(0.5)$ and $a_0 = |\dot{I}|_0$, this is

$$z = a - 4(\Delta - \Delta_0)^2|\dot{I}|_0.$$

The last term in this equation is a mess and not particularly easily measured, so treat it as an unknown excitation:

$$Gf \doteq -4(\Delta - \Delta_0)^2|\dot{I}|_0. \quad (12)$$

in which G is a property of the actuator and f is controlled by the switching sequence. The estimator will be subject to the same f , but with an estimated gain $\tilde{G}$.

After a little simple algebra, the intermediate variable a is eliminated to leave (for the actual actuator)

$$\dot{z} = -\alpha z + \alpha |\dot{I}| + G(\alpha f + \dot{f}) \quad (13)$$

and for the simulation,

$$\dot{\tilde{z}} = -\alpha \tilde{z} + \alpha |\tilde{I}| + \tilde{G}(\alpha f + \dot{f}) \quad (14)$$

In order to reflect the effects of x , y , and g on these dynamics, expand $|\dot{I}|$. From (9), a first-order Taylor's series expansion about $x=y=g=g_0=0$ gives

$$\begin{aligned}\dot{I} &\approx L_0^{-1}(V - \Omega I) - L_0^{-1} \left(\frac{dL}{dx}x + \frac{dL}{dy}y + \frac{dL}{dg}\Delta g \right) L_0^{-1}(V - \Omega I) \\ &\quad - L_0^{-1} \left(\frac{dL}{dx}\dot{x} + \frac{dL}{dy}\dot{y} + \frac{dL}{dg}\dot{g} \right) I.\end{aligned}$$

Assume that the amplifier voltage is much larger than the resistive drop due to coil resistance: $V \gg \Omega I$. Further, assume that the velocity-induced term (the last term) is negligible. With this

$$\dot{I} \approx L_0^{-1} \left[L_0 - \frac{dL}{dx} \Big|_0 x - \frac{dL}{dy} \Big|_0 y - \frac{dL}{dg} \Big|_0 \Delta g \right] L_0^{-1} V. \quad (15)$$

Define the coefficients of this series as

$$K_0 \doteq L_0^{-1} \quad (16)$$

$$K_x \doteq -L^{-1} \frac{dL}{dx} L^{-1} \Big|_0 \quad (17)$$

$$K_y \doteq -L^{-1} \frac{dL}{dy} L^{-1} \Big|_0 \quad (18)$$

$$K_g \doteq -L^{-1} \frac{dL}{dg} L^{-1} \Big|_0 \quad (19)$$

so that

$$\dot{I} \approx K_0 V + K_x Vx + K_y Vy + K_g V(g - g_0) \quad (20)$$

and

$$|\dot{I}| \approx |K_0 V + K_x Vx + K_y Vy + K_g V(g - g_0)|. \quad (21)$$

This absolute value function makes analytic evaluation somewhat difficult. However, if $K_0 V$ is enough bigger than the other terms (which it generally is) then this expression can be expanded in a series of the form

$$|\dot{I}| \approx k'_0 + k'_x x + k'_y y + k'_g (g - g_0) \quad (22)$$

where

$$k'_0(V) \doteq |K_0 V|$$

$$k'_x(V) \doteq \frac{|K_0 V + K_x Vx| - k'_0}{x}$$

$$k'_y(V) \doteq \frac{|K_0 V + K_y Vy| - k'_0}{y}$$

$$k'_g(V) \doteq \frac{|K_0 V + K_g V(g - g_0)| - k'_0}{g - g_0}.$$

The k_j are dependent upon the instantaneous collection of coil voltages, V . However, given that V lies in the set $v_i = \pm v_{ps}$ and the voltage states switch very rapidly and more or less independently, it may make sense to evaluate the *expected value* of these expressions by averaging over all possible voltage combinations:

$$k_j = \varepsilon(k_j) = \frac{1}{2^p} \sum_{i=1}^{2^p} k'_j(v_i) \quad (23)$$

and to assume that the filter dynamics (which are low-pass filtered) are governed by these expected values. The expected values of k_j turn out to be the diagonal elements of the corresponding K_j scaled by the power supply voltage and are independent of the displacement (x , y , or g) selected for evaluation of the derivatives as long as these displacements are less than g :

$$k_j = \text{diag}(K_j) v_{ps}. \quad (24)$$

The net result of all of this manipulation is a linearized model for the filter output as driven by the displacements which are to be estimated:

$$\dot{z} = -\alpha z + \alpha k_0 + \alpha [k_x k_y k_g] \begin{Bmatrix} x \\ y \\ g - g_0 \end{Bmatrix} + G(\alpha f + \dot{f})$$

for the actual stator and

$$\dot{\tilde{z}} = -\alpha \tilde{z} + \alpha k_0 + \alpha [k_x k_y k_g] \begin{Bmatrix} \tilde{x} \\ \tilde{y} \\ \tilde{g} - g_0 \end{Bmatrix} + \tilde{G}(\alpha f + \dot{f})$$

for the simulated stator.

The objective of the controller is to drive the error between z and $\tilde{z}$ to zero, so define error coordinates $e \doteq z - \tilde{z}$ to obtain

$$\dot{e} = -\alpha e + \alpha [k_x k_y k_g] \begin{Bmatrix} x - \tilde{x} \\ y - \tilde{y} \\ g - \tilde{g} \end{Bmatrix} + (G - \tilde{G})(\alpha f + \dot{f}). \quad (25)$$

Table 1 Sensing performance under various fault scenarios

Run	Coils Failed	X_{mag}	X_{phase}	$X_{\text{error rms}}$	Y_{mag}	Y_{phase}	$Y_{\text{error rms}}$
1	0	46.4 μm	−3.8 deg	0.7 μm	46.4 μm	−3.8 deg	0.7 μm
2	1, X-axis	43.9 μm	−2.7 deg	3.3 μm	46.1 μm	−2.1 deg	0.7 μm
3	1, Y-axis	46.1 μm	−2.1 deg	0.7 μm	43.9 μm	−2.7 deg	3.3 μm
4	1, off-axis	47.7 μm	−3.4 deg	9.5 μm	47.7 μm	−3.4 deg	9.5 μm
5	2, off-axis, adj.	46.9 μm	−3.1 deg	2.8 μm	47.0 μm	−3.1 deg	10.0 μm

In the limit as $e \rightarrow 0$, $\tilde{G} \rightarrow G$ so that the influence of f will be negligible: This is largely the purpose of using a parameter estimator.

Each error state e is accessible so state feedback can be employed directly in regulating e . The result is a MIMO LQR controller that minimizes the error over all voltage switch states and under various fault conditions by converging the values of the estimated inputs to those of the actual plant inputs.

Controller Construction

The linear error dynamic model presented in (25) contains a number of external inputs which are essentially unknown: x , y , g , and f . A simplistic approach to designing a controller with these signals present is to assume that they are zero-mean, random, Gaussian distributed signals. As such, it is not possible to predict future values of these signals even if the entire time history up to the present is known. Therefore, the controller is derived without regard to these signals and the plant model takes the simpler form:

$$\dot{e} = -\alpha e - \alpha[k_x \ k_y \ k_g] \begin{Bmatrix} \tilde{x} \\ \tilde{y} \\ \tilde{g} \end{Bmatrix}. \quad (26)$$

The control variables are the estimated rotor position $(\tilde{x}, \tilde{y})$ and mean radial air gap $(\tilde{g})$. A suitable cost function for construction of a linear quadratic regulator solution ([6]) weights the states against the control “effort.”

The parameters of this model (k_x , k_y , and k_g) are dependent upon the fault state. Potentially, it will be necessary to derive a different controller for each possible fault state. However, as presented in the following section, a single controller derived at nominal no-fault conditions will provide adequate tracking even for multiple coil failure modes.

To demonstrate construction of the controller, return to the three-pole example for which the plant model becomes

$$\dot{e} = -\alpha e - \alpha[k_x \ k_y \ k_g] \begin{Bmatrix} \tilde{x} \\ \tilde{y} \\ \tilde{g} \end{Bmatrix} \quad (27)$$

$$\begin{Bmatrix} y_1 \\ y_2 \\ y_3 \\ y_4 \\ y_5 \\ y_6 \end{Bmatrix} = \begin{bmatrix} \frac{1}{\epsilon_1} & 0 & 0 \\ 0 & \frac{1}{\epsilon_2} & 0 \\ 0 & 0 & \frac{1}{\epsilon_3} \\ 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} e + \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ \frac{1}{\delta_x} & 0 & 0 \\ 0 & \frac{1}{\delta_y} & 0 \\ 0 & 0 & \frac{1}{\delta_g} \end{bmatrix} \begin{Bmatrix} \tilde{x} \\ \tilde{y} \\ \tilde{g} \end{Bmatrix}$$

where $\alpha = 4\pi \times 10^3$ and K is a scaling constant used for controller calibration of outputs $\tilde{x}$, $\tilde{y}$, and $\tilde{g}$. The scalings ϵ_i are identical filter error magnitudes and δ_i are the maximum controller output magnitudes used in the cost functions y_{pi} . As discussed in [3] stability of the estimator sets an upper bound on the ratio of δ_i/ϵ_i while convergence rate sets a lower bound. In the present work, this ratio was selected to be as large as possible without encoun-

tering instability in the simulation. The ϵ_i are chosen to be very small (0.01 volts in the simulation) while the δ_i , were quite large (100 meters in the simulation).

The parameter estimation controller is computed as a linear quadratic regulator by formulating an algebraic Riccati equation and solving for the resulting state feedback matrix ([8]) so that

$$\begin{Bmatrix} \tilde{x} \\ \tilde{y} \\ \tilde{g} \end{Bmatrix} = K e. \quad (28)$$

Simulation Results

The relevant parameters of the magnetic bearing we simulated were $g_0 = 1.0$ mm, $A_g = 4.91$ cm², and $n_s = 200$. To remove the effect of turnon transients, the tracking controller is suppressed for 2 ms. At 2 ms, the estimator is enabled, causing the controller to produce transients of its own as it begins tracking the actual filter output from the nominal filter output at zero estimated displacement. Therefore, each simulation run lasted 10 ms “simulation time,” with the results evaluated for the final 2 ms to eliminate the effect of the initial system and controller transients.

In each simulation, the rotor is sinusoidally displaced 50 microns from center at 1 kHz for both axes. The first harmonic coefficients of the sine and cosine terms of the Fourier series performed on the estimated displacement were used to determine magnitude and phase. This two-term series was subtracted from the estimator data, and the RMS noise values were then determined for both axes.

In simulations involving failures, the current in the “actual” model and the applied voltage for the failed pole were set to zero. Neither the “simulation” model nor the controller was altered, however, in order to evaluate the performance of the simplest hardware configuration (one controller and bearing model used over a range of fault states). Table 1 lists the results of several illustrative simulations.

Figure 5 depicts the estimator output for case 5, where coils on either side of the y -axis coil are failed. Noticable features include a modest mean offset as well as some harmonic distortion most evident at the extrema of journal motion. This error is reflected in the increased RMS error figure of 10.0 microns for this run.

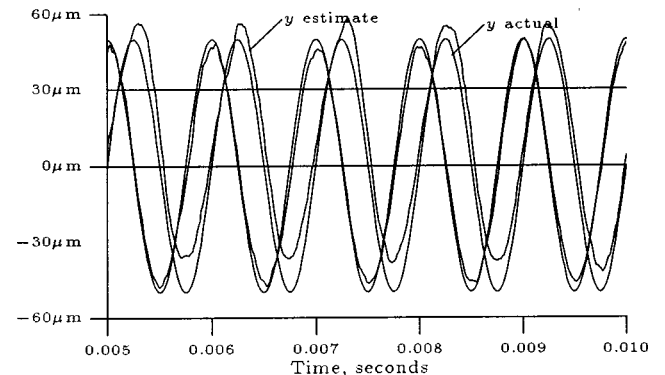


Fig. 5 Simulation results for run 5: two coils failed

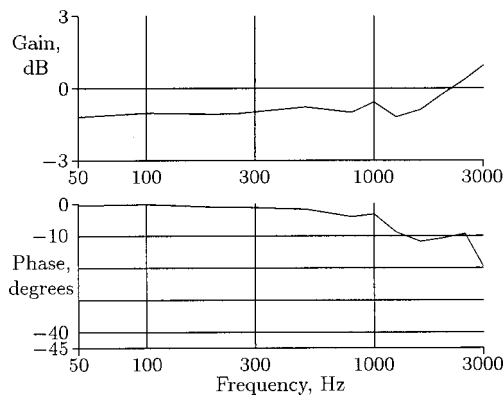


Fig. 6 No-fault controller Bode plot at 50 micron input displacement magnitude

The results demonstrate that there is little decrease in estimator performance under all single-coil fault conditions and under several double-coil fault conditions. The worst performance under any single-coil fault condition is a magnitude error of 5.4 percent and phase lag difference of -1.1 deg of the no-fault estimated position values, with only a slight decrease in position accuracy for the double-coil fault scenario. While noise levels increased significantly under the various fault conditions, the high-frequency components of the noise can be removed with proper filtering in the controller which reflects the physical limitations of the actual system. Offset errors introduced by fault conditions might be removed by simply establishing the correlation between the error and the fault, assuming that this correlation is consistent.

Figure 6 presents the Bode plot for the no-fault case over a range of frequencies, all at an input displacement magnitude of 50 microns. This demonstrates that under normal operating conditions, the controller provides acceptable performance over the entire range of typical operating frequencies. Because the simulation contains no noise, the controller tracks the position past the filter cutoff frequency of 2 kHz. With noise, the performance will begin to degrade near the cutoff frequency.

The geometric parameters for the bearing model were chosen to be consistent with the model developed in [6]. Better performance could be achieved with a geometry that increases the sensitivity of the filter output to displacements, such as decreasing the nominal gap length to increase the sensitivity of the inductance to displacement. It follows that the self-sensing technique must often be matched with a suitable application. High sensing performance requires special bearing geometries that maximize the range of current slope magnitudes delivered by the filters. The greater sensitivity to displacement, the higher accuracy of sensed position.

Discrete position sensors will, in general, offer even higher signal-to-noise ratios than can be delivered by bearing geometries and materials optimized for self-sensing because they do not have to simultaneously satisfy the performance requirements of actuation and sensing, which are somewhat in conflict. Thus, for applications which require very tight rotor-bearing tolerances or very stiff suspensions, discrete position sensors may be required.

Conclusions and Future Work

This paper has demonstrated that a self-sensing scheme can provide acceptable position accuracy and bandwidth under various fault conditions. Furthermore, it has examined the increase in noise and the loss of bandwidth under these fault conditions.

Of particular importance is the observation that a single estimator can suffice for wide range of fault scenarios. This is significant because the very high throughput requirements of the estimator (greater than 100 kHz) demand that it be implemented in analog circuitry. Consequently, it is advantageous to use a single controller for all fault states. The reasons include both space consider-

ations as well as cost and complexity of implementing controller switching hardware: This is not a simple matter of software selection of a different algorithm depending on fault state.

Future work for the controller includes designing filters that reflect the fundamental physical limitations of the system. In the present development, the signals x , y , and g were treated as random variables for the purpose of controller construction. However, these signals have reasonably bounded spectra: These bounds can be explicitly acknowledged in synthesizing the controller. Additionally, the present development doesn't include a noise model for the signals z . As such, the controller attempts to extend the sensor bandwidth to an extent which is probably not feasible in physical hardware (experimental experience with the single magnet problem supports this.) Again, as in the case of the displacement signals, estimates of bounds on the noise spectra can be made and incorporated into the controller synthesis. This would be expected to achieve a suitable balance between estimator bandwidth and noise rejection.

In addition, application of this general methodology to tristate switching amplifiers is viewed as a prerequisite to broad commercial application. There is no fundamental problem with applying this parameter estimation paradigm to tristate amplifiers, but significant effort must be expended in determining the appropriate signal processing to best extract the current slope information which is nearly buried in sensing noise for these amplifiers.

Finally, as noted by Noh [3], magnetic saturation presents a substantial challenge to self-sensing magnetic bearings. The parameter estimation scheme provides a perfect hook into solving this problem because the embedded model can potentially be changed to include an approximation to the saturation nonlinearity, which should then allow the estimator to account for saturation. However, this poses numerous technical challenges including economical circuit implementation of the magnetic nonlinearities, appropriate identification methods, and the stability of the estimation network with this new plant whose gain would now be substantially nonlinear.

Experimental testing is initially planned for a rigid-rotor, two-bearing assembly which includes disturbance application while operating in the bearing nonsaturation regime. Eventual testing will be performed into the saturation region under all types of amplifier fault conditions that are actuator tolerable.

Nomenclature

- A_g = cross-sectional area of flux path in air gap
- G = force feed-through gain
- g = actual average air gap length
- g_0 = nominal air gap length (room temperature zero speed)
- I = vector of bearing coil currents
- L = bearing inductance matrix
- N_I = magnetic flux linkage matrix
- N_V = flux-voltage relationship matrix
- n_s = number of coil turns around each pole
- R = bearing reluctance matrix
- V = vector of coil voltages
- w = total energy in a magnetic circuit
- v_{ps} = switching amplifier power supply voltage
- x = horizontal displacement of the bearing journal, normal to the axis of rotation
- y = vertical displacement of the bearing journal, normal to the axis of rotation
- z = envelope filter output
- α = reciprocal of filter time constant
- Δ = diagonal matrix of amplifier duty cycles
- Φ = vector of bearing magnetic fluxes
- μ_0 = magnetic permeability of a vacuum, $4\pi \times 10^{-7}$ Wb/Am
- Ω = diagonal matrix of coil resistances
- $(\cdot)$ = time derivative of the quantity $(\cdot)$
- $(\hat{\cdot})$ = estimate of $(\cdot)$

References

- [1] Iannello, V., "Sensorless Position Detector for an Active Magnetic Bearing," U.S. patent pending, application No. 08/138, 217.
- [2] Mizuno, T., Bleuler, H., Gähler, C., and Vischer, D., 1992, "Towards Practical Applications of Self-Sensing Magnetic Bearings," *Proceedings of the Third International Symposium on Magnetic Bearings*, Tokyo, Japan, Aug., Technomic Publishing Co., Lancaster, PA.
- [3] Noh, M. D., 1996, "Self-Sensing Magnetic Bearings Driven by a Switching Power Amplifier," Ph.D. dissertation, University of Virginia.
- [4] Vischer, D., and Bleuler, H., 1992, "A New Approach to Sensorless and Voltage Controlled AMBs Based on Network Theory Concepts," *Proceedings of the Second International Symposium on Magnetic Bearings*, Tokyo, July 12–14, Technomic Publishing Company, Lancaster, PA, pp. 301–306.
- [5] Demarest, K. R., 1998, *Engineering Electromagnetics*, Prentice-Hall, Englewood Cliffs, NJ.
- [6] Meeker, D. C., 1996, "Optimal Solutions to the Inverse Problem in Quadratic Magnetic Actuators," Ph.D. dissertation, University of Virginia.
- [7] Keith, F. J., Maslen, E. H., Humphris, R. R., and Williams, R. D., 1990, "Switching Power Amplifier Design for Magnetic Bearings," *Proceedings of the Second International Symposium on Magnetic Bearings*, Tokyo, Japan, July 12–14, Technomic Publishing Company, Lancaster, PA, pp. 211–218.
- [8] Green, M., and Limebeer, D. J. N., 1995, *Linear Robust Control*, Prentice-Hall, Englewood Cliffs, NJ.

Air Entrainment Versus Lubricant Vaporization in Squeeze Film Dampers: An Experimental Assessment of Their Fundamental Differences

S. E. Diaz

L. A. San Andrés

Mechanical Engineering Department,
Texas A&M University,
College Station, TX 77843-3123

Squeeze film dampers (SFDs) provide structural isolation and energy dissipation in air-breathing engines and process gas compressors. However, SFDs are prone to develop a flow regime where the ingestion of air leads to the formation of a bubbly lubricant. This pervasive phenomenon lacks proper physical understanding and sound analytical modeling, although actual practice demonstrates that it greatly reduces the damper force response. Measurements of film pressures in a test SFD describing circular centered orbits at whirl frequencies varying from 0 to 100 Hz are presented for fully flooded and vented discharge operating conditions. The experiments demonstrate that operation with low levels of external pressurization, moderate to large whirl frequencies, and lubricant discharge to ambient leads to the entrapment of air within the damper film lands. The experiments also elucidate fundamental differences in the generation of film pressures and forces for operation in a flooded condition that evidences vapor cavitation. Damping forces for the vented end with air entrainment are just 15 percent of the forces measured for the flooded damper. Further measurements at constant whirl frequencies demonstrate that increasing the lubricant pressure supply retards the onset of air entrainment. Classical fluid film cavitation models predict well the pressures and forces for the lubricant vapor cavitation condition. However, prevailing models fail to reproduce the dynamic forced response of vented (open-ended) SFDs where air entrainment makes a foamy lubricant, which limits severely the damper film pressures and forces.

[DOI: 10.1115/1.1383258]

Introduction

Excessive steady state synchronous vibration levels and subharmonic rotor instabilities are the two most common problems in rotordynamics. Squeeze film dampers (SFDs) are to date the only effective means to provide structural isolation and vibration dissipation in air breathing engines ([1]). An SFD comprises a thin lubricant film within the annular gap between a non-rotating journal and a stationary housing. The journal is typically connected to the rotor shaft through a ball bearing, though any other type of bearing can be used ([2]). When the rotor vibrates, the journal is forced to follow its whirling motion, thus squeezing the lubricant film. The fluid film develops hydrodynamic pressures in the zones where the film thickness locally decreases. However, when the local film thickness increases the lubricant flow may generate suction (negative dynamic) pressures that may lead to lubricant vaporization or air entrainment.

The Reynolds lubrication equation for incompressible fluids is hitherto used to predict the pressure field in hydrodynamic bearings, including SFDs ([3]). However, Childs [1] Walton et al. [4], and Zeidan and Vance [5,6] establish that the correlation between theory and experiment is considerably less compelling for dampers than for bearings due to the appearance of dynamic lubricant cavitation. It is customarily assumed that, since an ideal fluid is unable to sustain tension, the lubricant would undergo an instan-

taneous phase change once the fluid vapor pressure is reached. As a result, a vapor filled cavity interrupting the continuity of the film develops. Numerous physical models address the phenomenon of vapor cavitation with different levels of complexity. Braun and Hendricks [7], and Ku and Tichy [8] review these models as applied to the Reynolds equation with boundary conditions defining the onset of cavitation and the full film reformation zone. The simplest model, the half-Sommerfeld or π -film condition, replaces with a uniform (cavitation) pressure any negative pressure resulting from solution of the Reynolds equation. Next, the Swift-Stieber (or Reynolds) condition enforces a null pressure gradient at the cavitation inception boundary to account for flow continuity. The separation model, and the Floberg and Jakobsson-Floberg-Olsson models account for mass transport across the cavitation zone and allow for the occurrence of subambient pressures within the thin fluid film. These models differ from each other in the assumptions about the path followed by the fluid through the cavitation zone.

Zeidan and Vance [9–11] Walton et al. [4] and Hibner and Bansal [12] report a different film rupture mechanism in open-ended squeeze film dampers, as it occurs when the relative movement between the SFD journal and housing drags air into the film eventually leading to a bubbly mixture. Many authors refer to this phenomenon as gaseous cavitation. However, this term is a misnomer strictly applicable to instances where dissolved gases in the lubricant come out of solution at pressures below the saturation value for the noncondensable gas (air), thus forming a single large bubble. In SFDs, as well and in dynamically loaded journal bearings, the bubbly flow due to air entrainment forms a homogeneous mixture with small bubbles well dispersed throughout the film and persisting even in the high pressure zones (Dowson and Taylor

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-187. Manuscript received by IGTI Oct. 1998; final revision received by the ASME Headquarters Mar. 1999. Associate Editor: D. Wisler.

[13], Jacobson and Hamrock [14], Zeidan and Vance, [5,9] Sun et al. [15]). Foamy oil at the damper outlet evidences this pervasive operating condition.

Zeidan et al. [2] review the state of the art in SFDs and remark about the importance of the air entrainment phenomenon, as it considerably reduces the dynamic film forces and the overall damping capability of SFDs. There are no accurate models addressing to the effects of air ingestion in dynamically loaded bearings and SFDs, nor are there relevant experimental measurements that could aid in validating predictive analysis. Only recently, Diaz and San Andrés [16–18] provide experimental results on a test SFD describing circular centered orbits and correlate dynamic pressures and squeeze film forces to the volume concentration of air in a controlled bubbly mixture. The experiments evidence a dramatic reduction in film pressures and forces as the air volume fraction increases in the mixture.

The experimental results hereby presented evidence the paramount effect of air entrainment in the performance of SFDs operating with an open-end to ambient discharge, the most common configuration found in practical applications. The measurements evidence the fundamental differences in film pressure generation and forces for operation with a vented end versus a fully flooded condition where lubricant vaporization prevails. The test results also demonstrate the similarity between the dynamic pressure fields generated by air entrainment and those measured for operation with a controlled homogeneous bubbly mixture. The effect of increased supply pressure and lubricant through flow on the delay of air entrainment within the fluid film is also investigated.

Test Rig Description

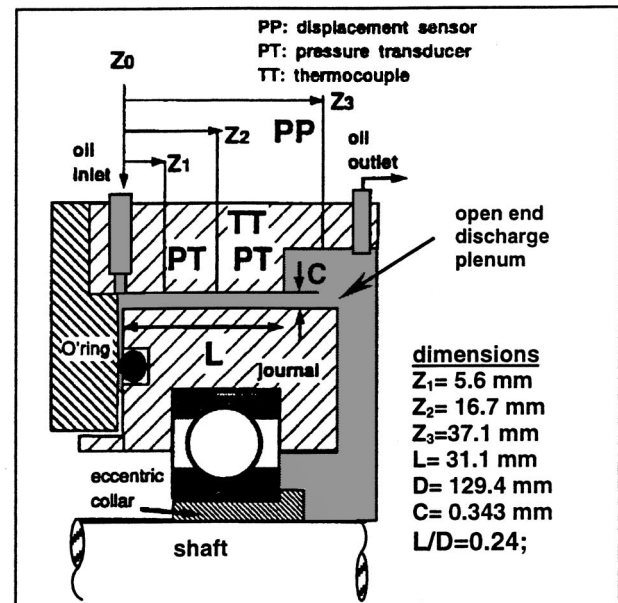
The experiments are conducted in an open-ended damper operating with a circular centered whirl orbit of amplitude equal to 50 percent of the film clearance. Diaz and San Andrés [16–18] describe the test apparatus, measurement procedure and uncertainty analysis. Figure 1 depicts the damper test section, flow loop, instrumentation and relevant nomenclature. The damper journal diameter (D) and axial length (L) are equal to 129.4 mm and 31.1 mm, respectively. The journal and its ball bearing are mounted eccentrically on a rigid shaft. A variable-speed DC motor and belt transmission drive the shaft supported on two precision ball bearings. At a pressurized condition of 7.0 bar the nominal radial clearance (c) and orbit radius (e) are equal to 0.343 mm and 0.216 mm, respectively. The left end of the damper is tightly sealed with an O-ring, while its right end opens to an exit plenum at ambient pressure. The lubricant enters the damper through two holes at the top (90 deg) and bottom (270 deg) of the damper left end.

The experiments are performed with an ISO VG 68 pure lubricant of density and viscosity equal to 0.87 gr/cm³ and 77.5 centipoise at 28°C, respectively. The lubricant is delivered to the test section via a gear pump and removed from the discharge plenum with an auxiliary pump. The lubricant flow rate and supply pressure are recorded at the inlet port with a gear flowmeter and a Bourdon-type pressure gauge, respectively. A K-type thermocouple detects the lubricant film temperature, a photoelectric tachometer senses the drive shaft speed (equal to the journal whirl frequency), and two eddy current displacement sensors record the motion of the journal. Six piezoelectric dynamic pressure sensors are flush mounted around the damper circumference at two axial planes (Z_1 and Z_2) as depicted in Figs. 1(a,b).

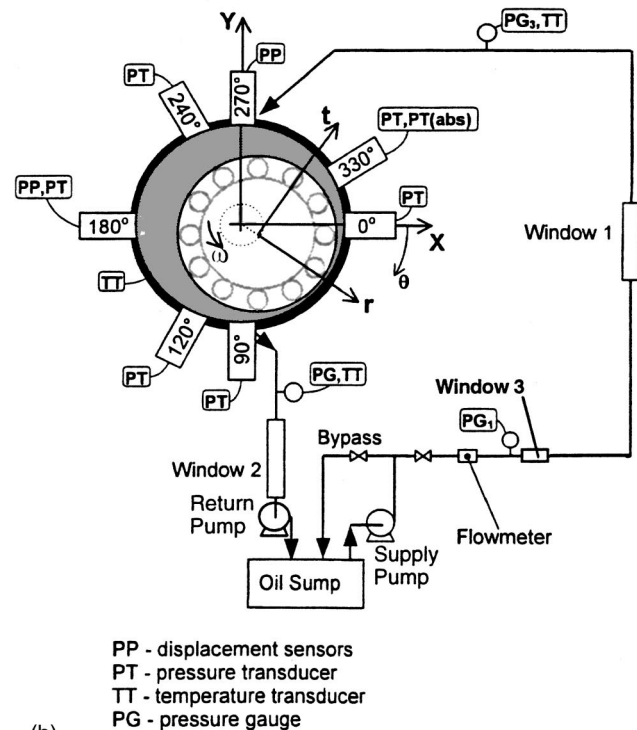
Experimental Procedure

The lubricant supply pressure and flow rate, film temperature, and synchronous whirl frequency are manually recorded in the experiments. The dynamic film pressure at the location (180 deg, Z_2), see Fig. 1, is measured with a piezoelectric transducer and digitally recorded with 16 bits of resolution at a sampling rate of 2048 Hz for a total length of 2 sec.

Four types of experiments have been conducted to evaluate the performance of the test SFD under different operating conditions.



(a)



(b)

Fig. 1 Test apparatus; (a) squeeze film damper detail, (b) schematic of test rig and instrumentation

Measurements of the pressure field and film forces are performed for fully flooded and open-end vented conditions and for increasing whirl frequencies while maintaining the same supply pressure. In the first type of measurements, the damper discharge plenum is fully flooded with lubricant at atmospheric pressure preventing the ingestion of external air. In this case, the fluid in the damper film lands shows vapor cavitation while the damper operates at a given whirl frequency. The second type of measurements exemplifies a more realistic operating condition where the lubricant discharges to an open plenum vented to atmospheric conditions. In this case

air entrains naturally into the squeeze film lands and the discharged lubricant has the appearance of a bubbly mixture resembling a foamy fluid.

In the measurements with increasing whirl frequencies, the supply pressure is set to a constant value of 2.3 bar, which corresponds to a lubricant flow rate of 0.4 l/min. In the vented to ambient condition tests the lubricant is evacuated from the plenum discharge chamber using a small diaphragm pump suctioning at a bottom port. On the other hand, the whole damper test section is submerged in a lubricant bath for the flooded condition tests. In this last case, the lubricant is evacuated from a top port directly to the oil sump. The plenum pressure is at atmospheric pressure for both test conditions. A transparent window on the damper housing discharge section allows verifying that the plenum remains either fully flooded or empty, depending on the type of test conducted. The SFD rig is brought rapidly to a maximum shaft speed of 6,000 rpm (100 Hz), and the measurements are performed while reducing the speed by steps of 500 rpm (8.33 Hz) each.

The other two types of measurements illustrate the performance of the open-end vented SFD for different lubricant flow rates (pressure supplies) and operating at two whirl frequencies, 8.33 Hz (500 rpm) and 16.67 Hz (1000 rpm). The measurements are performed while the supply pressure (or oil through flow) is increased in discrete steps until the return auxiliary pump is not able to evacuate all the lubricant and the discharge plenum starts to overflow.

The manufacturer values for the uncertainty of the measurement sensors have been verified. The individual errors are 1 percent of measurement for the oil flowmeter, 3 percent of full scale (10.3 bar) for the Bourdon pressure gauge, and 1 percent of full scale (100°C) for the thermocouples. The instrumentation uncertainty on the dynamic pressure measurements is mainly related to the A/D conversion, which is on the order of hundredths of a bar. However, the large temporal fluctuations of the pressures introduce a first-order uncertainty ([19]) that overcomes the instrumentation uncertainty by about two orders of magnitude. The discussion of test results addresses at length this issue.

Experimental Results

Figure 2 depicts the measured squeeze film pressures for the fully flooded and vented operating conditions. The solid lines represent the dynamic pressures and the dotted lines show the variation of the film thickness at the location of measurement and for five periods of journal whirl motion. In both tests, the supply pressure is 1.45 bar and the discharge plenum is nearly at atmospheric pressure. The oil flow rate is 0.12 liter/min and the film temperature is 27°C. The SFD journal describes circular centered orbits with an orbit radius of 0.180 mm at a whirl frequency of 75 Hz.

Figure 2(a) shows the typical pressure profile found when the pure lubricant undergoes a vapor cavitation process. Note that the pressure profile is smooth and shows nearly identical shapes for each period of motion. A (flat) constant pressure zone develops at nearly zero absolute pressure, and it corresponds to the rupture of the film and formation of a vapor filled cavity. The vapor cavity appears only during that portion of the journal motion cycle where the film gap increases. The vapor bubble collapses immediately as the local pressure raises above the lubricant vapor pressure. In general, correlations of measured pressures and vapor cavitation extent with predictions based on traditional film rupture models are satisfactory.

The pressure profile shown in Fig. 3(b) shows the effects of air entrainment for the vented to ambient condition. The periodic motion of the journal not only squeezes the lubricant but it also induces air ingestion and entrapment. At the times when the local film gap increases a suction pressure draws air into the thin film and leads to the formation of a bubbly mixture of air in oil, with bubbles persisting even in the high pressure zones. Note that the squeeze film pressures differ markedly for each period of journal

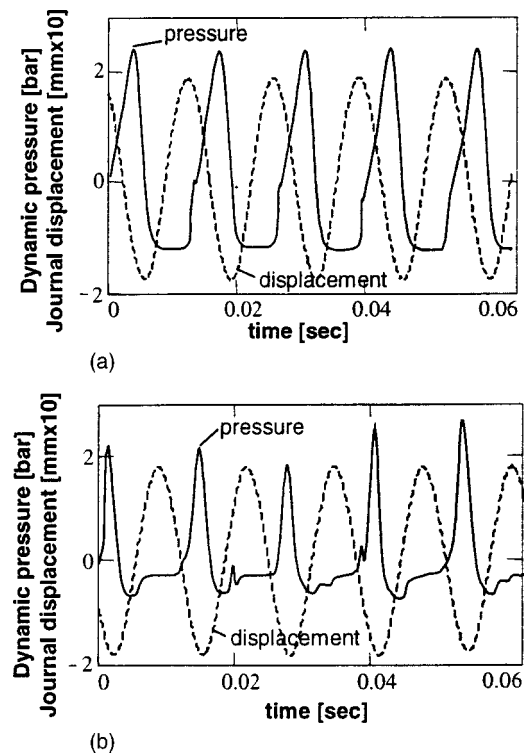
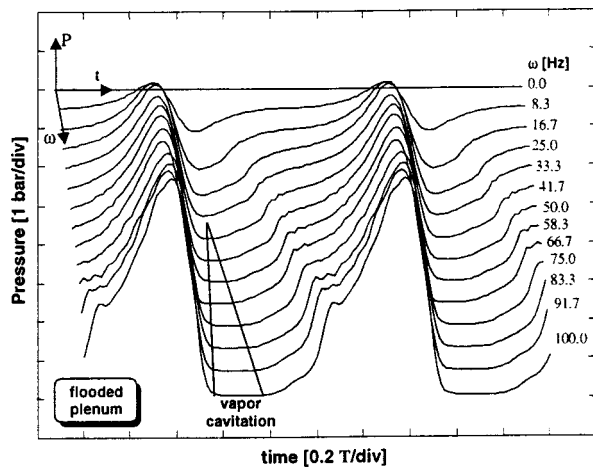


Fig. 2 Dynamic pressure and journal displacement versus time; (a) flooded plenum and (b) vented plenum

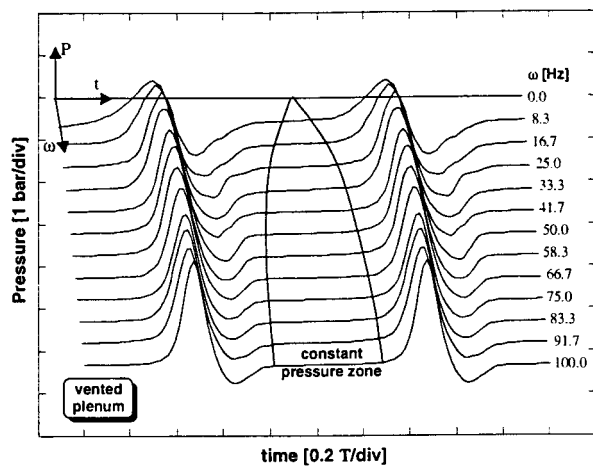
motion and even the peak pressures undergo large variations. A pressure flat zone at nearly ambient pressure, typically denoted as gaseous cavitation, initiates when the film goes from its minimum to its maximum values (negative squeeze action). The generation of subambient film pressures is also unique during the negative squeeze portion of the journal motion cycle.

A period-averaging scheme is applied to the measured pressures to filter random fluctuations, thus generating smooth periodic pressure fields representative of the overall behavior of the squeeze film pressures over many cycles of motion. Figures 3(a) and 3(b) compile the period-averaged dynamic pressures corresponding to measurements at different operating whirl frequencies for the flooded and vented open end, respectively. The vertical axis represents the pressure, the horizontal axis depicts the time normalized by the period of motion, and the off-plane axis denotes the whirl frequency ranging from 0 to 100 Hz (0–6000 rpm). For the flooded condition and at low whirl frequencies the level of external supply pressure prevents the onset of the fluid vapor cavitation zone. However, as the whirl frequency increases, the dynamic film pressures are low enough and approach the vaporization pressure, thus initiating the vapor cavitation zone. The extent of the vapor cavitation zone increases linearly with the journal whirl frequency.

On the other hand, the dynamic pressures corresponding to operation with a vented discharge condition (Fig. 3(c)) are most distinctive and resemble the pressures measured earlier in the same test rig operating with a controlled homogeneous bubbly mixture of air and oil ([16–18]). A constant pressure zone that interrupts the pressure rise from the minimum to the maximum peak develops and with a magnitude nearly equal to the ambient pressure. The extent of the flat pressure zone increases rapidly with the whirl frequency for low frequencies. However, as the operating frequency increases further, the extent of the constant pressure zone appears nearly constant. Diaz and San Andres [16,18] have correlated the extent of the constant pressure zone with the concentration of air in a test SFD operating with a controlled homogeneous bubbly mixture. Thus, the extent of the flat



(a)



(b)

Fig. 3 Development of the period-averaged pressure field and uniform pressure zone versus whirl frequency (flooded and vented end conditions); (a) flooded plenum, (b) vented plenum

pressure zone in the current measurements denotes increasing degrees of air entrainment as the whirl frequency increases.

Figure 4 provides a quantitative comparison of the peak-to-peak ($p-p$) pressures generated by the test SFD operating with vapor cavitation (flooded plenum) and with air entrainment (vented open end). The magnitude of the $p-p$ pressures for the vented condition represents an averaged value from the many periods of measurement. The vertical bars represent the first-order uncertainty as defined by Moffat [19], i.e., they correspond to the largest deviations (above and below) from the average value for the entire length of

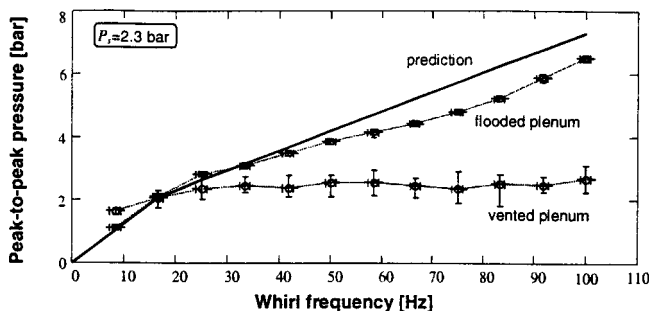


Fig. 4 Peak-to-peak pressure versus whirl frequency (flooded and vented end conditions and predictions)

the experiment. Figure 4 also includes an analytical prediction of the $p-p$ pressures and based on the simplest model available, the short length bearing with open ends, and given as

$$P_{\text{short}} = \frac{6\mu e\omega}{h^3} \sin(\theta)(Z^1 - L^2) + P_o \quad (1)$$

and accounting for vapor cavitation with the simple relation

$$P = \begin{cases} P_{\text{short}} & \text{if } P_{\text{short}} > 0 \\ 0 & \text{if } P_{\text{short}} \leq 0 \end{cases} \quad (2)$$

Refer to the Nomenclature for a proper definition of all variables.

The short length bearing model predicts a linear growth of the $p-p$ pressures with the journal whirl frequency. The rate of $p-p$ pressure rise reduces above the whirl frequency at which the lubricant vapor cavitation initiates (~ 17 Hz). The experimental pressures for the flooded SFD condition correlate very well with the predicted values. However, the $p-p$ pressures for the vented condition quickly reach a constant value, (nearly) independent of the whirl frequency and much lower than for the flooded condition. The first-order uncertainties for the flooded condition are rather small indicating a time invariant pressure field. On the other hand, the tests with air ingestion and entrapment lead to $p-p$ pressure fluctuations of up to 35 percent.

A SFD describing circular centered orbits and with a stationary¹ pressure field (synchronous with the shaft) generates invariant forces in a radial and tangential coordinate system rotating with the same speed as the shaft. A pure tangential force is distinctive of an ideal SFD since this force directly opposes any forward whirl motion. Radial forces could be centering denoting a stiffness-like effect, or outwards implying a fluid inertia effect. Dynamic damper forces (f_r, f_t) in the radial and tangential directions arise from integration of the pressure field around the journal surface. A kinematic relation between the *stationary* (though rotating) pressure field and the *dynamic* pressure field at a fixed location is used to transform the spatial integration around the journal circumference into integration in time. The relationship is

$$f_{(z)} = (f_r + i \cdot f_t)_{(z)} = \frac{-\omega}{n} \int_{t_0}^{t_0 + n \cdot T} P_{(t,z)} \cdot e^{-i \cdot [e(t-t_0)]} dt. \quad (3)$$

Refer to the Nomenclature for a proper definition of all variables.

Figure 5 depicts the radial and tangential forces per unit length at the axial location of measurement (Z_2) versus the journal whirl frequency. Circles and squares denote the measured forces for the vented and flooded conditions, respectively, and solid lines represent the forces predicted by the short length bearing model. Note that the radial forces are much lower than the tangential (damping) forces. The analytical predictions show the damping force to increase linearly with the whirl frequency, although with a reduced slope after vapor cavitation initiates. The calculated radial force is nil for a full-film condition, and increases linearly with the whirl frequency after vapor cavitation initiates. The predictions agree well with the forces derived from the experimental pressures under a flooded discharge operating condition. Conversely, for the vented plenum and where air entrains naturally into the fluid film, the damping (tangential) force actually decreases as the whirl frequency increases and becomes a mere fraction (15 percent) of the damping force obtained for the flooded SFD condition. At large whirl frequencies when air entrainment is more severe; the radial force becomes larger than the tangential force.

Figure 6 demonstrates the effects of the supply pressure (lubricant flow rate) on the generation of film pressures and the onset of air entrainment. Figure 6(a) compiles the period averaged pressures at an operating frequency equal to 16.67 Hz (1000 rpm), and

¹Here the stationary qualification means an invariant pressure field relative to an observer positioned in the spinning shaft.

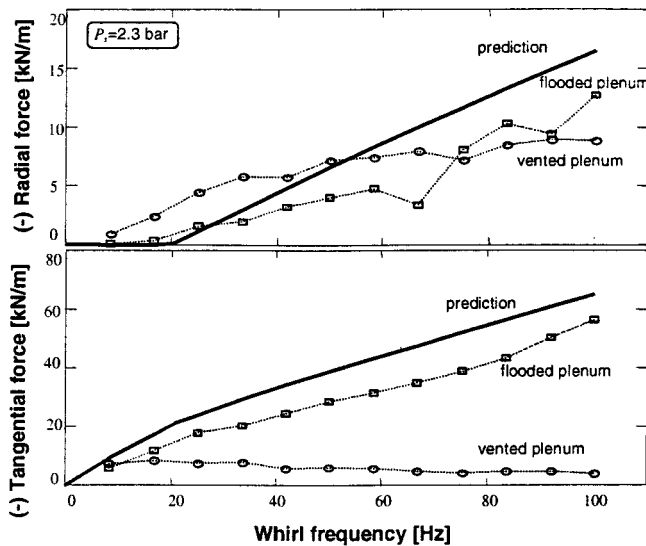
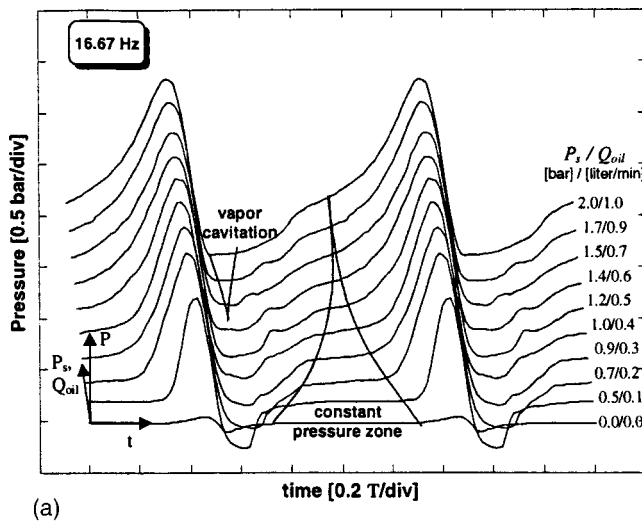
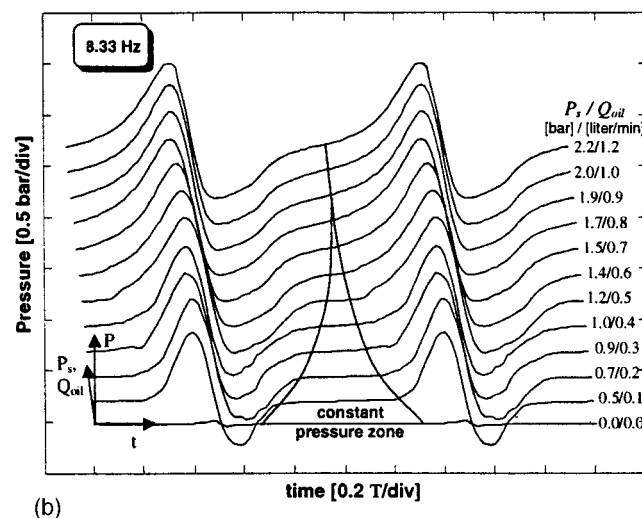


Fig. 5 Radial and tangential forces versus whirl frequency (flooded and vented end conditions and predictions)



(a)



(b)

Fig. 6 Development of the period-averaged pressure field and uniform pressure zone versus the lubricant flow and supply pressure; (a) whirl frequency 16.67 Hz (1000 rpm), (b) whirl frequency 8.33 Hz (500 rpm)

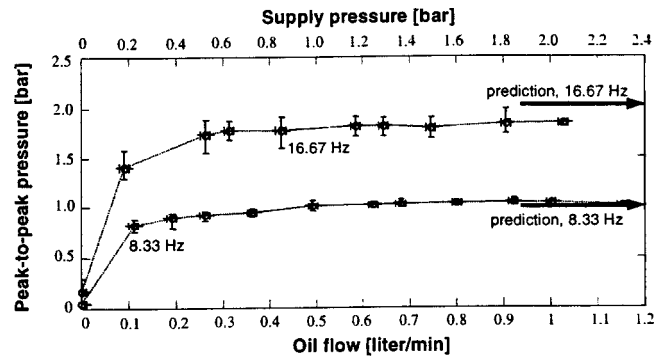


Fig. 7 Peak-to-peak pressures versus oil flow rate and supply pressure for two whirl frequencies

Fig. 6(b) depicts the pressures at 8.33 Hz (500 rpm). The SFD discharge plenum is vented to atmospheric pressure. In the figures, the vertical axis represents the pressure, the horizontal axis depicts the time normalized by the period of motion, and the in-plane axis gives the magnitude of supply pressure and lubricant flow rate. The measurements evidence that increasing the lubricant through flow rate reduces the extent of the constant pressure zone where air is entrapped. That is, the severity of air entrainment decreases allowing for the generation of larger squeeze film pressures and fluid film forces. Note that in the tests at 16.67 Hz, a lubricant vapor cavitation zone starts to develop for the largest flow rates.

Figure 7 depicts the period averaged p - p pressures corresponding to the dynamic pressures shown in Fig. 6 versus the oil flow rate and the supply pressure. The figure also includes the predicted theoretical values derived from the short bearing model given by Eq. (2). For both operating whirl frequencies, the p - p pressure grows rapidly with the supply pressure. For large flow rates the p - p pressures approach an asymptotic value, and correlate well with the simple analytical values. Note that the theoretical model does not predict fluid vaporization or air entrainment at these low frequencies. The experiments also evidence that larger lubricant flow rates are needed to control air entrainment as the whirl frequency increases.

Figure 8 depicts the test radial and tangential forces versus the oil flow rate and supply pressure for the whirl frequencies equal to

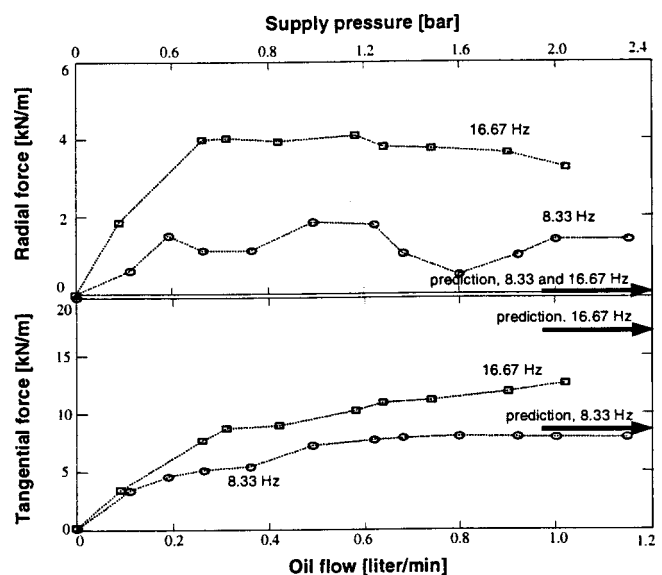


Fig. 8 Radial and tangential forces versus oil flow rate (supply pressure) for two whirl frequencies

8.33 and 16.67 Hz. The experiments at the lower whirl frequency reveal a rapid increase of the tangential film force towards the analytical asymptotic value. However, at the whirl frequency of 16.67 Hz, the tangential (damping) force grows continuously within the full range of test supply pressures. Unfortunately, the current test facility does not allow for higher levels of pressurization so it is not possible to confirm the asymptotic trend of the tangential force for large flow rates. The radial forces are small and appear to decrease with the supply pressure for the higher test whirl frequency (16.67 Hz). Note that in practice, higher supply pressures are not readily available nor are desired because of the increased cost and bulky hardware.

Conclusions

The force response of squeeze film dampers is affected by numerous factors, including geometric characteristics, lubricant type, whirl frequency, and supply and discharge mechanisms. Ingestion and entrapment of air is pervasive in SFDs operating with vented open ends at ambient pressure. The fluid film becomes a foamy-like mixture with air bubbles persisting in the regions of pressure generation above ambient. Current models for the rupture of thin films, gaseous and vapor cavitation, are strictly applicable for static loads, and their extension to transient or periodic loads gives accurate predictions only for bearings fully submerged in a lubricant bath.

Experiments have been conducted on a controlled orbit test SFD operating with two end conditions, one is fully flooded with lubricant, and the other one is just an open vent to ambient as most typical applications exemplify. Measurements of the dynamic film pressures and estimated fluid film forces are performed for increasing whirl frequencies at a fixed value of supply pressure. In the flooded SFD the film pressures are quasi-stationary and repetitive for each cycle of damper motion, and where the lubricant vapor cavitation zone increases linearly with the journal whirl frequency. On the other hand, for the vented condition, and as soon as air is ingested into the film lands, the film pressures show random variations and sudden spikes, zones of subambient pressures, and a discharge fluid appearing as a bubbly mixture. A constant pressure zone at ambient pressure also grows linearly with the whirl frequency.

Period-averaged peak-to-peak (p - p) pressures and film forces are calculated from the measured pressure fields. For the flooded condition, the p - p pressure increases linearly with the journal whirl frequency, and its rise rate reduces at the frequency where lubricant vapor cavitation initiates. The damping (tangential) force shows a similar trend. A radial (centering) force also appears when the fluid cavitates and its magnitude is proportional to the whirl frequency. In contrast, with a vented plenum condition, the p - p pressures quickly reach a constant value, (nearly) independent of the journal whirl frequency and much lower than for the flooded condition. The damping force actually decreases as the whirl frequency increases, and becomes a mere fraction (~ 15 percent) of the tangential force obtained from the flooded condition. At large whirl frequencies when air entrainment is more severe; the radial force becomes larger than the tangential force. Theoretical predictions based on a classical model predict well the pressures and forces for the flooded condition (vapor cavitation), but fail to reproduce the test values for the vented condition which develops into a foamy-like mixture.

Further experiments at low whirl frequencies demonstrate that increasing the supply pressure reduces the extent of air entrainment and increases the film p - p pressures and damping forces. However, the beneficial effect of external pressurization becomes less pronounced as the frequency of operation increases. A sound theoretical model accounting for air entrainment is needed for the reliable design of SFDs, thus reducing their costly trial and error tuning process. The test results suggest that considering the squeeze film flow as a homogeneous continuous mixture could lead to improved predictive analysis. Recently, Tao et al. [20]

present an initial model that compares favorably with controlled mixture SFD test results, though it requires of an empirical formulation for the bubbly fluid viscosity. Diaz [21] also offers another perspective to the analysis and presents a simple model, verified by the measurements, to predict the onset and extent of air entrainment and its paramount effect on SFD forced performance.

Acknowledgments

The support of the TAMU Turbomachinery Research Consortium is gratefully acknowledged. The first author thanks the financial aid from CONICIT (Consejo Nacional de Investigaciones Científicas y Tecnológicas) and Universidad Simón Bolívar, Venezuela.

Nomenclature

C	= damper radial clearance (0.343 mm)
D	= journal diameter (129.4 mm)
e	= journal orbit radius, nominal value (0.216 mm)
f_r, f_t	= radial and tangential film forces per unit length (N/m)
h	= film thickness, $C + e \cos(\omega t)$
i	= $\sqrt{-1}$ imaginary unit
L	= journal length (31.1 mm)
P	= film pressure (bar)
P_o, P_s	= exit plenum pressure (1.0 bar); supply pressure (bar)
P_{short}	= short length bearing solution for squeeze film pressure
Q_{oil}	= oil flow rate (liters/min)
r, t	= radial and tangential coordinates
t	= time (sec)
t_o	= time for minimum film thickness (sec)
T	= $2\pi/\omega$; period of journal motion (sec)
Z_j	= axial location of pressure measurements; Z_1 (5.16 mm), Z_2 (16.7 mm)
μ	= lubricant viscosity
θ	= angular coordinate in a rotating reference frame
ω	= journal whirl frequency (rad/sec)

References

- [1] Childs, D., 1993, *Turbomachinery Rotordynamics*, John Wiley and Sons, New York.
- [2] Zeidan, F. Y., Vance, J. M., and San Andrés, L. A., 1996 "Design and Application of Squeeze Film Dampers in Rotating Machinery," *Proceedings of the 25th Turbomachinery Symposium*, Texas A&M University, College Station, TX, pp. 169–188.
- [3] Vance, J. M., 1988, "Rotordynamics of Turbomachinery," John Wiley and Sons, New York.
- [4] Walton, J., Walowit, E., Zorzi, E., and Schrand, J., 1987, "Experimental Observation of Cavitating Squeeze Film Dampers," *ASME J. Tribol.*, **109**, pp. 290–295.
- [5] Zeidan, F. Y., and Vance, J. M., 1989, "Experimental Investigation of Cavitation Effects on The Squeeze Film Force Coefficients," *Rotating Machinery Dynamics*, DE-Vol. 18-1, ASME, New York, pp. 237–242.
- [6] Zeidan, F. Y., and Vance, J. M., 1990, "A Density Correlation for a Two-Phase Lubricant and its Effect on the Pressure Distribution," *STLE Tribol. Trans.*, **33**, pp. 641–647.
- [7] Braun, M. J., and Hendricks, R. C., 1984, "An Experimental Investigation of the Vaporous/Gaseous Cavity Characteristics of an Eccentric Journal Bearing," *ASLE Trans.*, **27**, No. 1, pp. 1–14.
- [8] Ku, C. P., and Tichy, J. A., 1990, "An Experimental and Theoretical Study of Cavitation in a Finite Submerged Squeeze Film Damper," *ASME J. Tribol.*, **112**, pp. 725–733.
- [9] Zeidan, F. Y., and Vance, J. M., 1989, "Cavitation Leading to a Two Phase Fluid in a Squeeze Film Damper," *STLE Tribol. Trans.*, **32**, No. 1, pp. 100–104.
- [10] Zeidan, F. Y., and Vance, J. M., 1990, "Cavitation Regimes in Squeeze Film Dampers and Their Effect on the Pressure Distribution," *STLE Tribol. Trans.*, **33**, pp. 447–453.
- [11] Zeidan, F. Y., and Vance, J. M., 1990c, "Cavitation and Air Entrainment Effects on the Response of Squeeze Film Supported Rotors," *ASME J. Tribol.*, **112**, pp. 347–353.
- [12] Hibner, D., and Bansal, P., 1979, "Effects of Fluid Compressibility on Viscous Damper Characteristics," *Proc. Conf. on the Stability and Dynamic Response of Rotors with Squeeze Film Bearings*, University of Virginia, Charlottesville, VA, pp. 116–132.

- [13] Dowson, D., Godet, M., and Taylor, C. M., 1974, eds, *Cavitation and Related Phenomena in Lubrication*, ImechE, London.
- [14] Jacobson, B. O., and Hamrock, B. J., 1983, "High-Speed Motion Picture Camera Experiments of Cavitation in Dynamically Loaded Journal Bearings," *ASME J. Tribol.*, **105**, pp. 446–452.
- [15] Sun, D. C., Brewe, D. E., and Abel, P. B., 1993, "Simultaneous Pressure Measurement and High-Speed Photography Study of Cavitation in a Dynamically Loaded Journal Bearing," *Trans. ASME*, **115**, pp. 88–95.
- [16] Diaz, S. E., and San Andrés, L., 1998, "Measurements of Pressure in a Squeeze Film Damper With an Air/Oil Bubbly Mixture," *STLE Tribol. Trans.*, **41**, No. 2, pp. 282–288.
- [17] Diaz, S., and San Andrés, L., 1998b, "Reduction of the Dynamic Load Capacity in a Squeeze Film Damper Operating with a Bubbly Lubricant," *ASME Paper 98-GT-109*.
- [18] Diaz, S., and San Andrés, L., 1998c, "Effects of Bubbly Flow on the Dynamic Pressure Fields of a Test Squeeze Film Damper," *ASME Paper FEDSM98-5070*.
- [19] Moffat, R. J., 1982, "Contributions to the Theory of Single-Sample Uncertainty Analysis," *ASME J. Fluid Eng.*, **104**, pp. 250–260.
- [20] Tao, L., Diaz, S., San Andrés, L., and Rajagopal, K., 1998, "Flow Analysis of Squeeze Film Dampers Operating With Bubbly Lubricants," *TAMU Turbomachinery Research Consortium Progress Report, TRC-SFD-1-98*, College Station, TX, May.
- [21] Diaz, S. E., 1999, "The Effect of Air Entrainment on the Forced Response of Squeeze Film Dampers: Experiments and Analysis," *Ph.D. dissertation*, Mechanical Eng. Dept. Texas A&M University, May.

Sliding Mode Control of Magnetic Bearings: A Hardware Perspective

E. Maslen

D. Montie

Department of Mechanical, Aerospace,
and Nuclear Engineering,
University of Virginia,
Thornton Hall, McCormick Road
Charlottesville, VA 22903-2442

The physical hardware of magnetic bearing systems has a very strong impact on the application of sliding mode control to this technology. Finite current slew rates affect the character of achievable reaching conditions; flexible rotors demand the use of state observers which then affect achievable robustness; finite controller throughput and use of switching power amplifiers has substantial impact on how chatter should be addressed. This paper formulates the sliding mode control problem for realistic rotor/magnetic actuator systems considering rotor flexibility and finite amplifier voltages. The structure of the resulting class of controllers is examined for both rigid and for flexible rotors. It is demonstrated that, fundamentally, these controllers are conventional linear state observer-based controllers acting through high-speed switching amplifiers as in conventional magnetic bearing technology. [DOI: 10.1115/1.1383771]

Introduction

A significant issue in the design of feedback controllers for any application is the ability to tolerate variability and unmodeled dynamics in the plant: magnetic bearing supported rotors are no exception. There are several control synthesis schemes which specifically address this issue and one of these is sliding mode control. Most widely used for magnetic bearings is the PID controller, often augmented with special filters to deal with higher order dynamics. A significant drawback to PID controllers is that the synthesis techniques are substantially *ad hoc* and provide no explicit means to deal with plant uncertainty other than uncertain actuator/sensor gain. Further, these *linear* techniques tell the designer nothing about what happens when any portion of the control loop becomes saturated, as can occur in magnetic bearings when driven hard by external loads.

Sliding mode control addresses all of these problems explicitly, allowing direct incorporation of forced response performance as well as parametric plant/load uncertainty. By dividing the response trajectory into two regimes: one where the control is essentially saturated and one where it is not, sliding mode directly addresses saturated as well as linear performance. Given these apparent advantages, it is useful to compare sliding mode to more conventional techniques to see exactly what it buys as a *synthesis* technique and as a *control* technique. This distinction is made because the thesis of this work is that sliding mode fundamentally offers no advantage over conventional techniques like augmented PID control in implementation, but can offer a useful perspective and collection of tools at the synthesis stage.

Numerous researchers have applied sliding mode control to magnetic bearings. Recently, Allaire and Sinha [1] used sliding mode techniques to develop controllers robust to several potential sources of uncertainty in the plant. Previously, Tian et al. [2] looked at discrete sliding mode control of flexible rotors including a disturbance model. Other papers ([3–5]) explore specific applications of the method. The present work is an effort to cast this approach into terms familiar to the industry so that sliding mode can be directly assessed and compared to more conventional approaches.

Contributed by the International Gas Turbine Institute (IGTI) 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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-205. Manuscript received by IGTI, Oct. 1998; final revision received by the ASME Headquarters, Mar. 1999. Associate Editor: D. Wisler.

Sliding Mode Control

The main idea behind sliding mode control ([6]) is to convert the control problem for multistate control systems to a simpler first-order control problem. Consider

$$\begin{aligned} \dot{x}^{(n)} &= f(\xi, u; t) \quad \xi^T = \{x \quad x^{(1)} \dots x^{(n-1)}\} \\ x &= h(\xi, u; t) \end{aligned} \quad (1)$$

in which the state vector, ξ , has dimension n while the control and output variables u and x are scalars. The notation $x^{(i)}$ denotes the i th time derivative of x . In general, the functions f and h are nonlinear.

The control objective is to find an input u which drives x to zero in some nice manner. The approach is to define a *sliding function*, $s(x; t)$ as

$$s(x; t) = \left\{ \prod_{i=1}^{n-1} \left(\frac{d}{dt} + \lambda_i \right) \right\} x \quad (2)$$

and then require that this sliding function approach zero:

$$\frac{d}{dt} \left(\frac{1}{2} s(x; t)^2 \right) \leq -\eta |s(x; t)| \quad (3)$$

Constraint (3) is called the *reaching condition*.

The function s and its derivative $\dot{s}$ are algebraic combinations of the state vector elements and the control variable, u . Consequently, satisfaction of (3) leads to a constraint on u which can be met by a rather free choice of functions for u . In general, this free choice will be limited by practical considerations so that (3) constitutes a formal synthesis procedure.

Of particular interest in (3) is that, if the plant uncertainties can be represented by free parameters with zero mean and fixed interval, then the solutions for u have discontinuities which overbound these intervals. To see this, expand (3) as

$$s(x; t) \dot{s}(x; t) \leq -\eta |s(x; t)| \quad (4)$$

and, noting that $s(x; t) = \text{sign}(s(x; t)) |s(x; t)|$, then (3) is equivalent to

$$-\text{sign}(s(x; t)) \dot{s}(x; t) \geq \eta.$$

If the function $\dot{s}(x; t)$ can be rearranged as

$$\dot{s}(x; t) = g(\xi) + r(u) + \beta$$

in which β is the uncertain interval, then the control u has the general form

$$u = -r^{-1} [g(\xi) + (\eta + \max|\beta|) \text{sign}(s(x; t))].$$

Both η and $\max|\beta|$ are constants so the control law is a combination of a potentially continuous function in the states and a switching function with a discontinuity at $s(x;t)=0$. A similar relationship can be developed in the case that the function r has uncertainty.

In general, the state variables must be estimated with a state observer. Assuming that the estimation error can be bounded, the general solution will have the form

$$u = -r^{-1}[g(\xi + \xi_e) + (\eta + \max|\beta|)\text{sign}[s + s_e]]$$

$$\beta = g(\xi + \xi_e) - g(\xi)$$

in which ξ_e is the state estimation error and s_e is the resulting error in estimation of the sliding function. In this case, the sliding condition can only be assured for $|s| > |s_e|$, where $\text{sign}(s) = \text{sign}(s + s_e)$. This bound describes an envelope surrounding the sliding surface $S: s=0$. Within this envelope, or *boundary layer*, the actual system behavior will depend strongly on the character of the estimation error, ξ_e . If the boundary layer is sufficiently small, this behavior may not matter to the engineer: it is sufficient that convergence to the boundary layer is guaranteed.

The particular structure of the sliding function (2) and the constraint (3) leads to the conclusion ([6]) that the condition $|s(x;t)| \leq |s_e(x;t)|$ is reached in finite time t_r , with a bound of $t_r < s(0)/\eta$. In this interval, the control is *saturated* in the sense that the control is often dominated by the term $r^{-1}(\eta + \max|\beta|)\text{sign}(s(x;t))$ which doesn't vary until $|s|$ reaches $|s_e|$. Further, if $s_e = 0$, then once $s(x;t)=0$, the control law switches at a theoretically unbounded rate to constrain the system to slide along the surface $s(x;t)=0$. In this mode, the plant dynamics are set by (2) which means that the plant order is effectively reduced by 1 and the remaining eigenvalues of the plant are replaced by $-\lambda_i$, $i=1 \dots n-1$.

The ensuing discussion demonstrates that feedback in magnetic bearings uses a minor loop to wipe out the electrodynamics of the actuator: This is the single state eliminated; and the plant eigenvalues are replaced or transformed by the λ_i .

Control of a Single Mass

A single mass controlled by opposing electromagnets as illustrated in Fig. 1 can be modeled by

$$\frac{d}{dt} \begin{Bmatrix} x \\ \dot{x} \end{Bmatrix} = \begin{bmatrix} 0 & 1 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} x \\ \dot{x} \end{Bmatrix} + \frac{1}{m} \begin{bmatrix} 0 & 0 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}. \quad (5)$$

The two forces, f_1 and f_2 , are generated by the fluxes, ϕ_1 and ϕ_2 , in the electromagnets as ([7])

$$f_1 = \frac{1}{A_g \mu_0} \phi_1^2, \quad f_2 = \frac{1}{A_g \mu_0} \phi_2^2 \quad (6)$$

in which A_g is the projected area of each air gap (all are assumed equal) and μ_0 is the permeability of free space. Equation (6) assumes that the magnetic flux is distributed uniformly over the air gap and ends abruptly at the edges of the pole pieces. The resulting error can be corrected for unsaturated magnetics and small gap variation by adjusting A_g .

The fluxes, in turn, are generated by applying voltages v_1 and v_2 to the actuator coils (each with N turns and a resistance R) ([7])

$$\dot{\phi}_1 = -\frac{R}{N} i_1 + \frac{1}{N} v_1, \quad \dot{\phi}_2 = -\frac{R}{N} i_2 + \frac{1}{N} v_2. \quad (7)$$

In the absence of leakage flux, this equation would be exact. However, some portion of the flux linked by the coil turns does not appear in the air gap: It leaks directly from one pole to the other. If the iron is operated well away from magnetic saturation, the ratio of leakage flux to gap flux is essentially constant and the model can be calibrated by a small adjustment of the attributed number of coil turns, N .

Finally, the currents i_1 and i_2 are associated with the fluxes by Ampère's loop law ([7])

$$i_1 = \frac{2(g-x)\phi_1}{\mu_0 N A_g}, \quad i_2 = \frac{2(g+x)\phi_2}{\mu_0 N A_g} \quad (8)$$

in which g is the nominal air gap and x is the displacement of the mass from center. Equation (8) ignores leakage, finite iron permeability, magnetic hysteresis, and eddy currents. If the material is kept from saturating, the effects of leakage and finite permeability can be compensated by adjusting the nominal gap, g , appropriately. The effects of eddy currents and hysteresis are harder to compensate and are neglected here.

Introducing the alternate state and control variables

$$\phi_a = \phi_1 - \phi_2, \quad \phi_b = \phi_1 + \phi_2$$

$$v_a = v_1 - v_2, \quad v_b = v_1 + v_2 \quad (9)$$

these equations can be combined into the single fourth-order differential equation

$$\frac{d}{dt} \begin{Bmatrix} x \\ \dot{x} \\ \phi_a \\ \phi_b \end{Bmatrix} = \begin{bmatrix} 0 & 1 & 0 & 0 \\ 0 & 0 & \frac{\phi_b}{m A_g \mu_0} & 0 \\ \kappa \frac{\phi_b}{g} & 0 & -\kappa & 0 \\ \kappa \frac{\phi_a}{g} & 0 & 0 & -\kappa \end{bmatrix} \begin{Bmatrix} x \\ \dot{x} \\ \phi_a \\ \phi_b \end{Bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & 0 \\ \frac{1}{N} & 0 \\ 0 & \frac{1}{N} \end{bmatrix} \begin{Bmatrix} v_a \\ v_b \end{Bmatrix} \quad (10)$$

in which $\kappa = 2Rg/(\mu_0 N^2 A_g)$. Note, very importantly, that the input variable is actuator coil voltage, not coil current. Each current is a nonlinear function of two states (x and ϕ_1 or ϕ_2) and so is useful as an estimator variable but cannot be commanded arbitrarily.

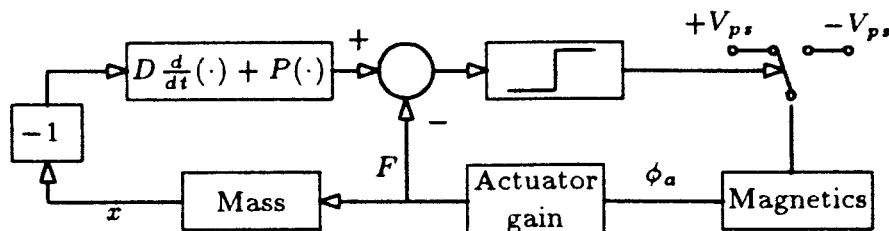


Fig. 1 Sliding mode control of a single mass in magnetic bearings

Treat the term in the lower left corner of the dynamic matrix as a small dynamic perturbation to the differential equation governing the evolution of ϕ_b , so that the problem becomes separated into two differential equations:

$$\frac{d}{dt} \begin{Bmatrix} x \\ \dot{x} \\ \phi_a \end{Bmatrix} = \begin{bmatrix} 0 & 1 & 0 \\ 0 & 0 & \frac{\phi_b}{mA_g\mu_0} \\ \kappa \frac{\phi_b}{g} & 0 & -\kappa \end{bmatrix} \begin{Bmatrix} x \\ \dot{x} \\ \phi_a \end{Bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{1}{N} \end{bmatrix} v_a \quad (11)$$

$$\dot{\phi}_b = -\kappa\phi_b + \frac{1}{N}v_b + \kappa \frac{\phi_a}{g}x. \quad (12)$$

Notice that the first equation contains ϕ_b as a parameter. The control strategy is to design a controller for the second differential equation which quickly brings ϕ_b to a fixed value and keeps it there in the face of moderate time varying perturbations. If this can be done, then the parametric dependency (11) is not a problem. Thus, there will be a *biasing* controller from v_b to ϕ_b and a *suspension* controller from v_a to ϕ_a , x , and $\dot{x}$.

Observers. Any control scheme for this system will require observers for any of the states which are not directly measurable. For the single mass problem, the states are x , $\dot{x}$, ϕ_a , and ϕ_b . Ideally, x is directly measurable as well as the currents i_1 and i_2 . Neglecting noise, $\dot{x}$ can be constructed from x by simple differentiation:

$$v = \dot{x}: V(s) = sX(s). \quad (13)$$

The two flux variables can be constructed from the measured position x and the two measured currents i_1 and i_2 :

$$\phi_1 = \frac{\mu_0 NA_g}{2(g-x)} i_1, \quad \phi_2 = \frac{\mu_0 NA_g}{2(g+x)} i_2 \quad (14)$$

so that

$$\begin{aligned} \phi_a &= \frac{\mu_0 NA_g}{2} \left(\frac{i_1}{g-x} - \frac{i_2}{g+x} \right) \\ \phi_b &= \frac{\mu_0 NA_g}{2} \left(\frac{i_1}{g-x} + \frac{i_2}{g+x} \right). \end{aligned} \quad (15)$$

In general, observing these variables is more difficult than indicated here. Keith [8] provides a thorough examination of methods for estimating the fluxes ϕ_a and ϕ_b . Numerous references ([5,9,10]) describe methods for estimating the states of the linear portion of the plant.

Bias Controller. The dynamics of the bias plant are governed by (12). Following [6], define a tracking error coordinate

$$\tilde{\phi}_b \doteq \phi_b - \phi_{b0}$$

in which ϕ_{b0} is a constant target bias flux density. Refer to (2) and note that that order here is 1 so the sliding function is $s_b = \tilde{\phi}_b$. The sliding surface is where the bias error ϕ_b is zero.

To derive the controller itself, the objective is to ensure that the *reaching condition* (3) is satisfied everywhere that $s_b \neq 0$: $s_b \dot{s}_b \leq -\eta |s_b|$, or

$$-\frac{\phi_b - \phi_{b0}}{|\phi_b - \phi_{b0}|} \left[\kappa \left(\phi_b - \phi_a \frac{x}{g} \right) - N v_b \right] \leq -\eta \quad (16)$$

in which the positive constant η bounds the rate of convergence to the sliding surface.

Many functions v_b can satisfy this condition, but it is most electrically efficient ([11]) to switch the coil voltages between any of three possible states: $+V_{ps}$, 0, and $-V_{ps}$ where V_{ps} is a fixed supply voltage. So, select the control law

$$v_b = -V_{ps} \text{sign}(\tilde{\phi}_b). \quad (17)$$

Substitute (17) into (16) to get the required power supply voltage:

$$\frac{V_{ps}}{N} \geq \eta - \text{sign}(\tilde{\phi}_b) \kappa \left(\phi_b - \frac{x}{g} \phi_a \right). \quad (18)$$

To determine the bound for V_{ps} , the unknown terms on the right side of the equation must be bounded. If ϕ_{b0} is half the saturation flux of the actuator (which maximizes linearity ([12])) then it is expected that $|\phi_a| < \phi_{b0}$. It is common to assume that $|x| < 0.5g$ and to ensure this with auxiliary bearings. Finally, assume that $|\phi_b| < 2\phi_{b0}$. With these bounds, the required power supply voltage in (18) is

$$V_{ps} \geq N(\eta + 2.5\kappa\phi_{b0}). \quad (19)$$

Consider a simple example in which $A_g = 0.0001 \text{ m}^2$, $g = 0.0005 \text{ m}$, $R = 1.0\Omega$, $N = 150$ turns, and $\phi_{b0} = 6 \times 10^{-5} \text{ Wb}$. This gives

$$\begin{aligned} V_{ps} &\geq 150\eta + \frac{5 \times 0.0005}{4\pi \times 10^{-7} \times 150 \times 0.0001} 6 \times 10^{-5} \\ &= 150\eta + 7.96. \end{aligned}$$

For a given initial condition $\tilde{\phi}_b(0)$, the time to reach the sliding surface is bounded by ([6])

$$t_r \leq \frac{\max|\tilde{\phi}_b(0)|}{\eta}.$$

Choose a bias control time constant of 0.1 millisecond and assume the initial error is bounded by ϕ_{b0} :

$$0.0001 \geq \frac{6 \times 10^{-5}}{\eta} \Rightarrow \eta \geq 0.6.$$

Any power supply voltage over 98 volts will satisfy the reaching condition, even though the coupling between ϕ_a , x , and ϕ_b was ignored.

Suspension Controller. The suspension controller is intended to keep the mass at $x=0$. Accordingly, since the order of the differential equation is 3, the sliding surface is defined by

$$\begin{aligned} s_a(x;t) &= \left(\frac{d}{dt} + \lambda_1 \right) \left(\frac{d}{dt} + \lambda_2 \right) x \\ &= \ddot{x} + (\lambda_1 + \lambda_2)\dot{x} + \lambda_1\lambda_2x. \end{aligned}$$

Following [6], select $\lambda_1 = \lambda_2 = \lambda_a$ where, for the moment, λ_a is a free positive parameter.

The reaching condition is

$$\text{sign}(\ddot{x} + 2\lambda_a\dot{x} + \lambda_a^2x)(\ddot{x} + 2\lambda_a\dot{x} + \lambda_a^2x) \leq -\eta.$$

Referring to (11),

$$\ddot{x} = \frac{\phi_b}{mA_g\mu_0} \phi_a, \quad \dot{\phi}_a = \frac{\kappa\phi_b}{g}x - \kappa\phi_a + \frac{1}{N}v_a.$$

Differentiating the first of these and substituting the second produces

$$\ddot{x} = \frac{\phi_b}{mA_g\mu_0} \left(\frac{\kappa\phi_b}{g}x - \kappa\phi_a + \frac{1}{N}v_a \right).$$

With this, the reaching condition can be expanded as

$$\text{sign}(s) \left(f(\phi_a, \dot{x}, x) + \frac{\phi_b}{mA_g\mu_0 N} v_a \right) \leq -\eta$$

in which

$$s(\phi_a, \dot{x}, x) = \frac{\phi_b}{mA_g\mu_0} \phi_a + 2\lambda_a\dot{x} + \lambda_a^2x$$

and

$$f(\phi_a, \dot{x}, x) = \frac{\kappa \phi_b^2}{m \mu_0 A_g} \left[\frac{x}{g} + \left(\frac{2\lambda_a}{\kappa} - 1 \right) \frac{\phi_a}{\phi_b} \right] + \lambda_a^2 \dot{x}.$$

Consequently, the control law must satisfy

$$-\text{sign}(s) \frac{\phi_b}{mA_g \mu_0 N} v_a \geq \eta + \text{sign}(s) f(\phi_a, \dot{x}, x).$$

As previously, a switching law is desired for v_a , i.e.,

$$v_a = -V_{ps} \text{sign}(s)$$

which gives the power supply requirement

$$V_{ps} \geq \frac{mA_g \mu_0 N}{\phi_b} (\eta + \text{sign}(s) f(\phi_a, \dot{x}, x)).$$

A bound for $f(\phi_a, \dot{x}, x)$ permits a robust guarantee of this reaching condition. This function is a linear combination of the states x , $\dot{x}$, and ϕ_a , so it can be bounded through bounds on the states. As before, $|x| < 0.5g$. Also, $|\phi_a| < \phi_{b0}$. Finally, $\dot{x}$ can be bounded by finding the largest force that can be exerted on the mass and determining how large the velocity could get within the available clearance. Consider a sinusoidal force $F \sin \omega t$ and resulting response,

$$\begin{aligned} x(t) &= -\frac{1}{m\omega^2} F \sin \omega t \Rightarrow |x| \\ &= \frac{F}{m\omega^2} \leq 0.5g \Rightarrow F(\omega) \\ &\leq \min(0.5gm\omega^2, F_{\max}) \end{aligned}$$

where $F_{\max}$ is the load capacity of the actuator. This permits bounding the velocity of the mass:

$$\dot{x}(t) = -\frac{1}{m\omega} F \cos \omega t \Rightarrow |\dot{x}| \leq \frac{1}{m\omega} \min(0.5gm\omega^2, F_{\max}).$$

This bound increases with increasing ω until $0.5gm\omega^2 = F_{\max}$ and decreases thereafter. The worst case occurs where $0.5gm\omega^2 = F_{\max}$:

$$|\dot{x}| \leq \sqrt{\frac{0.5gF_{\max}}{m}}.$$

The load capacity, $F_{\max}$, can be determined from (6):

$$F = f_1 - f_2 = \frac{\phi_b \phi_a}{A_g \mu_0} \Rightarrow F_{\max} = \frac{\phi_{b0}^2}{A_g \mu_0}. \quad (20)$$

To continue the previous example, compute $F_{\max}$ as

$$F_{\max} = \frac{(6.0 \times 10^{-5})^2}{0.0001 \times 4\pi \times 10^{-7}} = 28.6 \text{ N}.$$

Further, assume that the mass is 0.5 kG so that the actuator can develop an acceleration of nearly 6 g/s; the velocity bound is

$$|\dot{x}| \leq \sqrt{\frac{0.5 \times 0.0005 \times 28.6}{0.5}} = 0.12 \text{ m/sec}.$$

Finally, f is bounded by

$$|f(\phi_a, \dot{x}, x)| \leq 20264[0.5 + (0.0057\lambda_a - 1)] + 0.12\lambda_a^2.$$

The required power supply voltage is now determined by the two controller parameters, η and λ_a :

$$V_{ps} \geq 0.00016\eta + 3.183[0.5 + (0.0057\lambda_a - 1)] + 0.000019\lambda_a^2.$$

In physical terms, this bound represents an assessment of the slew rate requirements of the system ([13]). The power supply voltage bounds the rate at which the current can be changed; the terms on the right side of the bound assess the slew requirements of the controller for attainable states.

Assume that the power supply voltage must be less than 100 volts so that this limitation requires $\eta < 625,0000$. The reaching time for the suspension should be about ten times that of the bias control, so η is set by

$$0.01 \geq \frac{\max(|s_a(0)|)}{\eta} \approx \frac{58}{\eta} \Rightarrow \eta \geq 5800.$$

Using the values of $\eta = 5800$ and $V_{ps} = 100$ gives

$$100.6635 \geq 0.018\lambda_a + 0.0000188\lambda_a^2$$

which is satisfied by $\lambda_a \leq 1884$.

The control law emerges as

$$v_a = -100 \text{sign}(954930\phi_a + 3768\dot{x} + 3.55 \times 10^6 x).$$

From (20), the actuator force, F , is a simple function of the control flux, ϕ_a :

$$F = \frac{\phi_b}{\mu_0 A_g} \phi_a = 477465 \phi_a \text{ (in the present example)}$$

which means that the control law may be rewritten as

$$v_a = -100 \text{sign}(F + 1884\dot{x} + 1.77 \times 10^6 x).$$

On the switching surface, $s_a = 0$, so that

$$F = -1884\dot{x} - 1.77 \times 10^6 x \quad (21)$$

which represents simple PD feedback control of the position and velocity of the mass: The objective of v_a is to force the actuator (F) to produce a PD control law on x . This control is indicated in block form in Fig. 1.

To get an idea of the performance of the PD controller, normalize the terms in (21) by their maximum expected amplitudes:

$$\frac{F}{\max|F|} = -7.9 \frac{\dot{x}}{\max|\dot{x}|} - 15.5 \frac{x}{\max|x|}.$$

The important point to note here is that the basic assumption of the reaching condition is *violated*: There is a wide range of permissible states for which it is impossible to satisfy $s = 0$. If the only physical constraint on the problem were that $|v_a| \leq V_{ps}$, then it would be possible to satisfy the reaching condition throughout the feasible state space: This is the specific objective of the bounds used in deriving the control law. However, there is an additional constraint not explicitly considered in the synthesis: due to magnetic saturation, the actuator can't develop more than about 28.6 Newtons of force.

The problem can be fixed by reducing λ_a from 1884 to about 200, leading to the normalized sliding condition

$$\frac{F}{\max|F|} = -0.839 \frac{\dot{x}}{\max|\dot{x}|} - 0.175 \frac{x}{\max|x|} \quad (22)$$

which can be satisfied essentially everywhere that the states are feasible.

With this modified control law, the differential equation on the sliding surface is

$$\ddot{x} + 400\dot{x} + 40000x = 0.$$

The eigenvalues are $(-200, -200)$ (as expected: $\lambda_1 = \lambda_2 = 200$) so the system is critically damped, with a time constant of 5 milliseconds.

Net Controller. The two controllers derived so far operate in nonphysical control coordinates. To describe the controller in physical coordinates, refer to (9) so that

$$\begin{Bmatrix} v_1 \\ v_2 \end{Bmatrix} = -50 \begin{bmatrix} 1 & 1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} s_1 \\ s_2 \end{Bmatrix}$$

$$s_1 = \text{sign}[477465(\phi_1 - \phi_2) + 200\dot{x} + 40000x]$$

$$s_2 = \text{sign}[\phi_1 + \phi_2 - \phi_{b0}].$$

This implies that the two coil voltages, v_1 and v_2 lie in the discrete set $\{-100, 0, 100\}$.

Load Rejection. Adding a load changes the arithmetic only slightly. The differential equation for the suspension problem becomes

$$\frac{d}{dt} \begin{Bmatrix} x \\ \dot{x} \\ \phi_a \end{Bmatrix} = \begin{bmatrix} 0 & 1 & 0 \\ 0 & 0 & \frac{\phi_b}{mA_g\mu_0} \\ \frac{\kappa\phi_b}{g} & 0 & -\kappa \end{bmatrix} \begin{Bmatrix} x \\ \dot{x} \\ \phi_a \end{Bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{1}{N} \end{bmatrix} v_a + \begin{bmatrix} 0 \\ \frac{1}{m} \\ 0 \end{bmatrix} f. \quad (23)$$

Accordingly, the sliding function is

$$\text{sign}(s) \left(p(\phi_a, \dot{x}, x, f) + \frac{\phi_b}{mA_g\mu_0 N} v_a \right) \leq -\eta$$

$$s(\phi_a, \dot{x}, x, f) = \frac{\phi_b}{mA_g\mu_0} \phi_a + \frac{1}{m} f + 2\lambda_a \dot{x} + \lambda_a^2 x$$

$$p(\cdot) = \frac{\kappa\phi_b^2}{m\mu_0 A_g} \left[\frac{x}{g} + \frac{2\lambda_a - \kappa}{\kappa} \frac{\phi_a}{\phi_b} \right] + \lambda_a^2 \dot{x} + \frac{1}{m} \dot{f} + \frac{2\lambda_a}{m} f$$

Select the control law $v_a = -V_{ps}\text{sign}(s)$ which assumes that the states and the force can be estimated. With a reasonable model for the force (i.e., static, or sinusoidal with known frequency), estimating the force is reasonably easy and routine. The requirements for the control law are

$$V_{ps} \geq \frac{mA_g\mu_0 N}{\phi_b} (\eta + \max|p|).$$

As before, the function p can be bounded through physical arguments with added constraints introduced by required bounds on f and its rate of change.

If f is assumed to be constant, an observer is constructed by embedding f as a state in the differential equation

$$\frac{d}{dt} \begin{Bmatrix} x \\ \dot{x} \\ \phi_a \\ f \end{Bmatrix} = \begin{bmatrix} 0 & 1 & 0 & 0 \\ 0 & 0 & \frac{\phi_b}{mA_g\mu_0} & \frac{1}{m} \\ \kappa \frac{\phi_b}{g} & 0 & -\kappa & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} x \\ \dot{x} \\ \phi_a \\ f \end{Bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{1}{N} \\ 0 \end{bmatrix} v_a. \quad (24)$$

It is easily confirmed that this system is observable if the variable x is measured, so a Luenberger observer can be constructed ([9]) which simply reconstructs f (or $f, x, \dot{x}$, and ϕ_a). Interestingly, on the sliding surface, the resulting control law replaces the term in f with a term in $\int_0^t x \, dt$: The controller is PID. This is true regardless of the selected observer gain.

Chatter. A central problem of sliding mode control is *chatter*: As the system approaches the sliding surface, the control is discontinuous in the states. For the control laws developed so far, the resulting switching will occur, theoretically, at an infinite rate. This is undesirable not out of concern for excitation of higher unmodeled dynamics, but because the efficiency of the power

amplifiers driving the coils drops off dramatically as the switching rate increases. Consequently, some mechanism is needed to prevent this rapid switching.

The solution is to introduce a boundary layer. Outside the boundary layer, the nominal control law is retained so that

$$\frac{d}{dt} \frac{1}{2} s^2 \leq -\eta |s| : |s| > \Phi. \quad (25)$$

If the control law satisfies this constraint then the surfaces of the boundary layer ($|s| = \Phi$) are *attractive* and the system will converge to the boundary layer whenever it finds itself outside of the boundary layer.

Once inside the boundary layer, the control law is modified in some manner so that the behavior for small s is somehow more desirable. The approach favored by Slotine and Li [6] and implemented in [1] is to use a linear interpolation of the discontinuous elements of u from one side of the boundary layer to the other. This solution is undesirable in magnetic bearing systems because it is not compatible with using a switching device to control the voltage applied to the coils.

A very effective alternative solution is to introduce an intentional perturbation signal, $\sigma(t)$, into the measured flux, ϕ_a . This signal is bounded by the amplitude a_σ :

$$|\sigma(t)| \leq a_\sigma. \quad (26)$$

To see how this works, consider the bias control. As before, the reaching condition is

$$s\dot{s} = -(\phi_b - \phi_{b0}) \left(\kappa \phi_b - \frac{v_b}{N} - \frac{\kappa \phi_a x}{g} \right) \leq -\eta |\phi_b - \phi_{b0}|. \quad (27)$$

Select the control law

$$v_b = -V_{ps}\text{sign}(\phi_b - \phi_{b0} + \sigma(t)) \quad (28)$$

so that (27) is

$$\begin{aligned} & -(\phi_b - \phi_{b0})\text{sign}[\phi_b - \phi_{b0} + \sigma] \frac{1}{N} V_{ps} \\ & \leq -\eta |\phi_b - \phi_{b0}| + \frac{\kappa(\phi_b - \phi_{b0})}{g} (g\phi_b - x\phi_a). \end{aligned} \quad (29)$$

Introduce the boundary layer $|\phi_b - \phi_{b0}| = a_\sigma$. Outside the boundary layer

$$|\phi_b - \phi_{b0}| > a_\sigma \Rightarrow |\sigma(t)| \Rightarrow \text{sign}[\phi_b - \phi_{b0} + \sigma] = \text{sign}[\phi_b - \phi_{b0}].$$

With this, (29) becomes

$$\frac{V_{ps}}{N} \geq \eta - \text{sign}(\phi_b - \phi_{b0} + \sigma) \kappa \phi_{b0} \left(\frac{\phi_b}{\phi_{b0}} - \frac{\phi_a}{\phi_{b0}} \frac{x}{g} \right)$$

and a bound on V_{ps} is $V_{ps} \geq N(\eta + 2.5\kappa\phi_{b0})$ as previously in (19).

The objective of adding the perturbation signal $\sigma(t)$ was to ensure a finite switching rate within the boundary layer. Since switches occur everywhere that $\phi_b - \phi_{b0} + \sigma = 0$, this is equivalent to ensuring that $\phi_b - \phi_{b0} + \sigma = 0$ only at discrete points within the boundary: It is undesirable to have $\phi_b - \phi_{b0}$ track $-\sigma$. One way to do this is to pick a triangle wave with frequency f_σ for $\sigma(t)$. The magnitude of the time derivative is everywhere constant: $|\dot{\sigma}| = 4f_\sigma a_\sigma$. Therefore, if the magnitude of the derivative of $\phi_b - \phi_{b0}$ is constrained to be less than $4f_\sigma a_\sigma$ within the boundary layer, then the flux error can equal the perturbation signal only at discrete points and the switching rate will be finite.

For the bias controller,

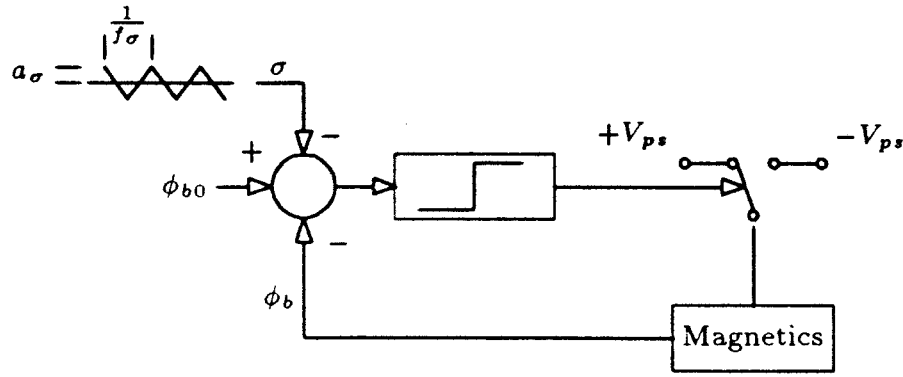


Fig. 2 Sliding mode control of the bias flux with chatter control: pulse width modulation

$$\left| \frac{d}{dt} (\phi_b - \phi_{b0}) \right|_{\max} = \left| \frac{\kappa (\phi_b g - \phi_a x)}{g} - \frac{1}{N} v_b + \dot{\phi}_{b0} \right|_{\max} \leq 2.5\kappa\phi_{b0} + \frac{1}{N} V_{ps} + |\dot{\phi}_{b0}|_{\max}$$

so that finite switching is achieved if

$$a_\sigma \geq \frac{1}{4f_\sigma} \left(\frac{V_{ps}}{N} + |\dot{\phi}_{b0}|_{\max} + 2.5\kappa\phi_{b0} \right).$$

The overall scheme of the bias controller with chatter control is indicated in Fig. 2. This is, of course, conventional two-state pulse width modulation applied to a flux control loop: such schemes are well established in the industry ([8,14]). The suspension control can be amended in a similar manner to include a perturbation signal and keep its switching rate finite.

Generalization to Flexible Rotors

To generalize the preceding discussion to flexible rotors, introduce the common notation for such a system

$$\begin{aligned} \frac{d}{dt} \begin{Bmatrix} \mathbf{x} \\ \dot{\mathbf{x}} \end{Bmatrix} &= \begin{bmatrix} 0 & I \\ A_1 & A_2 \end{bmatrix} \begin{Bmatrix} \mathbf{x} \\ \dot{\mathbf{x}} \end{Bmatrix} + \begin{bmatrix} 0 \\ B_f \end{bmatrix} \mathbf{f} + \begin{bmatrix} 0 \\ B \end{bmatrix} \mathbf{u} \\ \mathbf{y}_p &= C_1 \begin{Bmatrix} \mathbf{x} \\ \dot{\mathbf{x}} \end{Bmatrix} + D_{12} \mathbf{u} \\ \mathbf{y}_m &= [C_2 \ 0] \begin{Bmatrix} \mathbf{x} \\ \dot{\mathbf{x}} \end{Bmatrix} + D_{21} \mathbf{f}. \end{aligned} \quad (30)$$

First, assume that the performance measure, $y_p = \tilde{x}$, is a scalar, consistent with the preceding development so that $C_1 = \mathbf{c}^T$. Second, assume that $D_{12} = 0$. This is justified in that the sliding mode synthesis will take care of actuator capacity limitations explicitly, rather than needing to embed them in a performance function. Third, assume that any external loads may be subsumed into the problem in the manner of (24). Finally, assume that only one magnetic actuator acts on the system (maybe the rotor is pinned at one end). The first and last assumptions are introduced only to keep the discussion simple: It is reasonably straightforward to generalize the ensuing discussion to multiple input–multiple output systems as long as the number of inputs matches the number of outputs: the plant is *square*. With this, the control input matrix B becomes a vector with only one nonzero element, corresponding to the physical degree-of-freedom where the actuator acts. Denote this as

$$B = \frac{1}{m} \mathbf{b}; \mathbf{b} = \{0 \dots 010 \dots 0\}^T$$

in which m is the inverse scale of B , which has mass units.

Following the discussion of magnetic modeling and transformations introduced above, the overall dynamic model of rotor and suspension flux is

$$\begin{aligned} \frac{d}{dt} \begin{Bmatrix} \mathbf{x} \\ \dot{\mathbf{x}} \\ \mathbf{f} \\ h\phi_a \end{Bmatrix} &= \begin{bmatrix} 0 & I & 0 & 0 \\ A_1 & A_2 & B_f & \frac{1}{m} \mathbf{b} \\ 0 & 0 & A_f & 0 \\ n\mathbf{b}^T & 0 & 0 & -\kappa \end{bmatrix} \begin{Bmatrix} \mathbf{x} \\ \dot{\mathbf{x}} \\ \mathbf{f} \\ h\phi_a \end{Bmatrix} + \begin{bmatrix} 0 \\ 0 \\ 0 \\ \frac{h}{N} \end{bmatrix} v_a \\ \tilde{x} [\mathbf{c}^T \ 0 \ 0 \ 0] & \begin{Bmatrix} \mathbf{x} \\ \dot{\mathbf{x}} \\ \mathbf{f} \\ h\phi_a \end{Bmatrix} \end{aligned} \quad (31)$$

in which

$$h = \frac{\phi_{b0}}{A_g \mu_0} \quad \text{and} \quad \eta = \frac{2R}{N^2} h^2.$$

For the moment, rewrite (31) as

$$\begin{aligned} \frac{d}{dt} \xi &= A \xi + B v \\ \tilde{x} &= \mathbf{C}^T \xi. \end{aligned}$$

Construction of the sliding surface function $s(\tilde{x}; t)$ requires access to the performance scalar itself as well as its $n-1$ derivatives, where n is the dimension of the state vector ξ . It is easily demonstrated that a general formula for the k th derivative of $\tilde{x}$ is

$$\frac{d^k}{dt^k} \tilde{x} = \mathbf{C}^T \left(A^k \xi + \sum_{j=0}^{k-1} A^{k-1-j} B \frac{d^j}{dt^j} v \right).$$

The sliding function, then, is defined for single input–single output linear systems by

$$s(\tilde{x}; t) = \sum_{k=0}^{n-1} w_k \left(\mathbf{C}^T A^k \xi + \sum_{j=0}^{k-1} \mathbf{C}^T A^{k-1-j} B \frac{d^j}{dt^j} v \right), \quad (32)$$

Likewise,

$$\dot{s}(\tilde{x}; t) = \sum_{k=0}^{n-1} w_k \left(\mathbf{C}^T A^{k+1} \xi + \sum_{j=0}^k \mathbf{C}^T A^{k-j} B \frac{d^j}{dt^j} v \right). \quad (33)$$

When the parameters λ_i are chosen equal so that $\lambda_i = \lambda$, then the coefficients w_k are

$$w_k = \lambda^{n-1-k} \frac{(n-1)!}{k!(n-1-k)!}.$$

For more general choices of λ_i , the w_k can be computed in a direct manner as the k th term in the expansion of $\Pi(1+\lambda_i)$. In either case, $w_{n-1}=1$.

If the state system can be transformed so that A is lower Hessenberg ([15]). B is all zeros except the bottom element, and C^T is all zeros except the left element, then s is independent of v , and $\dot{s}$ contains v but not $\dot{v}$. This permits ready construction of the sliding mode controller because (32) and (33) will reduce to the forms

$$s(\tilde{x};t) = \alpha^T \tilde{\xi} \quad \text{and} \quad \dot{s}(\tilde{x};t) = \gamma^T \tilde{\xi} + \rho v.$$

Referring back to (31), since $\mathbf{b}$ has only one nonzero element, there is a permutation of the states $\mathbf{x}, \dot{\mathbf{x}}, \mathbf{f}$ so that this nonzero element occurs in the next to last row of the A matrix. Given this, it is only necessary to transform the upper left $(n-1, n-1)$ corner of the matrix in order to put the entire system in lower Hessenberg form. Consequently, the existing structure of the $\mathbf{B}$ vector is preserved with its only nonzero element at the bottom. Further, the last state remains ϕ_a , a fact which will be exploited shortly.

With a little additional normalization, it is possible to transform (31) into the form

$$\frac{d}{dt} \begin{Bmatrix} \zeta \\ h\phi_a \end{Bmatrix} = \begin{bmatrix} \begin{bmatrix} \mathbf{0} & I \end{bmatrix} & \mathbf{0} \\ \mathbf{p}^T & \frac{1}{m} \\ \mathbf{q}^T & -\kappa \end{bmatrix} \begin{Bmatrix} \zeta \\ h\phi_a \end{Bmatrix} + \begin{bmatrix} \mathbf{0} \\ h \\ \frac{h}{N} \end{bmatrix} v_a \quad (34)$$

$$\tilde{x} = \begin{bmatrix} \mathbf{g}^T & 0 \end{bmatrix} \begin{Bmatrix} \zeta \\ h\phi_a \end{Bmatrix}.$$

This accomplishes essentially all of the objectives except that the form of $\mathbf{C}$ is not right. Noting that the choice of output variable will change the zeros of the resulting closed-loop system but not the poles, the output is arbitrarily selected as

$$\tilde{x} = \begin{bmatrix} 1 & 0 & \dots & 0 & 0 \end{bmatrix} \begin{Bmatrix} \zeta \\ h\phi_a \end{Bmatrix}$$

where the physical significance of ζ is not obvious. The sliding function can be constructed as

$$s(\tilde{x};t) = \sum_{k=0}^{n-1} w_k \mathbf{C}^T A^k \tilde{\xi}.$$

The product $\mathbf{C}^T A^k \tilde{\xi}$ produces

$$\mathbf{C}^T A^k \tilde{\xi} = \begin{cases} \zeta_k = \tilde{x}^{(k)} & : k=0 \dots n-2 \\ \mathbf{p}\zeta + \frac{h}{m} \phi_a & : k=n-1. \end{cases}$$

On the sliding surface where $s=0$,

$$h\phi_a = -m \left(\mathbf{p}^T \zeta + \sum_{k=0}^{n-1} w_k \tilde{x}^{(k)} \right)$$

$$= -m \left(\sum_{k=0}^{n-1} (p_k + w_k) \tilde{x}^{(k)} \right).$$

Noting that the force applied to the rotor by the actuator is $\mathbf{F} = h\phi_a$, this shows that the control effort is some linear combination of the states of the rotor. That is, on the sliding surface, the dynamics of the actuator are canceled and the control solution is simply linear state feedback for the rotor, as would be obtained through any other synthesis technique. Further, if model uncertainty or sensor noise were considered, they would not affect the choice of control law along the sliding surface: They would only affect the extent of the boundary layer surrounding the sliding surface and the control used to reach this boundary layer.

As with the single mass example, the control law can be selected as

$$v = -V_{ps} \text{sign}(s).$$

With this, the requirements for V_{ps} are given by an overbounding condition on $\dot{s}$:

$$\dot{s}(\tilde{x};t) = \sum_{k=0}^{n-1} w_k (\mathbf{C}^T A^{k+1} \tilde{\xi} + \mathbf{C}^T A^k v)$$

$$= \mathbf{v}^T \tilde{\xi} + v_\phi \phi_a + v_v v$$

so that

$$V_{ps} \geq \frac{1}{v_v} (\eta + \max |\mathbf{v}^T \tilde{\xi} + v_\phi \phi_a|).$$

In general it will be difficult to estimate the required state bounds and an appropriate V_{ps} may need to be found by simulation.

Conclusions

Appropriate application of sliding mode control to magnetic bearing suspensions should reflect the physical limitations of the hardware. In particular, this means that the control variable must be recognized to be coil voltage rather than the commonly used current variables. Further, selection of the control function should recognize that the most efficient way to control the coil voltages is to switch them, rather than varying them continuously. With these restrictions in mind, sliding mode control solutions will generally look remarkably like more conventional linear synthesis solutions.

What advice does sliding mode offer to the designer of magnetic bearings? First, it suggests that the effects of modeling error can be covered by increasing the bearing amplifier voltage. Further, it says that estimation errors due either to modeling error or to sensor noise will result in an increased boundary layer. Unfortunately, increasing the amplifier voltage will also increase the boundary layer. How large can the boundary layer get before the system blows up? Theoretically, there is no limit. However, given that the model for the plant is generally only valid for a specific range of the state variables, a boundary layer which allows the states to exceed this range will produce unacceptable performance. For a plant of more than the slightest complexity, a thorough analytic evaluation of bounds on estimation error and the derivative of the sliding function will be prohibitively difficult.

Given that sliding mode analysis indicates the existence of a boundary layer (either due to plant model uncertainty or due to sensor noise), it is reasonable to ask how the system will behave when it is inside this boundary layer. If the boundary layer were obtained by linear interpolation of the control discontinuity across the layer, then it would be possible to make some claims about actual convergence to the sliding surface inside the boundary layer. However, since this linear interpolation isn't practical, there is little that can be said about this behavior. Certainly, this result has been noted experimentally where rotors often exhibit small limit cycles, which could be interpreted as rapid bouncing from one side of the boundary layer to the other. It is common in practice to refine ("tune") the control law to minimize or eliminate these limit cycles.

In the end, any linear feedback control which acts through a switching amplifier is an example of sliding mode control. The apparent benefit of the sliding mode approach lies primarily in its ability to directly assess the impact of modeling error and estimator noise on amplifier voltage requirements. It also provides some insight into the behavior and purpose of switching algorithms. As a synthesis method, it provides only a part of the solution because it doesn't provide a formal mechanism for selecting state gains: only for assessing them once selected. Selection of these gains should be done through other methods such as μ synthesis which

permit explicit tradeoffs between performance and robustness and directly incorporate the effect of sensor noise and model uncertainty. Once the control gains are synthesized, the selection of amplifier properties can be appropriately treated as a sliding mode problem, which will come down to assessment of required power supply voltage and the effect of switching frequency and dither amplitude on overall system performance.

References

- [1] Allaire, P., and Sinha, A., 1998, "Robust Sliding Mode Control of a Planar Rigid Rotor System on Magnetic Bearings," *Proceedings of the Sixth International Symposium on Magnetic Bearings*, Boston, Technomic Publishing, Co., Lancaster, PA, pp. 577–586.
- [2] Tain, H., Yamamoto, M., Kuwahara, O., and Hamabe, K., 1996, "Robust Control of High Speed Spindle-Magnetic Bearing System Using Sliding Mode Control and VSS Disturbance Observer," *Proceedings of the Fifth International Symposium on Magnetic Bearings*, Department of Electrical and Computer Engineering, Kanazawa University, Kanazawa, Japan, pp. 209–214.
- [3] Jiang, W., and Zhao, H., 1996, "Variable Structure Control for Active Magnetic Bearings," *Proceedings of the Fifth International Symposium on Magnetic Bearings*, Department of Electrical and Computer Engineering, Kanazawa University, Kanazawa, Japan, pp. 215–219.
- [4] Nonami, K., and Nishina, K., 1996, "Discrete Time Sliding Mode Control with Simple VSS Observer of Zero-Power Magnetic Bearing Systems," *Proceedings of the Fifth International Symposium on Magnetic Bearings*, Department of Electrical and Computer Engineering, Kanazawa University, Kanazawa, Japan, pp. 221–226.
- [5] Rundell, A., Drakunov, S., and DeCarlo, R., 1996, "A Sliding Mode Observer and Controller for Stabilization of Rotational Motion of a Vertical Shaft Magnetic Bearing," *IEEE Transactions on Control Systems Technology*, **4**, No. 5, pp. 598–608.
- [6] Slotine, J.-J., and Li, W., 1991, *Applied Nonlinear Control*, Prentice-Hall, Englewood Cliffs, NJ.
- [7] Lorrain, P., and Corson, D., 1970, *Electromagnetic Fields and Waves*, W. H. Freeman, San Francisco.
- [8] Keith, F., 1993, "Implicit Flux Feedback Control for Magnetic Bearings," Ph.D. dissertation, University of Virginia.
- [9] Sage, A., and White, C., 1977, *Optimum Systems Control*, Prentice-Hall, Englewood Cliffs, NJ.
- [10] Green, M., and Limebeer, D., 1995, *Linear Robust Control*, Prentice-Hall, Englewood Cliffs, NJ.
- [11] Maslen, E., Allaire, P., Noh, M., and Sortore, C., 1996, "Magnetic Bearing Design for Reduced Power Consumption," *ASME J. Tribol.*, **118**, pp. 839–846.
- [12] Maslen, E., Hermann, P., Scott, M., and Humphris, R., 1989, "Practical Limits to the Performance of Magnetic Bearings: Peak Force, Slew Rate, and Displacement Sensitivity," *ASME J. Tribol.*, **111**, pp. 331–336.
- [13] Weise, D., 1986, "Active Magnetic Bearings Provide Closed Loop Servo Control for Enhanced Dynamic Response," presented at the 27th IEEE Machine Tool Conference.
- [14] Bigelow, S., 1997, "All About Switching Power Supplies," *Electronics-Now*, **68**, pp. 40–47.
- [15] Horn, R., and Johnson, C., 1985, *Matrix Analysis*, Cambridge University Press, Cambridge, UK.

J. S. Rao
A. Pathak
A. Chawla

Department of Mechanical Engineering,
Indian Institute of Technology,
New Delhi 110016, India

Blade Life: A Comparison by Cumulative Damage Theories

A turbine blade is modeled as a rotating pretwisted beam and subjected to aerodynamic excitation from the flow path interference. The resonant stresses are determined using a modal analysis. With the help of steady steam bending and centrifugal stresses and the dynamic stresses, life estimates are made using the linear and nonlinear cumulative damage theories and a comparison of the results is presented. Based on these results, recommendations are made on the usefulness of different theories for practical application. [DOI: 10.1115/1.1384879]

Introduction

Turbine blades are prone to high cycle fatigue failures. These failures are essentially due to the resonance that occurs at the blade critical speeds during startup and shutdown conditions and in certain cases due to excursions of the machine near an operating speed which is close to one of resonances. The excitation is mainly from two sources: (1) due to the flow path interference between the stator and rotor blade rows at nozzle passing frequencies and (2) due to mechanical assembling errors at per rev harmonics.

Life estimation using stress based theories is a multifaceted technology that involves the following steps: (1) steady loads and mean stresses, (2) free vibrations, (3) excitation forces— aerodynamic and mechanical, (4) damping, (5) dynamic stresses, (6) failure surface, (7) load history, and (8) cumulative damage.

Basic theories of vibration of blades are now well established, (see Rao [1]). The flow path excitation due to interference between the stationary and moving blade rows is essentially an aerodynamic phenomenon. Three aspects are to be taken into account, namely, (i) potential interaction, (ii) vortex wake interaction, and (iii) viscous effects. Details of blade force estimation due to these effects is presented by Rao [2]. The mechanical excitation that occurs at per rev and harmonics is due to errors in manufacturing and mounting the stationary diaphragms and they cannot be modeled mathematically. They have to be determined from tests. Experimental results given by Levin [3] can be a source on this information in the absence of test data.

Damping is a crucial factor in the design of blades, as practically every blade is subjected to resonance during startup and shutdown conditions. In addition, one or two blade rows in a machine may be operating close to resonance conditions and the dynamic stresses should be controlled by appropriate damping treatment to achieve the desired life. A designer may adopt friction devices at disk-blade root junction, shrouds, lacing wires, etc., a typical example of additional damping devices is given by Sinha and Griffin [4]. Damping in a blade is very complex as it may arise out of material damping, friction damping, aerodynamic damping, and possibly impactive damping. A method of quantifying and measuring blade damping is proposed by Rao et al. [5], in which electromagnets are used to simulate the nozzles of a stage and the damping ratio is measured as a function of strain amplitude at a particular point on the blade and rotational speed for each mode of vibration. Similar excitation systems using water or

air jets are now being used in industry (see Benedict [6]). Nonlinear damping plays an important role in the estimation of blade stresses, (see [7,8]).

Ever since Wöhler [9] discussed the railway wagon axle failures in fatigue, this subject has gained importance in the design of machinery. The strength of the material under the action of completely reversed stress fatigue loads is determined from $S-N$ diagrams. These diagrams are not representative of an actual machine member and therefore the fatigue strength is to be modified to take into account the conditions prevailing for a specific machine subjected to fatigue (see [10]).

The influence of mean stress is described by various relations, linear or nonlinear, e.g., Goodman, Soderberg, Gerber, Marin, and Kececioglu, see [11]. Bagci [12] proposed a fourth-order relation, which is found to fit well with experimental data for steels. When the alternating stress is of variable amplitude, several cumulative damage theories are proposed, the most common of them being the linear one due to Palmgren and Miner [13,14]. The nonlinear theories proposed include Marco and Starkey [15], Corten and Dolan [16], Haibach [17], Marin [10], Henry [18], Gatts [19], and Manson [20]. In this paper, life estimates of a turbine blade are made and a comparison is given pointing out the relative merits of different approaches.

Blade Model

The rotating blade can be modeled as a pretwisted beam ([1]) or a plate ([21]), alternatively it can be modeled with solid elements ([1]). Here, the blade is modeled as a pretwisted rotating cantilever beam. Reissner's variational principle ([22]) is used so that the stresses are accurately determined. Since this model is well established ([7,8]) no details are presented here excepting the required geometric and material properties that are used in the analysis that follows. The blade considered is of uniform cross section without any pretwist.

Blade data:

Blade length = 11.1 cm

Disk radius = 18 cm

Distance of c.g. from center of chord in x -direction = 2.959 mm

Distance of c.g. from center of chord in y -direction = 15.371 mm

Distance of c.g. from center of flexure along x = 0.27488 mm

Distance of c.g. from center of flexure along y = -0.15782 mm

Area = 178.411 mm²

Second moment of area about yy -axis = 8008.62 mm⁴

Second moment of area about xx -axis = 3837.27 mm⁴

Product moment of area about xy -axis = 4632.18 mm⁴

Polar mass moment of inertia per unit length about center of flexure = 0.09491 kg-mm

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-287. Manuscript received by IGTI, Oct. 1998; final revision received by the ASME Headquarters, March 1999. Associate Editor: D. Wisler.

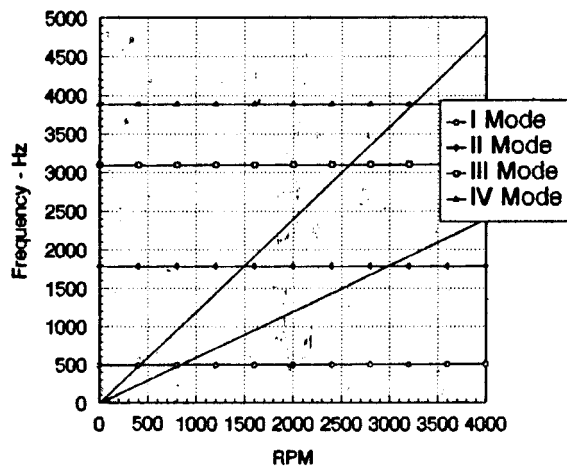


Fig. 1 Campbell diagram

Torsional rigidity = 281693224299.7 kg mm³/s²

The sectional properties in the above can be calculated directly from the blade profile using a computer program ([23])

Young's modulus = 200 GPa

Shear modulus = 72 GPa

Density = 0.0080000013 kg/cm³

The blade natural frequencies can be determined from a computer program, e.g., BLADE, given in [23]. The Campbell diagram for this blade is given in Fig. 1.

Stage Aerodynamic Data

The details of estimation of non steady forces in the stage are not given here, a reference to this can be taken from Rao and Rao [24]. The stator and rotor blades are modeled as parabolic cambered blades which will take into account all the effects of transverse as well as chord wise gusts and the effect of angle of attack. In the absence of viscous wake information, its effect which is second order in nature is neglected here. Only the potential interaction and the vortex wake interaction are considered. A computer program code named DELHI is used to determine the nonsteady forces and moments of the rotor blades. This program is integrated in the program BLADE, so that the stresses are determined simultaneously ([23]). The data required are

Semi-chord of rotor blade = 15 mm

Semi-chord of stator blade = 15 mm

Stagger angle of rotor blade = 45 deg

Stagger angle of stator blade = 45 deg

Distance of midpoint on the camber line from the center of chord of rotor blade = 3.0 mm

Distance... of stator blade = 3.0 mm

Distance between the trailing edge of the stator and leading edge of the rotor parallel to the turbine axis = 3.0 mm

Angle of attack on rotor blade = 6 deg

Angle of attack on stator blade = 6 deg

Steady rotor blade load normal to angle of attack passing through the chord center = 50.0 N/mm

Number of stator blades = 36

Blade Damping Model

Nonlinear damping model is assumed in this analysis for all the modes. The damping ratio is a function of strain amplitude at the root for different speeds in the range of 0 to 4000 rpm in each mode. At low speeds, Coulomb damping prevails due to friction between the mating surfaces at the blade root junction with the disk. With increasing speed, the blade root gets tighter in its groove because of centrifugal forces and has only material damp-

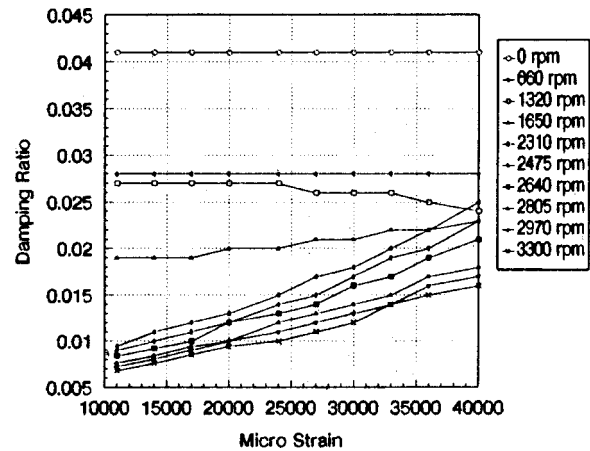


Fig. 2 Nonlinear damping—one mode

ing. Figure 2 shows the variation of damping ratio as a function of the strain amplitude for different speeds of rotation.

Stress Results

The critical speeds from the Campbell diagram, steady steam bending, centrifugal, and dynamic stresses and the equivalent damping ratio obtained at these speeds are given in Table 1.

The equivalent damping obtained at high speeds is low since the root gets tightened at the junction and friction damping disappears. The damping is also higher at 823.78 rpm compared with 411.48 rpm because of higher stress. The above results will be used in making life estimates by different methods which are briefly outlined below.

Failure Surface

We first define the failure surface before applying the cumulative damage theories for life estimation. The *S-N* diagram for the material is defined by

$$\log 0.9S_u = -A \log 10^3 + B \quad 10^3 < N < 10^6$$

$$\log S_e = -A \log 10^6 + B \quad N > 10^6$$

$$A = \frac{1}{3} \log \frac{0.9S_u}{S_e} \quad B = \log \frac{(0.9S_u)^2}{S_e}$$

where ultimate Tensile Strength $S_u = 900$ MPa and endurance Limit $S_e = 450$ MPa.

In order to obtain the *S-N* diagram for the blade, the fatigue strength modifying factor is adopted from [11] as follows. The surface factor k_a takes into account the actual surface finish. For forged steel with ultimate tensile strength equal to 900 MPa, $k_a = 0.35$.

The size effect factor suggested by Juvinall (see [11]) $k_b = 1$ for $d < 10$ mm where d is the effective thickness.

Table 1 Results from Program BLADE

Mode/N PF no.	Critical speed, rpm	Steady stress, MPa	Centri stress, MPa	Dyn stress, MPa	Damping ratio
I/I	823.78	5.704	1.189	55.41	0.00182
I/II	411.48	5.709	0.297	32.17	0.00126
II/I	2,975.37	5.629	15.52	49.27	0.00051
II/II	1,486.38	5.689	3.872	18.43	0.00053
III/II	2,582.77	5.648	11.69	78.17	0.00078
IV/II	3,236.14	5.615	18.35	37.83	0.00015

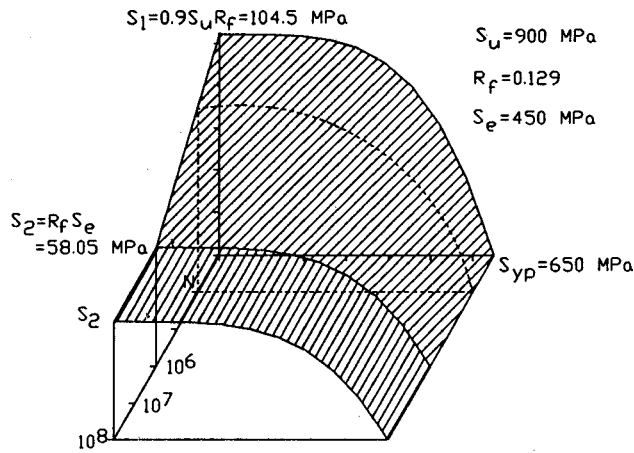


Fig. 3 Fatigue failure surface

The reliability factor k_c corresponding to eight percent standard deviation of the endurance limit for ninety five percent reliability is 0.868.

For most steels the temperature factor $k_t = 1.0$ for $T \leq 450^\circ\text{C}$.

The stress concentration factor $k_e = 1/(1 + q(K - 1))$ where q is the notch sensitivity factor and K is a factor dependent on geometric values of the member. A sharp notch of 0.2 cm radius at the root of the blade gives $q = 0.85$ and $K = 2.6$ and $k_e = 0.424$.

The factors k_a to k_e defined above are significant parameters that effect the fatigue strength of the blade. There are several other factors that influence the fatigue strength, e.g., chemical composition of the material, manufacturing process (rolling, drawing, etc.), heat treatment, fretting-corrosion, corrosion environment, stress state, residual stresses, and relaxation times. These effects are taken into account by a miscellaneous factor k_f and here it is taken to be unity. The fatigue strength modifying factor $R_f = k_a k_b k_c k_d k_e k_f$ works out to be 0.129.

The effect of mean stress is taken into account by the fourth order relation of Bagci [12]

$$\sigma_a = \sigma_N \left[1 - \left(\frac{\sigma_m}{\sigma_{yp}} \right)^4 \right]$$

where the yield strength $\sigma_{yp} = 650$ MPa.

The $S-N$ diagram is now drawn perpendicular to the plane of mean stress—alternating stress to obtain the fatigue failure surface as shown in Fig. 3. $S'_e = 0.9S_u$ represents the fatigue strength of the standard rotating beam specimen at 1000 cycles life. As the number of cycles for life is increased, the material fatigue strength

falls to one million cycles of life, where the fatigue strength of the material is taken as the endurance limit S_e . The fatigue strength modifying factor R_f is then used to obtain the blade fatigue strength $S_1 = 0.9R_f S_u$ for 10^3 cycles.

The endurance limit is also modified with the factor R_f to give $S_2 = R_f S_e$ and the new $S-N$ law is as shown in Fig. 3. The failure curve according to Bagci's law is drawn at every ordinate of the blade fatigue strength in the region of $10^3 - 10^6$ cycles and beyond to give the failure surface.

Cumulative Damage

The different cumulative damage theories are briefly discussed here which are then used to make life estimates.

1 Palmgren-Miner Linear Damage Theory ([13,14]). This is the most commonly used theory to predict the fatigue life of a blade subjected to variable stress amplitude. The dynamic stress around a critical speed can be discretized into a convenient number of steps, 20 on either side of the peak stress. Operation at a stress level S_i gives a life of N_i cycles. If the blade is subjected to n_i cycles, it suffers a damage fraction $D_i = n_i/N_i$. Failure is then predicted to take place when

$$\sum \frac{n_i}{N_i} \geq 1.$$

It asserts that the damage fraction at any stress level is linearly proportional to the ratio of number of cycles of operation to the total number of cycles that would produce failure at that stress level. As the blade is subjected to a mean stress, the $S-N$ plane is shifted to the location of the applied mean stress level on the fatigue failure surface.

One of the serious drawbacks of this theory is that it does not recognize the order of application of various stress levels and damage is assumed to accumulate at the same rate at a given stress level without a consideration to the past history. Experimental evidence shows that fatigue damage accumulates non linearly, depending on the alternating stress level.

2 Marco-Starkey Damage Theory ([15]). Marco and Starkey proposed that the damage for each level of reversed sinusoidal stress amplitude is given by

$$D = \left(\frac{n}{N} \right)^m$$

where the exponent m depends on the amplitude of the alternating stress. A specimen is deemed to have failed when D reaches a value c irrespective of the sequence in which the stresses are applied. Mathematically, failure will occur when $\sum n/N$ given below reaches a critical value c .

$$\sum \frac{n}{N} = c \int_0^1 \frac{1 + \frac{N_1}{N_2} + \frac{N_1}{N_3} + \dots + \frac{N_1}{N_i}}{1 + \frac{N_1}{N_2} r_2 D \left(\frac{r_2 - 1}{r_2} \right) + \frac{N_1}{N_3} r_3 D \left(\frac{r_3 - 1}{r_3} \right) + \dots + \frac{N_1}{N_i} r_i D \left(\frac{r_i - 1}{r_i} \right)} dD$$

where N_i are the cycles of completely reversed stresses S_i to produce failure with i denoting the order of application of stress levels; D is the damage; r_i is the ratio of exponents m_i/m_1 representing the two stress levels S_i & S_1 ; and m_i is the exponent in the damage equation associated with i th stress level.

The evaluation of the integral in the above equation is not easy, though the main difficulty in using this theory is the necessity of

generating a lot of experimental data m_i . Qualitatively, the procedure can be illustrated with the help of experimental data for two different stress levels σ_1 and σ_2 , with $\sigma_1 > \sigma_2$ as given in Fig. 4. Consider first the specimen to be loaded for $n/N = 0.5$ with a stress level σ_1 (O-A in Fig. 4) and then with a lower stress level of σ_2 (B-C in Fig. 4) until the damage takes place at $D = 1$. In this case

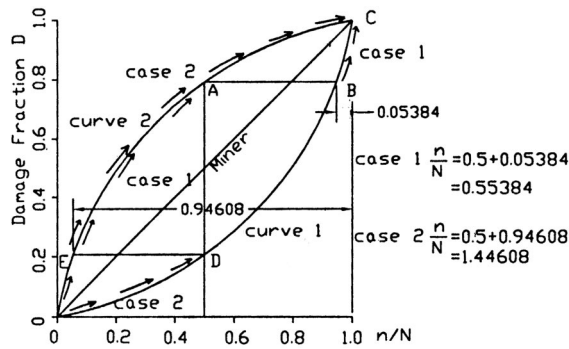


Fig. 4 Fatigue damage for two different cases

$$\sum \frac{n}{N} = \left(\frac{n}{N} \right)_{\sigma_1} + \left(\frac{n}{N} \right)_{\sigma_2} = 0.5 + 0.05384 = 0.55384.$$

Alternatively, we could load the specimen for $n/N = 0.5$ first with a lower stress level σ_2 (O-D in Fig. 4) and then with a higher stress level of σ_1 (E-C in Fig. 4) until the damage takes place at $D = 1$. Then

$$\sum \frac{n}{N} = \left(\frac{n}{N} \right)_{\sigma_2} + \left(\frac{n}{N} \right)_{\sigma_1} = 0.5 + 0.94608 = 1.44608.$$

When the higher stress level is applied first, the specimen seems to take only a few more cycles of lower stress level before damage takes place. On the contrary, when a lower level of stress is applied first, the specimen could take considerably lot more cycles at the higher stress level before damage takes place. Experimental evidence supports such a situation—in the first case the specimen is already fatigued at a high stress level and when the stress level is changed to a lower level, damage is reached quickly; in the second case the specimen is initially stressed at a lower level and relatively less effected in fatigue and therefore it can take a higher number of cycles when the stress level is altered to a higher value.

One of the problems with the application of Marco-Starkey's theory is the lack of material data for the exponent m for different values of stress levels. Considerable number of tests are required before this theory can be applied. However, a study of the sequential load test data for SAE 4340 steel obtained by Marco-Starkey given in Table 2 suggests a simple procedure to account for the non linearity in the damage.

In Table 2 when the stress level is increased from 550 MPa to 825 MPa in larger number of steps, N , i.e., in a continuously increasing manner, the average value of $\sum(n_i/N_i)$ decreased from

(1.14–1.17) to 1.05. Similarly, when the stress level is decreased from 825 to 550 MPa in larger number of steps, the average value of $\sum(n_i/N_i)$ decreased from (0.76–0.81) to 0.6.

Therefore, we may conclude that the effect of varying stress level is almost negligible when the stress level is gradually increased and that Palmgren-Miner's linear damage rule may be applied while the stress levels are continuously increasing as in the case of a blade approaching peak resonant amplitude.

For the period when the stress levels are decreasing from the peak amplitude, the average value of $\sum(n_i/N_i)$ in this period may be multiplied by a factor $f = 1/0.6 = 1\frac{2}{3}$, while summing up $\sum(n_i/N_i)$ in the entire period. This implies that the damage is higher when the stress levels are decreasing. While applying the above simplification, it should be ensured that the stress response around resonance may be discretized to at least 20 steps on either side of resonance.

3 Corten-Dolan Damage Theory ([16]). This theory recognizes the need for postulating the initiation of permanent damage, somewhat similar to fracture mechanics theory, wherein fracture should initiate first before propagating. Sub microscopic level voids are assumed to form for initiation of damage, these voids are termed as damage nuclei. The number of damage nuclei that form throughout the member is considered to increase as the stress level is increased. As the number of cycles of the applied stress amplitude increases, the rate at which damage takes place also increases. This rate of damage per cycle also increases with the magnitude of the stress amplitude. It is also assumed that the total damage constituting failure is a constant for all possible stress histories that could be applied to a member. Further, it is assumed that damage will continue to be propagated at stress levels that are lower than the minimum stress required to initiate damage. This assumption is similar to the fracture mechanics approach, wherein, fracture propagation takes place, once it is initiated, at stress levels that are much lower than the minimum stress required to initiate a crack. In other words, some aspects of fracture mechanics are incorporated in this theory, bringing it closer to the advanced concepts of fatigue failures, (see [25]). Corten and Dolan assumed that the damage per nucleus D' is given by $D' = rN^a$ where r is the coefficient of damage propagation rate which is a function of stress level, N is the number of cycles of stressing and a is the damage propagation exponent, which is also a function of stress level. If the total number of fatigue damage nuclei is m , then the total fatigue damage is given by $D = mrN^a$. For two stress levels. Corten and Dolan found an experimental relation between the stress dependent ratio $R = r_2/r_1$, the damage exponent a and the stress ratio through a material property d given by $R^{1/a} = (S_3/S_1)^d$. Using this property, they have shown that failure can be expected when

$$\frac{n_1}{N_1} + \frac{n_2}{N_1} \left(\frac{S_2}{S_1} \right)^d + \frac{n_3}{N_1} \left(\frac{S_3}{S_1} \right)^d + \dots + \frac{n_i}{N_1} \left(\frac{S_i}{S_1} \right)^d = 1$$

where N_1 is the number of cycles to failure at the highest stress amplitude S_1 , and n_i are the total number of cycles imposed at each stress level S_i . For steels the exponent d is found to range between 6.2 and 6.9 with a mean value 6.57. Corten and Dolan suggest a modification of S-N diagram to be defined by one line in the entire region given by

$$N = N_e \left(\frac{S}{S_e} \right)^{-k}$$

where N_e is the number of stress cycles endured at the fatigue limit S_e and k lies between 0.8 and 0.9 (taken as 0.85 in the present case) which yields a stress cycle with slope more than the classical S-N diagram, but continued into the range of fatigue strength. Haibach [17] suggests the usage of a double line S-N

Table 2 Sequential load test results for SAE 4340 Steel [15]

First stress level	Last stress level	No. of steps	Avg. $\sum \left(\frac{n_i}{N_i} \right)$	Max. $\sum \left(\frac{n_i}{N_i} \right)$	Min. $\sum \left(\frac{n_i}{N_i} \right)$	N
550	825	2	1.14	1.51	0.71	10
550	825	4	1.13	1.43	0.7	10
550	825	6	1.14	1.4	1	6
550	825	10	1.17	1.66	0.77	13
550	825	15	1.1	1.27	1	3
550	825	20	1.05	1.4	0.9	6
825	550	2	0.73	0.84	0.64	6
825	550	4	0.74	0.84	0.61	6
825	550	6	0.76	0.86	0.65	6
825	550	10	0.81	0.84	0.78	6
825	550	15	0.73	0.79	0.6	9
825	550	20	0.6	0.79	0.57	13

diagram, the first one similar to the classical $S-N$ line and the second one that continues into the region of fatigue strength with a different slope.

$$N = N_e \left(\frac{S}{S_e} \right)^{-k^*} \quad S \geq S_e$$

$$N = N_e \left(\frac{S}{S_e} \right)^{-(2k^*-1)} \quad S < S_e$$

The value of k^* in the above is taken as Corten and Dolan value $k = 0.85$.

4 Marin's Theory ([10]). Marin assumed that the equivalent number of cycles at a reference level, say S_1 , that would produce the same damage as n_i cycles of operation at S_i level of stress can be expressed as

$$n_{ie} = n_i \left(\frac{S_i}{S_1} \right)^y$$

where y is an exponent that can be determined experimentally. The damage ratio corresponding to each operation is written as $R_i = n_m / N_1$ so that the condition of failure is

$$R_1 + R_2 + R_3 + \dots + R_i = 1$$

i.e.,

$$\frac{n_1}{N_1} + \frac{n_2}{N_1} \left(\frac{S_2}{S_1} \right)^y + \frac{n_3}{N_1} \left(\frac{S_3}{S_1} \right)^y + \dots + \frac{n_i}{N_1} \left(\frac{S_i}{S_1} \right)^y = 1.$$

The above relation is same as that obtained by Corten-Dolan with Marin's exponent y equal to Corten-Dolan exponent d . Marin suggests that the $S-N$ relation can be expressed by

$$S^x N = k$$

i.e.,

$$S_i^x N_i = S_1^x N_1.$$

The failure criterion then becomes

$$\frac{n_1}{N_1} + \frac{n_2}{N_2} \left(\frac{S_2}{S_1} \right)^q + \frac{n_3}{N_3} \left(\frac{S_3}{S_1} \right)^q + \dots + \frac{n_i}{N_i} \left(\frac{S_i}{S_1} \right)^q = 1$$

where $q = y - x$. It is interesting to note that Marin's theory becomes Palmgren-Miner linear hypothesis, when q becomes zero, i.e., $y = x$ and Corten-Dolan theory, if $q = d$.

While applying this theory, y is taken as equal to d of Corten-Dolan and x in the above relation for the given $S-N$ diagram is 11.75, therefore $q = -5.18$.

5 Henry Cumulative Damage Theory. Henry [18] accounted for the fact that the fatigue limit gets reduced when a specimen suffers damage, i.e., the $S-N$ curve gets shifted as damage accumulates. The relation between the damage D and the fatigue limit is expressed as $D = S_e - S_{ed} / S_e$, where S_e is the original fatigue limit of the virgin material and S_{ed} is the fatigue limit after the damage reduced from S_e .

The $S-N$ curve itself, is assumed to be represented by an equilateral hyperbola referred to the stress axis S , and a line passing through S_e parallel to the N -axis as the asymptotes. Thus, $N = k_0 / S - S_e$ where k_0 is material constant. Henry assumed that no damage takes place by operation at cyclic stress levels below the fatigue limit and that after damage, the $S-N$ curve could be represented by a similar hyperbola, i.e., $N_r = k / S - S_{ed}$ where N_r is the number of remaining cycles to failure at stress level S and k is a material constant.

Based on some experimental data, it is further assumed that the material, constant k and the fatigue limits S_e satisfy the relation $k/k_0 = S_{ed}/S_e$. Now, if n cycles of stress amplitude S are applied to a specimen, the remaining life is given by $N_r = N - n = k/S - S_{ed}$. It can then be shown that

$$1 - \frac{n}{N} = \frac{S_{ed}}{S_e} \left(\frac{S - S_e}{S - S_{ed}} \right)$$

and

$$S_{ed} = \frac{S \left(1 - \frac{n}{N} \right)}{\left(\frac{S - S_e}{S_e} \right) + \left(1 - \frac{n}{N} \right)}$$

The above relation gives the current value of the fatigue damage S_{ed} after n cycles of stress amplitude S have been applied to a specimen whose original virgin fatigue limit is S_e . It can be expressed in terms of the damage equation to give

$$D = \frac{S_e - S_{ed}}{S_e} = 1 - \frac{S_{ed}}{S_e}$$

$$= \frac{\frac{n}{N}}{1 + \left(\frac{S_e}{S - S_e} \right) \left(1 - \frac{n}{N} \right)}.$$

The above theory due to Henry can be extended to a sequence of different alternating stress amplitudes in the order of applied stress levels. Each time of application of a stress level, the current fatigue limit S_{ed} should be updated. Thus, a sequence of the updated fatigue limits will be obtained, say $S_{ed1}, S_{ed2}, \dots$ after the application of n_1 cycles of stress level S_1 , then n_2 cycles of stress level $S_2, \dots$ and so on. Henry's theory therefore allows us to determine the diminishing fatigue limit as damage accumulates.

6 Gatts Cumulative Damage Theory. Gatts damage theory [19] is based on the dependence of fatigue strength and fatigue limits on the number of cycles of stress and that this change is proportional to a damage function $D(S)$, i.e., $dS_1/dn = -k_1 D(S)$ with S_i as the instantaneous value of the strength, n the number of stress cycles applied k_1 a constant of proportionality. The damage is expressed as a function of stress level $D(S) = (S - S_e)^p$ with p a material constant. To determine k_1 and p , Gatts related his damage function with the strain energy associated with strains and stresses exceeding the level of fatigue limit. The fatigue limit is then obtained as

$$kn = \frac{1}{S - S_{e0}} - \frac{1}{S - S_e}$$

where S_{e0} is the fatigue limit when $n = 0$, S_e is the fatigue limit, a function of stress history, S is the amplitude of applied cyclic stress, and n is the number of cycles of stress applied.

It was further assumed that the fatigue limit can be expressed in terms of instantaneous strength as $S_e = CS_i$ where C is a material constant and upon applying boundary conditions the following $S-N$ curve equation is obtained.

$$kN = \frac{1}{S - S_{e0}} - \frac{1}{S(1 - C)}$$

In nondimensional form the above relation is

$$KN = \frac{1}{\gamma - 1} - \frac{1}{\gamma(1 - C)}$$

where $\gamma = S/S_{e0}$, $\beta = n/N$, $\gamma_e = S_e/S_{e0}$ and $K = kS_e$.

The material constant C is usually found to be 0.5 for most of the steels. From the above Gatts has shown that

$$\gamma_s = \gamma \left\{ 1 - \frac{1}{\frac{\beta}{1 - C} + \frac{\gamma}{\gamma - 1} (1 - \beta)} \right\}.$$

For $C = 0$, the above reduces to

$$\gamma_e = \frac{\gamma(1-\beta)}{\gamma-\beta}$$

which is same as Henry's damage expression.

7 Manson Double Linear Damage Theory. In this theory ([20]) the concepts of fracture mechanics are implied by recognizing the two aspects, crack initiation and propagation during the damage suffered by a specimen. The crack initiation period is denoted N' and the crack propagation period is defined by the number of cycles for failure after the crack has been initiated and denoted by $N_p = PN_f^p$, then

$$N' = N_f - N_p = N_f - PN_f^p$$

where N_f is the total number of cycles for failure including crack initiation, P is propagation coefficient and p is propagation exponent.

Manson found $p=0.6$ and $P=14$ from several experimental results, thus

$$N_p = 14N_f^{0.6} \quad N' = N_f - 14N_f^{0.6}.$$

The above results are found to be fairly accurate when $N_f > 730$ cycles. For failures occurring with less than 730 cycles, it is found that the crack initiation takes place in the first cycle of loading, because of the high stress level that is involved with very short lives. Miner's rule is then adopted by Manson for the crack initiation and propagation phases separately. Fatigue nuclei of critical size are initiated when

$$\sum_{i=1}^m \frac{n_i}{N'_i} = 1$$

This equation is applied until, a crack is initiated. Fatigue cracks are then propagated to failure after cracks of critical size have been initiated and when

$$\sum_{j=1}^q \frac{n_j}{N_{pj}} = 1$$

In both of the above phases, n is the number of cycles applied at the i th or j th stress level. The above equation then determines the life for crack propagation period. Thus, Manson's damage rule is a double linear damage rule.

Life Estimates

The stress response at a critical speed is determined from the dynamic magnifier

Table 3 Life estimates from different theories

rpm	Life in Cycles $\times 10^{-6}$						
	1*	2	3	4	5	6	7
823	16.1	12.1	27.02* 14.07@ 13.589#	27	-	-	16.2
2975	54.4	40.8	88.57* 15.20@ 14.46#	90	-	-	54
2582	0.21	0.16	0.335* 8.592@ 8.592#	0.33	0.02	0.21	0.2
3236	2474	1835	4178.4* 39.25@ 36.8#	4174	-	-	1627

*classical S-N,

@Corten-Dolan S-N,

#Haibach

*numbers correspond to theories in this section

$$H(\omega) = \frac{1}{\sqrt{\left[1 - \left(\frac{\omega}{p}\right)^2\right]^2 + \left[2\xi\left(\frac{\omega}{p}\right)\right]^2}}$$

where ξ are given in Table 1. The life results converged when 36 steps are used for the stress response and an acceleration of 1000 rpm/min is assumed to determine the life and the results obtained are compared in Table 3.

At the critical speeds 411.68 and 1486.38 rpm, the mean stress and dynamic stress are low, therefore no damage occurs. It may also be noted that Henry and Gatts theories can be applied only when the alternating stress is above the endurance limit.

Conclusions

From the results given in Table 3, the following conclusions can be drawn.

- Macro-Starkey theory consistently predicted lower life in comparison to the linear theory. This theory recognizes the influence of variable stress amplitude with a consideration to the past history. In the present application this effect is accounted for the decreasing stress amplitudes, where it is found to be predominant. During the period of continuously increasing amplitude this effect is found to be less effective and if there is an effect it would have reduced the life further marginally. This theory is most promising and can be easily adopted with additional tests to determine the actual factor of a specific material in place of $1 \frac{2}{3}$ derived from Table 2 of Marco-Starkey for SAE 4340 steels.

- Marin's theory consistently predicted a higher life over the linear theory when y is assumed to be equal to Corten-Dolan value of d . In order to make use of this theory, it is necessary to obtain considerable experimental data on the exponent y .

- Manson's theory which recognizes the crack initiation and propagation periods, estimated life similar to Palmgren-Miner values. At 823.78 rpm the life is somewhat on higher side but the difference is negligible. At 2582.77 and 3236.14 rpms the life predicted is almost the same as the linear theory but on the lower side. At 2975.37 rpm the life is lower than the linear theory. Though this theory separates the crack initiation and propagation lives, it is essentially the same as Palmgren-Miner linear theory, since the conventional S-N diagram is generated by a complete failure of the specimen which includes both the aspects of crack initiation and propagation.

- Henry's theory predicted the lowest life at 2582.77 rpm, almost a tenth of the linear theory. Here the dynamic stress is the highest and above the endurance limit. This theory recognizes that the endurance limit gets reduced as the blade suffers damage and it seems to have a significant effect on the result. The modification suggested by Gatts is perhaps necessary since the life estimated is too conservative.

- Gatts' theory predicted lower life, of similar order of Palmgren-Miner theory, but higher than Marco-Starkey and Manson. Gatts' theory is same as Henry's theory if the factor C is made zero. Therefore, it appears that a lot of experimentation is needed to evaluate the material property C accurately for each application.

- Corten-Dolan relation with the conventional S-N law gave consistently higher life almost double that of Palmgren-Miner.

- When Corten-Dolan relation is used in conjunction with the modified S-N law according to Corten-Dolan-single line or Haibach double line, the difference between the results is somewhat negligible when compared with Palmgren-Miner at 823.78 rpm, where the alternating stress 55.41 MPa is close to the endurance limit. The life predicted by both Corten-Dolan and Haibach S-N laws is considerably lower at 2975.37 rpm where the alternating stress is below the endurance limit when compared with the linear theory. With the alternating stress well below the endurance limit at 3236.14 rpm, the predicted life is way below that of Palmgren-

Miner. Alternatively, when the alternating stress is clearly more than the endurance limit at 2582.77 rpm, the life predicted is very high almost 40 times that of Palmgren-Miner. It appears that combining different *S-N* laws with Corten-Dolan theory does not yield useful information except when the alternating stress is around the endurance limit.

References

- [1] Rao, J. S., 1991, *Turbomachine Blade Vibration*, John Wiley and Sons, New York.
- [2] Rao, J. S., 1994, *Turbomachine Unsteady Aerodynamics*, Wiley Eastern, New Delhi.
- [3] Levin, A., 1953, *Moving Blades and Disks of Steam Turbines*, The State Power Engineering Publishers, Moscow (in Russian).
- [4] Sinha, A., and Griffin, J., 1984, "Effects of Static Friction on the Forced Response of Frictionally Damped Turbine Blades," *ASME J. Eng. Gas Turbines Power*, **106**, p. 65.
- [5] Rao, J. S., Gupta, K., and Vyas, N. S., 1986, "Blade Damping Measurement in A Spin Rig With Nozzle Passing Excitation Simulated by Electromagnets," *Shock and Vibration Bulletin*, **56**, Part 2, p. 109.
- [6] Benedict, B., 1997, "Blade Vibration Measurements, Forced Vibration and Blade Life Estimation," Continuing Education Course, ASME, Orlando, June.
- [7] Vyas, N. S., and Rao, J. S., 1994, "Fatigue Life Estimation Procedure for A Turbine Blade Under Transient Loads," *ASME J. Eng. Gas Turbines Power*, **116**, p. 198.
- [8] Rao, J. S., and Vyas, N. S., 1996, "Determination of Blade Stresses Under Constant Speed and Transient Conditions With Nonlinear Damping," *ASME J. Eng. Gas Turbines Power*, **118**, No. 2, p. 424.
- [9] Wöhler, A., 1860, "Versuche Über die Festigkeit der Eisenbahn-wagen Achsen," *Zeitschrift für Bauwesen*.
- [10] Marin, J., 1962, *Mechanical Behavior of Materials*, Prentice-Hall, Englewood Cliffs, NJ.
- [11] Shigley, J. E., and Mitchell, L. D., 1983, *Mechanical Engineering Design*, McGraw-Hill, New York.
- [12] Bagci, C., 1981, "Fatigue Design of Machine Elements Using Bagci Line Defining the Fatigue Failure Surface Line (Mean Stress Diagram)," *Mech. Mach. Theory*, **16**, p. 339.
- [13] Palmgren, A., 1924, "Die Lebensdauer von Kugellagern," *ZV'DI*, **68**, p. 339.
- [14] Miner, M. A., 1945, "Cumulative Damage in Fatigue," *ASME Trans. J. Appl. Mech.*, **67**, p. A159.
- [15] Marco, S. M., and Starkey, W. L., 1954, "A Concept of Fatigue Damage," *Trans. ASME*, **76**, p. 627.
- [16] Corten, H. T., and Dolan, T. J., 1956, *Cumulative Fatigue Damage*, *Proc. Intl. Conf. on Fatigue of Metals*, ASME, and IME, p. 235.
- [17] Haibach, E., 1989, *Betriebsfestigkeit Verfahren und Daten zur Bauteilberechnung*, VDI-Verlag GmbH, Dusseldorf.
- [18] Henry, D. L., 1955, "Theory of Fatigue Damage, Accumulation in Steel," *Trans. ASME*, **77**, p. 913.
- [19] Gatts, R. R., 1961, "Application of a Cumulative Damage Concept to Fatigue," *Trans. ASME*, **83**, Series D, p. 529.
- [20] Manson, S. S., Frecke, J. C., and Ensign, C. R., 1967, "Application of a Double Linear Damage Rule to Cumulative Fatigue," *Fatigue Crack Propagation*, STP-415. American Society for Testing and Materials, Philadelphia, p. 384.
- [21] Rao, J. S., 1998, *Dynamics of Plates*, Marcel Dekker.
- [22] Rao, J. S., 1992, *Advanced Theory of Vibration*, John Wiley and Sons, New York.
- [23] Rao, J. S., Shin, L. D., and Vyas, N. S., 1996, "Blade Dynamics and Life Estimation—Computer Program BLADE," Power Research Institute, TaiPower Co., Taipei, Oct.
- [24] Rao, J. S., and Rao, V. V. R., 1987, "Effect of Downwash on the Unsteady Forces in Turbomachine Stage," *Proc. 11th Biennial ASME Conf. Vib. and Noise, Bladed-Disk Assemblies*, DE-Vol. 6, ASME, New York, p. 21.
- [25] Barsom, J. M., and Rolfe, S. T., 1987, *Fracture and Fatigue Control in Structures*, Prentice-Hall, Englewood Cliffs, NJ.

A Reduced-Order Model of Mistuning Using a Subset of Nominal System Modes

M.-T. Yang

J. H. Griffin

Department of Mechanical Engineering,
Carnegie Mellon University,
Pittsburgh, PA 15213

Reduced-order models have been reported in the literature that can be used to predict the harmonic response of mistuned bladed disks. It has been shown that in many cases they exhibit structural fidelity comparable to a finite element analysis of the full bladed disk system while offering a significant improvement in computational efficiency. In these models the blades and disk are treated as distinct substructures. This paper presents a new, simpler approach for developing reduced-order models in which the modes of the mistuned system are represented in terms of a subset of nominal system modes. It has the following attributes: the input requirements are relatively easy to generate; it accurately predicts mistuning effects in regions where frequency veering occurs; as the number of degrees-of-freedom increases it converges to the exact solution; it accurately predicts stresses as well as displacements; and it accurately models the deformation and stresses at the blades' bases. [DOI: 10.1115/1.1385197]

1 Introduction

The resonant amplitudes of turbine blades tend to be sensitive to small changes in the blades' properties. In the literature, this problem is often referred to as the *blade mistuning* problem and has been studied extensively by a number of researchers; for examples, refer to ([1–7]). Recently, researchers have been developing reduced-order models of mistuned bladed disks ([8–9]) in which the properties of the reduced-order model are calculated directly from finite element models of the blade and disk substructures. The advantage of the reduced-order models is that they can accurately represent the true geometry of the bladed disk system while incurring computational costs comparable to those of spring mass models.

In this paper a new approach is presented for developing reduced-order models of mistuned bladed disk vibration. This new approach represents the mistuned modes in terms of a limited sum or subset of “nominal” system modes. For brevity the approach will be referred to with the acronym SNM¹ for subset of nominal modes. Nominal system modes are defined as the modes of the system for a nominal blade and disk geometry. The nominal geometry could be one in which every blade and disk sector is identical, i.e., a tuned bladed disk, or it could be one in which the blade and disk sector geometry or the constraints are varied in a prescribed manner. For example, the latter approach could be used to specify a nominal system in which the blades are intentionally mistuned in a repeating pattern such as alternate mistuning. The SNM approach determines the effect of variations from the nominal design, such as those caused by uncontrolled manufacturing variations, on the vibratory response of the system. SNM differs from the earlier reduced-order models proposed in the literature

in that the blades and disk are treated as a single structure and not as separate substructures with different mathematical representations.

In practice the mistuning in a bladed disk system is often thought of in terms of variations in the frequencies of individual blades. One of the reasons that the earlier reduced-order models represented the blades as separate substructures was that it made it easier to use the frequencies of individual blades to characterize mistuning in their models. A methodology is presented in this paper which also allows the blade frequencies to be used to determine mistuning in the SNM approach.

The SNM approach is based on a concept developed in a recent paper by the authors, Yang and Griffin [10]. That paper discusses how the modes of the nominal system can be used to represent the modes of the system if the system's properties slightly change, for example, because of manufacturing variations. If the nominal structure has modes with closely spaced natural frequencies, then the altered structure will also have modes with closely spaced frequencies in the same frequency range. It is these closely spaced modes that are very sensitive to structural changes. The key result from the paper is that the closely spaced modes in the altered system can be approximated as a sum of the closely spaced nominal modes. This approximation results in a reduced-order formulation of the problem where the number of degrees-of-freedom is equal to the number of closely spaced modes. The Yang and Griffin [10] paper shows that neglecting nominal modes with more remote natural frequencies results in errors which are inversely proportional to the frequency difference. Thus, if more accuracy is required the error in the SNM reduced-order model can be reduced by increasing the number of nominal modes used in the representation.

The SNM approach works very naturally with bladed disk systems because bladed disks tend to have natural frequencies that fall into closely grouped clusters. For example, the lowest frequency modes in a bladed disk often correspond to modes in which the strain energy is stored mostly in the blades. In a tuned bladed disk, this family of modes is often referred to as the first bending family and the closely spaced frequencies correspond to different numbers of nodal diameters in the disk. If the disk is relatively stiff this set of clustered modes may be fairly well isolated. In this case, the SNM approach could accurately represent the first bending family of mistuned modes using only the first

¹A preliminary version of the results reported in this paper was presented at a DoD and industrial review meeting of the GUIde Consortium in August 1998. At that time, the authors of this paper referred to their method as the “Modal Domain Approach” or by the acronym MDA and that designation is still being used by some members of the gas turbine community to refer to what is called the SNM approach in this paper.

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-288. Manuscript received by IGTI Oct. 1998; final revision received by the ASME Headquarters March 1999. Associate Editor: D. Wisler.

bending family of tuned modes. If additional accuracy is needed then several families of tuned modes also could be included in the reduced-order model.

The representation of using nominal modes to express the solution is similar to a representation employed by Lin and Mignolet [7]. In their case, they transform the physical domain to a modal domain. The columns of their transformation matrix are the mode shapes of the tuned system when certain coupling terms are large and the mode shapes of the decoupled system when the coupling terms are small. However, the solution procedure that they use is quite different from the one discussed in this paper in that they directly solve for the forced response of the system using an impedance approach and an adaptive perturbation scheme. In the case of SNM, the solution is determined by first calculating the modes of the mistuned system by solving a “modal eigenvalue” problem ([10]). The eigenvalue problem is important in its own right since it is formulated so that it can readily include the motion dependent aerodynamic forces ([11]). When the aerodynamic terms are included the eigenvalues determine the aerodynamic damping as well as the frequency and composition of the mistuned modes. Since negative aerodynamic damping can cause flutter ([12]) the modal eigenvalue calculation provides a computationally efficient method for determining the effect of mistuning on the aerodynamic stability of the stage. If the response of the stage is stable the mistuned modes can be used to express the forced response of the system using modal superposition.

This paper is organized as follows. Section 2 provides the theoretical basis for SNM. In Section 3 a benchmark case is discussed in which the modes and forced response of a mistuned bladed disk are determined using a finite element code, SNM, and for comparison LMCC, a first generation reduced-order model from Yang and Griffin [9]. Some attributes of the new method are summarized in Section 4. Lastly, as part of the SNM methodology, a new procedure has been developed for calculating the mistuning in the reduced-order model from the frequencies of the mistuned blades. The procedure is summarized in the Appendix.

2 Theory

This section presents the mathematical basis of SNM. The first part transforms the equations of motion from the physical domain into one that is in terms of the nominal system modes. The equations are then rewritten in state-space form so that motion-dependent aerodynamic forces and gyroscopic forces can be included in the modal analysis. The number of degrees-of-freedom is then reduced by limiting the number of nominal modes used in the representation. The modal eigenvalue problem is then described. The modal eigenvalue problem determines mode shapes and complex eigenvalues of the mistuned system. It is a “modal” eigenvalue problem in that the mode shapes of the mistuned modes are represented as a weighted sum of nominal modes and the eigenvectors determined in the analysis are the weighting coefficients. The complex eigenvalue determines the natural frequency and aerodynamic damping of corresponding mistuned mode. This analysis determines the stability of the system since if the aerodynamic damping is negative then the mode may flutter. Expressions are then given for the forced response of the system in terms of the modes.

2.1 Transformation From Structural Domain to Modal Domain. Consider a mistuned bladed disk which rotates at a constant speed through a fluid that has a sinusoidal disturbance in space. By assuming a harmonic, steady-state response, the equation of motion of the system can be written as

$$[\mathbf{K}^o + \Delta\mathbf{K} + i\omega(\mathbf{C} + \mathbf{G}) - \omega^2(\mathbf{M}^o + \Delta\mathbf{M})]\mathbf{u} = \mathbf{f}_e + \mathbf{f}_m \quad (1)$$

where $\mathbf{K}^o$ and $\mathbf{M}^o$ are the stiffness and mass matrices for a perfect system of the nominal design, $\Delta\mathbf{K}$ and $\Delta\mathbf{M}$ are the variations in the stiffness and mass matrices, $\mathbf{C}$ and $\mathbf{G}$ the damping and gyro-

scopic matrices, $\mathbf{u}$ the amplitudes, $\mathbf{f}_e$ the prescribed excitation forces, $\mathbf{f}_m$ the motion-dependent aeroelastic forces, and ω the excitation frequency.

Consider also the modes ϕ_j^o and natural frequencies ν_j^o of the nominal system which satisfy

$$\mathbf{K}^o\Phi^o = \mathbf{M}^o\Phi^o\Lambda^o \quad (2)$$

where the nominal modal and eigenvalue matrices Φ^o and Λ^o have the form

$$\Phi^o = [\phi_1^o \phi_2^o \cdots \phi_N^o] \quad (3)$$

$$\Lambda^o = \text{diag}(\nu_1^{o^2}, \nu_2^{o^2}, \dots, \nu_N^{o^2}) \quad (4)$$

and N is the number of degrees-of-freedom of the system. Typically, Φ^o and Λ^o are calculated using a finite element analysis of the nominal system. The nominal modal matrix Φ^o can be either real (a standing wave) or complex (a traveling wave) depending on whether it is computed using a full bladed disk model or a sector model. It is assumed in this paper that Φ^o is complex since the traveling wave coordinate system is consistent with the approach typically used for the aerodynamic forces.²

Since the ϕ_j^o form a complete basis, the amplitude vector $\mathbf{u}$ can be expressed as a weighted sum of nominal modes, i.e.,

$$\mathbf{u} = \Phi^o\alpha \quad (5)$$

where

$$\alpha = [\alpha_1 \alpha_2 \cdots \alpha_N]^T \quad (6)$$

with α_j determining the amount that the j th nominal mode ϕ_j^o contributes to the response. Furthermore, the motion-dependent aeroelastic forces $\mathbf{f}_m$ can be written as

$$\mathbf{f}_m = \mathbf{P}^o\alpha \quad (7)$$

where

$$\mathbf{P}^o = [\mathbf{p}(\phi_1^o) \mathbf{p}(\phi_2^o) \cdots \mathbf{p}(\phi_N^o)] \quad (8)$$

and $\mathbf{p}(\phi_j^o)$ is the vector of unsteady aeroelastic forces caused by a unit amplitude vibration of the system in the mode ϕ_j^o .

Substituting (5) and (7) in (1) and premultiplying by Φ^{oH} , the Hermitian of Φ^o , the equation of motion of the mistuned system in terms of the nominal modes is

$$[\hat{\mathbf{K}}^o + \Delta\hat{\mathbf{K}} + i\omega(\hat{\mathbf{C}} + \hat{\mathbf{G}}) - \omega^2(\hat{\mathbf{M}}^o + \Delta\hat{\mathbf{M}}) + \hat{\mathbf{Z}}_a]\alpha = \hat{\mathbf{f}}_e \quad (9)$$

where $\hat{\mathbf{K}}^o$ and $\hat{\mathbf{M}}^o$ are the diagonal modal stiffness and mass matrices of the nominal system, $\Delta\hat{\mathbf{K}}$ and $\Delta\hat{\mathbf{M}}$ the variations in the modal stiffness and mass matrices

$$\Delta\hat{\mathbf{K}} = \Phi^{oH}\Delta\mathbf{K}\Phi^o \quad (10)$$

$$\Delta\hat{\mathbf{M}} = \Phi^{oH}\Delta\mathbf{M}\Phi^o, \quad (11)$$

$\hat{\mathbf{C}}$ and $\hat{\mathbf{G}}$ the modal damping and gyroscopic matrices,

$$\hat{\mathbf{C}} = \Phi^{oH}\mathbf{C}\Phi^o \quad (12)$$

$$\hat{\mathbf{G}} = \Phi^{oH}\mathbf{G}\Phi^o, \quad (13)$$

$\hat{\mathbf{Z}}_a$ the modal aerodynamic impedance matrix,

$$\hat{\mathbf{Z}}_a = -\Phi^{oH}\mathbf{P}^o, \quad (14)$$

$\hat{\mathbf{f}}_e$ the modal excitation force,

$$\hat{\mathbf{f}}_e = \Phi^{oH}\mathbf{f}_e, \quad (15)$$

²Using a bladed disk sector model with cyclic symmetric boundary conditions to compute complex modes is more efficient and has less numerical problems since the eigenvalues are more isolated.

and the vector of weighting coefficients α determines the amount of each nominal mode in the response.

Suppose that the modal aerodynamic impedance $\hat{Z}_a$ can be written in the following form ([13]):

$$\hat{Z}_a = \hat{K}_a + i\omega\hat{C}_a - \omega^2\hat{M}_a \quad (16)$$

where $\hat{K}_a$, $\hat{C}_a$, and $\hat{M}_a$ are the modal aerodynamic stiffness, damping, and inertia matrices that are, in general, complex and non-Hermitian. The modal equation of motion (9) can then be solved using standard methods by casting it in its state space form, i.e.,

$$(-\mathbf{A} + i\omega\mathbf{B})\mathbf{y} = \mathbf{q} \quad (17)$$

where

$$\mathbf{A} = \begin{bmatrix} 0 & I \\ -(\hat{K}^o + \Delta\hat{K} + \hat{K}_a) & -(\hat{C} + \hat{G} + \hat{C}_a) \end{bmatrix} \quad (18)$$

$$\mathbf{B} = \begin{bmatrix} I & 0 \\ 0 & \hat{M}^o + \Delta\hat{M} + \hat{M}_a \end{bmatrix} \quad (19)$$

$$\mathbf{y} = \begin{Bmatrix} \alpha \\ i\omega\alpha \end{Bmatrix} \quad (20)$$

$$\mathbf{q} = \begin{Bmatrix} 0 \\ \hat{f}_e \end{Bmatrix}. \quad (21)$$

Equation (17) is the state-space form of the equation of motion of the mistuned system in terms of nominal modes.

2.2 Reduced Set of Modes. Thus far, the modal equation of motion (17) does not show any particular advantage over the original equation of motion (1) in terms of computational costs, since both of them have the same number of degrees of freedom. Equation (17) could be even worse since the coefficient matrices $\mathbf{A}$ and $\mathbf{B}$ are not sparse matrices. However, as discussed in Section 1, it is known from an earlier study ([10]) that, given small variations in the system's properties (i.e., small $\hat{K}_a$, $\hat{C}_a$, $\hat{M}_a$, $\hat{G}$, $\Delta\hat{K}$, and $\Delta\hat{M}$), a mode of the structure will significantly change its shape only if the neighboring modes have frequencies which are very close. In this case, the resulting mistuned mode can be well approximated as a linear combination of the closely spaced nominal modes. Neglecting nominal modes with more remote natural frequencies results in errors which are inversely proportional to the frequency difference. Based on this understanding, it is then possible to use only a subset of the nominal modes to estimate the associated mistuned modes of interest.

Suppose ϕ_{s+1}^o , ϕ_{s+2}^o , ..., and ϕ_{s+n}^o are the subset of n nominal modes to be considered. Equation (17) then implies that the reduced-order equation of motion in terms of a subset of nominal system modes is

$$(-\mathbf{a} + i\omega\mathbf{b})\mathbf{y} = \mathbf{q} \quad (22)$$

where $\mathbf{a}$ and $\mathbf{b}$ are the reduced-order matrices of $\mathbf{A}$ and $\mathbf{B}$ with dimensions $2n$ by $2n$. The choice of n will be discussed in Section 3. The accuracy of a solution can be studied by examining how the results converge as n is increased. Clearly, the error goes to zero in the limit since the method provides an exact solution when all of the nominal modes are included in the representation.

2.3 Free Vibration (Flutter) Analysis. For free vibration, (22) implies a modal eigenvalue problem of the form

$$(-\mathbf{a} + i\nu_j\mathbf{b})\mathbf{r}_j = \mathbf{0} \quad j=1,2,\dots,2n \quad (23)$$

where ν_j is the j th eigenvalue and $\mathbf{r}_j$ is the associated state-space right eigenvector of the mistuned system. Equation (23) can be solved using standard numerical routines. The eigenvalues are, in general, complex, i.e.,

$$\nu_j = \nu_{R,j} + i\nu_{I,j}. \quad (24)$$

When $\nu_{I,j} > 0$, the j th mistuned mode has positive damping and is stable. When $\nu_{I,j} < 0$, the mode has negative damping and the system flutters since the transient response will grow exponentially as time increases.

2.4 Forced Response Analysis. Assuming that $\nu_{I,j} > 0$ for all $\mathbf{r}_j$, the solution to the forced response problem (22) can be written as a linear combination of the state space eigenvectors $\mathbf{r}_j$, i.e.,

$$\mathbf{y} = \sum_{j=1}^{2n} \beta_j \mathbf{r}_j \quad (25)$$

where β_j is the amplitude of the mistuned mode $\mathbf{r}_j$ induced by the harmonic excitation. Second, substitute (25) in (22) and utilize mode orthogonality to decouple the equations. A simple expression for β_j can then be written as

$$\beta_j = \frac{\mathbf{l}_j^T \mathbf{q}}{\mathbf{l}_j^T \mathbf{b} \mathbf{r}_j (\nu_j - \omega)} i \quad (26)$$

where $\mathbf{l}_j$ is the j th left eigenvector of the eigenvalue problem associated with (23) which satisfies

$$\mathbf{l}_j^T (-\mathbf{a} + i\nu_j \mathbf{b}) = \mathbf{0}^T. \quad (27)$$

Given expression (26) for β_j , the physical amplitude $\mathbf{u}$ can be calculated using (5), (20), and (25), i.e.,

$$\mathbf{u} = \Phi^o \mathbf{R}_d \boldsymbol{\beta} \quad (28)$$

where

$$\mathbf{R}_d = [\mathbf{r}_{d1} \mathbf{r}_{d2} \cdots \mathbf{r}_{d2n}] \quad (29)$$

and $\mathbf{r}_{dj}$ is the displacement part of the state-space eigenvector $\mathbf{r}_j$.

3 Numerical Results

A computer program SNM (Subset of Nominal Modes) was written to implement the theory presented in the last section. A flow diagram of the SNM is shown in Fig. 1. Note that the structure of the program is relatively simple and, as a result, the code was much easier to develop than the first generation program LMCC ([9]) which required separate representations for the various substructures.

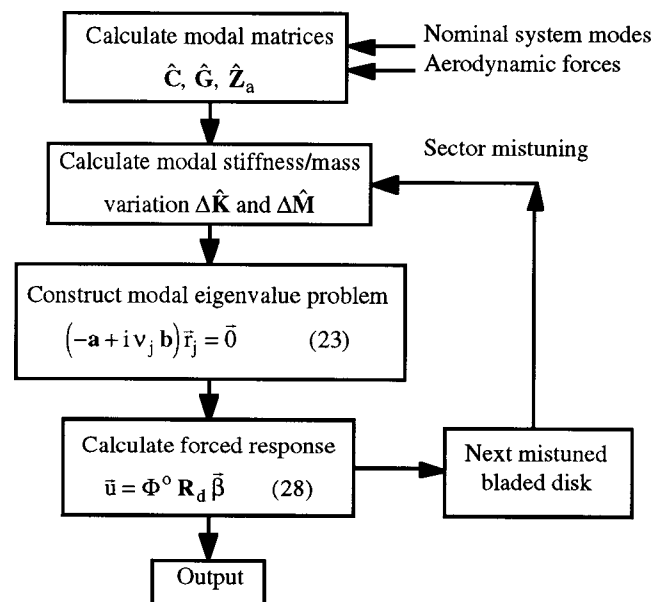


Fig. 1 Flow chart for the SNM algorithm

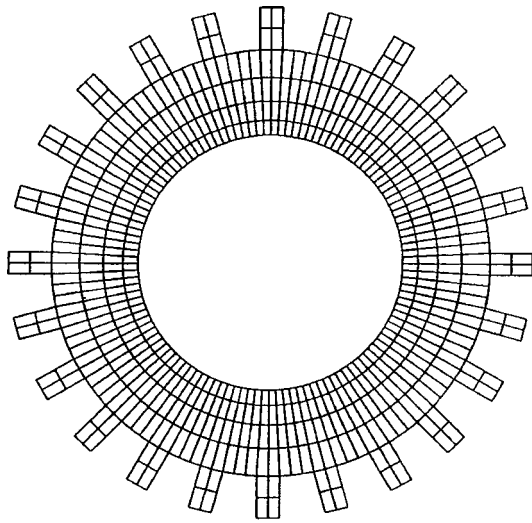


Fig. 2 Finite element model of bladed disk system

3.1 Benchmark Test Problem and Tuned System

Results. In this section SNM is used to compute the dynamic response of the bladed disk system, Fig. 2, originally analyzed by LMCC. The geometry of the test problem was chosen to represent a structure with low aspect ratio, plate-like blades. A coarse model of plate elements was used so that the entire system would not have too many degrees-of-freedom. This made it possible to run a benchmark, finite element analysis of the entire bladed disk without too much difficulty.

In this case the “nominal” system is “tuned,” i.e., every blade, disk sector is identical and blade frequency mistuning is introduced by changing the elastic modulus from one blade to the next. An efficient procedure for calculating the change in the modal stiffness matrix, $\Delta \mathbf{K}$ in Eq. (10), from the frequencies of the mistuned blades and the stiffness matrix of a nominal blade is given in the Appendix. In this example, the motion-dependent aeroelastic forces, the gyroscopic forces, and $\Delta \mathbf{M}$ in (1) are zero.

The geometry of the test problem is the same as the “thin disk” analyzed in the earlier work. When the disk is thin the disk modes tend to interact with the blade modes and more regions of frequency veering occur. Regions of frequency veering are wanted since they provide a more severe test of the accuracy of reduced-order models ([9]). The natural frequencies of the tuned system are shown in Fig. 3 as a function of the phase angle parameter, j , where the relative phase between adjacent sectors is equal to $2\pi j/N_b$ and N_b is the number of blades. Frequency veering is apparent in the way in which the frequencies tend to merge and then veer away from each other.

The SNM approach works well in bladed disk systems because the natural frequencies tend to occur in closely spaced clusters. This is because the modes tend to belong to families that share

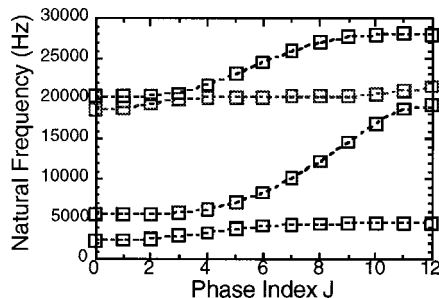


Fig. 3 Natural frequencies of tuned bladed disk

certain characteristics. In many systems, the families can be readily separated into blade and disk modes. A family of *blade modes* are system modes in which the strain energy is stored primarily in the blades. Typically, a family of blade modes have frequencies that fall into a narrow range that is close to a frequency that the blade would have if it were fully constrained at its base. Thus, blade modes are further classified by type, e.g., a “first bending” family or a “first torsion” family of modes. For example, in Fig. 3, the lowest frequency modes for j equal to 6 through 12 belong to the first bending family of modes. *Disk modes* are system modes that have their strain energy stored primarily in the disk. Often the frequencies of disk modes increase significantly as the phase angle parameter, j , increases. An example of a set of frequencies that belong to a family of disk modes would be the second highest frequencies for j equal to 6 through 11 in Fig. 3. Families of disk modes can be classified by the number of nodal circles. The lower frequency families have fewer nodal circles.

The bladed disk system used in this paper has a very flexible disk and, as a result, the frequencies of the disk mode families appear to cross over the frequencies of the blade modes. For example, the lowest frequency modes for j equal to 0 through 3 are disk modes whereas the lowest frequency modes for j equal to 6 through 12 are blade modes. Since for j equal to 4 and 5 the lowest frequency mode has to transition from being a disk mode to a blade mode, the actual mode shapes in the transition region tend to be a combination of modes from the two families. This phenomenon is often referred to as frequency or curve “veering” in the literature, e.g., Perkins and Mote [12], since a more careful analysis of the situation would show that the frequencies do not cross but, in fact, “veer” away from each other.³

In an earlier paper on reduced-order models, Yang and Griffin [9], the authors compared the exact natural frequencies of the tuned system with the natural frequencies of the tuned system calculated with their linear, mistuning computer code, LMCC. Their error was related to the number of blade modes and disk modes used to represent the substructures and the approximation in assuming that the bases of the blades are limited to rigid-body-type motions. This type of frequency comparison is not relevant in the approach introduced here since the exact, tuned system modes are now used as the basis of the representation. As a result, the new approach automatically gives exactly the same natural frequencies as the finite element model when the mistuning is set to zero in the reduced-order model. The current method reduces the number of degrees-of-freedom by limiting the number of tuned system modes. Thus, the type of error that is introduced does not affect the frequencies of the tuned system modes that are included in the representation, but instead is apparent in the absence of frequencies that are not included in the representation.

Another property of SNM is that it exactly represents the modes of the tuned system when the mistuning goes to zero. This is especially important in regions of frequency veering. For example, consider the tuned system modes associated with the fourth highest frequency for $j=3$ in Fig. 3. Because this frequency is very close to the third highest frequency for $j=3$, the associated mode shapes (it is a repeated frequency) are more difficult to determine since any error in the reduced-order model will be amplified by the closeness of the two sets of modes. The sensitivity of the tuned modes to small modeling errors is due to the same mathematical problem that causes mistuning, i.e., small variations in the structural model can cause large changes in the mode shapes when the tuned system modes are close together. Here, however, the small variations in the structural model result from errors in the reduced-order model, the changes in the mode

³Frequency veering can be seen in this example by using cyclic symmetric boundary constraints on a single-blade disk sector when calculating the system's natural frequency. In such an analysis, the phase difference across the sector can be taken as a continuous variable. One then finds that the frequencies do not cross but veer away from each other as the phase difference varies.

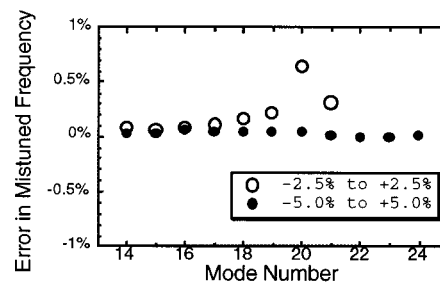
shapes are displayed in the radial rather than the circumferential direction, and the tuned system modes are close together because of veering.

Thus, one of the reasons that LMCC had difficulty in predicting the mistuned response of the system in regions of frequency veering ([9]) is that it had difficulty in accurately predicting the modes of the tuned system. This is not a source of error in the SNM reduced-order model since SNM automatically gives the same results as the finite element method when mistuning goes to zero.

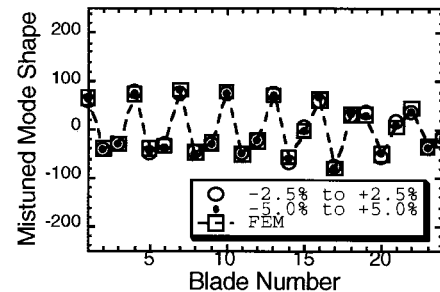
3.2 Mistuned System Results. The accuracy of SNM depends on the number of degrees-of-freedom used to represent the structure. The number of degrees-of-freedom in SNM is the same as the number of tuned system modes used in the representation. An issue explored in this section is how the number of tuned system modes used in SNM affects the accuracy of the solution to the forced response problem. This issue can be divided into two parts. The first is how the number of tuned system modes affects the estimate of the mistuned system modes. The second issue is how many mistuned system modes are needed to determine forced response.

Mistuned Modes. Since the frequencies of the bladed disk are only slightly perturbed when the system is mistuned, the mistuned modes also can be classified as belonging to same types of families as the tuned disk. For example, in the mistuned system the lowest frequency modes also belong to the first bending and the zero nodal circle mode families. As discussed in Sections 1 and 2, the paper on modal interaction by Yang and Griffin [10] shows that the degree to which tuned system modes participate in forming the modes of the mistuned system is inversely proportional to the differences in their natural frequencies. Thus, the lowest frequency mistuned modes are comprised primarily of the first two families of tuned modes. This point is illustrated by the results shown in Fig. 4.

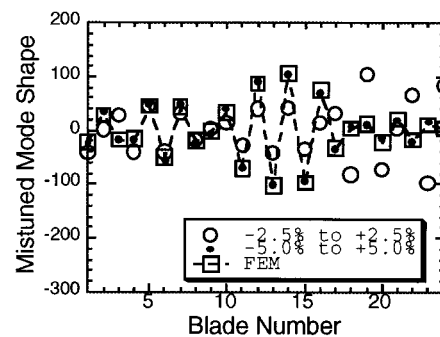
In Fig. 4(a), the estimated errors in the natural frequencies of the mistuned system are shown for a representative case from the first bending family of modes. The criteria used to determine which tuned modes to include in the reduced-order model was to include all modes with natural frequencies in a certain range, i.e., $[f_{ave}(1-p), f_{ave}(1+p)]$. In Fig. 4(a) the average frequency, f_{ave} , was set equal to the lowest frequency mode for j equal to 8 (Fig. 3) and p was taken as 2.5 percent and then increased to 5.0 percent. The number of frequencies shown in plots is the same as the number of modes used in the model. Thus, Fig. 4(a) indicates that the number of tuned modes in the model increased from 8 to 11 when p increased from 2.5 percent to 5.0 percent. The estimated error is defined as $(f_j^{fem} - f_j^{SNM})/f_j^{fem}$ where f_j^{fem} is the j th frequency calculated by a finite element analysis of a full mistuned bladed disk and f_j^{SNM} is the corresponding value calculated from the SNM model. The error is only an estimate since the eigenvalue solver in the finite element program may have significant errors when trying to calculate the modes and natural frequencies of a system in which the frequencies are very close together.⁴ As can be seen the estimated error is reduced when p is increased to 5 percent and more modes are used in SNM. A mistuned mode corresponding to the 16th natural frequency is depicted as a function of circumferential position in Fig. 4(b). Note that both reduced-order models, p equal to 2.5 percent and 5 percent, provide a reasonably accurate estimate of the mode shape. In this case the mistuned mode has a natural frequency that is in the center of the frequency range of tuned modes included in the representation. A second example of a mode shape, the 20th mode, is shown in Fig. 4(c). In this case, the mistuned mode has a natural frequency near the top of the range of frequencies included



(a) Estimated frequency error



(b) Mode shape of the 16th mode



(c) Mode shape of the 20th mode

Fig. 4 Representative results for the first bending family of modes

in the representation. Clearly, the error in the mode shape has been significantly reduced by increasing p from 2.5 percent to 5 percent in this case.

A more severe test of the SNM model are mistuned modes in the higher frequency, veering region near j equal to 4 in Fig. 3. A plot of estimated frequency error is shown for these modes in Fig. 5(a) for p equal to 2.5 percent and 5.0 percent. Again, the estimated error is reduced as more tuned system modes are included in the SNM model. A mode with a natural frequency near the center of the frequencies of the tuned modes used in the reduced order model is shown in Fig. 5(b). Again, both the 2.5 percent and 5 percent models provide a reasonably accurate representation of the mistuned mode. The results for a mode that has a natural frequency near the top of the range of frequencies included in the representation is shown in Fig. 5(c). Again, the error in the mode shape has been significantly reduced by increasing p from 2.5 percent to 5 percent. Note that the 5 percent model has only 28 degrees-of-freedom and, yet, gives very accurate results even in the veering region.

Forced Response. A second issue is how many mistuned modes are needed to accurately estimate the forced response when the system is excited at a resonant frequency? Standard modal analysis indicates that the degree to which a mode contributes to

⁴This is usually not a problem for tuned system calculations since analysts use cyclic symmetry boundary conditions to isolate modes.

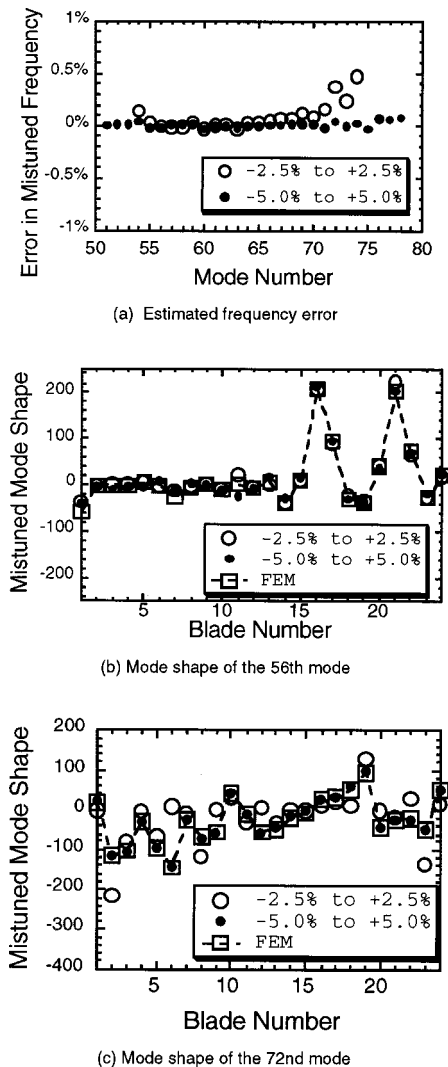


Fig. 5 Representative modal results for the veering region

the steady-state harmonic response of a system depends on three factors: the amount of damping, the difference between the mode's resonant frequency and the excitation frequency, and the generalized force exciting the mode. Typically, bladed disks are very lightly damped. As a result, when the excitation frequency is at a resonant frequency the mode that is excited will have a very high response. The contributions of other modes will die off in inverse proportion to the frequency difference. As a result, other families of modes that have distinctly different frequencies have little effect on the forced response of a particular mistuned family.

Additionally, the generalized force exciting a mode can also play an important role in determining if a mode will respond to a particular engine-order excitation. For example, for low-frequency tuned modes the generalized force exciting the mode is zero unless the engine order of the excitation equals the number of nodal diameters (the parameter j of Fig. 3) in the mode. When a system is weakly mistuned the mistuned modes often tend to exhibit a similar spatial oscillatory behavior. Typically, the higher the frequency of a mode in a family and the higher the engine order of the excitation the more spatial oscillations (and nodal lines) they will have around the wheel. The generalized force acting on a mode is proportional to the dot product of the mode shape and the vector representing the distribution of forces on the system. Consequently, in many cases a low-order engine excitation tends to

produce larger generalized forces on the lower frequency modes in a family, whereas, high-engine-order excitations tend to excite the higher frequency modes.

For these reasons, the number of mistuned modes needed to represent the forced response of the mistuned system is not very large. For systems in which the mode families are relatively isolated it would typically need to include only a single family of modes. In the case of veering when two families have a significant amount of interaction it would typically need to include only those two families of modes.

The forced response of the test problem was calculated using a modal summation, Eq. (28), in which the mistuned modes were determined by using SNM as well as the direct finite element method.⁵ The forced response was also calculated using LMCC. For the low-frequency modes both SNM and LMCC gave very satisfactory results. One difference was that LMCC results tend to exhibit a slight frequency shift⁶ which SNM does not have.

A more interesting case was an excitation of modes in the veering region, i.e., a 4th-engine-order excitation in the frequency range of 19,000 to 21,000 Hz. The tip amplitudes of all 24 blades are plotted as a function of the excitation frequency for this case in Fig. 6. The finite element results, Fig. 6(a), is based on summing the response of 90 mistuned modes. The SNM model, Fig. 6(b), has 28 degrees-of-freedom (28 tuned modes) whereas the LMCC model, Fig. 6(c), has 144 degrees-of-freedom. Clearly, SNM more accurately captures the forced response of the mistuned bladed disk than LMCC. Note that no attempt was made to "calibrate" either the SNM or the LMCC model. As reported in Yang and Griffin [9] the accuracy of the LMCC model can be improved by artificially adjusting the attachment stiffness in the blade model so that the resonant frequencies of the tuned LMCC model more closely match the natural frequencies of the finite element model in the veering region. The attraction of the SNM approach is that no modifications of the model are required to get accurate results in this case.

3.3 Numerical Efficiency. The SNM approach is more efficient than the earlier LMCC algorithm for two reasons. The first is that it requires fewer degrees-of-freedom to accurately represent the system. If N_b is the number of blades then LMCC reduces the number of independent degrees-of-freedom in the system to six N_b , the number of rigid-body motions at the base of the blades. As discussed previously, SNM typically requires at most one or two times N_b modes to accurately represent the same system. Thus, the number of degrees-of-freedom is usually a factor of three or more smaller in SNM than in LMCC.

A second reason that SNM is more efficient than LMCC is that it calculates the forced response in a different manner. SNM calculates forced response by summing the mistuned modes, Eq. (28). The modes are found by solving a reduced-order eigenvalue problem. LMCC solves a six N_b system of equations at each excitation frequency. In the simulations reported in this paper SNM was approximately 30 times faster than LMCC.

The relative efficiency of SNM or LMCC when compared with a direct finite element solution of a full mistuned bladed disk depends on the application. In developing either reduced-order model there is the initial computational overhead of determining the tuned system modes that are used as input. Since the tuned system modes need to be calculated only one time, the real computational advantage in using reduced-order models occurs when simulating a large number of mistuned bladed disks either to generate the statistical response of the mistuned system or for optimization purposes. In Yang and Griffin [9], the authors estimate

⁵The damping ratio in each mistuned mode was assumed to be 0.25 percent of critical damping.

⁶The frequency increase in LMCC results has been attributed to the increase in stiffness associated with the constraint that the blade's base undergoes rigid-body-type motions.

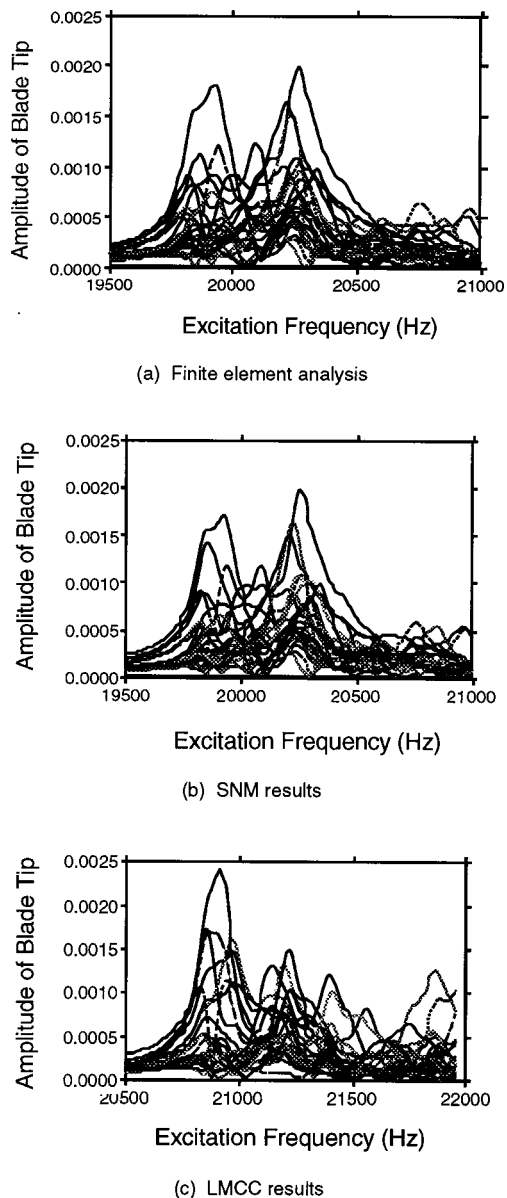


Fig. 6 Representative forced response amplitudes in the veering region

that for a 100 bladed disk simulation LMCC would be one to two orders of magnitude faster than direct finite element simulation.

4. Conclusions

A new reduced-order model of mistuned bladed disks has been developed in which the modes of the mistuned system are represented in terms of a subset of nominal system modes, SNM. The SNM approach has a number of attractive features.

1 The input data are relatively easy to generate. For example, only a finite element analysis of a single blade, disk sector is needed to generate the tuned system modes as input to SNM. In addition, since only nominal modes are used as a basis for the representation, it is necessary to determine the motion-dependent aeroelastic forces only for these modes. In contrast, in LMCC the aerodynamic influence coefficients also have to be calculated for rigid-body blade motions since they are also used to represent the system's response.

2 The mathematical formulation exactly yields the finite element solution as the amount of mistuning goes to zero. This is

especially important when frequency veering occurs since other reduced-order models can have difficulty in predicting the tuned system modes and natural frequencies in this case.

3 As the number of degrees-of-freedom in the SNM model increases the results from a mistuned system calculation converges to the exact solution. Consequently, the accuracy of the SNM model can be checked by observing how the results converge as more degrees-of-freedom are used in the SNM model. This is especially important in bladed disks with a large number of degrees-of-freedom in the finite element model since it is difficult to run a finite element analysis of a full mistuned bladed disk as a benchmark.

4 The fact that the SNM approach first solves for the eigenvalues and modes of the mistuned system is appealing because:

- it provides a very natural formulation for solving the mistuned aerodynamic eigenvalue problem (Eq. (23)) to determine how mistuning affects flutter.
- it is easy to calculate the stresses in the bladed disk. This is because the mistuned modes are calculated as a weighted sum of nominal system modes. The modal stresses of the mistuned modes are the same weighted sum of the nominal system's modal stresses.
- it is computationally very efficient since the forced response is expressed as a sum of mistuned modes that are independent of the excitation frequency (at least over the limited speed range associated with a particular engine-order crossing⁷).

5 The SNM approach automatically captures the deformation at the base of the blade and the manner in which it transfers dynamic forces to neighboring blades. In contrast, LMCC imposes rigid interfaces between the blades and disk that only transmit non-self-equilibrated forces and moments. This increase in accuracy should be important when analyzing the effect of mistuning on the response of higher frequency blade mode families in which the dynamic forces tend to be more self-equilibrated.

6 The SNM approach uses "nominal modes" to represent the response and the nominal system modes do not have to be tuned modes. For example, the system could have some type of imposed periodicity which may be taken into account in defining the nominal modes. The periodicity could be from periodic boundary constraints or imposed manufacturing variations. In this case, the nominal modes would be calculated from a single periodic sector that could include multiple blades. In fact, the modes of a completely mistuned system could be used as nominal modes and SNM is used to determine the effect of additional random mistuning on the system's response. In this case the nominal modes would have to be calculated once from a finite element model of the full, mistuned stage.

An important area of research that still needs to be addressed is one of system identification. Typically, in the past only a limited amount of information, the blades' frequencies, has been used to characterize mistuning. Reduced-order models make it possible to more precisely model the effects of real physical variations in a computationally efficient manner. Thus, the issues are: what physical variations are of primary importance, how can they be determined experimentally in an efficient manner, and how can they be efficiently translated into input that can be used in the reduced order models?

Acknowledgments

This work was supported by the U.S. Navy as part of a GUIde Consortium integrated research program on high cycle fatigue in gas turbine engines.

⁷If necessary, it would be relatively easy to include the effect of engine speed changes in the analysis either by calculating the nominal modes at different engine speeds and interpolating or by including the changes as prescribed perturbations in Eq. (1).

Appendix

Efficient Calculation of $\Delta\hat{\mathbf{K}}$. If a cyclic symmetric structure is chosen as the nominal system, the computation of the modal stiffness variation $\Delta\hat{\mathbf{K}}$, modal mass variation $\Delta\hat{\mathbf{M}}$, and modal aerodynamic impedance $\hat{\mathbf{Z}}_a$ can be simplified by taking into account its cyclic symmetry. This Appendix will illustrate how to calculate $\Delta\hat{\mathbf{K}}$ only since $\Delta\hat{\mathbf{M}}$ and $\hat{\mathbf{Z}}_a$ can be computed in a similar fashion.

Suppose the nominal system is of rotational periodicity m , the modal matrix of the nominal system Φ^o can be written as

$$\Phi^o = \begin{bmatrix} \Phi_1^o \\ \vdots \\ \Phi_m^o \end{bmatrix} \quad (A1)$$

where Φ_j^o is the modal matrix of the j th cyclic symmetric sector of the nominal system. Since the mode shapes of all sectors are the same except for phase differences, the sector modal matrices can be expressed as

$$\Phi_j^o = \Phi_1^o \mathbf{E}^{j-1} \quad j = 1, \dots, m \quad (A2)$$

where

$$\mathbf{E} = \text{diag}(e_s) \quad (A3)$$

$$e_s = \exp\left(i \frac{2\pi s}{m}\right) \quad s = 0, \dots, m-1 \quad (A4)$$

Furthermore, the stiffness variation of the whole structure $\Delta\mathbf{K}$ can be written in terms of those of the individual sectors $\Delta\mathbf{K}_j$

$$\Delta\mathbf{K} = \text{diag}(\Delta\mathbf{K}_j) \quad j = 1, \dots, m.^8 \quad (A5)$$

Substituting (A2) and (A5) in (10), the modal stiffness variation $\Delta\hat{\mathbf{K}}$ becomes

$$\Delta\hat{\mathbf{K}} = \sum_{j=1}^m (\Phi_1^o \mathbf{E}^{j-1})^H \Delta\mathbf{K}_j \Phi_1^o \mathbf{E}^{j-1}. \quad (A6)$$

Suppose the stiffness variation of a sector can be characterized by q parameters

$$\Delta\mathbf{K}_j = \sum_{r=1}^q \Delta c_j^{(r)} \mathbf{K}^{(r)} \quad j = 1, \dots, m \quad (A7)$$

then, substituting (A7) in (A6) gives

$$\Delta\hat{\mathbf{K}} = \sum_{j=1}^m \sum_{r=1}^m \Delta c_j^{(r)} (\mathbf{E}^{j-1})^H \hat{\mathbf{K}}^{(r)} \mathbf{E}^{j-1} \quad (A8)$$

⁸It is assumed here that the coupling stiffness between sectors does not change. This formulation can be extended to include that effect if it proved to be important.

where

$$\hat{\mathbf{K}}^{(r)} = \Phi_1^{oH} \mathbf{K}^{(r)} \Phi_1^o \quad r = 1, \dots, q. \quad (A9)$$

In the case of blade frequency mistuning as studied in this paper, q is equal to 1, $\Delta c_j^{(1)}$ is the percentage change of the Young's modulus of the j th blade (the blade's frequency is proportional to the square root of Young's modulus), and

$$\mathbf{K}^{(1)} = \begin{bmatrix} \mathbf{K}_b & 0 \\ 0 & 0 \end{bmatrix} \quad (A10)$$

where $\mathbf{K}_b$ is the stiffness matrix of the nominal blade.

It should be noticed that $\hat{\mathbf{K}}^{(r)}$ needs to be calculated only once before running Monte Carlo simulations and the mistuning for each new bladed disk is incorporated by changing $\Delta c_j^{(1)}$. Therefore, using (A8) to calculate $\Delta\hat{\mathbf{K}}$ for each additional bladed disk requires very low computational cost.

References

- [1] Whitehead, D. S., 1966, "Effect of Mistuning on the Vibration of Turbomachine Blades Induced by Wakes," *J. Mech. Eng. Sci.*, **8**, pp. 15–21.
- [2] Ewins, D. J., and Han, Z. S., 1984, "Resonant Vibration Levels of a Mistuned Bladed Disk," *J. Sound Vib.*, **106**, pp. 211–215.
- [3] Fabunmi, J. A., 1980, "Forced Vibrations of a Single Stage Axial Compressor Rotor," *ASME J. Eng. Gas Turbines Power*, **102**, pp. 322–328.
- [4] Griffin, J. H., and Hoosac, T. M., 1984, "Model Development and Statistical Investigation of Turbine Blade Mistuning," *ASME J. Vibr. Acoust.*, **106**, pp. 204–210.
- [5] Sinha, A., and Chen, S., 1989, "A Higher Order Technique to Compute the Statistics of Forced Response of a Mistuned Bladed Disk Assembly," *J. Sound Vib.*, **130**, No. 2, pp. 207–221.
- [6] Wei, S. T., and Pierre, C., 1988, "Localization Phenomena in Mistuned Assemblies With Cyclic Symmetry, Part II: Forced Vibrations," *ASME J. Vibr. Acoust.*, **110**, No. 4, pp. 429–438.
- [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] Kruse, M. J., and Pierre, C., 1996, "Forced Response of Mistuned Bladed Disks Using Reduced-Order Modeling," *Proceedings of the AIAA/ASME/ASCE/AHS/ASC Structures Dynamics, and Materials Conference and Exhibit*, Salt Lake City, UT, AIAA Paper No. A96-26801, AIAA, Reston, VA, pp. 1938–1950.
- [9] Yang, M.-T., and Griffin, J. H., 1997, "A Reduced Order Approach for the Vibration of Mistuned Bladed Disk Assemblies," *ASME J. Eng. Gas Turbines Power*, **119**, pp. 161–167.
- [10] Yang, M.-T., and Griffin, J. H., 1997, "A Normalized Modal Eigenvalue Approach for Resolving Modal Interaction," *ASME J. Eng. Gas Turbines Power*, **119**, pp. 647–650.
- [11] Crawley, E. F., 1988, "Aeroelastic Formulation for Tuned and Mistuned Rotors," *AGARD Manual on Aeroelasticity in Axial-Flow Turbomachines, Vol. 2, Structural Dynamics and Aeroelasticity*, M. F. Platzer and F. O. Carta, eds., pp. 19-1–19-24.
- [12] Pidaparti, R. M. V., and Afolabi, D., 1996, "The Role of Eigenvectors in Aeroelastic Analysis," *J. Sound Vib.*, **193**, No. 4, pp. 934–940.
- [13] Hall, K. C., Florea, R., and Lanzkron, P. J., 1995, "A Reduced Order Model of Unsteady Flows in Turbomachinery," *ASME J. Turbomach.*, **117**, pp. 375–383.

Periodic Response of Blades Having Three-Dimensional Nonlinear Shroud Constraints

J. J. Chen

C. H. Menq¹

e-mail: menq.1@osu.edu

Coordinate Metrology and Measurement
Laboratory,
Department of Mechanical Engineering,
The Ohio State University,
Columbus, OH 43210

In this paper, a three-dimensional shroud contact model is employed to predict the periodic response of blades having three-dimensional nonlinear shroud constraint. When subjected to periodic excitation, the resulting relative motion at the shroud contact is assumed to be periodic in three-dimensional space. Based on the three-dimensional shroud contact model, analytical criteria are used to determine the transitions between stick, slip, and separation of the contact interface and are used to simulate hysteresis loops of the induced constrained force, when experiencing periodic relative motion. The constrained force can be considered as a feedback force that influences the response of the shrouded blade. By using the multiharmonic balance method along with Fast Fourier Transform, the constrained force can be approximated by a series of harmonic functions so as to predict the periodic response of a shrouded blade. This approach results in a set of nonlinear algebraic equations, which can be solved iteratively to yield the periodic response of blades having three-dimensional nonlinear shroud constraint. In order to validate the proposed approach, the predicted results are compared with those of the direct-time integration method. The resonant frequency shift, the damping effect, and the jump phenomenon due to nonlinear shroud constraint are examined. The implications of the developed solution procedure to the design of shroud contact are also discussed. [DOI: 10.1115/1.1385828]

Introduction

In a shrouded blade system, the protruding shrouds constrain the blade motions not only along the contact plane but also along the normal direction of the plane, resulting in very complex contact kinematics. The in-plane tangential relative motion is often two-dimensional, and it can induce stick-slip friction [1–4]. On the other hand, the normal relative motion can cause variation of the contact normal load and, in extreme circumstances, separation of contact interface [5,6]. In previous studies of the shrouded blade systems, Yang and Menq [2] proposed a two-dimensional model for the contact kinematics. This model retains the normal component of the relative motion that causes normal load variation while degenerating the in-plane tangential component of the relative motion into linear motion. In their study, the assumed blade motion has only two components, namely axial and tangential components. In order to take the radial component into account, Yang et al. [7] proposed a simplified three-dimensional shroud contact kinematics, in which the two-dimensional in-plane tangential relative motion is assumed to be elliptical and is decomposed into two linear motions along the principal major and minor axes of the ellipse. A variable normal load friction force model [2] was then applied separately to each individual linear motion to estimate the equivalent stiffness and damping of the shroud contact.

Yang and Menq [6] further proposed a three-dimensional shroud contact model, in which the joined effect of the two-dimensional tangential relative motion and the normal relative motion is examined. They developed a set of analytical criteria to determine the transitions among stick, slip, and separation, when experiencing variable normal load. With these transition criteria,

the constrained force can be predicted for any given three-dimensional cyclic relative motion across the contact interface. In this paper, the three-dimensional shroud contact model [6] is employed to obtain the constrained force at the shroud contact of a shrouded blade system. The bladed system is assumed tuned and the assumed blade motion has three components, namely axial, tangential, and radial components. In the shroud contact model, a contact plane is defined and its orientation is assumed invariant. When subjected to periodic excitation, the resulting three-dimensional relative motion at a shroud contact as well as the constrained force are assumed to be periodic. The constrained force can be considered as a feedback force that influences the response of the shrouded blade. By using the multiharmonic balance method along with fast Fourier transform, the constrained force can be approximated by a series of harmonic functions. In the calculation of the nonlinear forced response of a shrouded blade, all the linear degrees-of-freedom can be condensed to receptance and the modeling of shroud contact can be separated from the complex structure model. This approach results in a set of nonlinear algebraic equations, which can be solved iteratively to yield the periodic forced response of blades having three-dimensional nonlinear shroud constraint.

Three-Dimensional Shroud Contact

Figure 1 shows a shrouded blade system with two neighboring blades contacting each other through the protruding shrouds. When subjected to cyclic excitation, the vibratory motion of the shrouded blade can be assumed to be periodic, and the resulting relative motion across the shroud contact is also periodic in the three-dimensional space. In modeling the shroud contact, a “substructure” can be used to represent the friction interface that contains the contact plane and small portions of the two neighboring shrouds, and the substructure can be modeled as two massless elastic elements that are held together by a preload n_0 . The points A and B are the outermost points of these two elastic elements; and the difference of their respective motions can describe the three-dimensional periodic relative motion of the two neighboring shrouds. The periodic relative motion is often not parallel to the

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-289. Manuscript received by IGTI, October 1998; final revision received by the ASME Headquarters, March 1999. Associate Editor: D. Wisler.

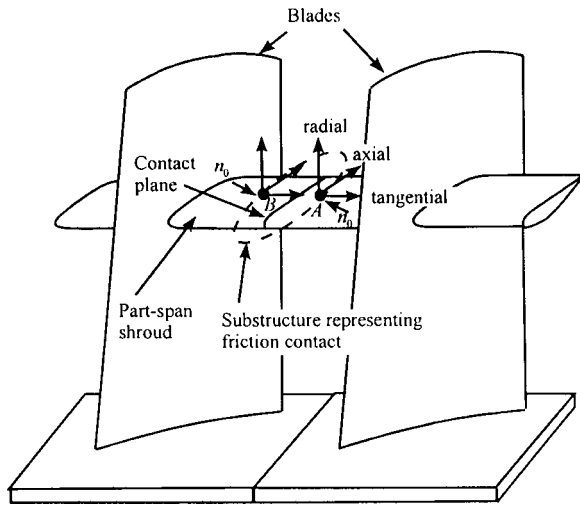


Fig. 1 Shroud contact of two neighboring shrouded blades

contact plane. In order to analyze the induced friction, the periodic relative motion in the three-dimensional space can be decomposed into an in-plane periodic motion on the contact plane and a periodically varying component normal to the contact plane.

Shroud Contact Geometry. Two shrouded blades contact each other through their protruding shrouds as are shown in Fig. 2, and the xyz-coordinate system (called blade coordinate system) is defined in accordance with the tangential (x), axial (y), and radial (z) directions. The contact plane of the three-dimensional shroud contact is defined by two angles ψ (called shroud angle) and ϕ (called inclination angle). A vuw -coordinate system (called shroud coordinate system) can be defined, where v -axis is along the normal direction of the contact plane, and u and w -axes are on the contact plane. In this paper, the blade coordinate system is defined by using three unit vectors, namely $[\hat{x} \hat{y} \hat{z}]$, and the shroud coordinate system $[\hat{v} \hat{u} \hat{w}]$. These two coordinate systems can be related to each other as follows:

$$[\hat{v} \hat{u} \hat{w}] = [\hat{x} \hat{y} \hat{z}] T_0 \quad (1)$$

where T_0 is the coordinate transformation matrix [7].

$$T_0 = \begin{bmatrix} \cos \psi \cos \phi & -\sin \psi & \cos \psi \sin \phi \\ \sin \psi \cos \phi & \cos \psi & \sin \psi \sin \phi \\ -\sin \phi & 0 & \cos \phi \end{bmatrix} \quad (2)$$

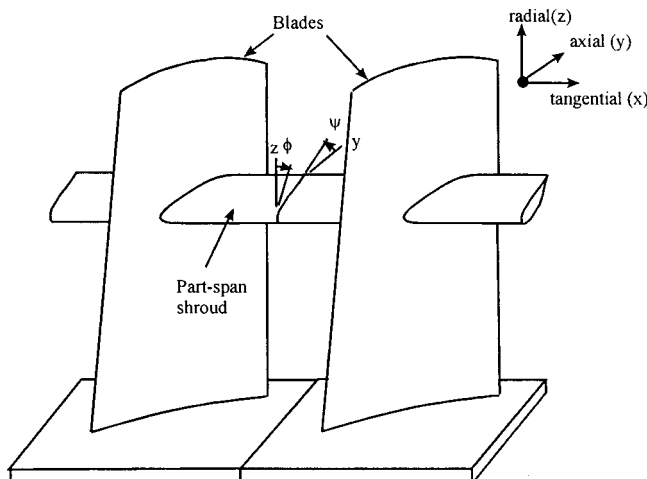


Fig. 2 Contact geometry of a shrouded blade system

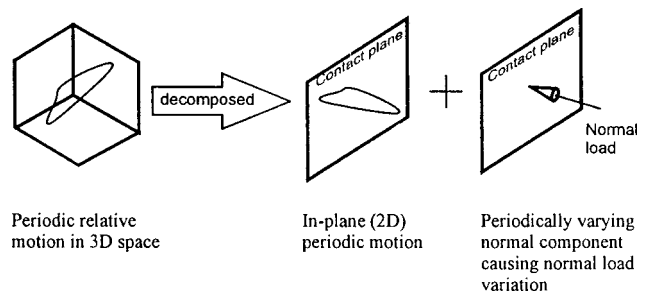


Fig. 3 Decomposition of three-dimensional relative motion

In current design practice, the inclination angle ϕ is often set to be zero.

The decomposition of the three-dimensional relative motion is shown in Fig. 3 schematically. The in-plane periodic motion can induce stick-slip friction, and thus can attenuate the periodic response of the shrouded blades. On the other hand, the normal component tends to alter the normal load across the interface; and this effect, in extreme circumstances, may lead to separation of the interface. It should be noted that the variable normal load is taken as the sum of the initial contact pressure at equilibrium plus a term that is proportional to the periodically varying normal component of the relative motion. Since this decomposition is to transfer the relative motion from the blade coordinate system to the shroud coordinate system, it can be carried out by performing a coordinate transformation on the three-dimensional relative motion by using Eq. (1).

Three-Dimensional Shroud Contact Model. In the three-dimensional shroud contact model proposed by Yang and Menq [6], the contact interface between two vibrating shrouds can be modeled as a substructure that contains a massless elastic element and a friction contact point, as depicted in Fig. 4. In this model, the elastic element accounts for the shear and normal stiffness of the substructure, and it is characterized by a 2×2 stiffness matrix K_u for the shear stiffness and a spring constant k_v for the normal stiffness. The friction contact point, that is assumed to obey the Coulomb friction law with the friction coefficient μ when in contact with Body 2, can undergo tangential stick-slip motion, and may experience intermittent separation from Body 2 when the normal relative motion (v) becomes large. The motion of the con-

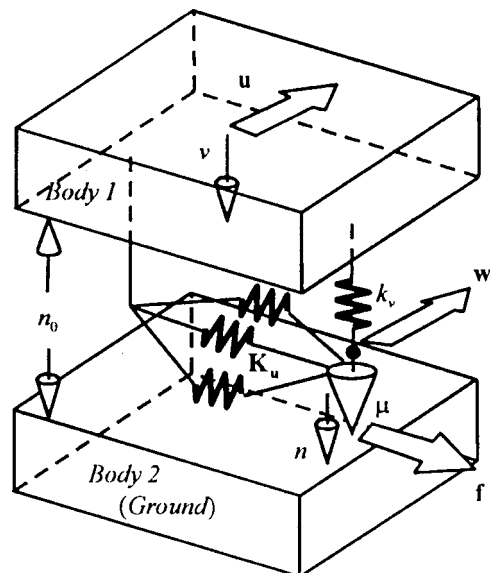


Fig. 4 A three-dimensional shroud contact model

tact point is denoted as $\mathbf{w}$ in this model. The contact interface is assumed to have either a preload or an initial gap (as designated by n_0). This model allows a negative preload to represent the situation when the interface has an initial gap; the equivalent preload across the interface with a gap e is calculated as $-k_v e$. In this model, $\mathbf{u}$ and v are the input tangential relative motion ($[u \ w]^T$) and normal relative motion (v) of the contact interface, respectively, and they can be evaluated as the motion of *Body 1* with respect to *Body 2*, where v , u , and w are the relative motions in the shroud coordinate system.

Constrained Force. The constrained force consists of two components: the induced friction force on the contact plane and the variable normal force. Since the friction force is completely characterized by the relative motion, it will not lose generality to assume one of the contacting surfaces is the ground. With this assumption, the input tangential relative motion $\mathbf{u}$, the slip motion of the contact point $\mathbf{w}$, and the induced friction force $\mathbf{f}$ are vectors parallel to the ground; the normal relative motion v and the normal load n are scalars. The friction force, acting on the ground, can be expressed as

$$\mathbf{f} = \mathbf{K}_u(\mathbf{u} - \mathbf{w}). \quad (3)$$

The normal load is taken as the sum of the preload n_0 plus the variation caused by v , and it can be expressed as

$$n = \begin{cases} n_0 + k_v v & \text{when } v \geq -n_0/k_v \\ 0 & \text{when } v < -n_0/k_v \end{cases}. \quad (4)$$

In this paper, the orientation of the contact plane is assumed to be invariant. This assumption is reasonable if the amplitude of shroud relative motion is relatively small when compared to the overall dimension of the shroud interface.

Stick, Slip, and Separation. Depending on the amplitude and phase of the in-plane relative motion and the normal load component of the vibratory motion, the friction contact will either stick, slip, or separate during a cycle of oscillation. The stick and slip conditions can be expressed as follows [6]:

$$\text{stick condition} \quad |\mathbf{f} = \mathbf{K}_u(\mathbf{u} - \mathbf{u}_0) + \mathbf{f}_0| < \mu n \quad \dot{\mathbf{w}} = 0 \quad (5)$$

$$\text{slip condition} \quad \mathbf{f} = \mu n \frac{\dot{\mathbf{w}}}{|\dot{\mathbf{w}}|} \quad \dot{\mathbf{w}} \neq 0 \quad (6)$$

where $\mathbf{u}_0$ and $\mathbf{f}_0$ are the initial values of $\mathbf{u}$ and $\mathbf{f}$ at the beginning of the stick state. During the cycle of motion, the applied variable normal load may vanish to cause the interface to separate; consequently, the friction force is not present.

Stick/Slip/Separation Transition Criteria. In order to evaluate the resulting periodic constrained force at the shroud contact, analytical criteria are employed to determine the transitions between stick, slip, and separation when experiencing variable normal load. The analytical criteria can be summarized as follows.

(i) *Stick-to-Slip Transition.* This transition occurs when the friction force on the tangential plane reaches the varying slip load μn . That is,

$$|\mathbf{f} = \mathbf{K}_u(\mathbf{u} - \mathbf{u}_s) + \mathbf{f}_0| = \mu n. \quad (7)$$

To ensure that the magnitude of the friction force has a tendency to exceed the slip load, the following constraint is imposed:

$$|\dot{\mathbf{f}}| > \mu \dot{n}. \quad (8)$$

(ii) *Slip-to-Stick Transition.* During the slipping state, according to the Coulomb friction law, the friction force can be solved from an initial value problem:

$$\dot{\mathbf{f}} = \mathbf{K}_u \left(\dot{\mathbf{u}} - \frac{\mathbf{f}^T \mathbf{K}_u \dot{\mathbf{u}} - \mu^2 n \dot{n}}{\mathbf{f}^T \mathbf{K}_u \mathbf{f}} \mathbf{f} \right). \quad (9)$$

The slip-to-stick transition occurs when the velocity of the relative motion $\dot{\mathbf{w}}$ equals 0, which implies

$$\mathbf{f}^T \mathbf{K}_u \dot{\mathbf{u}} - \mu^2 n \dot{n} = 0. \quad (10)$$

Since the initial friction force at the beginning of the slip condition is known, the initial value problem of Eq. (9) can be solved by using a numerical integration scheme such as the fourth-order Runge-Kutta method to obtain the friction force $\mathbf{f}$. Once the friction force is obtained, the criterion of Eq. (10) can be used to check the occurrence of the slip-to-stick transition.

(iii) *Stick/Slip-to-Separation Transition.* The transition from stick/slip to separation occurs when the normal load vanishes. In addition, the normal load should be decreasing at this moment to ensure the occurrence of the separation. Hence the transition criteria can be formulated as

$$n = 0, \quad \dot{n} < 0. \quad (11)$$

(iv) *Separation-to-Stick/Slip Transition.* Similarly to the above criterion, the separation ends when the normal load is about to develop on the contact plane. Therefore, the moment of this transition can be determined by the criterion

$$n = 0, \quad \dot{n} \geq 0. \quad (12)$$

When the normal load and the friction force begin to develop on the contact plane at the end of the separation, their rate of change at the moment determine whether the following state is either stick or slip.

$$\dot{\mathbf{u}}^T \mathbf{K}_u^T \mathbf{K}_u \dot{\mathbf{u}} < \mu^2 \dot{n}^2 \Rightarrow \text{stick begins} \quad (13)$$

$$\dot{\mathbf{u}}^T \mathbf{K}_u^T \mathbf{K}_u \dot{\mathbf{u}} \geq \mu^2 \dot{n}^2 \Rightarrow \text{slip begins}. \quad (14)$$

It should be pointed out that the incipient slipping condition can be regarded as a one-dimensional case, because the friction force is not present at this moment and the slip action will be developed along $\dot{\mathbf{u}}$. Thus, according to the Coulomb friction law, the rate of change of the developing friction force can be expressed as

$$\dot{\mathbf{f}} = \mu \dot{n} \frac{\dot{\mathbf{u}}}{|\dot{\mathbf{u}}|}. \quad (15)$$

Once the friction force develops, it can be further determined by solving the initial value problem of Eq. (9).

In this paper, these criteria are used to simulate hysteresis loops of the friction contact, when experiencing periodic relative motion. With these hysteresis loops, the resulting constrained force can be characterized by the relative motion between two neighboring shrouds. By using fast Fourier transform, the constrained force can be approximated by a series of harmonic functions and employed in multiharmonic balance method to solve for the periodic response.

Periodic Response of A Shrouded Blade System

In the analysis of a shrouded blade system, a great simplification can be obtained by assuming that the bladed system is tuned, namely each blade of the system has exactly the same dynamic characteristics. In addition, the excitation of interest is that induced by the blades rotating through circumferential variations in the flow field. It can be shown that in effect each blade is exposed to a periodic excitation having the same amplitude but differing in phase by an amount which is proportional to the blade's angular location on the disk. It is assumed that the forced response of the bladed system is also periodic and has the same fundamental period as the excitation.² Thus the motion of the blade as well as the nonlinear constrained force can be represented by infinite Fourier series. By truncating these series after the m th terms, an approximate solution assuming that the forced response is periodic can be

²Subharmonic components can also be included in the approach presented in this paper. However, they are ignored in the current manuscript.

derived. In this approach, each blade vibrates in the same manner but with a proportional interblade phase difference ($k\varphi$) for k th harmonic component from its adjacent blades. The interblade phase angle is defined as follows:

$$\varphi = \frac{2\pi E}{N} \quad (16)$$

where N is the total number of the blades in the system and E is the engine order of the excitation on the system.

The equation of motion of a shrouded blade subjected to a periodic excitation can be expressed as follows:

$$\mathbf{m}\ddot{\mathbf{x}} + \mathbf{c}\dot{\mathbf{x}} + \mathbf{k}\mathbf{x} = \mathbf{f}_e - \mathbf{f}_N \quad (17)$$

where $\mathbf{x}$ is the nodal displacement vector, $\mathbf{m}$ is the mass matrix, $\mathbf{c}$ is the damping matrix, $\mathbf{k}$ is the stiffness matrix, $\mathbf{f}_e$ is the external periodic excitation, and $\mathbf{f}_N$ is the nonlinear constrained force which is a nonlinear function of the relative motion at the shroud contact. The finite element model is three-dimensional and if the model contains n nodes, all the matrices will be $3n \times 3n$ matrices, and all the vectors will be $3n$ -element vectors.

The external periodic excitation can be expressed as follows:

$$\mathbf{f}_e(t) = \sum_{k=0}^m \mathbf{f}_{e,k} e^{jk\omega t} \quad (18)$$

where ω is the fundamental excitation frequency and $\mathbf{f}_{e,k}$ is a complex vector representing the amplitude and phase for the k th harmonic component of the excitation. Here the external periodic excitation is assumed to include up to m th harmonic component. It should be pointed out that except for the elements associated with those shroud contact points the other elements of $\mathbf{f}_N$ are zeros. It is clear that the nonlinear aspect of the dynamic problem is embedded in the nonlinear friction force $\mathbf{f}_N$. By using the modal analysis method, the mode shape matrix can be obtained and is denoted as Φ . Using the mode shape matrix and the associated modal information, the receptance of the blade can be derived as

$$\mathbf{r} = [r_{pl,k}] = \sum_{i=1}^{3n} (\Phi_i \Lambda_{i,k} \Phi_i^T) \quad (19)$$

and

$$\Lambda_{i,k} = [(\mathbf{k}_i - k^2 \omega^2 \mathbf{m}_i) + j(k\omega \mathbf{c}_i)]^{-1} \quad (20)$$

where $r_{pl,k}$ is defined as the steady-state response of the p th node due to unit k th harmonic excitation force at the l th node, Φ_i is the i th mode shape, $\mathbf{m}_i$ is the i th modal mass, $\mathbf{k}_i$ is the i th modal stiffness, and $\mathbf{c}_i$ is the i th modal damping.

In this paper, the periodic three-dimensional motion in the blade coordinate system can be represented as follows:

$$\mathbf{x} = \begin{bmatrix} x \\ y \\ z \end{bmatrix} \approx \begin{bmatrix} \sum_{k=0}^m \mathbf{A}_{x,k} e^{jk\omega t} \\ \sum_{k=0}^m \mathbf{A}_{y,k} e^{jk\omega t} \\ \sum_{k=0}^m \mathbf{A}_{z,k} e^{jk\omega t} \end{bmatrix} \quad (21)$$

where ω is the fundamental oscillating frequency, and $\mathbf{A}_{i,k}$, $i = x, y, z$, are the complex vectors representing the amplitude and phase angle of the k th harmonic component along tangential, axial and radial axes. For a shrouded blade, several pairs of shroud contact points can be defined. For each pair of shroud contact points, one is on the right and the other left and their motions are denoted as $[\mathbf{x}_r \ \mathbf{x}_l]^T$. For convenience, this vector can be arranged as

$$[\mathbf{x}_r \ \mathbf{x}_l]^T = [(\mathbf{x}_r \ \mathbf{x}_l)_k]^T = [\mathbf{x}_{r,0} \mathbf{x}_{l,0} \mathbf{x}_{r,1} \mathbf{x}_{l,1} \cdots \mathbf{x}_{r,m} \mathbf{x}_{l,m}]^T. \quad (22)$$

Relative Motion at Shroud Contact. Since the shrouded blade system is assumed tuned, the condition of cyclic symmetry can be applied when deriving the relative motion of a shroud contact. Take the relative motion between the point B of the right shroud and the point A of the left shroud in Fig. 1 as an example. First, the motions of the two contact points of the first shrouded blade (left one) are defined and they are $[\mathbf{x}_r \ \mathbf{x}_l]^T$. Therefore, the motion of point B is now $\mathbf{x}_r$ and the motion of point A differs from $\mathbf{x}_l$ with the proportional interblade phase angle ($k\varphi$) for the k th harmonic component. As a result, the relative motion of the two neighboring shrouds, $\mathbf{w}_r$, can be derived as

$$\mathbf{w}_r = \mathbf{T}_1 [(\mathbf{x}_r \ \mathbf{x}_l)_k]^T, \quad k=0,1,\dots,m \quad (23)$$

where $\mathbf{T}_1$ is the interblade relative displacement transformation matrix.

$$\mathbf{T}_1 = [\text{diag}(\mathbf{T}_{1,0}, \mathbf{T}_{1,1}, \dots, \mathbf{T}_{1,m})] \quad (24)$$

$$\mathbf{T}_{1,k} = [\mathbf{I}_{3 \times 3} - e^{-jk\varphi} \mathbf{I}_{3 \times 3}], \quad k=0,1,\dots,m. \quad (25)$$

It is worthy noting that only the relative motion at the right shroud contact point is derived. Since the shrouded blade system is assumed tuned, the relative motion and the resulting constrained force of the left shroud contact point can be related to those of the right shroud contact point by using the condition of cyclic symmetry. Since both $\mathbf{x}_r$ and $\mathbf{x}_l$ are periodic motions, the resulting relative motion $\mathbf{w}_r$ also has periodic trajectory in the three dimensional space.

Since this relative motion $\mathbf{w}_r$ is in the blade coordinate system, it can be transformed to the shroud coordinate system by using the transformation defined in Eq. (1):

$$\mathbf{u}_r = \mathbf{T}_{BS}^T \mathbf{w}_r \quad (26)$$

$$\mathbf{T}_{BS} = [\text{diag}(\mathbf{T}_{0,0}, \mathbf{T}_{0,1}, \dots, \mathbf{T}_{0,m})], \mathbf{T}_{0,k} = \mathbf{T}_0, \quad k=0,1,\dots,m \quad (27)$$

where $\mathbf{T}_{BS}$ is the coordinate transformation matrix for three dimensional periodic relative motion. The three dimensional relative motion in the shroud coordinate system can be expressed as follows:

$$\mathbf{u}_r = \begin{bmatrix} v \\ u \\ w \end{bmatrix} \approx \begin{bmatrix} \sum_{k=0}^m \mathbf{A}_{v,k} e^{jk\omega t} \\ \sum_{k=0}^m \mathbf{A}_{u,k} e^{jk\omega t} \\ \sum_{k=0}^m \mathbf{A}_{w,k} e^{jk\omega t} \end{bmatrix} \quad (28)$$

where $\mathbf{A}_{i,k}$, $i=v,u,w$, are the complex vectors representing the amplitude and phase angle of the k th harmonic component along v , u and w -axes.

Constrained Force at Shroud Contact. After the decomposition, the u and w -components of the relative motion follow a periodic trajectory on the contact plane and can induce the stick-slip friction, while the v -component can cause the normal load across the interface to vary dynamically. It is apparent that the u and w motions are coupled together when inducing stick-slip friction. Using the three dimensional contact model proposed by Yang and Menq [6] the constrained force at the right shroud contact point can be determined. By using fast Fourier transform, the constrained force can be approximated by a series of harmonic functions and can be expressed as follows:

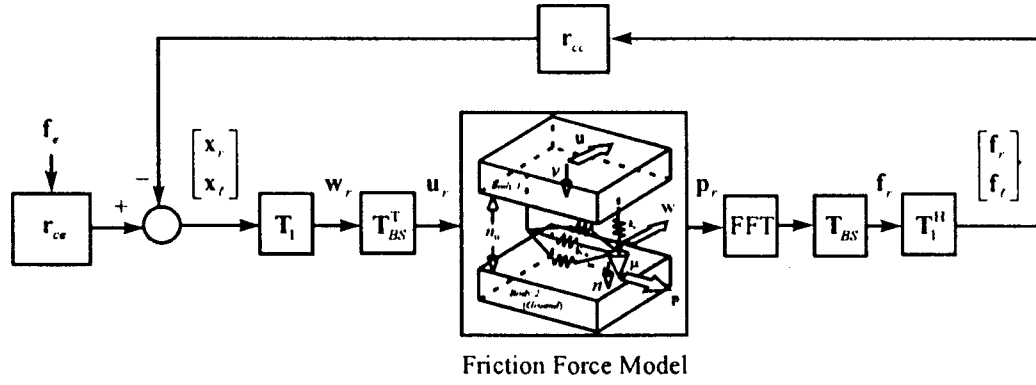


Fig. 5 Nonlinear feedback loop of a shrouded blade

$$\mathbf{p}_r = \begin{bmatrix} \mathbf{p}_{rv}(v) \\ \mathbf{p}_{ru}(v, u, w) \\ \mathbf{p}_{rw}(v, u, w) \end{bmatrix} \approx \begin{bmatrix} \sum_{k=0}^m \mathbf{p}_{rv,k} e^{jk\omega t} \\ \sum_{k=0}^m \mathbf{p}_{ru,k} e^{jk\omega t} \\ \sum_{k=0}^m \mathbf{p}_{rw,k} e^{jk\omega t} \end{bmatrix} \quad (29)$$

where $p_{ri,k}$, $i=v, u, w$, are the complex Fourier coefficients of the k th harmonic component along v , u and w -axes. Then, the constrained force can be transformed back to the blade coordinate system:

$$\mathbf{f}_r = \mathbf{T}_{BS} \mathbf{p}_r. \quad (30)$$

Furthermore, the constrained forces at a pair of shroud contact points can be related to the force at the right shroud contact point using the interblade relative displacement transformation matrix.

$$\begin{bmatrix} \mathbf{f}_r \\ \mathbf{f}_f \end{bmatrix} = \mathbf{T}_1^H \mathbf{f}_r \quad (31)$$

where $\mathbf{T}_1^H$ is the complex conjugate transpose of $\mathbf{T}_1$.

Nonlinear Algebraic Equations. When the blade is constrained by its neighboring blades through shroud contacts, the resulting constrained forces are characterized by the displacements of a pair of contact points, $[\mathbf{x}_r, \mathbf{x}_f]^T$, and they can be considered as feedback forces that influence the response of the blade. This feedback effect along with the contact kinematics is shown in Fig. 5. From the nonlinear feedback loop shown in Fig. 5, it is evident that in the calculation of nonlinear forced response of a shrouded blade, all the linear degrees-of-freedom can be condensed to receptance and the modeling of friction contact can be separated from the complex structure model. This approach results in a set of nonlinear algebraic equations, which can be formulated as follows:

$$\mathbf{u}_r = \mathbf{T}_{BS}^T \mathbf{T}_1 \{ \mathbf{r}_{ce} \mathbf{f}_e - \mathbf{r}_{cc} \mathbf{T}_1^H \mathbf{T}_{BS} \mathbf{p}_r(\mathbf{u}_r) \} \quad (32)$$

where $\mathbf{r}_{ce}$ is the receptance at the shroud contact points due to unit harmonic excitation force, and $\mathbf{r}_{cc}$ is the receptance at the shroud contact points due to unit constrained force. For simplicity of demonstration, each shrouded blade is assumed to contain a pair of shroud contact points. By using the fast Fourier transform, the constrained force can be approximated by harmonic functions having the same fundamental frequency as the external periodic excitation, and its amplitude and phase are nonlinear functions of the relative motions of the pair of contact points. By using the multiharmonic balance method, the nonlinear algebraic equations become

$$\mathbf{u}_{r,k} = \mathbf{T}_{ce,k} \mathbf{f}_{e,k} - \mathbf{T}_{cc,k} \mathbf{p}_{r,k}(\mathbf{u}_r), \quad k=0,1,2,\dots,m \quad (33)$$

where

$$\mathbf{T}_{ce,k} = \mathbf{T}_{0,k}^T \mathbf{T}_{1,k} \mathbf{r}_{ce,k} \quad (34)$$

$$\mathbf{T}_{cc,k} = \mathbf{T}_{0,k}^T \mathbf{T}_{1,k} \mathbf{r}_{cc,k} \mathbf{T}_{1,k}^H \mathbf{T}_{0,k}. \quad (35)$$

This set of nonlinear algebraic equations has the unknown $\{\mathbf{u}_{r,k}\}$, and can be solved iteratively by using Newton-Raphson algorithm. With the solution $\{\mathbf{u}_{r,k}\}$, the constrained force can be obtained by using Eq. (29). The periodic response of the shrouded blade system can be calculated by using the resulting constrained force together with the receptance.

Comparison With Direct-Time Integration Method

In order to verify the solution procedure presented in this paper, the predicted results are compared to those of direct time integration method. Since it is very time consuming when using direct-time integration method for a bladed system, a simplified system is considered. A blade with shroud-to-ground friction damper, as shown in Fig. 6, is tested to validate the proposed solution procedure. In this test, the three dimensional shroud contact model is employed to predict the periodic response of a shrouded blade, which is constrained by the ground. In this study, the first five vibration modes of the shrouded blade are employed to calculate the receptances and three harmonic terms are included in the calculation of nonlinear forced response. Various levels of preload, ranging from fully separate to fully stuck case, are applied and the predicted results are shown as solid curves in Fig. 7, along with

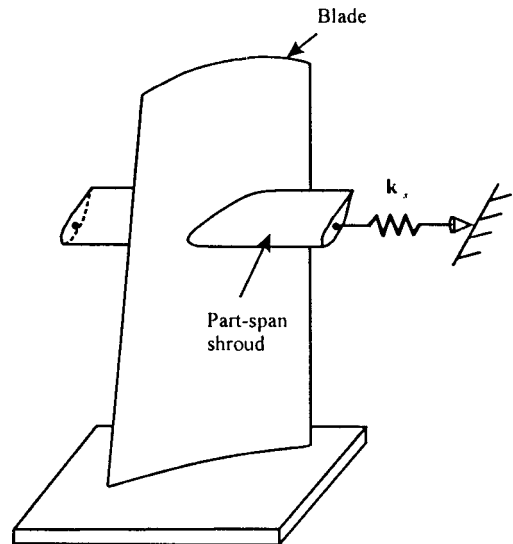


Fig. 6 A blade with shroud-to-ground friction damper

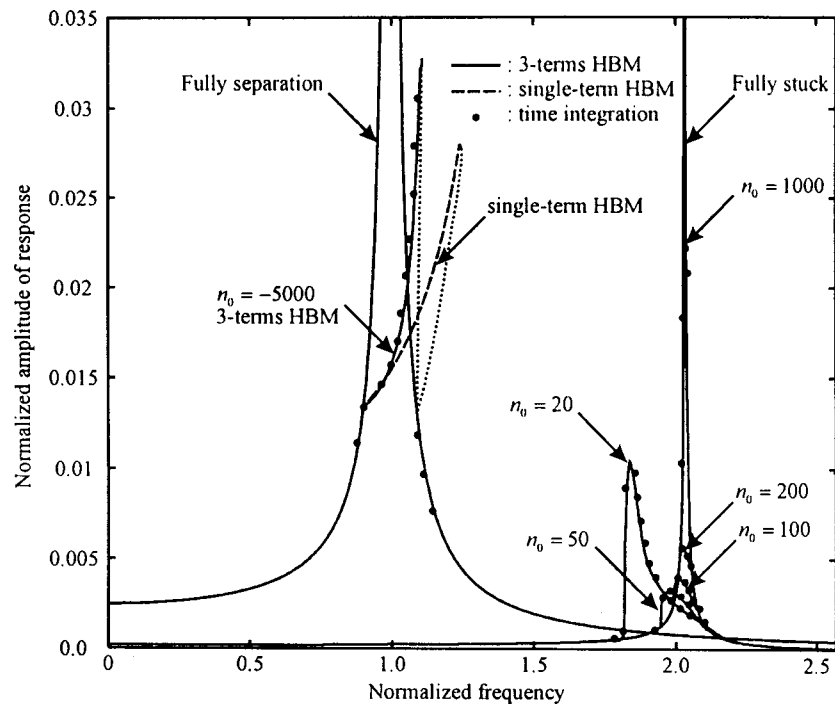


Fig. 7 Periodic response of a blade with shroud-to-ground friction damper

discrete data points, which are the results of the direct time integration method. In the figure, the frequency is normalized with respect to the first mode natural frequency, and the amplitude is normalized with respect to the peak value of the fully separation case. It is seen that the multiharmonic balance method can accurately predict the periodic response of the shrouded blade when the shroud contact is either preloaded or having an initial gap. In the same figure, the predicted resonance response for the case $n_0 = -5000$ using single-term harmonic balance method is also compared. It is apparent that in this case the nonlinear spring force of the shroud constraint exhibits a hardening effect that causes a jump phenomenon and the single-term harmonic balance method can not accurately predict the resonance response.

Periodic Response of a Shrouded Blade System

The proposed method is applied to predict the periodic response of a shrouded blade system. In this shrouded blade system, the first 20 vibration modes are employed in the analysis.

Shift of Resonant Frequency. It is known that when changing the preload there exist two limit cases, namely the fully separate case and fully stuck case [6]. Since the nonlinear contact force does not appear in the analysis, both cases are linear problems. The fully separate case occurs when the interface has such a large initial gap that the vibrating neighboring shrouds can not make contact with each other. Since the contact force is not present, the resonant response corresponding to the natural frequencies of the system can be clearly seen. On the other hand, when the preload of the interface exceeds a level depending on the external excitation, the shroud contact interface remains fully stuck. In this case, the shroud contact does not dissipate energy. However, it provides additional stiffness, which arises from the shroud constraint, to the system to cause higher resonant frequencies.

The frequency shifts of the first three vibration modes of the shrouded blade system subjected to nodal force external excitation are shown in Fig. 8. In the figure the frequency is normalized with respect to the first mode natural frequency, and the amplitude is normalized with respect to the peak of the first mode for the fully

separation case. Since the resulting responses along the three axes are similar, only the normalized amplitude of the response along the axial direction is presented in the figure. In the first vibration mode, the resonant peak shifts from fully separation at normalized frequency 1.0 to fully stuck at normalized frequency 2.08, and the amplitude of the peak resonance is reduced by 54.5 percent. For the second vibration mode, the resonant peak shifts from fully separation at normalized frequency 2.14 to fully stuck at normalized frequency 4.79, and the amplitude of the peak resonance is reduced by 93.3 percent. For the third vibration mode, the resonant peak shifts from fully separation at normalized frequency 2.33 to fully stuck at normalized frequency 6.85, and the amplitude of the peak resonance is reduced by 89.1 percent.

Reduction of Resonant Peak. In between the two linear cases, the constraint force consists of nonlinear friction force and the variable normal load. The significance of the variation of the contact normal load depends on the direction of the resulting relative motion at the shroud contact and the orientation of the contact plane. If the variation of the contact normal load is not significant, the effect of a shroud constraint is not very different from that of a platform damper. This is demonstrated by the attenuation effect of the induced friction on the resonant response of the first vibration mode. Figure 9 shows the tracking curves of the first vibration mode of the shrouded blade system when changing the contact preload. Two sets of curves are shown. The solid curves are the predicted results using the three-terms harmonic balance method and the dashed curves the single-term harmonic balance method. It is seen that as the preload increases, the resonant peak decreases until the minimum response is reached at $n_0 = 2$. Beyond this preload, the damping effect tends to reduce gradually towards the fully stuck case. The preload that gives the minimum response is known as the optimal preload.

Since the induced friction force has super-harmonic components, it is possible that the super-harmonics of the shrouded blade system can be excited and internal resonance can occur. As can be seen in Fig. 9, when the preload is 0, the internal resonance can be observed at normalized frequencies 0.75 and 1.45 based on the results using three-terms harmonic balance method. It appears that

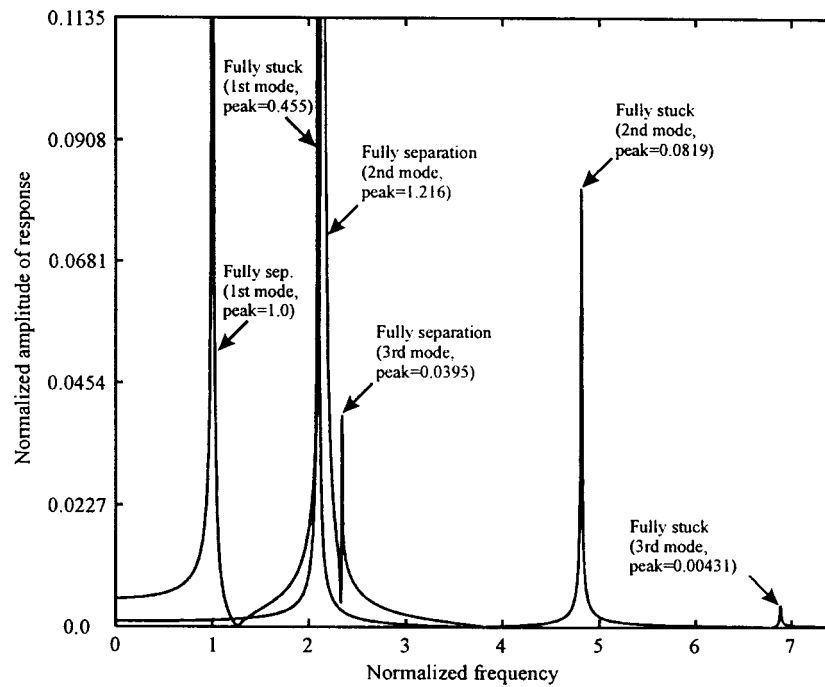


Fig. 8 Periodic response of a shrouded blade system: resonant shift of resonant frequency

the single-term harmonic balance method tends to overestimate the resonant peak although for the first vibration mode the problem is not obvious.

Jump Phenomenon. In addition to its influence on the friction characteristic, the variable normal load can directly impose

nonlinear stiffness on the system. This nonlinear stiffness arises from the intermittent separation of the contact surface during the course of vibration. It has been known that this nonlinearity can result in a multivalued response that can lead to a jump phenomenon [8]. A multivalued response can be obtained by using the

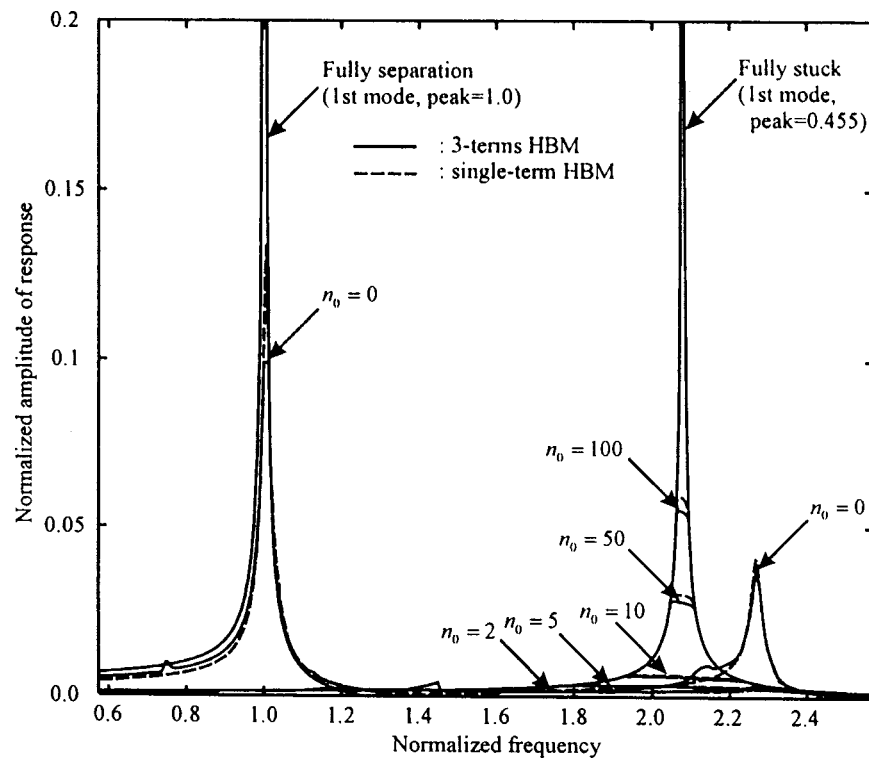


Fig. 9 Periodic response of a shrouded blade system: attenuation of resonant amplitude

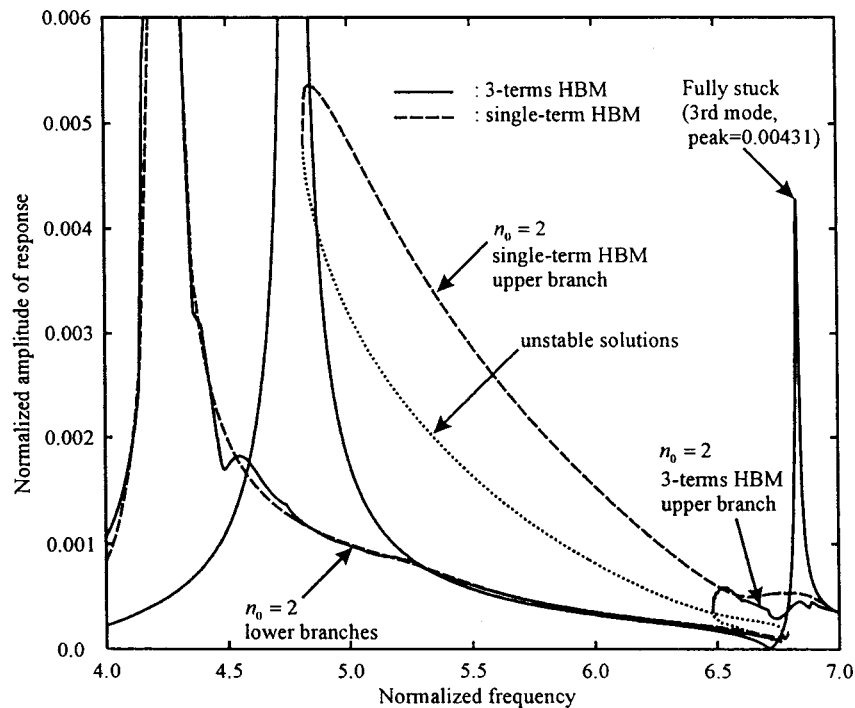


Fig. 10 Periodic response of a shrouded blade system: jump phenomenon at third vibration mode

standard continuation technique [9]. In this study, a jump phenomenon can be observed at the third vibration mode, as shown in Fig. 10, when a moderate preload ($n_0 = 2$) is applied. The increase in the resonant amplitude causes the preloaded interface to separate, and as a result, the interface can not provide stiffness to the system temporarily. The overall effect of the temporary separation is similar to the effect of a “softening spring” that gives rise to the response with a resonance peak bending towards lower frequencies. It should also be noted that one of the multiple solutions from the harmonic balance method shown as the dotted curve is unstable [8]; while separated by the unstable response, the stable response consists of two curves, which are referred to as the upper and lower branches. In this figure, the resonant response predicted by the single-term harmonic balance method is compared to that by the three-terms harmonic balance method. It is seen that the single-term harmonic balance method well overestimates the resonant response at the region near jump. Furthermore, the internal resonance can be clearly observed from the predicted results using three-terms harmonic balance method.

Conclusion

In this paper, a three dimensional shroud contact model is employed to predict the periodic response of blades having three dimensional nonlinear shroud constraint. When subjected to periodic excitation, the resulting relative motion at the shroud contact is assumed to be periodic in three-dimensional space. Based on the three dimensional shroud contact model, analytical criteria are used to determine the transitions between stick, slip, and separation of the contact interface and are used to simulate hysteresis loops of the induced constrained force, when experiencing periodic relative motion. The constrained force can be considered as a feedback force that influences the response of the shrouded blade. By using the multiharmonic balance method along with fast Fourier transform, the constrained force can be approximated by a series of harmonic functions so as to predict the periodic response of a shrouded blade. This approach results in a set of nonlinear

algebraic equations, which can be solved iteratively to yield the periodic response of blades having three dimensional nonlinear shroud constraint.

The predicted nonlinear response shows three distinct features: (1) shifted resonant frequency due to the additional spring constant introduced by the shroud constraint, (2) damped resonant response due to the additional friction damping introduced by frictional slip, (3) multivalued response leading to a jump phenomenon due to intermittent interface separation. The predictive ability of the proposed approach has important implications to the design of the shroud contact. In the design of the shroud contact, the preload is one of the important parameters to control the effectiveness of the shroud contact. Since the attenuation effect of the shroud contact on resonant vibration can be accurately predicted over a wide range of preload using the proposed approach, the designer can achieve the optimal preload to maximize the performance of the shroud contact in dissipating vibratory energy. Moreover, the proposed approach can also facilitate the design of shroud angle, which is another important parameter to be considered in the design of the shroud contact.

Acknowledgment

This material is based on work supported by the GUIde Consortium and the U.S. Air Force under contract No. F33615-96-C-2664. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the authors and do not necessarily reflect the views of the GUIde Consortium or the Air Force.

References

- [1] Meng, C. H., and Yang, B. D., 1998, “Nonlinear Spring Resistance and Friction Damping of Frictional Constraint Having Two-Dimensional Motion,” *J. Sound Vib.*, **217**, No. 1, pp. 127–143.
- [2] Yang, B. D., and Meng, C. H., 1997, “Modeling of Friction Contact and Its Application to the Design of Shroud Contact,” *ASME J. Eng. Gas Turbines Power*, **119**, No. 4, pp. 958–963.
- [3] Griffin, J. H., and Meng, C. H., 1991, “Friction Damping of Circular Motion

- and Its Implications to Vibration Control,” ASME J. Vibr. Acoust., **113**, pp. 225–229.
- [4] Menq, C. H., Chidamparam, P., and Griffin, J. H., 1991, “Friction Damping of Two-Dimensional Motion and Its Application in Vibration Control,” J. Sound Vib., **144**, No. 3, pp. 427–447.
- [5] Menq, C. H., Griffin, J. H., and Bielak, J., 1986, “The Influence of a Variable Normal Load on the Forced Vibration of a Frictionally Damped Structure,” ASME J. Eng. Gas Turbines Power, **108**, pp. 300–305.
- [6] Yang, B. D., and Menq, C. H., 1998, “Characterization of three-dimensional Contact Kinematics and Prediction of Resonant Response of Structures Having three-dimensional Frictional Constraint,” J. Sound Vib., **217**, No. 5, pp. 909–925.
- [7] Yang, B. D., Chen, J. J., and Menq, C. H., 1998, “Prediction of Resonant Response of Shrouded Blades With three-dimensional Shroud Constraint,” ASME paper no. 98-GT-485.
- [8] Thomson, W. T., 1988, *Theory of Vibration with Applications*, Prentice-Hall, Englewood Cliffs, NJ.
- [9] Allgower, E. L., and Georg, K., 1990, *Introduction to Numerical Continuation Methods*, Springer-Verlag, Berlin.

Rotordynamic Force Prediction of Whirling Centrifugal Impeller Shroud Passages Using Computational Fluid Dynamic Techniques

J. J. Moore

A. B. Palazzolo

Department of Mechanical Engineering,
Texas A&M University,
College Station, TX 77843-3123

The demand for higher efficiencies and performance of modern centrifugal turbomachinery requires improved knowledge of critical design factors in strength of materials, aerodynamics, and rotordynamics. While tremendous strides in finite element stress analysis and computational fluid dynamics (CFD) have addressed the first two areas, the lack of accurate prediction tools for centrifugal impellers typically leaves rotordynamics out of the design loop. While several authors have analyzed the rotordynamic forces arising from shrouded centrifugal impellers, there has been no study to couple the secondary shroud passage with the three-dimensional primary flow model. The strong interaction between these domains makes this approach advantageous. The current study utilizes CFD techniques to analyze the full three-dimensional viscous, primary/secondary flow field in a centrifugal pump impeller to determine rotordynamic forces. Multiple quasi-steady solutions of an eccentric three-dimensional model at different precessional frequency ratios yield the rotordynamic impedance forces. Performing a second-order least-squares analysis generates the skew-symmetric stiffness, damping, and mass matrices. The results show good correlation with experiment for both performance and rotordynamic forces. [DOI: 10.1115/1.1385829]

Introduction

Fluid dynamic forces play a significant role in many turbomachines including single and multistage centrifugal compressors and pumps. The trend for more efficient designs with higher power densities have made an even greater challenge to the modern rotordynamicist. Much of the effort have focused on bearings and seals providing good design tools for these components. Centrifugal impellers typically play a strong role in the resulting stability of multi-stage machines. However, the source and mechanism of these destabilizing forces are probably the least understood. Many designers are forced to rely on empirically based guidelines such as those provided by Wachel et al. [1] Bolleter et al. [2], and Jery et al. [3]. The goal of this study is to improve the prediction of rotordynamic forces arising from the secondary flow passage of shrouded centrifugal impellers using modern CFD techniques.

Review of Experimental Data. Jery et al. [3] investigated the impedance forces from a three-dimensional centrifugal pump impeller ($R_2=16.2$ cm, $\beta_2=-65$ deg, five blades, $N_s=0.57$) over a range of precessional frequency ratios from -1.0 to 1.0 . A trapezoidal volute with no diffuser was utilized. The resulting curves match a parabolic shape, yielding negative direct stiffness (K) and positive cross-coupled stiffness (k). The shroud clearance in this data is unrealistically high compared to most turbopump designs and underestimates the rotordynamic contribution from the shroud. Adkins and Brennen [4] indicate that shroud accounts for 80 percent of the direct stiffness of the entire impeller force

and 33 percent of the cross-coupled stiffness; however, these percentages will be higher for typical shroud clearances. Clearances of 2.5 to 9.0 mm are observed in four modern rocket engine turbopumps including the space shuttle main engines (SSME) ([5]).

Ohashi et al. [6] presented additional data for a two-dimensional and three-dimensional impeller ($R_2=33$ and 35 cm, $\beta_2=-68$, and -66 deg, six blades, respectively) in a vaneless diffuser and a three-dimensional impeller with vaned diffuser. Both the two-dimensional and three-dimensional data with vaneless diffuser showed essentially no destabilizing force (k) at the design flow. The results of the three-dimensional impeller (w/large shroud clearance) showed that the fluid forces with the vaned diffuser is about 50 percent greater than with a vaneless diffuser but still resulted in zero cross-coupled stiffness. The effect of shroud clearance was investigated by adding a spacer yielding a 2.7–3.0 mm shroud clearance. The tighter clearance raised the precessional frequency ratio at the stability threshold (WFR) from 0 to about 0.5. According to this data, the shroud is the dominant source of direct and cross-coupled stiffness for impellers with typical shroud clearances.

Bolleter et al. [7] utilized an impeller test rig based on a horizontal “rocking-arm” principle to test a 13.8 in. (350 mm) boiler feed pump impeller ($n_q=33$) with vaned diffusers. In order to eliminate the effect of the wear-ring seal, a face seal was employed. These authors [2] went on to test three additional impeller configurations and investigate the effect of impeller to shroud wall clearance at entrance (so called “Gap A”) and the impeller to diffuser vane clearance (“Gap B”). At the same speed with design flow, Bolleter et al. [7] shows three times the cross-coupled stiffness (nondimensional) compared to the data of Jery et al. [3] clearly due to the smaller shroud clearance. The data of Bolleter et al. [7] is utilized in the present study since it represents a more common impeller geometry and is accompanied with detailed performance measurements.

Guinzburg [8] presents results for a simulated shroud only pas-

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-334. Manuscript received by IGTI, October 1998; final revision received by the ASME Headquarters, March 1999. Associate Editor: D. Wisler.

sage using an external pump loop and tests for varying precessional frequency ratios, shroud and exit seal clearances, whirl radii, leakage flow rates, and speeds. Sivo et al. [9] continues this work by investigating the effect of meridional swirl brakes installed in the test section. Uy [5] makes similar measurements for a curved shroud passage.

Review of Theoretical Studies. Many authors have focused on the impeller/diffuser interaction including Colding-Jørgensen [10] using two-dimensional potential flow theory, Adkins et al. [4] using a quasi one dimensional model including unsteadiness, and Fongang et al. [11] with an improved potential flow model. A recent study utilizing an unsteady two-dimensional CFD calculation was performed by Torbergsen and White [12] and makes fluid dynamic comparisons to previously published data. While rotor-dynamic impedance forces were calculated, no comparison is made to experiment.

Other authors have studied the rotordynamic forces arising from only the shroud passage. Childs [13] expanded his incompressible, turbulent, concentric bulk flow seal model for constant radius to a variable radius model by casting the axial momentum in the pathwise, meridional direction. The resulting radial impedance curves generated by the bulk flow analysis showed substantial “resonances” for inlet swirls exceeding 0.5. The resonant peaks are attributed to the centrifugal acceleration term in the path momentum equations. Comparisons are made to the results of Bolleter [7] as will be shown later.

Contrary to the assumptions built into the bulk-flow theory, the shroud region typically contains many recirculation zones. Several authors have utilized the capabilities of computational fluid dynamics (CFD) to model the shroud region. Baskharone, Daniel, and Hensel [14] also modeled the isolated impeller shroud region for the Sulzer impeller (with face seal). Using a three-dimensional model, the discretized finite element flow governing equations are perturbed algebraically using a virtual eccentricity. A single-block grid was utilized that only includes only a portion of the primary flow path (up and down stream of the impeller) and results in a grid with high skewness requiring radiused corners. These results will also be compared with the present study Wyman [15] eliminates the need to prescribe boundary conditions leading into the shroud region by including the primary flow path in the model. However, only a two-dimensional model is developed and no rotordynamic coefficients are presented.

Several other authors have utilized CFD for modeling annular seals including Dietzen and Nordman [16], Rhode, Hensel, and Guidry [17] and Arghir and Frêne [18]. All of these authors employ a coordinate transformation from the three-dimensional eccentric domain into a two-dimensional axisymmetric one. These techniques require a zeroth-order solution and first-order calculations at the different precessional frequency ratios. The procedure is efficient but the coordinate transformation is only valid for constant radius seal geometries preventing the modeling of impeller shrouds. The CFD code SCISEAL, developed by Athavale et al. [19], utilizes a three-dimensional whirling method and solves for the quasi-steady flow field in an eccentric three-dimensional annular passage. This approach is more computationally intensive but is more general in the class of problems that may be solved. Moore and Palazzolo [20] use this code to make comparison to three-dimensional laser Doppler anemometer and rotordynamic measurements for grooved liquid annular seals. Their results show the κ - ε turbulence model with wall functions yielding close prediction of mean velocity profiles and reasonable calculation of rotordynamic force coefficients.

The present study utilizes a similar three-dimensional whirling method to analyze a three-dimensional eccentric shroud model that is coupled to the primary impeller flow. Being a steady solution, significantly less CPU time is required compared to unsteady approaches. Potentially the results of an impeller/diffuser interaction model could be added to the current shroud model to obtain the total force acting on the impeller.

Computational Fluid Dynamics Code Description

This study utilizes the commercial CFD code TASCflow from AEA [21]. The Reynolds-averaged Navier-Stokes equations (shown in Eq. (1) using index notation) are solved in strong-conservative form utilizing an algebraic multigrid solver and is applicable to both incompressible and compressible flows. The code allows solutions for both rotating and stationary frames of reference, using body-fitted structured grids of multidomains.

$$U_j \frac{\partial U_i}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} (-P \delta_{ij} + 2\mu S_{ij} - \overline{\rho u_i u_j}) \quad (1)$$

Both compressible and incompressible flow fields may be modeled using the standard κ - ε turbulence model. The turbulent Reynolds stresses are approximated using the Boussinesq eddy viscosity principle given in Eq. (2).

$$-\overline{\rho u_i u_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho \kappa \delta_{ij} \quad (2)$$

The turbulent, or eddy, viscosity is related to the turbulent kinetic energy (κ) and the dissipation rate (ε) by

$$\mu_t = \rho C_\mu \frac{\kappa^2}{\varepsilon} \quad (3)$$

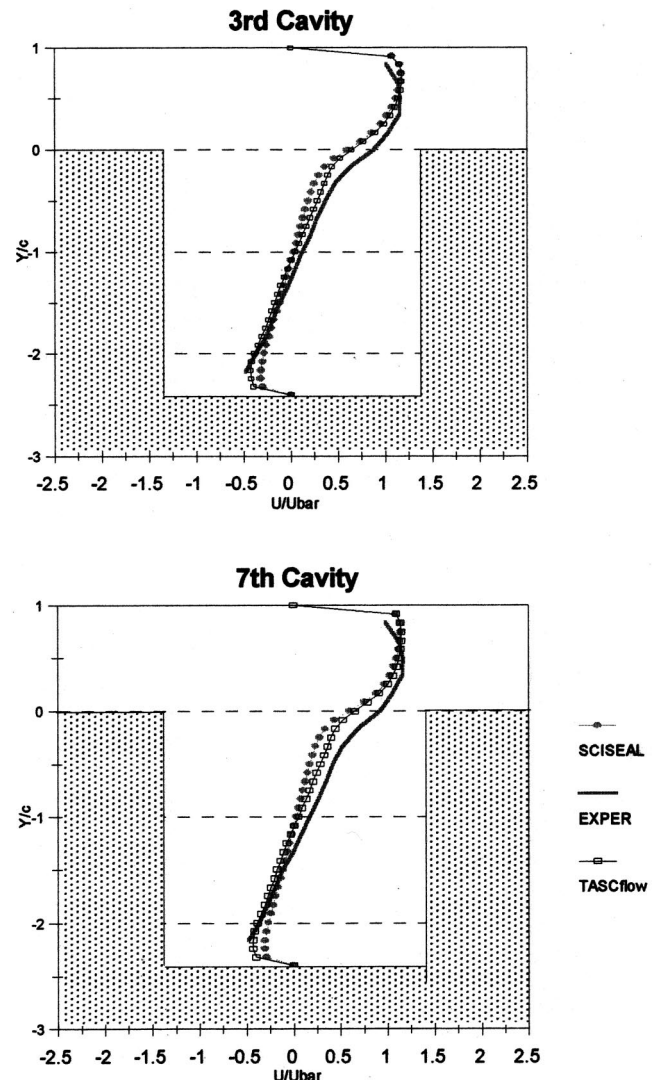


Fig. 1 Axial velocity profiles, grooved liquid annular seal

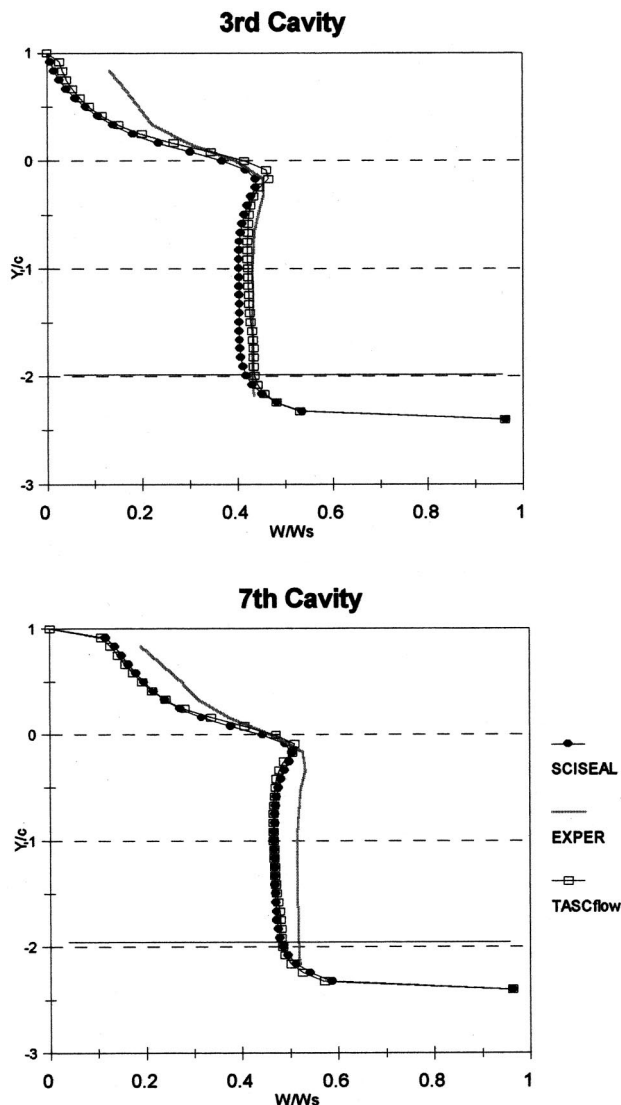


Fig. 2 Circumferential velocity profiles, grooved liquid annular seal

where the turbulent kinetic energy is defined as

$$\kappa = \frac{1}{2} u_k u_k. \quad (4)$$

The law of the wall formulations models the sharp velocity gradients near the wall

$$u^+ = y^+ \quad \text{for } y^+ < 11.5 \quad (5)$$

$$u^+ = \frac{U_\tau}{K} \ln(E y^+) \quad \text{for } y^+ > 11.5 \quad (6)$$

where

$$y^+ = \frac{y U_\tau}{\nu}, \quad u^+ = \frac{u}{U_\tau} \quad (7)$$

and

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}. \quad (8)$$

In order to obtain accurate wall shear stress, care is taken to ensure the near-wall grid points are placed in the logarithmic region (ideally $20 \leq y^+ \leq 200$). A full description of the theoretical as-

pects of the code may be obtained from AEA [21]. Raw [22] provides a complete description of the multi-grid solver.

CFD Comparison to three-dimensional LDA Measurements

In order to validate the code for annular flows, comparisons are made to the three-dimensional laser Doppler anemometer (LDA) measurements of Morrison et al. [23] for a grooved liquid annular seal. This flow field is Couette in nature (due to the spinning journal) and contains regions of strong recirculation and high turbulence intensity. The seal tested is an eight-tooth, teeth-on-rotor design (radius=82.05 mm, clearance=1.27 mm, tooth width=1.524 mm, and cavity=3.048 mm square) spinning at 3600 rpm with a leakage rate of 4.86 1/s. The grid used is identical to that used by Moore and Palazzolo [20] who verified grid independence, allowing direct comparison to both experiment and SCISEAL predictions.

Figure 1 compares the axial velocity profiles at a section in the middle of the third and seventh seal cavities. The ordinate represents the radial position (Y) nondimensionalized with the clearance (c), while the abscissa is the axial velocity nondimensionalized by the mean velocity in the through-flow jet ($U_{bar} = 7.4$ m/s). The through-flow jet, shear layer, and recirculation zones are clearly evident in this plot. The CFD computation predicts a steeper boundary layer near that stator (top) wall but yields essentially identical peak velocities as the experiment. TASCflow shows slight improvement over SCISEAL in capturing the strength of the recirculation in the cavity. Figure 2 shows similar plots at the same sections but for circumferential (swirl) velocity. Accurate prediction of swirl velocity is crucial to successful rotordynamic prediction. In the third cavity, both codes demonstrate good swirl prediction for both magnitude and distribution, with TASCflow showing slightly better agreement. By the seventh cavity, both codes slightly under predict the measured swirl. This study demonstrates that good mean flow prediction is possible using the standard κ - ϵ turbulence model with wall functions, even for swirling annular flows with strong recirculations.

Rotordynamic Solution Technique

A software interface to the TASCflow code is developed to generate the perturbed mesh and post process the results, yielding rotordynamic information. A full three-dimensional combined primary/secondary flow model is built using a multiblock body-fitted mesh and solved by performing a multiple frame of reference (MFR) solution, where a different frame of reference is used in different parts of the domain. A sliding interface is required at the boundaries between these domains.

The whirling, unsteady problem is transformed into a steady one by solving the three-dimensional eccentric flow field in the frame of reference attached to the whirling rotor (Fig. 3). In this rotating frame, the stator wall moves in the opposite direction (for positive whirl). The rotor surface moves with, against, or not at all relative to the whirling frame of reference, depending on the precessional frequency ratio (PFR) defined as the ratio of rotor whirl to rotor spin. Quasi-steady solutions are obtained at each PFR. A PFR equal to unity is termed synchronous whirl, where the shroud is whirling at the same frequency it rotates. A PFR of zero indicates a static displacement of the shroud, which then simply spins.

The choice of eccentricity is arbitrary but is typically kept near 10 percent of the shroud clearance to capture the linear, small motion characteristics. This value was arrived at based on previous modeling of annular seals ([20]). Larger eccentricities are possible and may be utilized to predict non-linear characteristics. The eccentricity vector is along the Y -axis.

A solution is obtained at multiple PFR values typically ranging from 0.0 to 2.0. The Cartesian area components of each cell face in the grid are stored. The X and Y area components on the (outer) surface of the shroud are multiplied by the local static pressure then summed for all cells on the shroud surface yielding F_l and F_n , respectively, as shown in Eqs. (9) and (10).

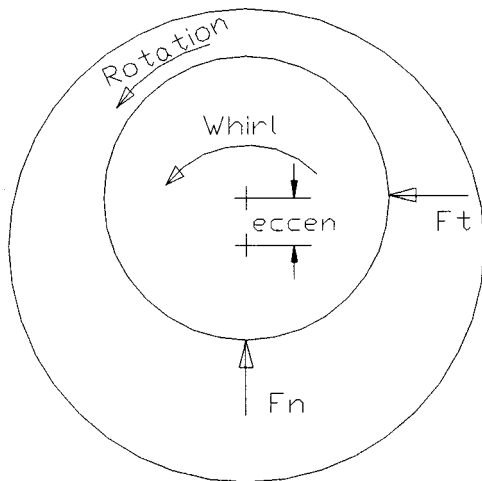


Fig. 3 Schematic of whirling rotor

$$F_n = \sum_{\text{shroud}} P A_y \quad (9)$$

and

$$F_t = \sum_{\text{shroud}} P A_x \quad (10)$$

A similar procedure is performed on all surfaces of the primary impeller passage (including blades) resulting in impedances for the impeller, showing the effect from shroud circumferential pressure distributions. The shear stress contribution at the wall contributes less than 1 percent of the total impedance force and is ignored. These impedance forces may be normalized by ϵ yielding

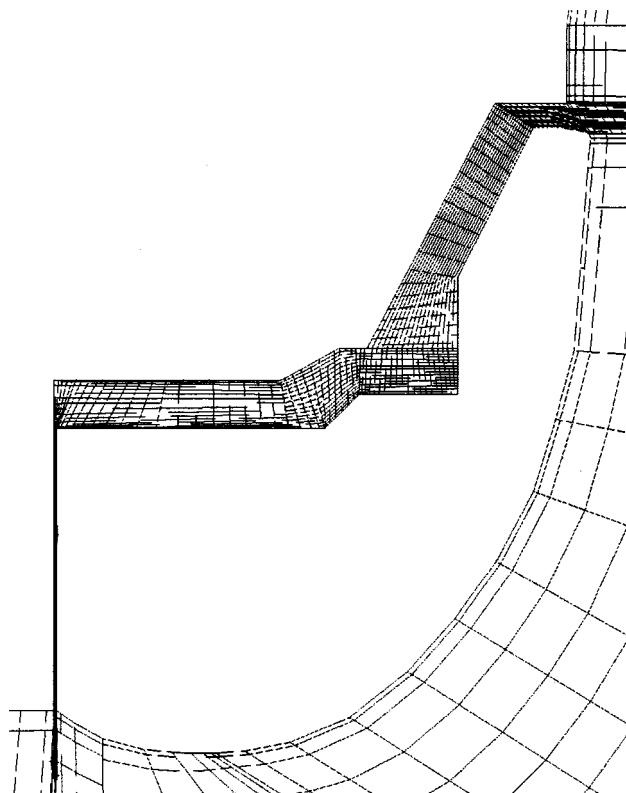


Fig. 5 Mesh of shroud region

the impedance both normal and tangential to the whirl orbit. A comparison of these impedance forces to a linear, second-order model is shown in Eqs. (11) and (12).

$$\bar{F}_a = \frac{F_a}{\epsilon} = -K - c\Omega + M\Omega^2 \quad (11)$$

$$\bar{F}_t = \frac{F_t}{\epsilon} = k - C\Omega - m\Omega^2 \quad (12)$$

Each of the above equations represent one equation and three unknowns. The force coefficients are evaluated by determining the impedance force at multiple precessional frequencies (Ω). A minimum of three is required. For improved accuracy over a wide range of precessional frequencies, more than three are calculated (seven for this study), and a curve-fit to the linear second-order model is performed. The coefficients of the curve-fit yield the impeller's stiffness, damping, and mass force coefficients (K, k, C, c, M, m). The generality of this approach allows modeling of all variety of geometries including both seals and impeller shrouds.

Since the shroud region is solved in the whirling frame of reference (Ω) while the primary impeller passage is always solved in the rotating frame (ω), a sliding interface is employed. There are several possible locations of the sliding interface, but the best results were achieved placing the sliding interface across the primary flow just up and downstream of the impeller (see Fig. 4). Therefore, the inlet region leading into the impeller, face seal, shroud region, and diffuser section are all solved in the whirling frame of reference. The bladed impeller region is always solved in the rotating frame of reference (synchronous with impeller rotation).

The shroud region is made up of two grid blocks to minimize grid skewness and optimize node placement as shown in Fig. 5. The face seal, inlet region, and diffuser section each contain a separate grid block all connected using a structured interface. Since only shroud forces are of interest in this study, only the

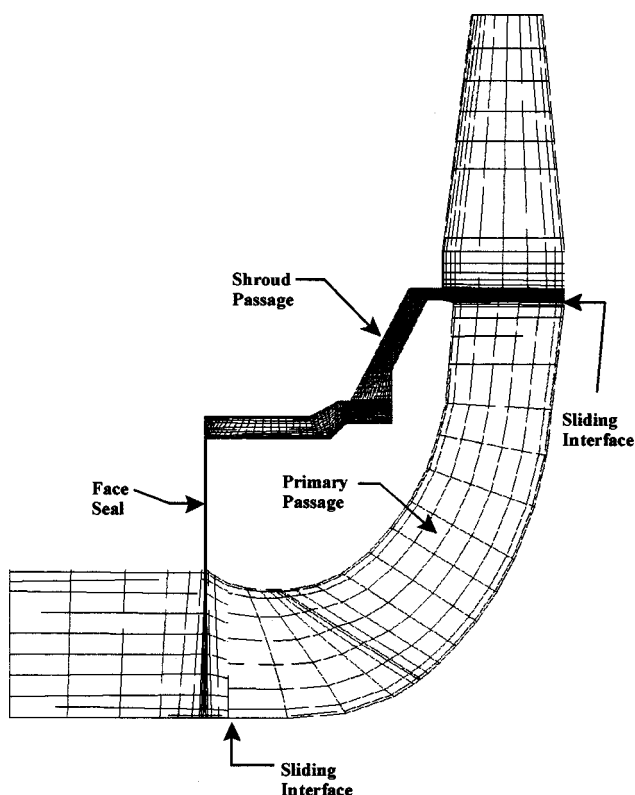


Fig. 4 Meridional plane of combined primary/secondary model

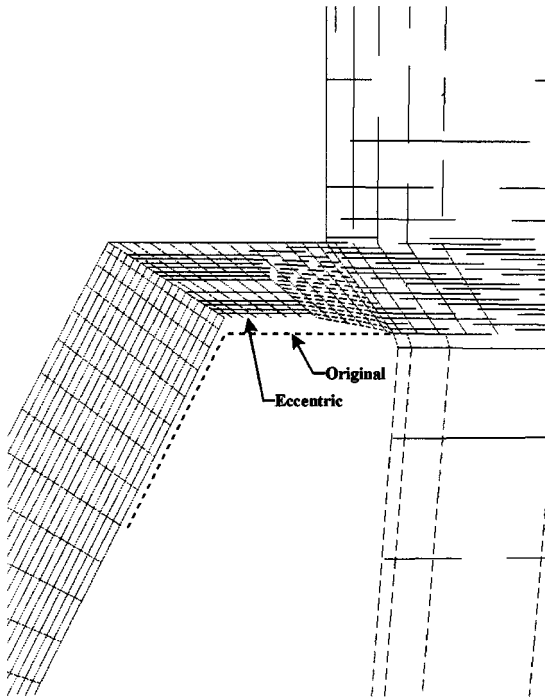


Fig. 6 Interface with impeller exit showing eccentric shroud

shroud region is made eccentric (see Fig. 6). As assumed by previous authors, the diffuser vanes are ignored in this model. The diffuser width is adjusted to mimic the actual flow area in the diffuser.

Boundary Conditions. Total pressure is specified at the inlet while velocity is specified (in absolute coordinates) at the diffuser exit and is done to ensure a circumferentially uniform flow field. The values are specified in cylindrical coordinates based on averaged velocities obtained from the centered case ($\epsilon=0$) using a mass flow boundary condition. Any nonuniformities in the model besides the eccentric shroud will yield false radial forces and affect the rotordynamic force prediction. Smooth, no-slip boundary conditions are assumed at all walls using wall functions.

TASCflow offers two sliding interface options: stage and frozen rotor. Stage performs circumferential averaging of the flow quantities, while frozen rotor performs a local transformation at each interface node. Strict conservation of all flow properties is maintained with both methods. The frozen rotor approach was chosen for both sliding interfaces so as to not artificially constrain the circumferential pressure field and is used exclusively in this study. This sliding interface does not require one-to-one matching of nodal points simplifying grid construction.

The spinning wall in the shroud region requires special treatment. Its relative rotation velocity (as observed in the whirling frame of reference) is a function of the precessional speed:

$$\omega_{\text{shroud}} = \omega - \Omega. \quad (13)$$

Its rotation vector is displaced by the eccentricity vector and is $(0, 0.6e-3, -1)$, since an eccentricity (ϵ) of 0.6 mm in the Y-direction is used for this study and impeller rotation is about the -Z-axis.

Eccentric Grid Generation. Accurate rotordynamic prediction requires a quality eccentric grid. First, a two-dimensional axisymmetric mesh in the meridional (R - Z) plane is created as shown in Fig. 7. Nodes are distributed in the body fitted mesh to adequately resolve the important flow details. Care is taken in placement of the near wall nodes so as to not violate wall function criteria ($y^+ > 11.5$). This two-dimensional grid is saved in a

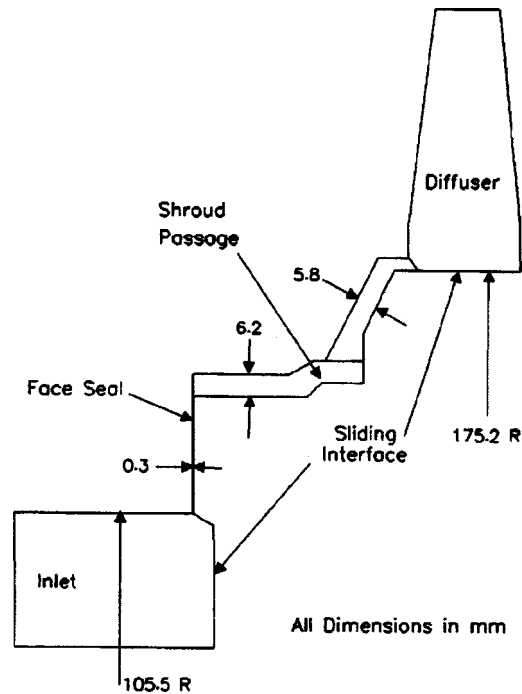


Fig. 7 Layout of shroud model

PLOT3D format and read by a FORTRAN code. The grid is made eccentric by a specified value using control points in the grid coordinate directions (i and j). Only the three grid blocks representing the shroud are made eccentric. Appropriate transition regions are required near the impeller tip (Fig. 6). For more detail on the grid eccentricity algorithm, see Bhattacharya [24].

The impeller model is created with Turbogrid ([21]) using geometry information obtained from Sulzer Brothers, Ltd. The edge of the grid is centered between the impeller blades and represents an exact periodic section (360 deg/seven blades). Six additional grids are created from this grid by rotation about the spin axis ($-Z$) to create a full 360 deg impeller model (Fig. 8). The impeller blades are created by “blocking out” cells in those regions. Although appearing coarse, this particular grid is the result of a grid density study using a single impeller passage, where the predicted head rise is compared to measurements. A coarse, medium, and finer shroud grid is utilized in this study with identical impeller grids and is summarized in Table 1.

Note that the I, J, K local coordinate directions for the impeller are pathwise (I), circumferential (J), and cross-path (K), while all other blocks are pathwise, cross-path, and circumferential. All

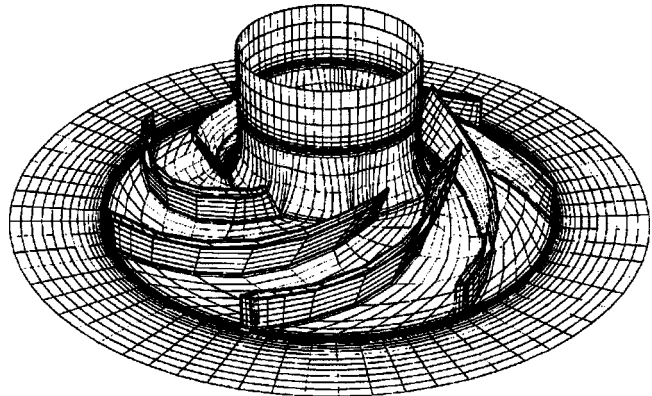


Fig. 8 360-deg seven-bladed impeller model

Table 1 Summary of computational grids

Grid Block	Coarse (C1)	Medium (Med)	Fine (F1)
Inlet	15×12×50	15×12×70	15×12×70
7 imp. blocks	22×15×10	22×15×10	22×15×10
Diffuser	28×13×50	28×13×70	34×13×70
Upper shroud	23×13×50	23×13×70	36×19×70
Lower shroud	38×13×50	38×13×70	56×17×70
Face seal	16×6×50	16×6×70	16×6×70
No. of nodes	94750	123410	187880
CPU time per PFR (min)	152	175	263

three models of reasonable in size for modern workstations. The CPU time listed is for an SGI MIPS 10000, 195 MHz processor. Notice that the CPU times varies linearly with model size, a favorable characteristic of the multigrid solver.

Description of Experimental Setup

Bolleter et al.[2] utilize a hydraulic exciter that inputs a swept sine into the system. Impeller forces are measured using a six-bridge strain gauge arrangement that is fed into a rotating amplifier and out of the rotor through slip rings. Impedance functions are determined using the measured response. Since vaned diffusers are used (with volute far down stream), symmetry of the force coefficients is assumed. Synchronous time averaging (STA) is employed to reduce the effect of forces not coherent with the rotor motion. The impedance functions are determined from the STA forces and displacements. A frequency range from 10–150 percent of running speed is used. Data is sampled at 32 points/rev for 32 revolutions, and 100 averages are used. Tests are performed at 2000 rpm with cold water (30°C) and is the data utilized in the present study. Other tests are performed at higher speeds (4000 rpm) and water temperatures up to 160°C. Table 2 gives impeller dimensions and characteristics.

Performance Predictions

While the total head rise (impeller+diffuser) was measured to be 68 meters, the head rise across the impeller is 48.25 m. Table 3 compares the predicted head with experiment.

The predicted head correlates well with experiment for all three grids. The experimental efficiency is a stage value including both the impeller and diffuser, while the prediction is just for the impeller. The average inlet swirl ratio (W_{rat}) at the entrance to the shroud (impeller tip) compares well to measured Pitot tube measurements and changes little after the medium grid.

Table 2 Sulzer Impeller Data

Geometry Information	Dimension
Impeller blade tip diameter	350 mm
Impeller shroud diameter	350.4 mm
Impeller discharge width	28 mm
Number of impeller blades	7
Rotation speed	2000 rpm
Design flow	130 l/s
n_q	33

Table 3 Sulzer performance data

Grid	Impeller Isentropic Head (m)	Isentropic Efficiency (%)	Avg. Tang. Swirl (Shroud Inlet)
Coarse grid	49.02	93.1	0.52
Medium grid	49.11	93.2	0.51
Fine grid	49.08	93.20	0.52
Experiment	48.25	84	~0.5

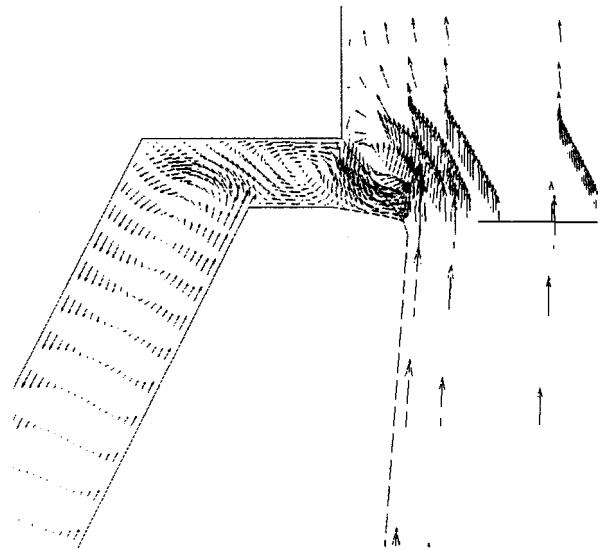
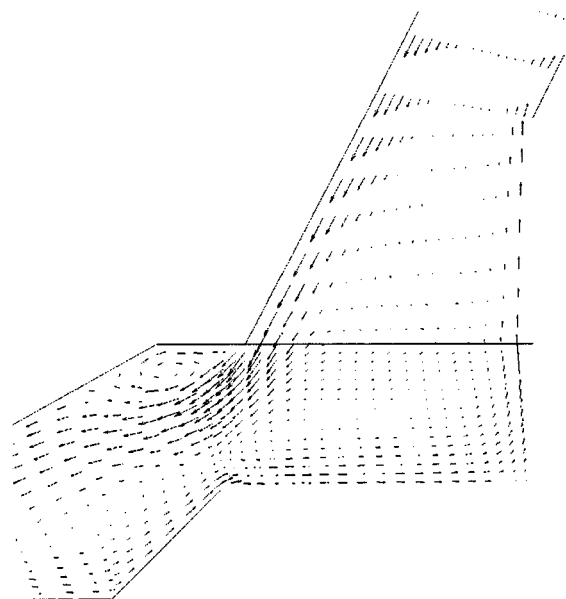
**Fig. 9 Computed velocity vectors at impeller tip (F1 grid)**

Figure 9 is a vector plot in the meridional plane at the impeller tip and demonstrates the complex flow field at the shroud entrance, showing the presence of several recirculation zones. The viscous pumping action near the shroud wall causes reverse flow toward the tip, while pressure forces drive the flow downward near the stator wall. Figures 10 and 11 show several more recirculation zones near the “elbow” of the shroud and at the face seal entrance. The relative velocities from each domain are transformed into the absolute (fixed) frame of reference. Figure 12 plots the absolute tangential swirl contours. Notice that the swirl velocity is a maximum near the tip of the shroud and decreases while traveling down the shroud path (toward the seal). This decrease conflicts with free vortex principles (conservation of angular momentum), which predicts an increase in swirl velocity as the radius is decreased. Viscous effects from the stator wall cause this deceleration despite the shroud rotation promoting swirl. Clearly, any abstraction (i.e., inviscid or bulk-flow assumptions) that ignores the viscous effects in the shroud will not capture the true physics of the flow field.

**Fig. 10 Velocity vectors at shroud elbow (F1 grid)**

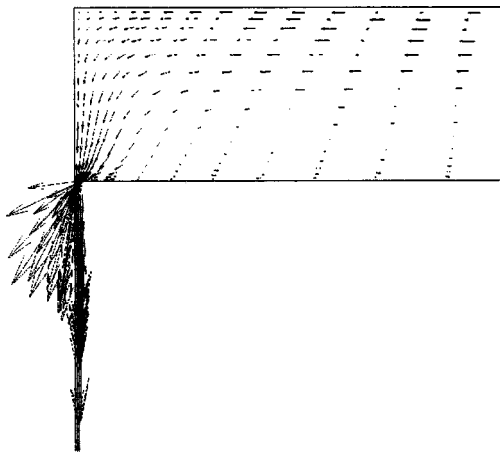


Fig. 11 Velocity vectors at seal entrance (F1 grid)

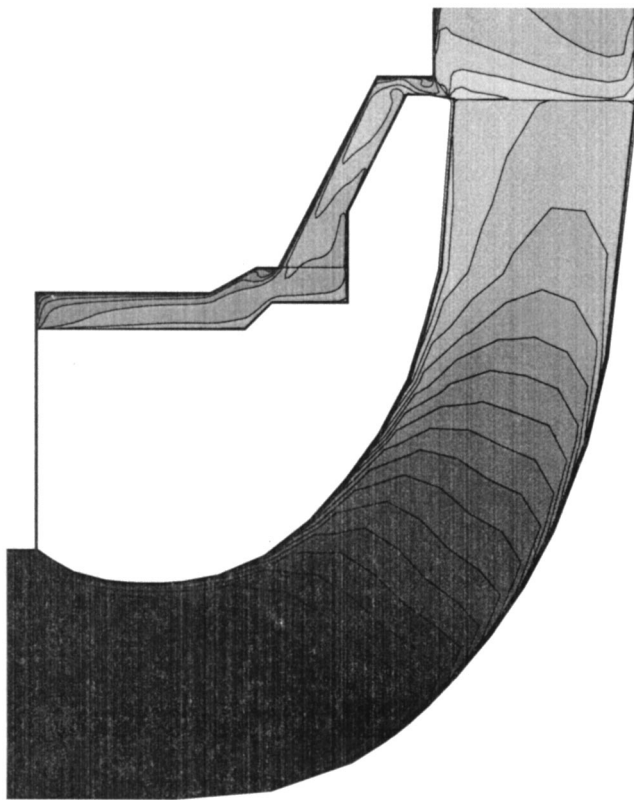


Fig. 12 Absolute tangential velocity contours (F1 grid)

Rotordynamic Prediction

To validate the pressure integration algorithm and to verify full convergence, a zero-eccentricity case was first run to confirm that the impedance forces are indeed zero. The shroud is then made eccentric and recalculated. The pressure distribution at the entrance to the impeller is plotted in Fig. 13 and shows the fundamental pressure field giving rise to the rotordynamic forces plus a higher frequency (7X) due to the wakes off the impeller blades. The predicted impedance force for the fine grid (F1) (at PFR values of 0, 0.25, 0.5, 0.75, 1.0, 1.5, and 2.0) along with the curve-fits for all three grids at are plotted in Fig. 14. Mesh independence is established for the medium grid density, which shows identical results with the fine grid. The plot shows reasonable Fr prediction at the lower frequencies but underpredicts the radial

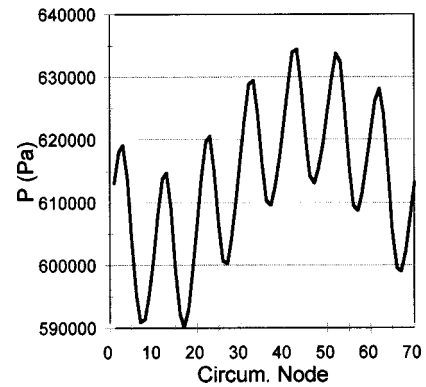


Fig. 13 Shroud inlet pressure distribution

impedance at higher PFR's, since the large inertial of the fluid inside the primary passage of the impeller is not considered. The tangential impedance shows good correlation at the subsynchronous precessional frequencies ($PFR < 1$). These results are in agreement with the experimental investigation of Ohashi et al.[6] which demonstrated most of the direct and cross-coupled stiffness arises from the shroud passage. The predicted PFR at the neutral stability point ($WFR = 0.62$) is identical with experiment. The rotordynamic coefficients resulting from these curve-fits are summarized in Table 4 and are compared to those of Baskharone et al.[14] and Childs [13].

The contribution of the impeller (due to eccentric shroud) is relative small but does further improve the predictions. The results show good stiffness prediction and slightly over predictions the

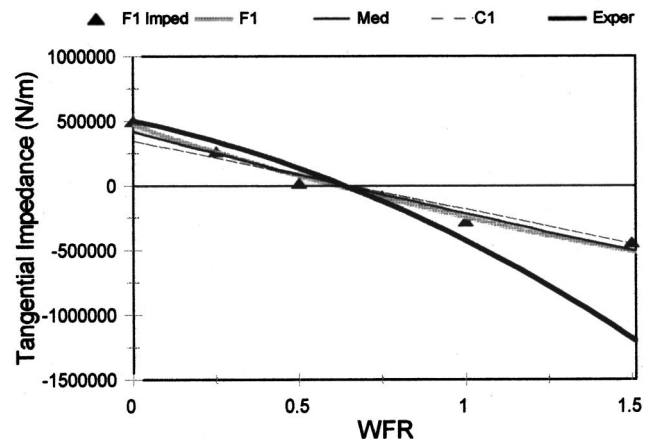
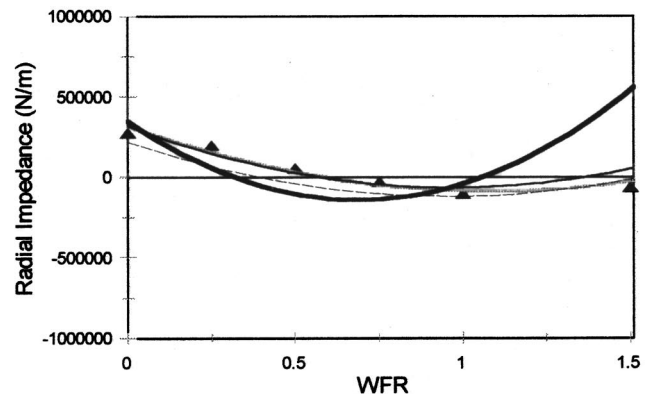


Fig. 14 Effect of grid density of impedance curves

Table 4 Summary of predicted rotordynamic force coefficients

Mesh/Components	K KN/m	k KN/m	c KN-c/m	c KN-c/m	M kg	m kg
Coarse grid (shroud only)	-180	370	2.80	3.04	8.11	-1.28
Coarse grid (shroud+imp)	-223	350	2.52	3.37	8.34	0.07
Med. grid (shroud only)	-286	448	3.59	3.62	9.30	-2.67
Med. grid (shroud+imp)	-323	420	3.26	3.79	9.32	-1.05
Fine grid (shroud only)	-284	501	4.40	3.35	7.77	-4.55
Fine grid (shroud+imp)	-324	471	4.05	3.59	7.92	-2.92
Experiment	-353	506	2.58	6.80	23.6	8.85
Experimental uncertainty	22	24	0.16	0.17	0.56	0.62
Baskharone et al. [14]	-268	165	1.64	2.92	5.03	3.03
Childs [13] ($W_{rat} \approx 0.5$)	-42	288	2.02	2.29	8.96	-0.009

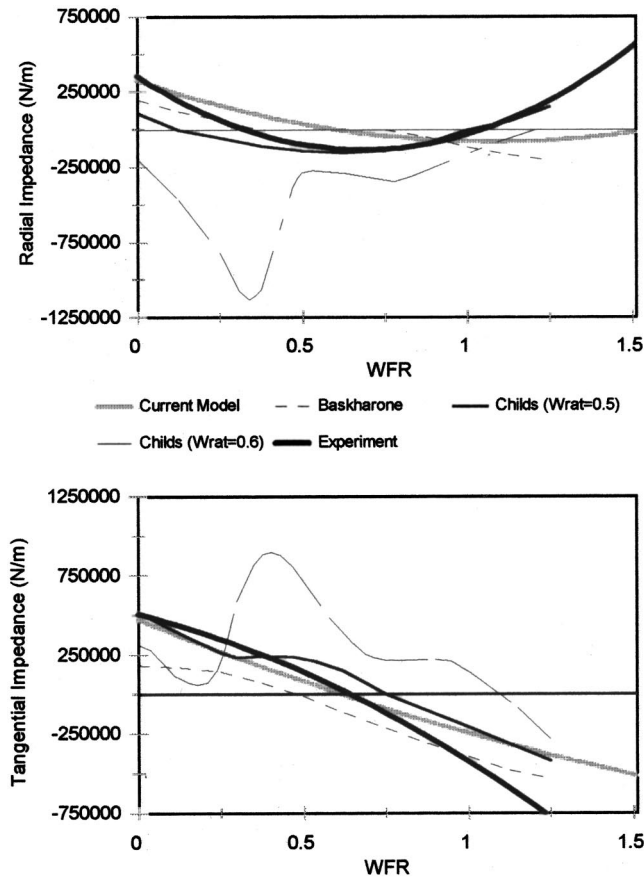


Fig. 15 Comparison of impedance curves

direct damping. The cross-coupled and inertial coefficients are under predicted since whirling forces from the impeller primary flow passage are not considered.

Figure 15 compares the F1 impedance curves with those of Baskharone and Childs. Overall, an improvement over these previous studies is realized especially at the lower precessional frequencies. The fluctuating curve represents the bulk-flow model with inlet swirl ratio (W_{rat}) equal to 0.6. These “resonances” previously described appear to be an anomaly unique to the bulk-flow model.

Summary and Conclusions

A commercial CFD code is adapted to solve for rotordynamic coefficients in a shrouded pump impeller. The code demonstrates close prediction of mean velocity profiles in the complex flow field of a grooved annular (labyrinth) seal. A finite whirl analysis is performed on a shrouded pump impeller by solving the coupled primary/secondary flow field at different precessional frequencies.

Integration of the resulting pressure field in both the shroud and impeller regions yields the rotordynamic impedances. A curve-fit to a linear second-order model yields the rotordynamic force coefficients. The coupled model captures the complex flow field entering the shroud at the impeller tip, eliminating the need to specify these boundary conditions. Comparisons to the experimental results of a boiler feed pump impeller show good overall correlation especially in the subsynchronous regime, which is important for good rotordynamic stability assessment. The 360 deg model employed here may be easily generated from traditional “pie” models allowing a full concurrent design approach with the performance, structural, and rotordynamic analysis.

Acknowledgment

The authors would like to thank Marshall Space Flight Center for funding this research. Special thanks also goes to Mr. Arno Frei at Sulzer Brothers, Ltd., for providing additional details of the experimental impeller testing.

Nomenclature

- A_x, A_y = Cartesian area components of cell face
- $\bar{E}$ = surface roughness parameter=9.0
- F_n, F_t = normal (radial) and tang, reaction force (N)
- $\bar{F}_n, \bar{F}_t$ = normal (radial) and tang, impedance (F/ϵ) (N/m)
- H = pump head (ft or m)
- K, C, M = direct stiffness, damping, mass (N/m, N-s/m, kg)
- k, c, m = cross-coupled stiffness, damping, mass (N/m, N-s/m, kg)
- N = speed (rpm)
- $N_s = NQ^{1/2}/H^{3/4}$ = specific speed (using rpm, gpm, ft)
- n_q = specific speed (using rpm, m³/s, m)
- P = local static pressure (Pa)
- Q = flow (gpm or m³/s)
- R_2 = impeller tip radius (m)
- Re = Reynolds number= $2\rho U_c/\mu$
- S_{ij} = mean strain rate [$1/s$]= $1/2(\partial U_i/\partial x_j + \partial U_j/\partial x_i)$
- u = velocity component tangent to wall (m/s)
- u_i = fluctuation velocity component (m/s)
- U_i = mean velocity component (m/s)
- U_τ = shear velocity (m/s)
- PFR = precessional frequency ratio= Ω/ω
- WFR = PFR at the neutral stability point
- y = distance from wall to first nodal point (m)
- β_2 = exit blade angle (meas. from radial)
- δ_{ij} = Kronecker delta
- ϵ = eccentricity (m)
- ϵ = turbulent kinetic energy dissipation (m²/s³)
- κ = turbulent kinetic energy (m²/s²)
- K = von Kármán constant=0.4
- μ = molecular viscosity (Pa-s)
- μ_t = turbulent (eddy) viscosity (Pa-s)
- τ_w = wall shear stress (Pa)
- ω = rotation speed (rad/sec)
- Ω = precessional speed (rad/sec)=PFR $\cdot\omega$

References

- [1] Wachel, J. C., and von Nimitz, W. W., 1981, "Ensuring the Reliability of Offshore Gas Compressor Systems," *J. Pet. Technol.*, **33**, No. 11, pp. 2252–2260.
- [2] Bolleter, U., Leibundgut, E., and Sturchler, R., 1989, "Hydraulic Interaction and Excitation Forces of High Head Pump Impellers," *Pumping Machinery-1989*, Proceedings of the Third Joint ASCE/ASME Mechanics Conference, La Jolla, CA, NASA Science and Technical Branch, pp. 187–194.
- [3] Jerry, B., Acosta, A. J., Brennen, C. E., and Caughey, T. K., 1984, "Hydrodynamic Impeller Stiffness, Damping, and Inertia in the Rotordynamics of Centrifugal Pumps," *Rotordynamic Instability Problems in High-Performance Turbomachinery*, NASA CP2338, proceedings of a workshop held at Texas A & M University, NASA Science and Technical Branch, pp. 137–160.
- [4] Adkins, D. R., and Brennen, C. E., 1988, "Analysis of Hydrodynamic Radial Forces on Centrifugal Pump Impellers," *ASME J. Fluids Eng.*, **110**, No. 1, pp. 20–28.
- [5] Uy, R. V., 1998, "Studies of Rotordynamic Forces Generated by Annular Flows," Ph.D. dissertation, California Institute of Technology, Mechanical Engineering Department.
- [6] Ohashi, H., Sakurai, A., and Nishihama, J., 1988, "Influence of Impeller and Diffuser Geometries on the Lateral Fluid Forces of Whirling Centrifugal Impeller," *Rotordynamic Instability Problems in High-Performance Turbomachinery*, NASA CP3026, proceedings of a workshop held at Texas A&M University, NASA Science and Technical Branch, pp. 285–306.
- [7] Bolleter, U., Wyss, A., Welte, L., and Sturchler, R., 1987, "Measurement of Hydrodynamic Interaction Matrices of Boiler Feed Pump Impellers," *ASME J. Vib. Acoust.*, **109**, pp. 144–151.
- [8] Guinzburg, A., 1992, "Rotordynamic Forces Generated By Discharge-to-Suction Leakage Flows in Centrifugal Pumps," Ph.D. dissertation, California Institute of Technology, Mechanical Engineering Department.
- [9] Sivo, J. M., Acosta, A. J., Brennen, C. E., and Caughey, T. K., 1995, "The Influence of Swirl Brakes on the Rotordynamic Forces Generated by Discharge-to-Suction Leakage Flows in Centrifugal Pumps," *ASME J. Fluids Eng.*, **117**, pp. 104–108.
- [10] Colding-Jorgensen, J., 1980, "Effect of Fluid Forces on Rotor Stability of Centrifugal Compressors and Pumps," in *Rotordynamic Instability in High Performance Turbomachinery*, Texas A & M University.
- [11] Fongang, R., Colding-Jørgensen, J., and Nordmann, R., 1996, "Investigation of Hydrodynamic Forces on Rotating and Whirling Centrifugal Pump Impellers," presented at the International Gas Turbine and Aeroengine Congress & Exhibition, Birmingham, UK, June 1–13.
- [12] Torbergsen, E., and White, M. F., 1998, "Numerical and Experimental Study of Impeller/Diffuser Interactions in Centrifugal Pumps," *Proceedings of the ISROMAC Conference*, Vol. C, Honolulu, HI, Feb. 22–26, Bird Rock Publishing House, pp. 1349–1358.
- [13] Childs, D., 1989, "Fluid-Structure Interaction Forces at Pump Impeller-Shroud Surfaces for Rotordynamic Calculations," *Trans. ASME*, **111**, pp. 216–223.
- [14] Baskharone, E. A., Daniel, A. S., and Hensel, S. J., 1994, "Rotordynamic Effects of the Shroud-to-Housing Leakage Flow in Centrifugal Pumps," *ASME J. Fluids Eng.*, **116**, pp. 558–563.
- [15] Wyman, N. J., 1994, "Effects of the Primary Passage on the Flow Through the Secondary Passage of a Shrouded-Impeller Pump," Master's thesis, Mechanical Engineering Department, Texas A & M University.
- [16] Dietzen, F. J., and Nordmann, R., 1987, "Calculating Rotordynamic Coefficients of Seals by Finite Difference Techniques," *ASME J. Tribol.*, **109**, pp. 388–394.
- [17] Rhode, D. L., Hensel, S. J., and Guidry, M. J., 1992, "Three-Dimensional Computations of Rotordynamic Force Distribution in a Labyrinth Seal," *Tribol. Trans.*, **36**, No. 3, pp. 461–469.
- [18] Arghir, M., and Frene, J., 1997, "Rotordynamic Coefficients of Circumferentially-Grooved Liquid Seals Using the Average Navier-Stokes Equations," *ASME J. Tribol.*, **119**, pp. 556–567.
- [19] Athavale, M. M., Przekwas, A. J., Hendricks, R. C., and Liang, A., 1994, "SCISEAL: A three-dimensional CFD Code for Accurate Analysis of Fluid Flow and Forces in Seals," *Advance ETO Propulsion Conference*, May.
- [20] Moore, J. J., and Palazzolo, A. B., 1998, "CFD Comparison to Three-Dimensional Laser Anemometer and Rotordynamic Force Measurements for Grooved Liquid Annular Seals," presented at the ASME/STLE International Tribology Conference, Oct. 25–29, Toronto, Ontario, Canada.
- [21] AEA, 1997, *TASCFLOW Documentation Ver. 2.7*, AEA Technologies, Waterloo, Ontario, Canada.
- [22] Raw, M., 1996, "Robustness of Coupled Algebraic Multigrid for the Navier-Stokes Equations," AIAA paper.
- [23] Morrison, G. L., Johnson, M. C., and Tattersson, G. B., 1991, "three-dimensional Laser Anemometer Measurements in a Labyrinth Seal," *ASME J. Eng. Gas Turbines Power*, **113**, No. 1, pp. 119–125.
- [24] Bhattacharya, A., 1997, "CFD Based Rotordynamic Coefficients for Labyrinth Seals and Impeller Shroud Leakage Paths," Master's thesis, Mechanical Engineering Department, Texas A & M University.

Underplatform Dampers for Turbine Blades: Theoretical Modeling, Analysis, and Comparison With Experimental Data

K. Y. Sanliturk

Faculty of Mechanical Engineering,
Istanbul Technical University,
80191, Gumussuyu,
Istanbul, Turkey

D. J. Ewins

A. B. Stanbridge

Mechanical Engineering Department
Center of Vibration Engineering,
Imperial College of Science,
Technology and Medicine,
London, SW7 2BX, UK

This paper describes a theoretical model for analyzing the dynamic characteristics of wedge-shaped underplatform dampers for turbine blades, with the objective that this model can be used to minimize the need for conducting expensive experiments for optimizing such dampers. The theoretical model presented in the paper has several distinct features to achieve this objective including: (i) it makes use of experimentally measured contact characteristics (hysteresis loops) for description of the basic contact behavior of a given material combination with representative surface finish, (ii) the damper motion between the blade platform locations is determined according to the motion of the platforms, (iii) three-dimensional damper motion is included in the model, and (iv) normal load variation across the contact surfaces during vibration is included, thereby accommodating contact opening and closing during vibration. A dedicated nonlinear vibration analysis program has been developed for this study and predictions have been verified against experimental data obtained from two test rigs. Two cantilever beams were used to simulate turbine blades with real underplatform dampers in the first experiment. The second experiment comprised real turbine blades with real underplatform damper. Correlation of the predictions and the experimental results revealed that the analysis can predict (i) the optimum damping condition, (ii) the amount of response reduction, and (iii) the natural frequency shift caused by friction dampers, all with acceptable accuracy. It has also been shown that the most commonly used underplatform dampers in practice are prone to rolling motion, an effect which reduces the damping in certain modes of vibration usually described as the lower nodal diameter bladed-disk modes.

[DOI: 10.1115/1.1385830]

1 Introduction

The friction damping concept is frequently applied in turbomachinery applications, especially at hot locations, to reduce resonance stresses. A typical application of dry friction damping in gas turbines is the so-called "friction damper," "cottage-roof damper," or "underplatform damper," which is loaded by centrifugal force against the underside of the platforms of two adjacent blades. The main design criterion for such devices is to determine the optimum damper configuration or the damper mass or both in order to reduce the dynamic stresses by the maximum possible extent. For example, if the damper mass is too small for a given configuration, the friction force will not be large enough to dissipate sufficient energy. On the other hand, if the damper mass is too large, it will "stick," limiting the relative motion across the interface and thus the amount of energy dissipation. In both cases, the friction damper will be inefficient and between these two extremes there is an optimum size. A good review of the friction damping concept in turbomachinery applications is given by Griffin [1].

The so-called "cottage-roof damper" or "underplatform damper" is physically a very simple device usually a simple-

shaped piece of metal loaded on the underside of adjacent blades. The theoretical analysis and the optimization of these simple devices have been quite difficult, however, due to marked nonlinearity and to uncertainties about the contact characteristics and damper behavior. Substantial effort has been devoted to understanding and modeling basic contact characteristics, usually in the form of friction force-displacement hysteresis loops [2–7].

In addition to those studies related to the basic contact characteristics between two contacting surfaces, several friction damper models and analysis methods have been proposed in the past. The simplest, yet the most commonly used, friction damper model in the literature is the adaptation of a basic macroslip contact model to represent underplatform dampers [8–11]. This simple model is usually combined with a single-degree-of-freedom blade model, the friction damper being attached between the SDOF system and ground. These studies have revealed various important aspects of the friction damper characteristics and yielded some qualitative answers, especially on damping optimization.

More recent research efforts on friction dampers for turbine blades have introduced more details of the damper geometry. Menq et al. [12] have developed a theoretical model for bar-shaped underplatform dampers and that model was used for the forced response analysis of turbine blades [13,14]. Most of the commercially used dampers are more complex than the bar-shaped underplatform dampers. Pfeiffer and Hajek [15] proposed a method to analyze stick-slip vibrations in general and studied a curved wedged damper for turbine blades. The behavior of curved wedge-shaped dampers was also studied by others, usually by approximating the contact behavior using Hertzian contact theory

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-335. Manuscript received by IGTI October 1998; final revision received by the ASME Headquarters, March 1999. Associate Editor: D. Wisler.

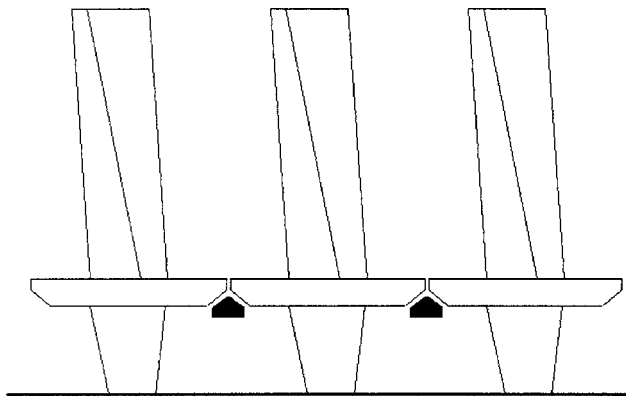


Fig. 1 Schematic illustration of underplatform dampers between adjacent blades

[16–18]. The most commonly used type of friction damper in industry is the wedged-shaped damper with flat contact surfaces, as illustrated in Fig. 1 which is also the least studied although important contributions have been made in modeling and analysis of this type of damper [19]. Various analysis methods have also been developed and reported in the literature, mainly for friction damper optimization in turbomachinery applications [11,20–23].

Although significant advances have been made in theoretical modeling of friction dampers and the analysis methods for damper optimization, turbomachine manufacturers still rely on previous experience and empirical data rather than computer-based predictions alone for friction damper optimization. This has been mainly due to the over-simplification introduced in the models regarding the basic contact behavior and/or damper geometry and the inability to analyze representative-size models due to excessive computational cost.

This paper proposes a methodology which combines three essential aspects of friction damping optimization in turbomachinery application. These include (i) utilizing experimentally measured hysteresis loops to describe the basic contact behavior between contacting surfaces in the analysis, (ii) development of a theoretical model for the motion of the wedge-shaped friction dampers and the forces generated at the contact interfaces, and (iii) development of an efficient analysis method which enables the analyst to use realistic finite elements models to describe the dynamic behavior of real turbine blades. The analysis methodology in (iii) is based on a combination of the harmonic balance method and a structural modification approach. The main motivation behind this work is to develop a friction damper optimization technique which is capable of dealing with real geometries.

As our paper addresses almost the same problem studied in [19], it is appropriate here to describe some of the similarities as well as the differences. The work reported here is similar to [19] in the sense that both aim to develop a prediction capability for the optimization of wedge-shapes dampers, the kinematics of the wedge-shaped damper are based on the motion of the platform nodes and the harmonic balance method is applied for the nonlinear analysis. The main differences are that our approach here includes three-dimensional platform and two-dimensional contact motions, microslip type of interface definition, a very efficient numerical analysis procedure and also includes experimental test case using real turbine blades.

2 Contact Model

In contrast to most research work in the literature, the contact model used in this work is based on an empirical model whose properties are obtained from experimental data. The experimental test rig reported in [7] for the measurement of hysteresis data has been slightly modified to measure the contact behavior of under-

platform dampers. This was achieved by rubbing a real underplatform damper against a block which was made of the same material as that of the blade with similar surface finishes. These measurements were carried out at constant normal load in order to provide the basic contact properties, although the analysis includes normal load variation during the blade response analysis as will be addressed in Section 3.2. It is noted that the overall contact behavior is quite similar to the “point” contact measurements reported in [7]. Furthermore, it has been found that the hybrid model proposed in [7] is generally adequate to represent the measured hysteresis loops, a typical example of which is illustrated in Fig. 2(a). Also, an array of macroslip elements, as shown in Fig. 2(b) can successfully be used to reproduce the observed microslip behavior [24].

3 Friction Damper Model

Underplatform dampers are physically very simple devices, yet their nonlinear behavior is quite complicated and its analysis can be extremely difficult if all the details of the damper characteristics are to be included in the analysis. These difficulties arise due to many complicated factors: for example, the temperature, frequency, and surface roughness effects, the real contact locations and their variation during vibration are just some of them. In spite of the physical simplicity of these dampers, the effects of these and other factors have not yet been fully understood. Accordingly, based on engineering judgement, some simplifying assumptions as listed below have been made here in order to reduce the problem to a manageable level:

- damper flexibility and inertia effects are negligible,
- damper contact on each side can be represented as a point contact with three translational degrees-of-freedom,
- left and right surfaces are identical, and
- the blade motion is harmonic.

In addition to the assumptions above, the first stage of the formulation below will have an additional assumption that

- damper and platform surfaces remain in parallel and in contact at all times.

Experimental results, however, indicated that this last assumption is often not valid for the wedge-shaped dampers studied here, and so some corrections are introduced later.

The friction damper model presented below is based on a given amplitude of vibration and so the model parameters must be recalculated until convergence is achieved due to marked nonlinearity of these devices. The analysis method based on iterative approach is given in Section 4.

3.1 Relative Motion Across Damper Surfaces. The theoretical formulation of the cottage roof damper motion as presented here is based on a model as shown in Fig. 3 where a local coordinate system is attached to one of the platforms and the instantaneous relative motion of the other platform is described by

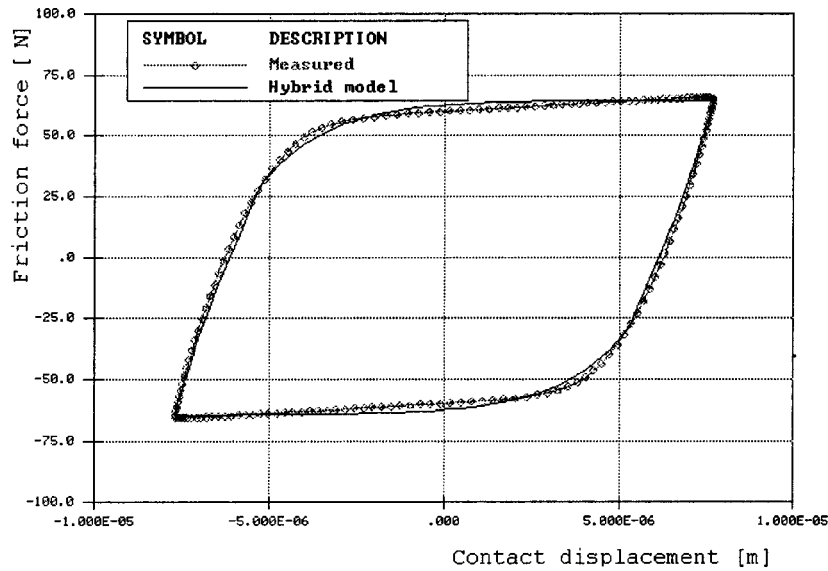
$$\mathbf{r}_{xyz} = i\mathbf{r}_x + j\mathbf{r}_y + k\mathbf{r}_z \quad (1)$$

where

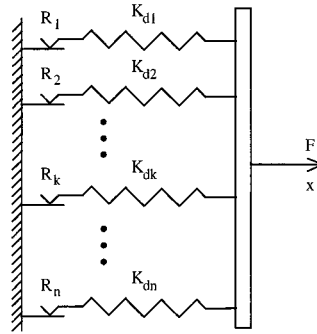
$$\begin{aligned} r_x &= |X| \cos(\omega t + \phi_x) \\ r_y &= |Y| \cos(\omega t + \phi_y) \\ r_z &= |Z| \cos(\omega t + \phi_z) \end{aligned} \quad (2a)$$

and

$$\begin{aligned} X &= X_L - X_R \\ Y &= Y_L - Y_R \\ Z &= Z_L - Z_R. \end{aligned} \quad (2b)$$



(a)



(b)

Fig. 2 (a) Typical measured hysteresis loop under constant normal load, (b) representation of measured behavior by an array of macroslip elements

$X_L, Y_L, Z_L, X_R, Y_R, Z_R$ above are complex quantities representing both amplitude and phases at platform nodes. Similarly, X, Y , and Z are the relative platform displacements with respect to the local coordinate system, ϕ_x, ϕ_y , and ϕ_z being the corresponding phase angles. Furthermore, ω is the angular velocity and t is time. It should be stated at the outset that r_x, r_y, r_z and other parameters which are functions of them are not constant values but functions of the angular displacement, ωt . The damper formulation presented in this paper requires calculation of these dis-

placements (and the associated forces in the next section) for at least one vibration cycle. However, ωt is dropped in the equations for brevity.

The relative platform motion in Fig. 4 is three-dimensional, having components in all three local directions. The relative displacements of the underplatform damper with respect to the platform surfaces shown in Fig. 5 are calculated based on the assumption that the centrifugal force acting on the damper mass is large enough to keep the damper in contact with the platform surfaces at all times.

Accordingly, a displacement triangle in Fig. 6 can be drawn

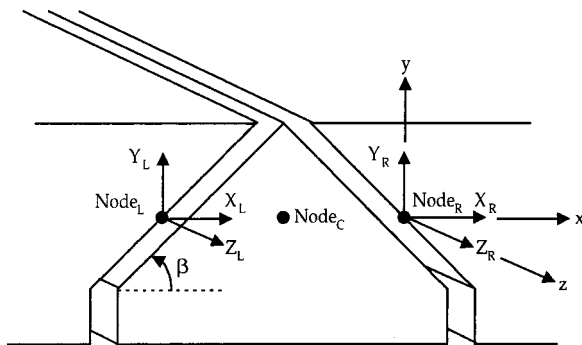


Fig. 3 Platform motion in three-dimensional space. (β is the platform angle, Node_L and Node_R are the platform structural nodes.)

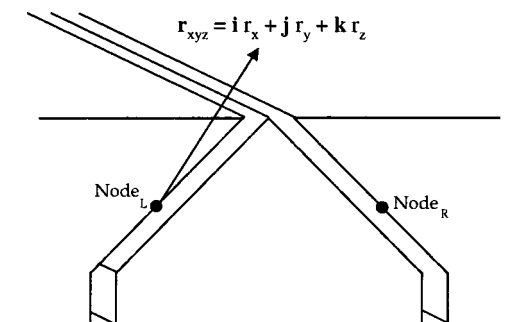


Fig. 4 Relative platform displacements

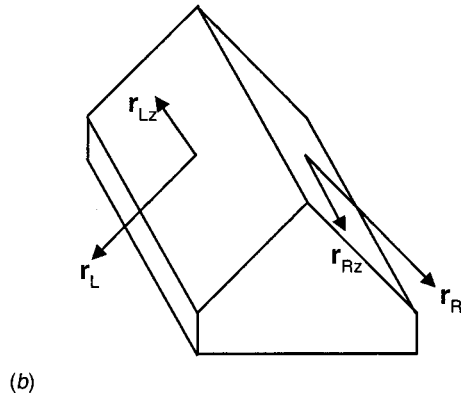
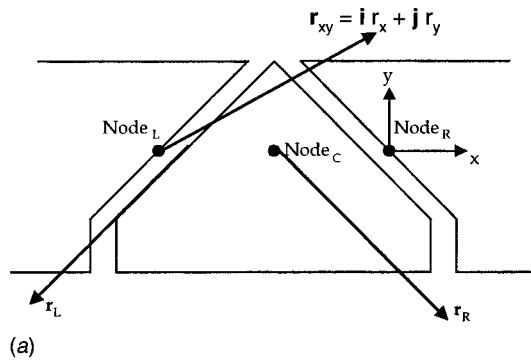


Fig. 5 Relative contact displacements on damper surfaces

relating the relative displacements of the platform in x - y plane ($\mathbf{r}_{xy}$) and the relative contact displacements $\mathbf{r}_L$ and $\mathbf{r}_R$ in Fig. 5(a). Since the directions of all the vectors in Fig. 5(a) and the magnitude of the $\mathbf{r}_{xy}$ are known, the magnitudes of the $\mathbf{r}_L$ and $\mathbf{r}_R$ are calculated as

$$|\mathbf{r}_L| = \frac{|\mathbf{r}_{xy}|(\sin(\alpha) + \cos(\alpha)\tan(\beta))}{2\sin(\beta)} \quad (3a)$$

$$|\mathbf{r}_R| = \frac{|\mathbf{r}_{xy}|\cos(\alpha)}{\cos(\beta)} - |\mathbf{r}_L|. \quad (3b)$$

Another component of the relative contact motion is in the local z -direction as indicated by $\mathbf{r}_{Lz}$ and $\mathbf{r}_{Rz}$ in Fig. 5(b) and there is some uncertainty regarding how the $\mathbf{r}_z$ is shared between the left and the right-hand side of the damper. It is assumed in this paper that the relative platform motion in the z -direction is shared equally between left and right side of the damper, i.e.,

$$|\mathbf{r}_{Lz}| = |\mathbf{r}_{Rz}| = |\mathbf{r}_z|/2. \quad (3c)$$

The reasoning behind above assumption is that minimum energy will be dissipated if the two sides share the relative displacements

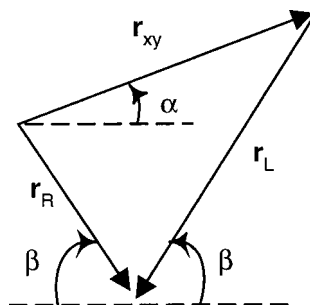


Fig. 6 Displacement triangle relating $\mathbf{r}_{xy}$ to $\mathbf{r}_L$ and $\mathbf{r}_R$

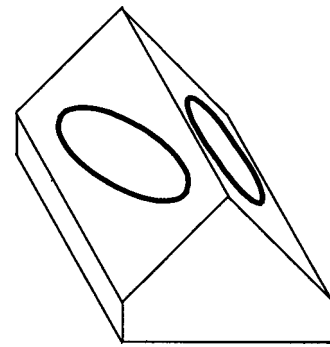


Fig. 7 Damper displacements with respect to the platform surfaces

equally. In general, there is a phase difference between the relative contact displacements in the x - y plane and in the z -direction and this results in elliptical relative contact motion as illustrated in Fig. 7. The reader may refer to [24] for details of how to analyze those interfaces where the relative contact motion is elliptical as well as how to relate a one-dimensional microslip model to elliptical motion at the interface.

3.2 Damper Forces. Determination of the friction forces across the damper surfaces requires contact displacement and normal load as well as the contact characteristics. The relative contact displacement can be calculated using the procedure summarized above.

The forces on a typical underplatform damper are quite complex in spite of the simplifying assumptions and in spite of the simplicity of the “static” forces involved in Fig. 8 where CF represents centrifugal force acting on the damper and N_{Ls} and N_{Rs} are the static reaction forces in the absence of any vibration. These reaction forces can easily be calculated using the static equilibrium conditions as depicted in Fig. 8(a) and given by

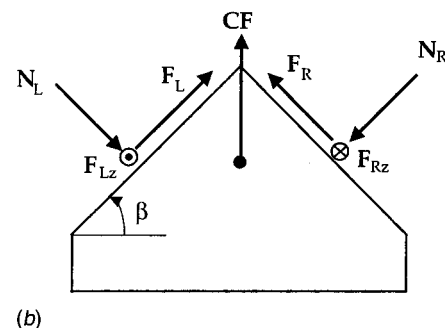
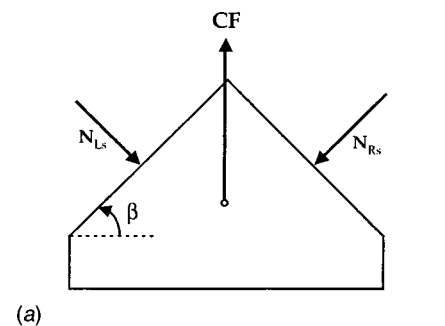


Fig. 8 Forces acting on a damper; (a) static case, (b) dynamic case

$$N_{Ls} = N_{Rs} = N_s = \frac{CF}{2 \cos(\beta)}. \quad (4)$$

The results, not presented here, indicate that the normal forces during vibration can deviate significantly from those of the mean “static” values. The main difficulty in including normal load variations is that the normal loads on the damper surfaces depend on friction forces and vice versa. In Fig. 8(b), N_L and N_R are different from each other, as are the corresponding friction forces on each side of the damper. As the damper inertia is neglected, the total force acting on a cottage roof damper must be zero for equilibrium, hence one can write equilibrium equations in local x - and y -directions, respectively, at any arbitrary time during a vibration cycle as follows:

$$N_L \sin(\beta) + F_L \cos(\beta) = N_R \sin(\beta) + F_R \cos(\beta) \quad x\text{-dir} \quad (5a)$$

$$N_L \cos(\beta) + N_R \cos(\beta) = F_L \sin(\beta) + F_R \sin(\beta) + CF \quad y\text{-dir}. \quad (5b)$$

It is worth mentioning here that similar to the relative displacements, the dynamically varying forces are functions of the angular displacement, ωt , although ωt is dropped from these equations. Using Eq. (5), one can write the normal loads on each side of the damper as a function of CF and the friction forces F_L and F_R , as given below.

$$N_L = \frac{(F_L + F_R)}{2} \tan(\beta) + \frac{CF}{2 \cos(\beta)} - \frac{(F_L - F_R)}{2 \tan(\beta)} \quad (6a)$$

$$N_R = \frac{(F_L + F_R)}{2} \tan(\beta) + \frac{CF}{2 \cos(\beta)} + \frac{(F_L - F_R)}{2 \tan(\beta)}. \quad (6b)$$

The dynamic equilibrium condition above provides only two equations and there are four unknowns to be determined, namely: N_L , N_R , F_L , and F_R . (The friction forces in local z -direction, F_{Lz} and F_{Rz} are perpendicular to the normal loads hence they are excluded in determining the normal loads here). Although the normal loads on the contact surfaces and the corresponding friction forces are not independent, it is very difficult to write an explicit relationship between them since this relationship is nonlinear and depends on the previous history of the contact motion. There are two possible solutions to overcome this problem: (i) iterate until convergence is achieved at every increment of a vibration cycle and, (ii) update the normal load successively at every increment of a vibration cycle without iteration. Both of these strategies, (i) and (ii) above, can be used to calculate the normal forces as well as the friction forces. It has been found, however, that option (i) is fairly expensive compared to option (ii) as option (ii) can yield acceptable accuracy. Therefore, only the second approach, also called the successive approximation approach, is described here.

In the so-called successive approximation approach, the normal loads are updated according to the friction forces calculated in a previous step, without involving any iteration. Various steps involved in this approach are summarized below.

Step 1 Start with static normal load (set the angular displacement to zero: $\theta = \omega t = 0$)

$$N_L = N_R = \frac{CF}{2 \cos(\beta)}. \quad (4')$$

Step 2 Calculate F_L , F_R , F_{Lz} , F_{Rz} based on N_L , N_R .

Step 3 Calculate new normal loads using equilibrium equation

$$N_L = \frac{(F_L + F_R)}{2} \tan(\beta) + \frac{CF}{2 \cos(\beta)} - \frac{(F_L - F_R)}{2 \tan(\beta)} \quad (6a)$$

$$N_R = \frac{(F_L + F_R)}{2} \tan(\beta) + \frac{CF}{2 \cos(\beta)} + \frac{(F_L - F_R)}{2 \tan(\beta)}. \quad (6b)$$

Step 4 If $\theta = 4\pi$, exit the cycle, otherwise set $\theta = \theta + \Delta\theta$ and go to Step 2.

It is obvious that there is some approximation involved in this approach. The normal loads used to calculate the current friction forces are in fact “correct” values for the previous step. That is to say that “correct” normal loads lag the friction forces by one angular increment and the error introduced due to this is found to be less than 5 percent when $\Delta\theta = \pi/36$. The 5 percent quoted here is the maximum error value obtained after comparing the correct and the approximate results for many combinations and levels of platform motion and contact parameters.

Whether the iterative or the successive approximation approach is employed, it is necessary to be able to calculate the friction forces while normal loads vary during a vibration cycle, i.e., Step 2 above. This is carried out as follows. The hybrid type of point contact model proposed in [7] is represented by an array of macroslip element as shown in Fig. 2(b) [24]. The initial properties of the model—namely, the individual stiffness values (K_{dk}) and the corresponding limiting friction forces (R_k)—are calculated based on measured contact stiffness and the coefficient of friction and the static normal load N_s for a given maximum amplitude of vibration. These sliders are then traced, as described in Ref. [24], across the damper surfaces while the individual limiting friction forces are adjusted as the normal loads on each side of the damper vary during a vibration cycle. For example, for the left side

$$R_k = R_{ks} \frac{N_L}{N_s} \quad k = 1, 2, \dots, n \quad (7)$$

where $R_{k,r}$ is the limiting friction force for the k th element based on static normal load and n is the number of sliders used in the model. This tracing process is carried out for a few cycles (usually two) until the trajectories of the sliders are stabilised and this allows the calculation of the friction forces (F_L, F_{Lz}, F_R, F_{Rz}) as well as the normal loads (N_L, N_R) acting on the damper.

4 Analysis Method

The analysis method proposed in this paper is a combination of the harmonic balance method and a structural modification approach and the proposed approach here is designed particularly for analyzing large models with localized nonlinearities very efficiently.

The behavior of the friction damper is analyzed at a given relative response amplitude between the damper connection points and the individual dampers are linearised as equivalent complex stiffnesses, representing both restoring and energy dissipation characters. This is addressed in Section 4.1 below for the under-platform damper described in Section 3. Section 4.2 describes the second stage where these equivalent complex stiffnesses are added to the otherwise linear system to include the effect of the friction dampers. The iterative nature of the problem as well as the procedure to analyze nonlinear systems at active coordinates only, are also described. It must be noted, however, that the nonlinear analysis procedure presented here is not limited to the friction dampers only. The methodology can easily be applied for the analysis of structures with other types of nonlinearities.

4.1 HBM Linearization of Damper Forces. After the normal and the corresponding friction forces are determined, the resulting forces applied to Node_L in Fig. 9 can be decomposed into x -, y - and z -directions as:

$$F_{Lx}(\theta) = F_L(\theta) \cos(\beta) + N_L(\theta) \sin(\beta) \quad (8a)$$

$$F_{Ly}(\theta) = F_L(\theta) \sin(\beta) - N_L(\theta) \cos(\beta) \quad (8b)$$

$$F_{Lz}(\theta) = F_{Lz}(\theta) \quad (8c)$$

A similar set of equations can also be written for at Node_R. Note that F_L, N_L (and others) in the above equations are expressed as $F_L(\theta)$ and $N_L(\theta)$ in order to emphasise that they are not constant but are functions of θ and they are defined for a vibration cycle.

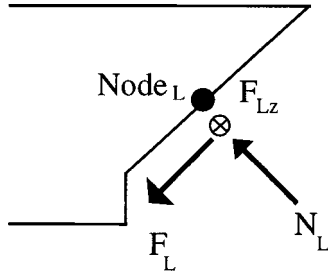


Fig. 9 Dynamic forces acting on the left platform

The first-order components of the resulting forces in the x -, y - and z -directions are calculated and scaled to obtain a complex stiffness representation of these forces in a manner similar to that for the one-dimensional case, as in [23]. For example, the effect of the friction damper in the x -direction is represented by a complex stiffness ($k_x^* = k_x^r + ik_x^i$) between the platform nodes as

$$k_x^r = \frac{1}{\pi|X|} \int_0^{2\pi} F_{Lx}(\theta) \cos(\theta + \phi_x) d\theta$$

$$k_x^i = \frac{-1}{\pi|X|} \int_0^{2\pi} F_{Lx}(\theta) \sin(\theta + \phi_x) d\theta. \quad (9a)$$

Similarly, the equivalent stiffness in y and z -directions are

$$k_y^r = \frac{1}{\pi|Y|} \int_0^{2\pi} F_{Ly}(\theta) \cos(\theta + \phi_y) d\theta,$$

$$k_y^i = \frac{-1}{\pi|Y|} \int_0^{2\pi} F_{Ly}(\theta) \sin(\theta + \phi_y) d\theta \quad (9b)$$

$$k_z^r = \frac{1}{\pi|Z|} \int_0^{2\pi} F_{Lz}(\theta) \cos(\theta + \phi_z) d\theta,$$

$$k_z^i = \frac{-1}{\pi|Z|} \int_0^{2\pi} F_{Lz}(\theta) \sin(\theta + \phi_z) d\theta. \quad (9c)$$

As $F_{Lx}(\theta)$, $F_{Ly}(\theta)$, and $F_{Lz}(\theta)$ are known at discrete values of θ , numerical integration can be carried out easily for the integrals in above equations.

4.2 Analysis Method. The solution procedure adopted in the frequency domain is based on finding the response amplitudes iteratively, the starting point being the response levels of the underlying linear system. The behavior of the friction dampers is analyzed at a given relative response amplitude between the damper connection points and the individual dampers are represented as equivalent complex stiffnesses, representing both restoring and energy dissipation characteristics as described above. The equivalent complex stiffnesses are then added to the otherwise linear system and the response levels of the modified system are calculated again, the procedure being repeated until convergence is achieved. The response levels obtained at current frequency are used as initial guesses for the next frequency increment.

A very efficient analysis method is proposed here for the analysis of systems with localized nonlinearities via a structural modification approach. An important feature of the method here is that it treats the linear and the nonlinear parts of a structure separately, the linear part being the original structure and the nonlinear part representing the modifications. The nonlinear elements (modifications) are linearised as equivalent stiffnesses at given amplitude of vibration using the first-order HBM as in the previous section. Expressing the nonlinearities as an impedance matrix multiplied by the displacement amplitude vector, was proposed earlier in [25–27] where the method developed by Ozguven [28] for the

harmonic response analysis of nonproportionally damped linear structures was adopted for harmonic response analysis of nonlinear structures. The mathematical background of the approach in this current paper is different; it is based on the Sherman-Morrison formula [29] and it is more general and more efficient for studying nonlinear systems than the method in [28]. The Sherman-Morrison formula has been used in the past [30] to calculate the frequency response of a (linear) modified structure. However, to the best of the authors' knowledge, this paper is the first to propose this method for the analysis of nonlinear systems and it has very good potential for the analysis of wide range of structures. It is shown in [29] that the Sherman-Morrison formula allows a direct inversion of the modified matrix efficiently using the data related to the initial matrix and to the modification. A brief summary of the Sherman-Morrison formula is given below.

Let $[A]^{-1}$ be the inverse of a nonsingular square matrix, $[A]$. If the inverse of a modified matrix, $[A']^{-1}$, is needed where $[A']$ is of the form

$$[A'] = [A] + \{u\}\{v\}^T. \quad (10)$$

This can be calculated using the Sherman-Morrison formula without any matrix inversion as

$$[A']^{-1} = [A]^{-1} - \frac{([A]^{-1}\{u\})(\{v\}^T[A]^{-1})}{1 + \lambda} \quad (11)$$

where

$$\lambda = \{v\}^T[A]^{-1}\{u\}. \quad (12)$$

The generalization of Eq. (11) is also available and is known as Sherman-Morrison-Woodbury formula which considers the modification as a product of two rectangular matrices such as $[U] \times [V]^T$. A more detailed coverage including the history of these formulas and the numerical aspects are discussed in [29].

It is proposed in this paper to adopt the Sherman-Morrison formula to calculate the nonlinear response levels of structures with localized nonlinearities rather than the linear modification analysis as reported in [30]. This is achieved as follows. Suppose that the linear structure is given by its dynamic stiffness matrix $[Z]$ and its frequency response function matrix $[\alpha]$, $[\alpha] = [Z]^{-1}$, and the modification matrix to be made to $[Z]$ is $[\Delta]$. The dynamic stiffness matrix of the modified system $[Z']$, can then be written as

$$[Z'] = [Z] + [\Delta]. \quad (13)$$

If the modification matrix is written in the form

$$[\Delta] = \{u\}\{v\}^T, \quad (14)$$

the FRF matrix of the modified system $[\beta]$ can be computed from

$$[\beta] = [Z']^{-1} = [\alpha] - \frac{([\alpha]\{u\})(\{v\}^T[\alpha])}{1 + \{v\}^T[\alpha]\{u\}} \quad (15)$$

which allows the FRF matrix of the modified system to be calculated without any matrix inversion. It should be noted that if the total modification matrix $[\Delta]$ cannot be written as a multiplication of two vectors as in Eq. (13), it can be decomposed into several, say p , modification matrices, such as

$$[\Delta] = [\Delta_1] + [\Delta_2] + [\Delta_3] + \dots + [\Delta_p] \quad (16)$$

where $[\Delta_i] = \{u_i\}\{v_i\}^T$. This allows the FRF of the system to be calculated by considering each $[\Delta_i]$ individually. For example, the spring modification in local x , y and z -directions in Eq. (9) can be included one at a time. It should be stated, however, that it is necessary to transform the individual stiffnesses from the local to a global coordinate system (GCS) as these coordinate systems, in general, do not coincide. This transformation also results in individual modification matrices being in the form of $\{u_i\}\{v_i\}^T$ as illustrated below in the case of x -direction:

$$[k]_{6 \times 6} = k_x^* \begin{bmatrix} e^2 & fe & ge & -e^2 & -fe & -ge \\ ef & f^2 & gf & -ef & -f^2 & -gf \\ eg & fg & g^2 & -eg & -fg & -g^2 \\ -e^2 & -fe & -ge & e^2 & fe & ge \\ -ef & -f^2 & -gf & ef & f^2 & gf \\ -eg & -fg & -g^2 & eg & fg & g^2 \end{bmatrix}$$

$$= \begin{bmatrix} e \\ f \\ g \\ -e \\ -f \\ -g \end{bmatrix} \begin{bmatrix} e \cdot k_x^* \\ f \cdot k_x^* \\ g \cdot k_x^* \\ -e \cdot k_x^* \\ -f \cdot k_x^* \\ -g \cdot k_x^* \end{bmatrix} = \{u_i\} \{v_i\}^T \quad (17)$$

where k_x^* is the equivalent complex stiffness in the local x -direction, $[k]_{6 \times 6}$ is the modification matrix corresponding to the translations at platform nodes with respect to the GCS, and e, f, g are the direction cosines of the local x -direction.

Another distinct feature of the technique proposed here is that the analysis can be restricted to active coordinates only; the active coordinates being the nonlinear coordinates, forcing coordinates and those coordinates where response levels are needed. This is achieved by using the natural frequencies and mode shapes of the linear system in order to describe the frequency response functions at active coordinates via modal summation. This feature has made it possible to analyse industrial cases, as the analysis does not depend on the size of the original linear model.

5 Predictions and Comparison With Experimental Data

5.1 Case 1: Simplified Blade-Damper-Blade Assembly.

The test rig, schematically illustrated in Fig. 10(a) essentially comprised two beams representing blades with representative platforms so as to accommodate an industrial wedge-shaped underplatform damper. The cantilever beams were clamped together via two clamping blocks and the whole assembly was in turn clamped to a large seismic block, the clamping arrangement being identical to that described in Ref. [31]. An industrial underplatform damper was installed between the blades about 33 percent up from the built-in ends and mass loading was applied so as to represent centrifugal load. Stepped-sine frequency response testing was carried out using constant input force levels. Nonlinear response levels were measured for various damper loads within a frequency range which covered the first two bending modes of the assembly, i.e., the in-phase (IP) and out-of-phase (OOP) bending modes of the two-beam assembly.

The linear structure was modelled using two-dimensional isoparametric plane elements, including the curvature at the root, as shown in Fig. 10(b). The clamping blocks were not included in the finite element model. Instead, fixed-displacement boundary conditions, shown in Fig. 10, were imposed so as to represent the linear behavior of the system. Correlation of the predictions and the measurements for the linear system was found to be quite good as illustrated in Fig. 11. It is important to note in Fig. 11 that there are two close bending modes around 530 Hz, i.e., in-phase (IP) and out-of-phase (OOP) bending modes, respectively.

The first set of results was calculated for a range of damper loads as in the experiments, the excitation force being maintained at 1.0 N amplitude. For comparison purposes, the predicted nonlinear response amplitudes, at the tips of the blades, were overlaid with measured values, as shown in Fig. 12 which contains a series of initially predicted and measured responses at different damper loads, but for the same level of excitation. Inspection of the results in Fig. 12 reveals that the theoretical predictions are rather different to the measured response levels. The most important point to note is that the measured natural frequency shift for the first mode

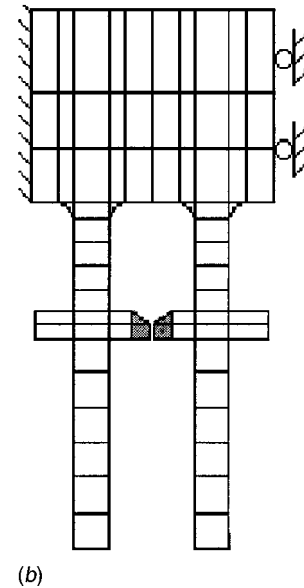
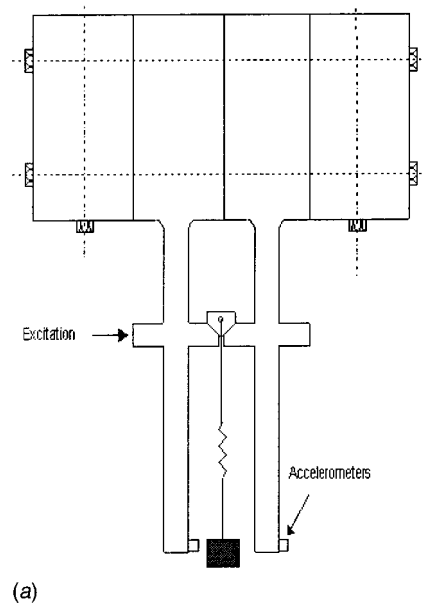
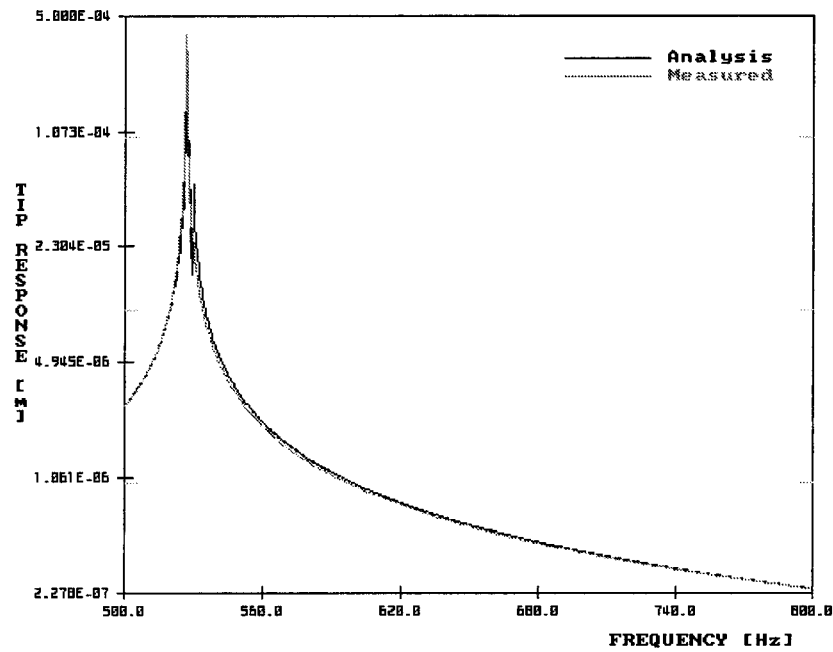
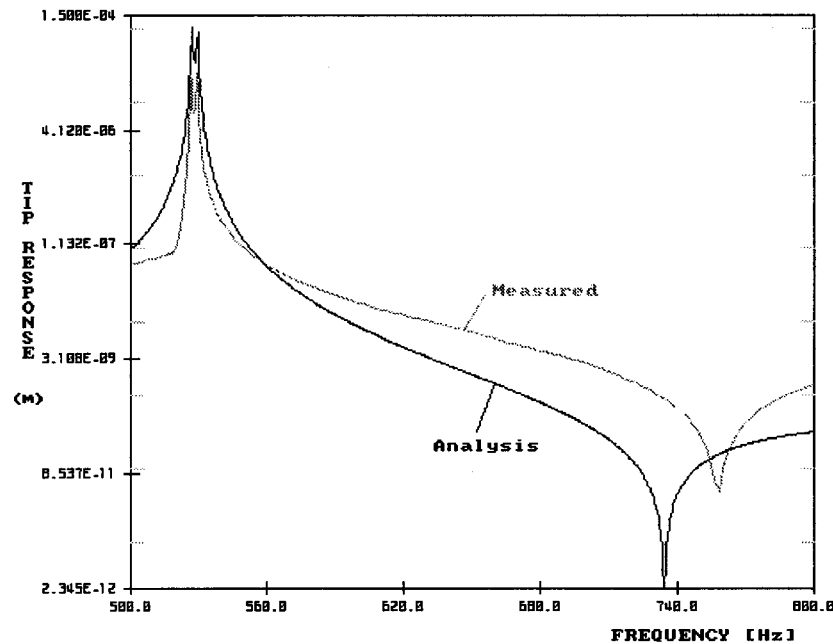


Fig. 10 (a) Experimental set-up for simplified blade-damper assembly, (b) finite element model for simplified blade-damper-blade assembly

(IP mode) is very small, about 4 percent, compared with that for the out-of-phase (OOP) mode. However, the theoretical predictions suggest more than 50 percent natural frequency shift for the same mode when the damper is almost locked at a 100 N normal load. This finding initiated a series of investigations including experimental visualisation of the underplatform dampers under vibration. The results, not presented here, revealed that one of the assumptions made during the theoretical formulation was not valid, i.e., that the damper and the platform surfaces remain parallel and in contact at all times. It has been found that the wedge-shaped dampers cannot always remain parallel to the platform surfaces, as illustrated schematically in Fig. 13(a) where the *rolling* behavior becomes dominant when the relative platform displacement is predominantly radial, as in the case of in-phase (IP) bending modes. The implication of this rolling behavior is that the calculated value for the effective complex stiffness in the local y -direction (k_y^*) is not realistic. This rolling situation has not been



(a)



(b)

Fig. 11 (a) Comparison of measured and predicted response levels (directly excited blade), (b) comparison of measured and predicted response levels (indirectly excited blade)

observed, however, when the platforms are subjected to out-of-phase bending vibration hence the theoretical model for complex stiffness in the local x -direction (k_x^*) is valid.

Modeling the rolling behavior depicted in Fig. 13(a) is very difficult and requires much more complicated theoretical models, not only because of the high sensitivity of the rolling effect to the gap between the adjacent platforms but also because of the difficulty in finding the effective contact parameters as a function of the rolling angle, i.e., as a function of contact opening. To date, such a complete theoretical model is not available. In the absence of such a complete theoretical model, the approach adopted here aims to give an allowance for the rolling effect, in the form of a

correction factor as in Eq. (18), based on empirical data combined with a method of estimating a critical platform angle below which the damper motion is predominantly rolling,

$$k_{y,c}^* = f(\beta, \mu) k_y^* \quad (18)$$

where $k_{y,c}^*$ is the corrected stiffness in the radial direction and $f(\beta, \mu)$ is the correction factor which is a function of the platform angle and the coefficient of friction and varies between 0 and 1.0. The physical reasoning behind such a correction factor in Eq. (18) is that the damper will slide and follow the imposed platform motion more easily when the roof angle is very small (large plat-

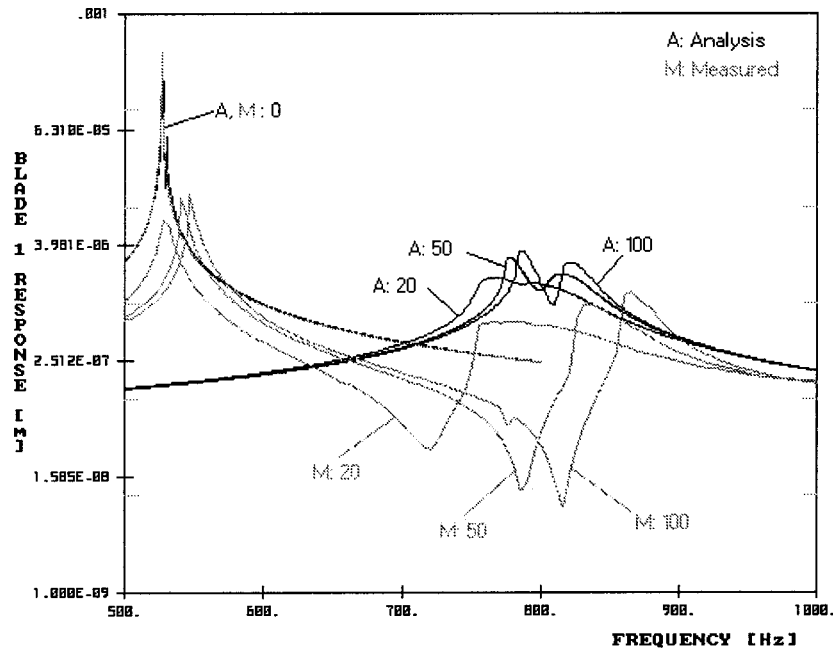


Fig. 12 Initial predictions (damper load=0, 20, 50, 100 N)

form angle). In another words, the damper will predominantly slide rather than roll to follow the platform motion, as in Fig. 13(b), if the platform angle is greater than a critical value. $\beta_{cr} = \tan^{-1}(\mu)$. On the other hand, if the platform angle is less than the critical platform angle, for a given coefficient of friction, the damper will tend to roll instead of slide, approaching pure rolling as the platform angle approaches zero degrees (this corresponds to the correction factor being zero). Therefore, there is good justification for the conclusion that the platform angle and the coefficient of friction are critical parameters which should be included in such an approach. The correction factor, $f(\beta, \mu)$ above, can also be written as a function of normalized platform angle alone, normalization being with respect to critical platform angle. This makes it possible to write the function in terms of β_N only, where β_N is the normalized platform angle. The experimental results of the simplified blade-damper assembly corresponding to the in-

phase bending mode are used to obtain empirical data and the results indicated that the correction factor could be even less than 0.1 for platform angles representative of those dampers currently in use. In the absence of any reliable method, the results obtained using the correction factor are quite useful to identify the extent of rolling for such dampers. Nevertheless, the authors of this paper are aware that a more complicated and complete model needs to be developed in order to avoid the need for the correction factor adopted in this paper. This is one of the areas where the current wedge damper model can be improved significantly.

The theoretical predictions presented in Fig. 12 were recalculated using the correction factor allowing for the rolling behavior and the results are presented in Fig. 14 together with corresponding experimental data. It is immediately seen that the refined analysis can now predict the natural frequency shift for both IP and OOP modes as well as the amount of response reduction and the optimum damping condition. It is also important to note that

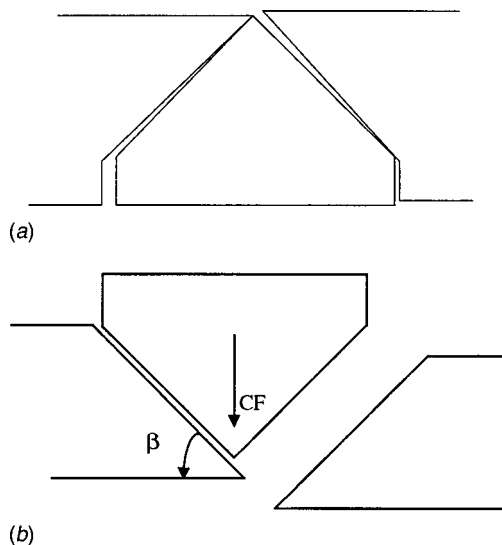


Fig. 13 (a) Relative platform motion in radial direction can cause rolling, (b) damper tends to slide if $\beta > \tan^{-1}(\mu)$

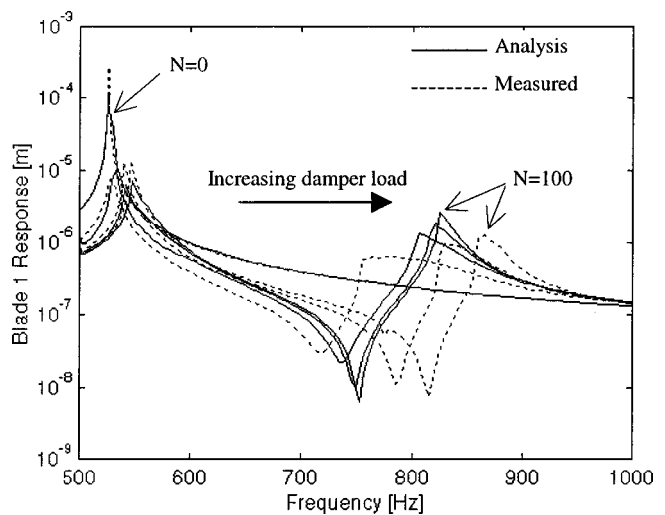


Fig. 14 Comparison of measured and predictions after incorporating rolling effect. (Damper load=0, 20, 50, 100 N.)

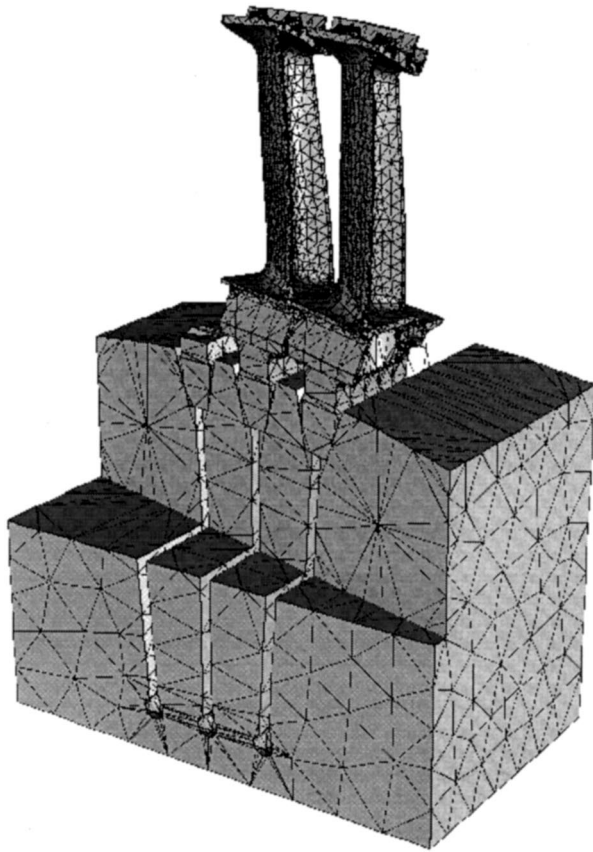


Fig. 15 Finite element model for the linear part of two-blade assembly

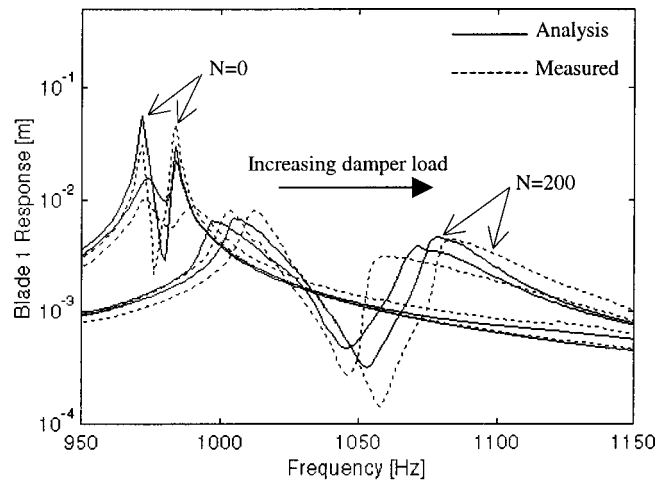
the damper is not as effective in damping the IP mode as it is for the OOP mode. Another point worth stating here is that the curved wedge-shaped dampers do not provide any damping nor natural frequency shift for IP bending mode [18].

5.2 Case 2: Real Blade-Damper-Blade Assembly. As illustrated in Fig. 15, this second case study comprised two real turbine blades and a wedge-shaped underplatform damper. The blades were clamped in a slotted block which had root serrations. This holding block was in turn clamped between two large steel blocks, not shown in Fig. 15. Experimental procedure was very similar to the previous case: Damper normal force representing the centrifugal force was achieved via gravitational loading, excitation was applied at about one-third of the blade length from the root, and measurements were taken close the blade tips under constant excitation force of 1N for various values of damper normal force.

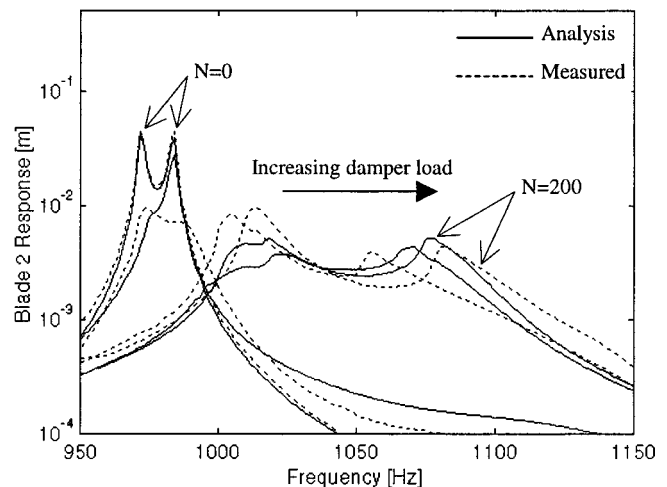
In spite of the geometry being very much complicated than in the previous case, where two simple beams were used, the dynamic behavior of this assembly with real blades and damper was very similar to that of the simple assembly in Section 5.1. As before, the assembly had pairs of in-phase (IP) and out-of-phase (OOP) modes, though the measurements were restricted to a frequency range covering the first IP and OOP bending modes.

Unlike the two-beam assembly, the linear three-dimensional finite element model shown in Fig. 15 for this case was very large, more than 200,000 degrees-of-freedom. However, this was not a drawback as the nonlinear analysis is carried out at active coordinates only as described earlier. Some initial adjustment of the linear model, in terms of material properties and modal damping, was necessary to align the linear model so that linear predictions matched the measurements without the friction damper.

Various input data including measured contact properties,



(a)



(b)

Fig. 16 (a) Correlation of predictions and experimental data. Directly excited blade, $F_{ex}=1.0$ N, damper load=0, 10, 100, 200 N. (b) Correlation of predictions and experimental data. Indirectly excited blade, $F_{ex}=1.0$ N, damper load=0, 10, 100, 200 N.

damper normal load, damper nodes/orientation and platform angle were used to define the damper for nonlinear analysis. The same relationship between the correction factor for rolling and the normalized platform angle obtained from simplified blade-damper assembly study was used here. Response levels were calculated at measurement locations for various values of damper normal loads, keeping the excitation constant as in the experiment. Correlation of the measured and the predicted response levels is presented in Fig. 16 in the form of a series of plots where the damper load is gradually increased. It is seen again that the theoretical predictions correlate very well with the measurements, validating the underplatform damper model and the analysis techniques developed here. An important point to note is that the IP bending mode is affected less by comparison with the OOP bending mode, as found in the previous case. However, the underplatform damper seems to provide a significant amount of damping for the IP mode as well for this assembly, possibly due the IP mode of vibration involving sliding in the local z -direction.

6 Concluding Remarks

1 A theoretical model for wedge-shaped underplatform dampers for turbine blades has been developed. Measured contact parameters can be used to provide the data required to describe contact characteristics.

2 A very efficient frequency domain response analysis method has been proposed. This is based on a combination of harmonic balance method and structural modification approaches. The method allows the calculation of nonlinear response levels at active coordinates only, making it possible to apply the analysis for industrial cases.

3 Wedge-shaped dampers are prone to rolling motion when subjected to radial platform motion and this can reduce the damping in certain modes of vibration usually described as the lower nodal diameter bladed disk modes. Further work is needed to develop a theoretical model for this rolling behavior.

4 The theoretical analysis proposed here can predict the nonlinear response behavior of assemblies with friction dampers with acceptable accuracy. However, an empirical correction factor is needed during this analysis for certain modes of vibration.

Acknowledgments

The authors are grateful to their sponsors, Rolls-Royce plc, and Volvo Aero Corporation, for their financial support and for permission to publish this work. The authors also thank to J. S. Green of Rolls-Royce, Derby, for generating the finite element mesh for the real blade-damper-blade assembly.

References

- [1] Griffin, J. H., 1990, "A Review of Friction Damping of Turbine Blade Vibration," *International Journal of Turbo and Jet Engines*, **7**, pp. 297–307.
- [2] Goodman, L. E., and Brown, C. B., 1962, "Energy Dissipation in Contact Friction: Constant Normal and Cyclic Tangential Loading," *ASME J. Appl. Mech.*, **29**, pp. 17–22.
- [3] Iwan, W. D., 1967, "On a Class of Models for the Yielding Behavior of Continuous and Composite System," *ASME J. Appl. Mech.*, **34**, pp. 612–617.
- [4] Rogers, P. F., and Boothroyd, G., 1975, "Damping at Metallic Interfaces Subjected to Oscillating Tangential Loads," *J. Eng. Ind.*, pp. 1087–1093.
- [5] Burtekin, M., Cowley, A., and Back, N., 1978, "An Elastic Mechanism for the Micro-sliding Characteristics Between Contacting Machined Surfaces," *J. Mech. Eng. Sci.*, **20**, No. 3, pp. 121–127.
- [6] Shoukry, S. N., 1985, "A Mathematical Model for the Stiffness of Fixed Joints Between Machine Parts," *Proc. Int. NUMETA Conf.*, J. Middleton and G. N. Pande, eds., Jan., Swansea, UK, A. A. Balkema, Rotterdam, pp. 851–858.
- [7] Sanliturk, K. Y., Stanbridge, A. B., and Ewins, D. J., 1995, "Friction Dampers: Measurement, Modelling and Application to Blade Vibration Control," *Proc. Des. Eng. Conf.*, Vol. 3, Part B, DE-Vol. 84-2, ASME, New York, pp. 1377–1382.
- [8] Menq, C.-H., and Griffin, J. H., 1985, "A Comparison of Transient and Steady State Finite Element Analysis of the Forced Response of a Frictionally Damped Beam," *ASME J. Vib. Acoust.*, **107**, pp. 19–25.
- [9] Chen, S., and Sinha, A., 1990, "Probabilistic Method to Compute the Optimal Slip Load for a Mistuned Bladed Disk Assembly With Friction Dampers," *ASME J. Vib. Acoust.*, **112**, pp. 214–221.
- [10] Wang, J. H., and Chen, W. K., 1992, "Investigation of the Vibration of a Blade with Friction Damper by HBM," presented at the ASME Int. Gas Turbine Aeroeng. Congress & Exhibition, Cologne, Germany, June 1–4, 92-GT-8.
- [11] Cameron, T. M., Griffin, J. F., Kielb, R. E., and Hoosac, T. M., 1990, "An Integrated Friction Damper Design," *ASME J. Vib. Acoust.*, **112**, pp. 175–182.
- [12] Menq, C.-H., Bielak, J., and Griffin, J. H., 1986, "The Influence of Microslip on Vibratory Response, Part I: A New Microslip Model," *J. Sound Vib.*, **107**, No. 2, pp. 279–293.
- [13] Menq, C.-H., Griffin, J. H., and Bielak, J., 1986, "The Influence of Microslip on Vibratory Response, Part II: A Comparison With Experimental Result," *J. Sound Vib.*, **107**, No. 2, pp. 295–307.
- [14] Csaba, G., 1995, "Friction Damping of Turbine Blade Vibrations Using a Microslip Model," *Journal of Machine Vibration*, **4**, pp. 2–7.
- [15] Pfeiffer, F., and Hajek, M., 1992, "Stick-Slip Motion of Turbine Blade Dampers," *Philosophical Transactions of the Royal Society, London*, **A338**, No. 1651, pp. 503–517.
- [16] Sextro, W., Popp, K., and Wolter, T., 1997, "Improved Reliability of Bladed Disks to Friction Dampers," presented at the ASME Int. Gas Turbine Aeroeng Congress & Exhibition, Orlando, June 2–5, ASME paper no. 97-GT-189.
- [17] Csaba, G., and Andersson, M., 1997, "Optimization of Friction Damper Weight, Simulation and Experiments," presented at the ASME TurboExpo 97, Orlando, FL, June 2–5, 97-GT-115.
- [18] Csaba, G., 1999, "Modelling of a Microslip Friction Damper Subjected to Translation and Rotation," presented at the ASME International Gas Turbine Conference, June Indianapolis, IN.
- [19] Yang, B. D., and Menq, C. H., 1997, "Characterization of Contact Kinematics and Application to the Design of Wedge Dampers in Turbomachinery Blading, Part I and Part II," presented at the ASME TurboExpo 97, Orlando, FL, June 2–5, 97-GT-19 and 97-GT-20.
- [20] Pierre, C., Ferri, A. A., and Dowel, E. H., 1985, "Multi-Harmonic Analysis of Dry Friction Damped Systems Using an Incremental Harmonic Balance Method," *ASME J. Appl. Mech.*, **52**, pp. 958–964.
- [21] Cameron, T. M., and Griffin, J. H., 1989, "An Alternating Frequency/Time Domain Method for Calculating the Steady-State Response of Non-linear Dynamic Systems," *ASME J. Appl. Mech.*, **56**, pp. 149–154.
- [22] Ostachowicz, W., 1989, "The Harmonic balance Method for Determining the Vibration Parameters in Damped Dynamic Systems," *J. Sound Vib.*, **131**, No. 3, pp. 465–473.
- [23] Sanliturk, K. Y., Imregun, M., and Ewins, D. J., 1997, "Harmonic Balance Vibration Analysis of Turbine Blades With Friction Dampers," *ASME J. Vib. Acoust.*, **119**, pp. 96–103.
- [24] Sanliturk, K. Y., and Ewins, D. J., 1996, "Modelling Two-Dimensional Friction Contact and Its Application Using Harmonic Balance Method," *J. Sound Vib.*, **193**, No. 2, pp. 511–523.
- [25] Budak, E., and Ozguven, H. N., 1990, "A Method for Harmonic Response of Structures With Synmetrical Non-linearities," *Proc. 15th International Seminar on Modal Analysis and Structural Dynamics*, P. Sas, ed., Leuven, Belgium, Sept. Katholieke Universiteit Leuven, Leuven, Belgium, pp. 901–915.
- [26] Budak, E., and Ozguven, H. N., 1993, "Iterative Receptance Method for Determining Harmonic Response of Structures With Symmetrical Non-linearities," *Mech. Syst. Signal Process.*, **7**, No. 1, pp. 75–87.
- [27] Tanrikulu, O., Kuran, B., Ozguven, H. N., and Imregun, M., 1993, "Forced Harmonic Response Analysis of Nonlinear Structures Using Describing Functions," *AIAA J.*, **31**, No. 7, pp. 1313–1230.
- [28] Ozguven, H. N., 1987, "A New Method for Harmonic Response of Nonproportionally Damped Structures Using Undamped Modal Data," *J. Sound Vib.*, **117**, No. 2, pp. 313–328.
- [29] Hager, W. G., 1989, "Updating the Inverse of a Matrix," *SIAM Rev.*, **31**, No. 2, pp. 221–239.
- [30] Level, P., Moraux, D., Drazetic, P., and Tison, T., 1996, "On a Direct Inversion of the Impedance Matrix in Response Reanalysis," *Commun. Numer. Methods Eng.*, **12**, pp. 151–159.
- [31] Sanliturk, K. Y., Stanbridge, A. B., and Ewins, D. J., 1996, "Friction Damper Optimization for Turbine Blades: Experiments and Theoretical Predictions," *Proc. 6th Int. Conf. on Vibrations in Rotating Machinery*, Oxford, Sept. 9–12, I. MECH E, London, pp. 171–186, C500/019/96.

Friction Damper Optimization: Simulation of Rainbow Tests

K. Y. Sanliturk

Faculty and Mechanical Engineering,
Istanbul Technical University,
80191, Gumussuyu, Istanbul, Turkey

D. J. Ewins

Mechanical Engineering Department,
Centre of Vibration Engineering,
Imperial College of Science, Technology,
and Medicine,
London SW7 7BX, UK

R. Elliott

J. S. Green

Rolls-Royce Plc,
Derby DE24 8BJ, UK

Friction dampers have been used to reduce turbine blade vibration levels for a considerable period of time. However, optimal design of these dampers has been quite difficult due both to a lack of adequate theoretical predictions and to difficulties in conducting reliable experiments. One of the difficulties of damper weight optimization via the experimental route has been the inevitable effects of mistuning. Also, conducting separate experiments for different damper weights involves excessive cost. Therefore, current practice in the turbomachinery industry has been to conduct so-called "rainbow tests" where friction dampers with different weights are placed between blades with a predefined configuration. However, it has been observed that some rainbow test results have been difficult to interpret and have been inconclusive for determining the optimum damper weight for a given bladed-disk assembly. A new method of analysis—a combination of the harmonic balance method and structural modification approaches—is presented in this paper for the analysis of structures with friction interfaces and the method is applied to search for qualitative answers about the so-called "rainbow tests" in turbomachinery applications. A simple lumped-parameter model of a bladed-disk model was used and different damper weights were modeled using friction elements with different characteristics. Resonance response levels were obtained for bladed disks with various numbers of blades under various engine-order excitations. It was found that rainbow tests, where friction dampers with different weights are used on the same bladed-disk assembly, can be used to find the optimum damper weight if the mode of vibration concerned has weak blade-to-blade coupling (the case where the disk is almost rigid and blades vibrate almost independently from each other). Otherwise, it is very difficult to draw any reliable conclusion from such expensive experiments. [DOI: 10.1115/1.1391278]

1 Introduction

Friction dampers have been widely used in turbomachinery applications for a considerable period of time in order to provide mechanical damping to reduce resonance stresses. A typical application of the dry friction damping concept in gas turbines is the so-called "friction damper," or underplatform damper, which is loaded by centrifugal force against the underside of the platforms of two adjacent blades. The main design criterion for such dampers is to determine the optimum damper mass for a given configuration in order to reduce the dynamic stresses by the maximum possible extent. If the damper mass is too small, the friction force will not be large enough to dissipate sufficient energy. On the other hand, if the damper mass is too large, it will "stick," limiting the relative motion across the interface and thus the amount of energy dissipation. In both cases, the friction damper will be inefficient and between these two extremes there is an optimum mass.

Although a substantial effort has been devoted to understanding, modeling, and optimization of friction dampers for turbomachinery applications including: (i) modeling the basic contact characteristics, usually in the form of friction force-displacement hysteresis loops ([1–6]), (ii) modeling the friction damper element incorporating the basic contact characteristics ([7–9]) in (i), and (iii) developing analysis methods and application of these for friction damper optimization in practice ([10–15]). Although significant advances have been made in all these three categories it is still difficult to rely on computer-based predictions alone for assessing the response amplitude of turbomachinery blading and for optimizing the friction interfaces. This is due mainly to the

marked nonlinearity of the contact mechanisms and the uncertainties of the actual dynamic forces acting on the blades. This situation has led the aero-engine manufacturers to rely mainly on previous experience and empirical data obtained from either simplified test rigs comprising a single or group of blades, or from more realistic, albeit more expensive, spin tests using a complete bladed-disk assembly which includes most of the important factors. Although most current research is focussed on validating theoretical models and methods so as to minimize these expensive experiments it is still common practice to carry out spin tests on complete bladed-disk assemblies in order to assess the effectiveness of the underplatform dampers. Conducting separate spin tests for different damper weights is, however, rarely employed in industry, due to excessive costs. Instead, so-called rainbow tests are adopted, where friction dampers with different weights are placed between adjacent blades with a predefined configuration as schematically illustrated in Fig. 1 for a special case of three different types of damper being installed. The experimental route, however, also has numerous difficulties in conducting reliable friction damping tests. More often than not, results obtained from such experiments have proven to be inconclusive for determining the optimum damper weight for a given bladed-disk assembly. One of the difficulties of damper weight optimization via the experimental route has been the inevitable effects of mistuning ([16–21]). There are mainly two problems associated with mistuning during damper optimization: (i) the difficulty of distinguishing the effect of mistuning from that of the dampers and (ii) in general, not being able to instrument every blade on an assembly, which makes it very likely that the maximum response levels experienced by a single blade will not be detected. Some recent measurement techniques using noncontact measurement systems allow (in principle) monitoring all the blades on a bladed-disk assembly although the interpretation of the results is also hampered by mistuning effects ([19]). Most of the research published in the literature is focussed on investigating the effects of stiffness and mass mistuning, resulting in variations of individual blade

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, Indianapolis, IN, June 7–10, 1999; ASME Paper 99-GT-336. Manuscript received by IGTI Oct. 1998; final revision received by the ASME Headquarters Mar. 1999. Associate Editor: D. Wisler.

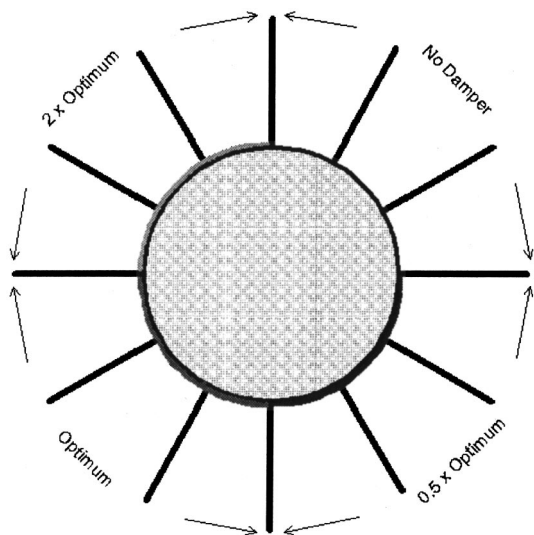


Fig. 1 Schematic illustration of "rainbow" tests

frequencies, some including the effect of uniform dry friction ([22]). One of the conclusions of [22] is that friction damped systems are more prone to localized vibrations. A number of researchers have also studied the effect of damping variations when all the blades within a bladed-disk are damped ([23,24]).

A recent study ([25]) investigated, using a linear analysis approach, whether accurate damping measurements for a single blade could be obtained by testing a bladed-disk with only a few blades damped. This paper is an extension of [25] and it seeks to identify those circumstances under which rainbow tests can be used for determining the optimum damping condition. Furthermore, it tries to establish how many blades need to be instrumented in such tests. The additional important and unique features of this paper are (i) the friction dampers are modeled as nonlinear friction elements, the analysis procedure being based on the harmonic balance method and (ii) a method based on the structural modification approach is used in conjunction with the harmonic balance method to analyze structures with friction interfaces efficiently.

2 Model Description

The lumped-parameter model used in this study is a variation of the bladed-disk model originally proposed by Dye and Henry [26], as shown in Fig. 2. A single mass (m) is used to model the blade while the other mass (M_d) represents the effective mass at the platform location and includes the sectorial mass of the disk as well as a proportion of the blade's mass. The dashpot attached between ground and the blade mass represents aerodynamic damping. The flexibility of the blade and the disk are also in-

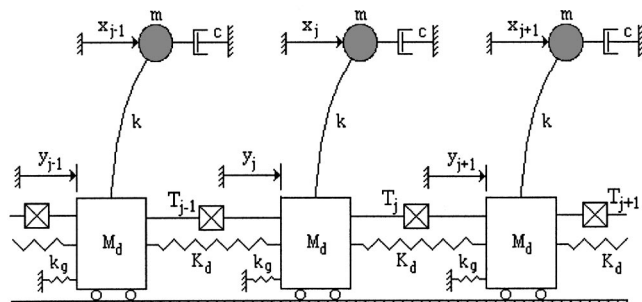


Fig. 2 Bladed-disk model with friction elements

cluded and friction dampers are introduced between the lumped masses, M_d , indicated by a crossed box. The current friction dampers in practice are usually wedged shaped and their vibration characteristics are much more complicated than the simple one-dimensional damper model used here presented here. This simple model is, however, considered appropriate for the objective of this paper. A realistic model for wedge-shaped dampers, which is based on both measurements and theory, is addressed in [27]. The system shown in Fig. 2 can be described by the familiar equation as

$$[M]\{\ddot{q}\} + [C]\{\dot{q}\} + [K]\{q\} = \{f(t)\} + \{r(t)\} \quad (1)$$

where M , C , K are the mass, viscous damping, and stiffness matrices; and $f(t)$ and $r(t)$ are the external and friction forces due to damping, respectively. Assuming harmonic motion leads to the well-known counterpart of Eq. (1), as

$$([K] + i\omega[C] - \omega^2[M])\{Q\} = \{F\} + \{R\} \quad (2)$$

where ω is the frequency and $\{F\}$ and $\{R\}$ are the Fourier Transforms of $f(t)$ and $r(t)$, respectively. The external forcing considered here, $\{F\}$ is of the engine-order type, its magnitude being unity as in [25], applied to the blade coordinates only while $\{R\}$ represents the first harmonic components of the resulting friction forces which are applied to the platform coordinates.

3 Analysis Method

The new analysis method proposed in this paper is a combination of the harmonic balance method and a structural modification approach. These two approaches, and how to combine them for an efficient analysis tool to analyze structures, are described below. The solution algorithm adopted in the frequency domain is based on finding the response amplitudes iteratively, the starting point being the response levels of the underlying linear system. The behavior of the friction dampers is analyzed at a given relative response amplitude between the damper connection points and the individual dampers are represented as equivalent complex stiffnesses, representing both restoring and energy dissipation characters. The equivalent complex stiffnesses are then added to the

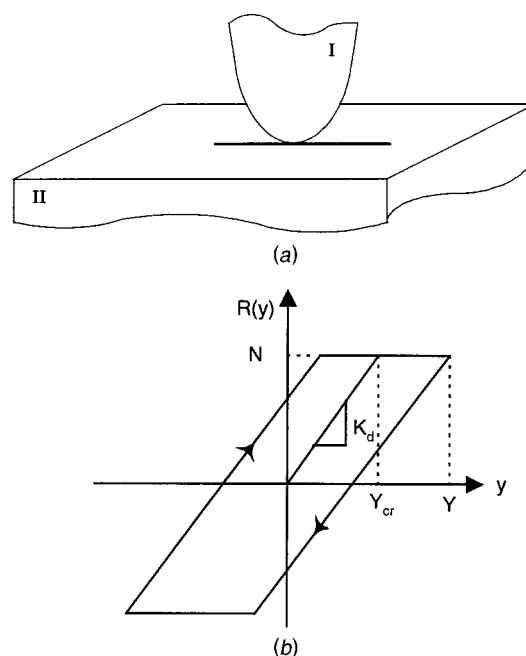


Fig. 3 (a) One-dimensional friction damper model, (b) force-displacement model of macroslip model

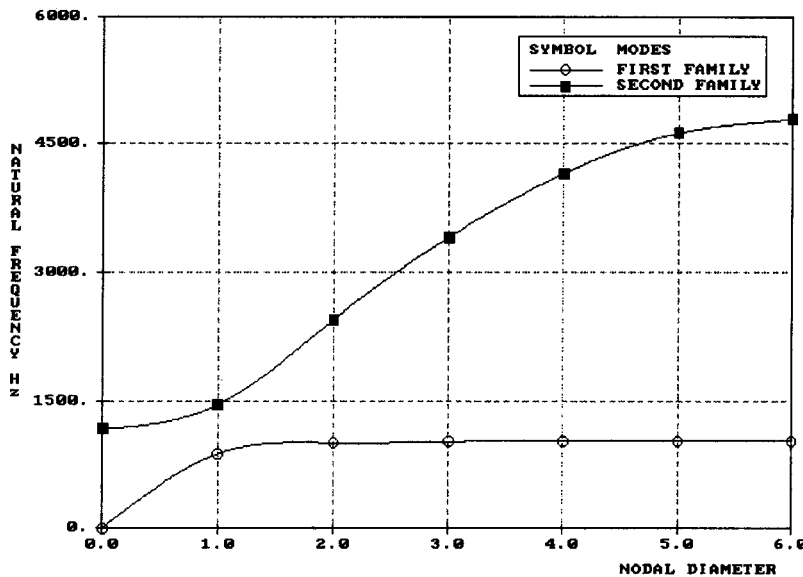


Fig. 4 Natural frequencies of the first and second family modes

otherwise linear system and the response levels of the modified system are calculated again, the procedure being repeated until convergence is achieved.

The following subsections are describing these two stages—representing a friction damper as a complex stiffness at a given relative response amplitude and—modifying the otherwise linear system to include frictional effects.

3.1 Harmonic Balance Approach. The friction damper element indicated by a crossed box in Fig. 2 is in fact a simplified representation of the situation depicted in Fig. 3(a) where two surfaces rub against each other along a line, parts I and II, representing the platforms of the neighboring blades. The macro-slip friction model of Fig. 3(b) has been used extensively in the analysis of various nonlinear systems and it will also be used in this paper although real contact characteristics can be quite different than that of macro-slip model ([6]). It should be noted, however, that the analysis method in this paper is equally applicable to any other type of hysteresis loop model.

In the traditional analysis of structures with friction joints, the nonlinear friction forces are calculated in an iterative fashion and are usually considered as external forces. As in [9,15] this paper converts these forces into amplitude (and phase)-dependent equivalent stiffness parameters for CPU reduction as well as numerical stability.

Now, let us consider the nonlinear force-displacement relationship of a friction damper element, denoted by T_j in Fig. 2. As mentioned earlier, the characterization of the damper is carried out at a given relative displacement as

$$Z_j = Y_{j+1} - Y_j \quad (3)$$

where Y_{j+1} and Y_j are complex quantities representing the platforms of neighboring blades. Z_j is also a complex quantity whose motion for a cycle can be described more explicitly as

$$z_j(t) = |Z_j| \cos(\omega t + \phi_j) = |Z_j| \cos(\theta_j) \quad (4)$$

where ϕ_j is the phase angle and $\theta_j = \omega t + \phi_j$. Now, let us consider the nonlinear force-displacement relationship of a friction damper, acting between coordinates y_{j+1} and y_j , given by (the subscripts will be omitted for clarity)

$$R = R(z) \quad (5)$$

where R is the friction force acting between the neighboring platforms. Then, the linearized stiffness coefficient, or the describing function of friction element, k_{eq}^* , can be written as ([9])

$$k_{eq}^*(Z) = k_{eq}^r(Z) + i k_{eq}^i(Z) \quad (6)$$

where k_{eq}^r and k_{eq}^i are the amplitude-dependent real and imaginary parts of the equivalent stiffness, respectively, as given by

$$k_{eq}^r = \frac{1}{\pi|Z|} \int_0^{2\pi} R(|Z|\cos(\theta)) \cos(\theta) d\theta \quad (7a)$$

$$k_{eq}^i = \frac{-1}{\pi|Z|} \int_0^{2\pi} R(|Z|\cos(\theta)) \sin(\theta) d\theta. \quad (7b)$$

If the system is tuned and the friction dampers are identical, all blades will experience the same level of vibration and the equivalent complex stiffness would be the same for all the friction dampers. However, friction dampers with different characteristics will result in blades having different vibration levels and these will, in turn, result in individual dampers having different equivalent stiffness parameters.

3.2 An Efficient Analysis Method for Structures With Friction Joints. The so-called Sherman-Morrison formula has already been proposed in the literature ([28]) to calculate the frequency response of a (linear) modified structure. It is shown in [28] that the Sherman-Morrison identity allows a direct inversion of the modified matrix efficiently using the data related to the initial matrix and to the modification. A brief summary of the Sherman-Morrison formula is appropriate here.

Let $[A]^{-1}$ be the inverse of a nonsingular square matrix, $[A]$. If the inverse of a modified matrix, $[A']^{-1}$, is needed where $[A']$ is of the form

Table 1 Linear structural parameters

$\frac{1}{2\pi} \sqrt{k/m} = 1033.7 \text{ Hz}$	$\frac{\sqrt{K_d/M_d}}{\sqrt{k/m}} = 2.3$
$k = 2.109 \text{ MN/m}$	$K_d = 40.17 \text{ MN/m}$
$k_g = 63.9 \text{ N/m}$	$c = 3.247 \text{ Ns/m}$

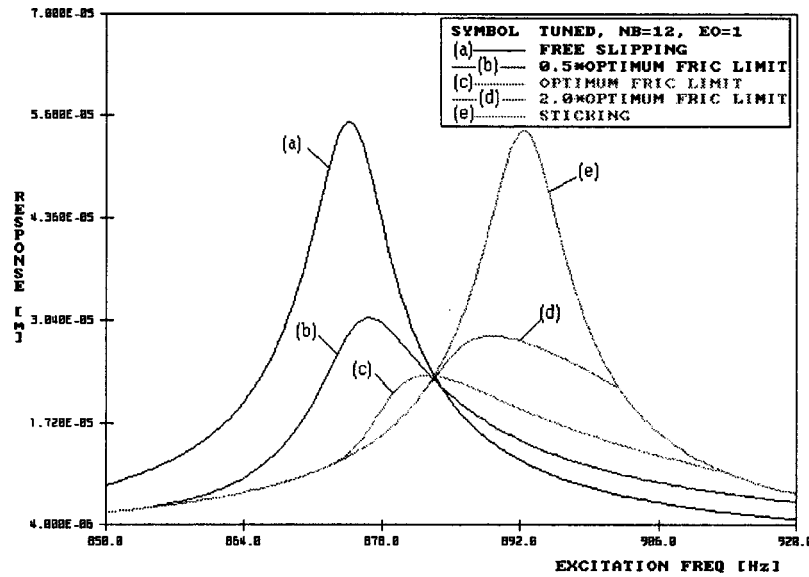


Fig. 5 Tuned response levels for various friction limits

$$[A'] = [A] + \{u\}\{v\}^T, \quad (8)$$

it can be calculated using the Sherman-Morrison formula as

$$[A']^{-1} = [A]^{-1} - \frac{([A]^{-1}\{u\})(\{v\}^T[A]^{-1})}{1 + \lambda} \quad (9)$$

where

$$\lambda = \{v\}^T[A]^{-1}\{u\}. \quad (10)$$

It should be noted that if $[A]^{-1}$ is known, Eq. (9) does not require any further matrix inversion to find the inverse of $[A']^{-1}$. The generalization of Eq. (9) is also available and is known as Sherman-Morrison-Woodbury formula which considers a modification as a product of two rectangular matrices such as $[U] \times [V]^T$. A more detailed coverage of this approach and the numerical aspects are discussed in [29].

The aim of this paper is quite different in the sense that the purpose of the analysis is to calculate the nonlinear response lev-

els of structures with localized nonlinearities rather than the linear modification analysis as reported in the above literature. The analysis method presented here treats the linear and the nonlinear parts of a structure separately, the linear part being the structure excluding the nonlinear parts. The nonlinear part is considered as a linearized modification to the original system, the linearized parameters being obtained using the harmonic balance method as described in the previous section. Suppose that the linear structure is given by its dynamic stiffness matrix $[Z]$ and its frequency response function matrix $[\alpha]$, $[\alpha] = [Z]^{-1}$, and the modification matrix to be made to $[Z]$ is $[\Delta]$. The dynamic stiffness matrix of the modified system $[Z']$ can then be written as

$$[Z'] = [Z] + [\Delta]. \quad (11)$$

If the modification matrix is written of the form

$$[\Delta] = \{u\}\{v\}^T, \quad (12)$$

the FRF matrix of the modified system $[\beta]$ can be computed from

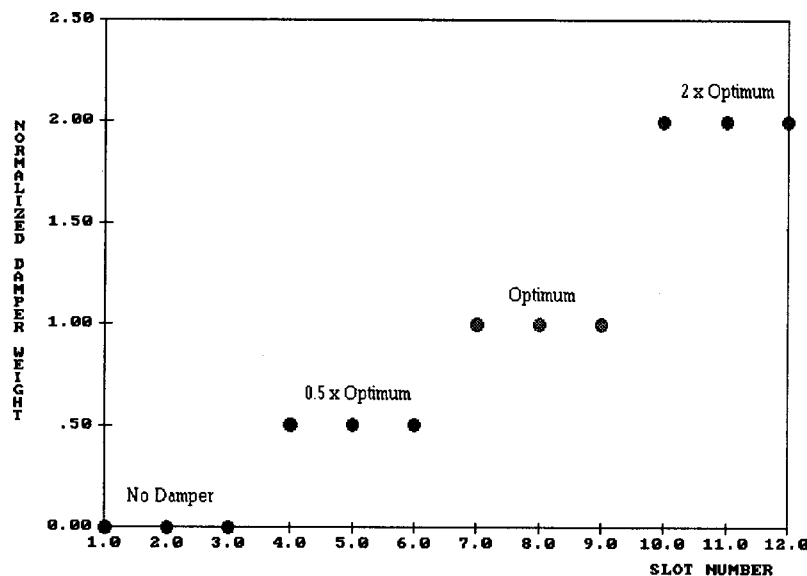
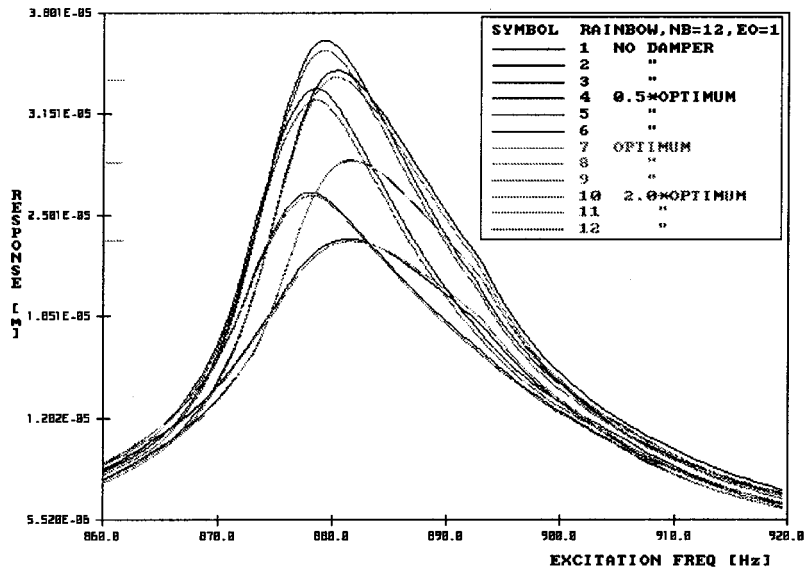
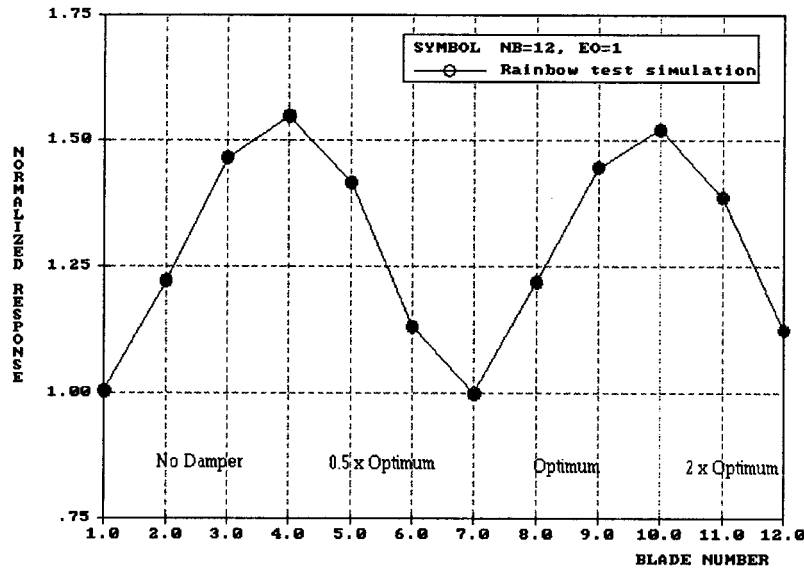


Fig. 6 Distribution of friction dampers



(a)



(b)

Fig. 7 (a) Blades' response levels versus excitation frequency ($N=12$, $EO=1$); (b) blades' maximum response levels with friction dampers ($N=12$, $EO=1$)

$$[\beta] = [Z']^{-1} = [\alpha] - \frac{([\alpha]\{u\})(\{v\}^T[\alpha])}{1 + \{v\}^T[\alpha]\{u\}} \quad (13)$$

which allows the FRF matrix of the modified system to be calculated without any matrix inversion. It should be noted that if the total modification matrix $[\Delta]$ cannot be written as a multiplication of two vectors as in Eq. (12), it can be decomposed into several, say p , modification matrices, such as

$$[\Delta] = [\Delta_1] + [\Delta_2] + [\Delta_3] + \dots + [\Delta_p] \quad (14)$$

where $[\Delta_i] = \{u_i\}\{v_i\}^T$. This allows the FRF of the system to be calculated by considering each $[\Delta_i]$ individually.

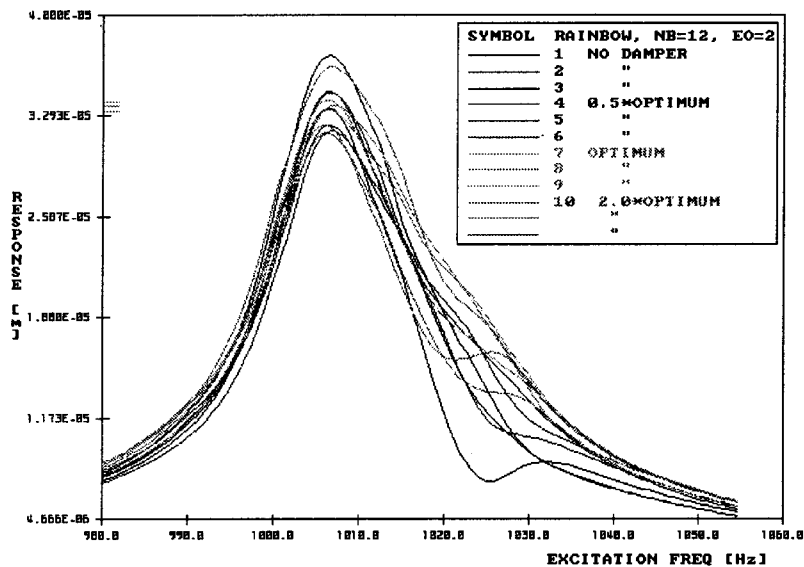
It is also possible that the solutions can be obtained very efficiently at active coordinates only, active coordinates being the nonlinear coordinates, excitation coordinates, and the other coordinates where the response levels are needed. This approach allows local solutions to be obtained no matter how large the whole

model is, as addressed in [27] including the application of this method to realistic turbine blades with friction dampers and validation of the predictions by experiments.

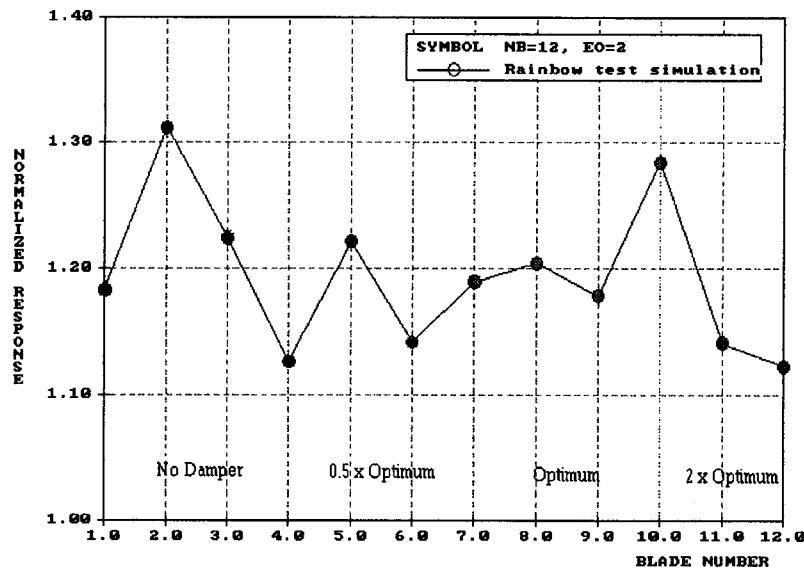
4 Analysis and Results

As the primary objective of this work is to investigate qualitative aspects of rainbow tests and to demonstrate the new analysis procedure, it was convenient to keep the number of blades reasonably small. Therefore, the bladed-disk system studied here had 12 blades, although a case with 24 blades is also studied in order to investigate the effect of varying the number of blades. However, the blade-to-blade coupling ratio was selected such that the bladed-disk studied here had a similar first family characteristic (see Fig. 4) to that of a realistic turbine stage. The corresponding linear structural parameters are listed in Table 1.

Having determined the linear structural model, the next step



(a)



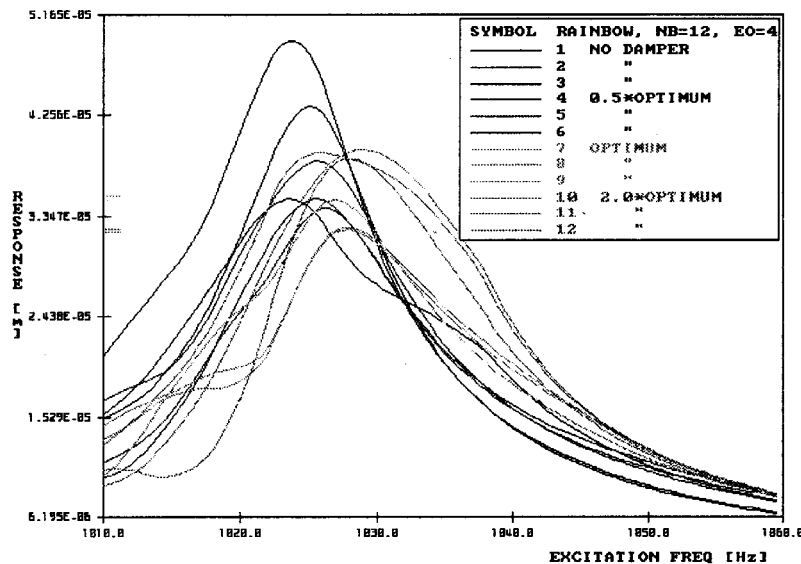
(b)

Fig. 8 (a) Blades' response levels versus excitation frequency ($N=12$, $EO=2$); (b) blades' maximum response levels with friction dampers ($N=12$, $EO=2$)

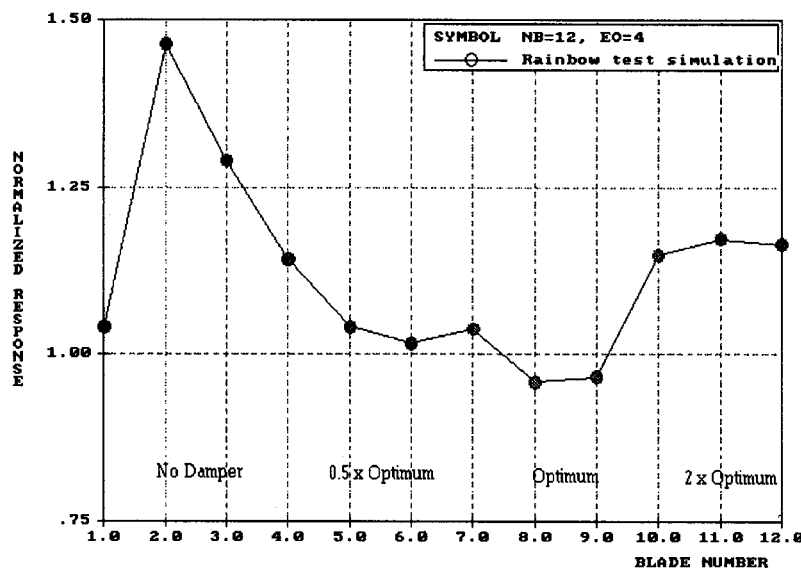
was to determine the parameters for the friction dampers. Macro-slip elements were selected to simulate friction dampers since it was easier to relate friction limits to damper weights. The contact stiffness of the macro-slip element was assumed to be the same for all dampers, and was chosen to give a clear natural frequency shift between slipping and sticking cases. A typical response plot is illustrated in Fig. 5 (free-slipping curve) for a tuned 12-bladed disk subjected to a first engine order (1EO) excitation (referring to Fig. 4, this particular mode of vibration can be classified as a mode with relatively strong blade-to-blade coupling). It is worth here to emphasize the definition of weak and strong blade-to-blade coupling used in this paper as some researchers use the same terminology for different meanings. The authors of this paper prefer to use the definition of "weak blade-to-blade coupling" for those cases where the disk is almost rigid and the assembly natural frequencies approach the cantilevered blade alone frequency (i.e., the stiffer the disk, the weaker the coupling hence zero coupling means rigid disk). Strong blade-to-blade coupling,

on the other hand, refers to the other extreme where the disk is quite flexible. One way of quantifying the degree of coupling strength is to define a coupling ratio CR , $CR = (1 - (\omega_{as}/\omega_n)^2)$, where ω_{as} is the tuned bladed-disk assembly frequency for a given nodal diameter mode of vibration and ω_n is the blade alone cantilever frequency.

The corresponding response levels of the tuned system were calculated for various friction limits (normal load times friction coefficient) in order to determine the optimum damper friction limit which is directly related to optimum damper weight. The analysis was done using a dedicated program based on the analysis procedure summarized in the previous section. It should be stressed that knowing the optimum friction damper beforehand is an essential part of this simulation since the rest of the study aims to determine whether the forced response levels of the blades with different friction dampers can be used to identify this known op-



(a)



(b)

Fig. 9 (a) Blades' response levels versus excitation frequency ($N=12$, $EO=4$); (b) blades' maximum response levels with friction dampers ($N=12$, $EO=4$)

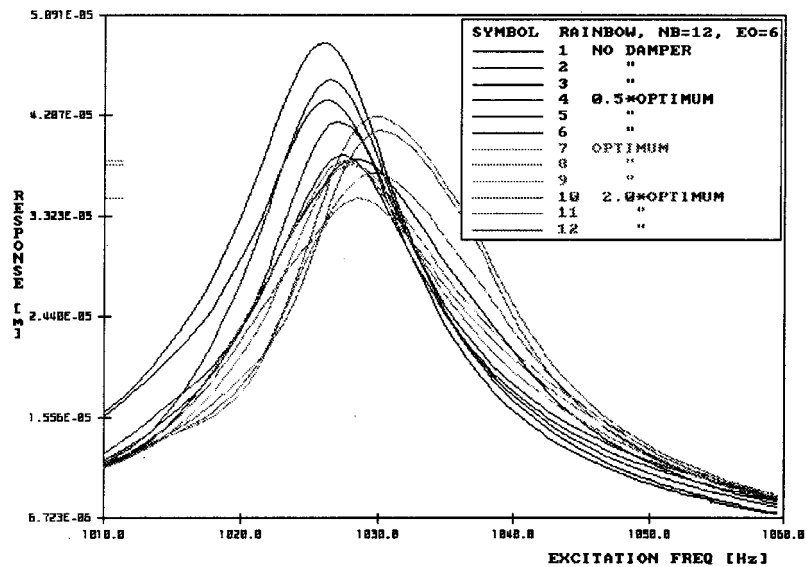
timus damper. The tuned system response amplitudes for the cases of the optimum damper weight, half of the optimum and twice the optimum, are illustrated in Fig. 5.

A particular rainbow test configuration studied here is illustrated in Fig. 6. It is seen that there are three different sizes of damper around the disk: optimum, half optimum, and twice the optimum in terms of damper weight. It should also be noted that some blades do not have any dampers at all. This type of simulation is quite representative of the actual tests in practice as it is common during this sort of test to install relatively heavy, normal, and light dampers so as to determine which one will produce the maximum damping. It should be stressed that the optimum damper in this simulation is known beforehand, the whole idea is to examine whether the measured results can identify this fact.

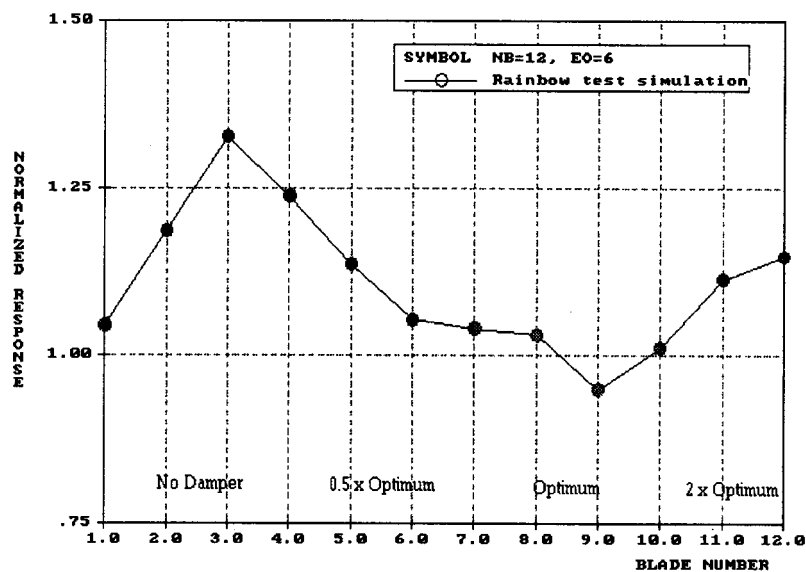
As can be seen in Fig. 6, the dampers are identified as "No Damper," "0.5×Optimum," "Optimum," and "2×Optimum" and this convention will be used throughout the rest of the plots so that results are presented in a consistent manner.

Most of the results were obtained for a 12-bladed disk, a typical set of results being illustrated in Fig. 7(a) for 1 engine-order (1EO) excitation. It is seen that the response levels of those blades with optimum dampers do not give any indication that those blades are the ones with optimum friction dampers (the maximum response levels of those blades with optimum dampers are identified by marks along the vertical axis in Fig. 7(a) and in other similar plots). The maximum response levels of individual blades in Fig. 7(a) were found and the results are presented in Fig. 7(b) (the maximum response levels were normalized to the tuned maximum response level with optimum friction damper). It is clear that there is no correlation between response amplitude and the damper weight but there is a strong suggestion that the maximum response amplitudes are determined by the mode shape rather than the distribution of dampers around the disk for this particular case.

A similar analysis was performed for the same bladed-disk assembly under different EO excitations. Each time, the optimum



(a)



(b)

Fig. 10 (a) Blades' response levels versus excitation frequency ($N=12$, $EO=6$); (b) blades' maximum response levels with friction dampers ($N=12$, $EO=6$)

damper weight was determined by analysing the tuned system under various friction limit conditions (different damper weights), and the response levels of the blades corresponding to the friction damper distribution, as shown in Fig. 5, were determined. The results, similar to those in Fig. 7, are presented in Figs. 8–10, the difference being the order of the EO excitation. Inspection of these figures (Figs. 7 to 10) reveals that there is no relationship between blade response levels and the damper weights for low EO excitations. However, as the order of the excitation increases (see Figs. 9 and 10), a pattern starts to emerge, showing that those blades with optimum friction dampers tend to experience minimum response levels. It should be noted that a specified EO excitation predominantly excites the corresponding nodal diameter modes and blade-to-blade coupling decreases with increasing nodal diameter mode of vibration. Therefore, the argument above in terms of EO can be equally valid in terms of nodal diameter mode of vibration, or corresponding strength of blade-to-blade coupling.

So far, all the results presented here were for 12-bladed disk under various EO excitations. Similar calculations were also performed for a 24-bladed disk under 2 EO and 12 EO excitations so as to verify that the previous findings are not specific to the particular bladed disk studied here. Results presented in Fig. 11 fully support the findings from the 12-bladed disk study. This is not surprising since there are very strong indications that the underlying parameter for forced vibration characteristics of a bladed disk is the coupling between the disk and the blade rather than number of blades or the order of the excitation alone ([21]).

All the findings in this study suggest that rainbow tests, where friction dampers with different weights are used on the same bladed-disk assembly, can be used to find the optimum damper weight if and only if the mode of vibration concerned has very weak blade-to-blade coupling (the case where the disk is almost rigid and blades vibrate almost independently from each other). Otherwise, it seems that it may be very difficult to draw any reliable conclusion from such expensive experiments. It is inter-

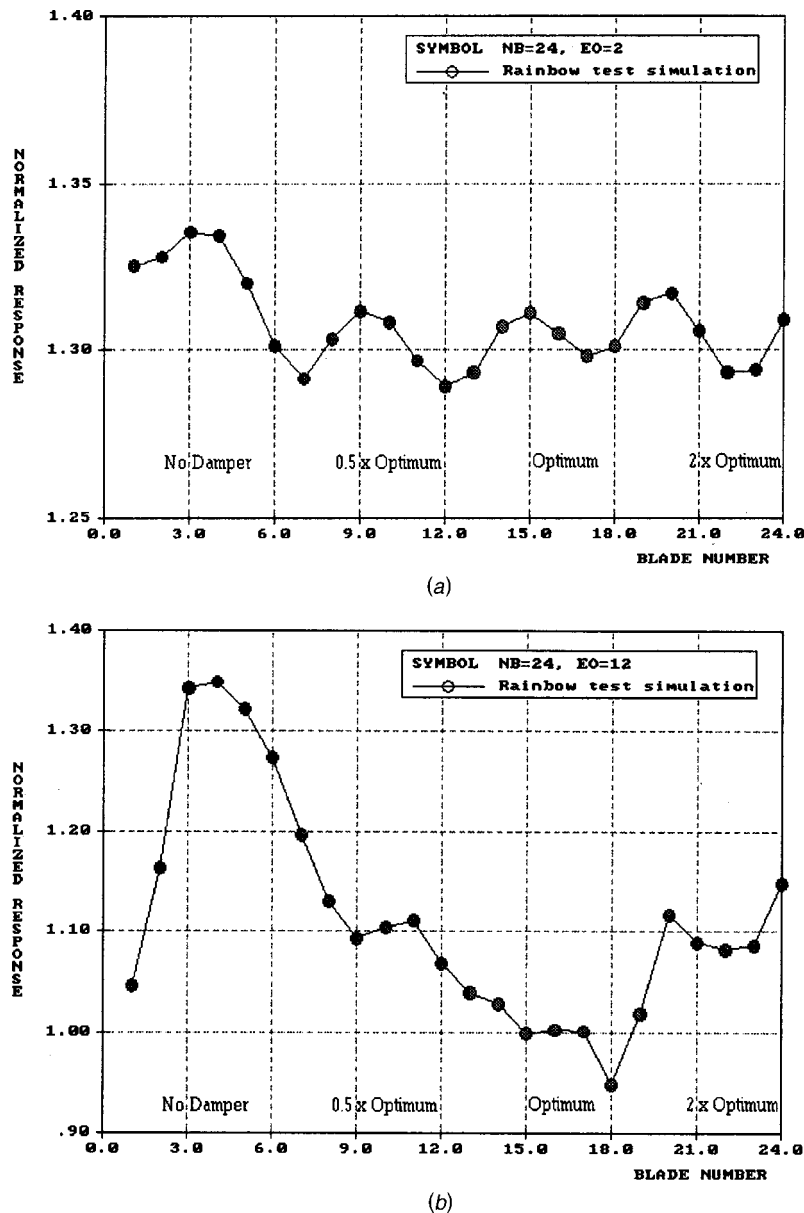


Fig. 11 (a) Blades' maximum response levels with friction dampers ($N=24$, $EO=2$); (b) blades' maximum response levels with friction dampers ($N=24$, $EO=12$)

esting to note that the nonlinear analysis method here yielded the same conclusion as that of [25] even though their simulation was based on a linear analysis.

Another important qualitative finding of this study is related to the number of blades that will require instrumenting in such experiments. The response levels of all the blades were assumed to be "measured" in this simulation in order to identify the conditions where rainbow tests can and cannot produce satisfactory results. In practice, however, it is hardly possible to monitor all the blades around the disk, especially for disks with large number of blades. The results presented in this paper also give some guide as to which and how many blades need instrumentation. Inspection of all the results suggest that at least one blade from each group needs to be instrumented although instrumenting two blades from each group is expected to yield a more reliable assessment of the results. Furthermore, it is better to instrument those blades which are close to the middle of a group of dampers around the circumference. It is expected, however, that this finding may not

be applicable to other situations where the distribution of dampers are quite different than what is examined here, nor for those cases where the effect of other sources of mistuning is stronger than that provided by the friction dampers.

It is worth restating here that the mistuning due to the differences between blades' mechanical properties are deliberately excluded from the study reported in this paper. The main reason for this exclusion was to establish an upper limit of what to expect from rainbow test results under idealistic conditions. It is, however, necessary to include these effects in order to make more realistic assessment of such expensive tests as the additional mistuning effects are inevitable in practical tests. It is quite likely that the results of a rainbow test may not indicate the optimum damper weight, even for lightly coupled bladed disks, when the additional mistuning level exceeds a certain threshold. The simulation of this situation requires a better damper model as well as more realistic (empirical) contact parameters for the friction dampers, so that the

relative importance of the nonlinear damping and blade-alone mistuning can be identified and their effects can be distinguished from each other.

5 Conclusions

A new method has been presented for the analysis of structures with friction interfaces. This method is a combination of the harmonic balance method and a structural modification approach and is based on modifying the otherwise-linear structure with the amplitude-dependent equivalent complex stiffness representing both restoring and energy dissipation characteristics of joints.

This method of analysis has been applied to investigate whether, and under which conditions, the so-called "rainbow" tests can be used for damper optimization purposes in turbomachinery applications. Results of the rainbow-test simulation presented in this paper suggest that these tests can be used to determine the optimum damper weight only if there is a weak blade-to-blade coupling for the mode of vibration concerned. It has also been found that instrumenting two blades from each group of dampers is expected to yield reliable assessment of the results when other sources of mistuning are negligible.

Although the results of this investigation have established the condition when rainbow tests can be used to identify optimum damper weight, a more detailed and representative analysis need to be carried out in order to establish the effect of additional mistuning present in these tests.

Acknowledgments

The authors are grateful to their sponsor, Rolls-Royce plc, for their financial support and for permission to publish this work.

References

- [1] Goodman, L. E., and Brown, C. B., 1962, "Energy Dissipation in Contact Friction: Constant Normal and Cyclic Tangential Loading," *ASME J. Appl. Mech.*, **29**, pp. 17–22.
- [2] Iwan, W. D., 1967, "On a Class of Models for the Yielding Behaviour of Continuous and Composite Systems," *ASME J. Appl. Mech.*, **34**, pp. 612–617.
- [3] Rogers, P. F., and Boothroyd, G., 1975, "Damping at Metallic Interfaces Subjected to Oscillating Tangential Loads," *J. Eng. Ind.*, **97**, pp. 1087–1093.
- [4] Burtekin, M., Cowley, A., and Back, N., 1978, "An Elastic Mechanism for the Micro-sliding Characteristics Between Contacting Machined Surfaces," *J. Mech. Eng. Sci.*, **20**, No. 3, pp. 121–127.
- [5] Shoukry, S. N., 1985, "A Mathematical Model for the Stiffness of Fixed Joints Between Machine Parts," J. Middleton and G. N. Pande, eds., *Proc. Int. NUMETA Conf.*, Swansea, U.K., A. A. Balkema, Rotterdam, pp. 851–858.
- [6] Sanliturk, K. Y., Stanbridge, A. B., and Ewins, D. J., 1995, "Friction Dampers: Measurement, Modelling and Application to Blade Vibration Control," *Proc. Des. Eng. Conf.*, Vol. 3, Part B, DE-Vol. 84-2, ASME, New York, pp. 1377–1382.

- [7] Pfeiffer, F., and Hajek, M., 1992, "Stick-Slip Motion of Turbine Blade Dampers," *Philos. Trans. R. Soc. London, Ser. A*, **338**, No. 1651, pp. 503–517.
- [8] Meng, C.-H., Bielak, J., and Griffin, J. H., 1986, "The Influence of Microslip on Vibratory Response, Part I: A New Microslip Model," *J. Sound Vib.*, **107**, No. 2, pp. 279–293.
- [9] Sanliturk, K. Y., and Ewins, D. J., 1996, "Modelling Two-Dimensional Friction Contact and Its Application Using Harmonic Balance Method," *J. Sound Vib.*, **193**, No. 2, pp. 511–523.
- [10] Meng, C.-H., and Griffin, J. H., 1985, "A Comparison of Transient and Steady State Finite Element Analysis of the Forced Response of a Frictionally Damped Beam," *ASME J. Vibr. Acoust.*, **107**, pp. 19–25.
- [11] Pierre, C., Ferri, A. A., and Dowel, E. H., 1985, "Multi-Harmonic Analysis of Dry Friction Damped Systems Using an Incremental Harmonic Balance Method," *ASME J. Appl. Mech.*, **52**, pp. 958–964.
- [12] Cameron, T. M., and Griffin, J. H., 1989, "An Alternating Frequency/Time Domain Method for Calculating the Steady-State Response of Non-linear Dynamic Systems," *ASME J. Appl. Mech.*, **56**, pp. 149–154.
- [13] Ostachowicz, W., 1989, "The Harmonic Balance Method for Determining the Vibration Parameters in Damped Dynamic Systems," *J. Sound Vib.*, **131**, No. 3, pp. 465–473.
- [14] Cameron, T. M., Griffin, J. F., Kielb, R. E., and Hoosac, T. M., 1990, "An Integrated Friction Damper Design," *ASME J. Vibr. Acoust.*, **112**, pp. 175–182.
- [15] Sanliturk, K. Y., Imregun, M., and Ewins, D. J., 1997, "Harmonic Balance Vibration Analysis of Turbine Blades With Friction Dampers," *ASME J. Vibr. Acoust.*, **119**, pp. 96–103.
- [16] Muszynska, A., and Jones, D. I. G., 1983, "On Tuned Bladed Disk Dynamics: Some Aspects of Friction Related Mistuning," *J. Sound Vib.*, **86**, No. 1, pp. 107–128.
- [17] Ewins, D. J., and Han, Z. S., 1984, "Resonant Vibration Levels of a Mistuned Bladed Disk," *ASME J. Vibr. Acoust.*, **106**, pp. 211–217.
- [18] Sanliturk, K. Y., Imregun, M., and Ewins, D. J., 1992, "Statistical Analysis of Random Mistuning of Bladed Assemblies," *IMECH Proc. Int. Conf. on Vib. in Rotating Machinery*, Univ. of Bath, UK., Sept. 7–10, pp. 51–58.
- [19] Schaber, U., 1997, "Non-Contact Vibration Measurements of Mistuned Coupled Blades," Paper No. 97-GT-190.
- [20] Chen, S., and Sinha, A., 1990, "Probabilistic Method to Compute the Optimal Slip Load for a Mistuned Bladed Disk Assembly With Friction Dampers," *ASME J. Vibr. Acoust.*, **112**, pp. 214–221.
- [21] Sanliturk, K. Y., 1992, "Vibration Analysis of Mistuned Bladed Systems," Ph.D. thesis, Mechanical Eng. Dept., Imperial College, University of London.
- [22] Wei, S.-T., and Pierre, C., 1989, "Effects of Dry Friction Damping on the Occurrence of Localised Forced Vibrations in Nearly Cyclic Structures," *J. Sound Vib.*, **129**, No. 2, pp. 379–416.
- [23] 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**, No. 2, pp. 525–543.
- [24] Sextro, W., Popp, K., and Wolter, I., 1997, "Improved Reliability of Bladed Disc to Friction Dampers," *ASME Paper No. 97-GT-189*.
- [25] Hollkamp, J. J., Hustedde, C. L., and Gordon, R. W., 1997, "Effects of Damping on a Blade-Disk Assembly," *AIAA Paper No. AIAA-97-1395*.
- [26] Dye, R. C. F., and Henry, T. A., 1969, "Vibration Amplitudes of Compressor Blades Resulting From Scatter in Blade Natural Frequencies," *ASME J. Eng. Gas Turbines Power*, **91**, pp. 182–188.
- [27] Sanliturk, K. Y., Ewins, D. J., and Stanbridge, A. B., 1999, "Underplatform Dampers for Turbine Blades: Theoretical Modeling, Analysis and Comparison with Experimental Data," *ASME J. Eng. Gas Turbines Power*, **123**, pp. 919–929.
- [28] Level, P., Moraux, D., Drazetic, P., and Tison, T., 1996, "On a Direct Inversion of the Impedance Matrix in Response Reanalysis," *Communications in Numerical Methods in Engineering*, **12**, pp. 151–159.
- [29] Hager, W. G., 1989, "Updating the Inverse of a Matrix," *SIAM Rev.*, **31**, No. 2, pp. 221–239.

Experimental Investigation of Mode Localization and Forced Response Amplitude Magnification for a Mistuned Bladed Disk

J. Judge

Graduate Student

C. Pierre

Professor
Fellow ASME

Department of Mechanical Engineering,
The University of Michigan,
Ann Arbor, MI 48109-2125

O. Mehmed

Senior Research Engineer,
NASA Glenn Research Center,
Cleveland, OH 44135

The results of an experimental investigation on the effects of random blade mistuning on the forced dynamic response of bladed disks are reported. The primary aim of the experiment is to gain understanding of the phenomena of mode localization and forced response blade amplitude magnification in bladed disks. A stationary, nominally periodic, 12-bladed disk with simple geometry is subjected to a traveling-wave out-of-plane "engine order" excitation delivered via phase-shifted control signals sent to piezoelectric actuators mounted on the blades. The bladed disk is then mistuned by the addition of small, unequal weights to the blade tips, and it is again subjected to a traveling wave excitation. The experimental data is used to verify analytical predictions about the occurrence of localized mode shapes, increases in forced response amplitude, and changes in resonant frequency due to the presence of mistuning. Very good agreement between experimental measurements and finite element analysis is obtained. The out-of-plane response is compared and contrasted with the previously reported in-plane mode localization behavior of the same test specimen. This work also represents an important extension of previous experimental study by investigating a frequency regime in which modal density is lower but disk-blade interaction is significantly greater. [DOI: 10.1115/1.1377872]

1 Introduction

The design and analysis of nominally cyclic structures such as bladed disks is often done under the assumption that the structure as a whole is comprised of perfectly identical substructures, or sectors. This assumption of cyclic symmetry allows the dynamic response of the entire structure to be determined from a finite element analysis of a single sector, yielding considerable computational savings. Unfortunately, any actual bladed disk will have small imperfections due to manufacturing tolerances and uneven wear while in use. These small differences between sectors, called mistuning, can have a large impact on dynamic behavior, leading to a response that is qualitatively different from that of an ideal, tuned structure. In particular, the phenomena of *peak splitting* and *mode localization* can lead to *increased resonant amplitudes* of blade vibration when the bladed disk is in use. The greater fatigue caused by these vibrations can in turn lead to significantly shorter life spans than predicted by an analysis which assumes perfect symmetry.

The first detailed work on mistuning phenomena was done by Tobias and Arnold [1], who described the splitting of single resonant peaks into "dual modes" when mistuning was present. Whitehead [2] and Dye and Henry [3] showed that random mistuning can cause forced response amplitudes to be significantly greater than in a perfectly tuned, ideal case. The first evidence of mode localization (although it was not recognized as such at the time) was given by Ewins ([4–6]), whose experimental investigation of bladed disks revealed complex modes of vibration which occurred when individual blades of an assembly were detuned.

Fabunmi [7] and Irretier [8] also observed modes distinctly different from perfectly tuned mode shapes; these modes have come to be known as localized modes. Wei and Pierre ([9,10]) investigated a model which contained a single degree-of-freedom per sector, and documented and explained the occurrence of localized modes and forced response magnification in systems having weak inter-blade coupling (in which motions of each blade have a small but non-negligible effect on motions of other blades). They also concluded that structures with high modal density are more susceptible to mode localization than those with more widely spaced modes.

The phenomenon of peak splitting was demonstrated experimentally by Stange and MacBain [11], who used holographic interferometry and strain gauge measurements to observe dual modes in a nominally tuned bladed disk as well as in one that had been mistuned intentionally. Jay et al. [12] and Jay and Burns [13] experimentally observed both traveling and standing wave response to traveling wave excitation. MacBain and Whaley [14] developed an analytical model to predict the maximum resonant response due to the interaction of the two modes in a pair. Other studies of amplitude magnification ([4,15–18]) have also demonstrated that while some mistuning causes an increase in maximum response, further mistuning often reverses the effect and results in lower amplitudes. This has been shown experimentally by Mehmed and Murthy [19] who intentionally mistuned an eight-bladed propfan and observed a decrease in peak amplitudes. Numerous studies have investigated this potentially beneficial effect of mistuning ([20–23]). Recently, Castanier and Pierre [24,25] investigated the feasibility of reducing negative effects of random mistuning by adding intentional mistuning.

Much of the early theoretical work on mistuning was done using lumped parameter models with few degrees-of-freedom per sector. This was because finite element analysis of entire mistuned structures is computationally expensive, especially when one considers that study of random mistuning often requires statistical

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-358. Manuscript received by IGTI Oct. 1999; final revision received by ASME Headquarters Oct. 2000. Associate Editor: D. Wisler.

analysis of a large number of mistuning patterns. Castanier et al. [26] and Kruse and Pierre [17] introduced a technique for developing reduced order models of bladed disks directly from finite element models, which allowed computational studies of entire mistuned disks to be carried out efficiently for the first time. An experimental investigation of a 12-bladed disk with simple geometry was carried out by Kruse and Pierre [17,18] in order to verify the existence of localized modes in bladed disk assemblies and to provide a correlation between experimental, finite element, and reduced order model predictions for both free and forced response of bladed disks. The work focused on the in-plane blade bending response, in a frequency region with very high modal density and weak coupling between blades through the disk. Because of the simple geometry of the experimental blisk and the nonrotating nature of the experiment, only structural coupling of the blades through the disk was considered (blades of a real rotor are also coupled by aerodynamic effects). While good correlation was observed between theoretical and experimental free-vibration modes, the agreement for the response amplitude of blades subject to forced engine order excitation was not satisfactory.

This paper presents an experimental investigation of the same 12-bladed disk used by Kruse and Pierre. The focus is on a frequency region with stronger structural interblade coupling and lower modal density: the modes dominated by out-of-plane bending motion of the blades. The purpose is to validate theoretical predictions about the occurrence of localized modes and increases in forced response amplitudes, and to gain greater insight into the mechanisms by which the phenomena occur. This work and the preceding experiment by Kruse and Pierre represent a first use of intentional mistuning in a pattern chosen specifically to illustrate negative effects of mistuning. There are three main contributions of this work. First, it extends the previously reported experiment to a regime with distinctly different characteristics and documents the occurrence of mode localization in this regime. Second, it provides the first good correlation between theory and experiment for the forced response blade amplitudes of a bladed disk subject to engine order excitation. Third, the behavior observed provides deeper insight into the mechanisms which lead to mode localization and increases in maximum amplitude of vibration.

Section 2 describes the experimental apparatus and procedure, including the characteristics of the nominally tuned bladed disk first investigated, the traveling-wave engine order excitation system, and the means used to introduce mistuning. Section 3 compares theoretical predictions for a perfectly tuned system with the experimental results from the nominally tuned (but, in reality, slightly mistuned) bladed disk. In Section 4, the theoretically predicted and experimentally observed behavior of the mistuned case are compared and examined.

2 The Experimental Apparatus and Procedure

2.1 Experimental Setup. A photograph of the experimental setup is shown in Fig. 1. It consists of a flat 12-bladed disk with simple geometry, held stationary by a clamp at its center. The test specimen is a *blisk*: a bladed disk in which the disk and blades are manufactured as a single piece. The blisk was machined from a 3.24-mm thick precision-ground 4140 steel plate. The disk has a diameter of 300 mm and the blades are 92 mm long and 10.14 mm wide. The clamp consists of two spindles that interlock through a 25.4-mm hole in the bladed disk's center and are themselves clamped by a 20-ton capacity bearing press. The bladed disk was manufactured to precise tolerances in an effort to provide a specimen as close to perfectly tuned as possible (see Kruse and Pierre [27] for a full description of the disk's manufacture).

Typical values of damping for steel are a loss factor η between 2×10^{-4} and 1×10^{-3} (Q factor between 1000 and 5000). Two methods were used to estimate the exact damping of the experimental blisk: a logarithmic decrement measurement and an estimation of the half-power bandwidth. Both types of measurements were made with the specimen mounted in the apparatus, with the

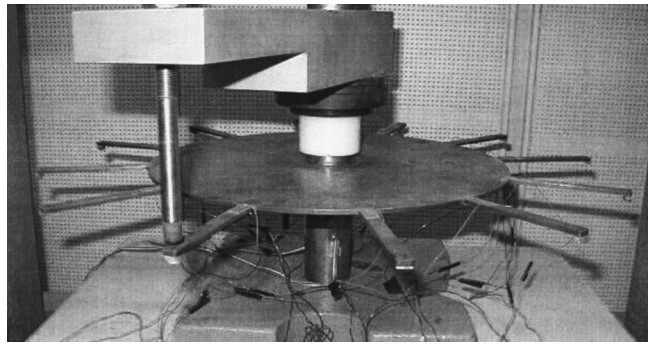


Fig. 1 Experimental blisk

piezoelectric actuators (see below) in place. The logarithmic decrement revealed a value of λ between 0.0010 and 0.0012, which corresponds to η between 3.2×10^{-4} and 3.8×10^{-4} . The half-power bandwidth estimation gave values near the middle of this same range. The damping was thus taken to be $\eta = 3.5 \times 10^{-4}$ for the purposes of modeling the experiment numerically; this number includes both structural damping and any additional damping due to ambient air.

Excitation was provided using piezoelectric actuators glued to the surface at the root of each blade. Measurements were taken using an eddy-current proximity probe, which was held in a fixture that could be rotated to move the probe over the tip of each blade. Figure 2 shows a close-up of one blade. The piezoelectric actuator is visible at the blade root, in the upper right of the photo. The proximity probe is positioned over the tip of the blade, on the left. Also visible is a lead weight glued to the underside of the blade tip. Various amounts of mass were added to each blade in this manner once the tuned portion of the experiment was complete, in order to add mistuning to the system by changing the natural frequencies of the blades.

The piezoelectric actuators provided two types of excitation. First, the blades were excited one at a time. Because damping in the blisk is very small, the resonant peaks are very narrow, and thus the resonant forced response of the structure to this single blade excitation can be taken as a means of determining the natural frequencies and mode shapes. Second, all the piezoelectric actuators were driven simultaneously at the same amplitude, but with varying phase relative to a common source. This allowed the

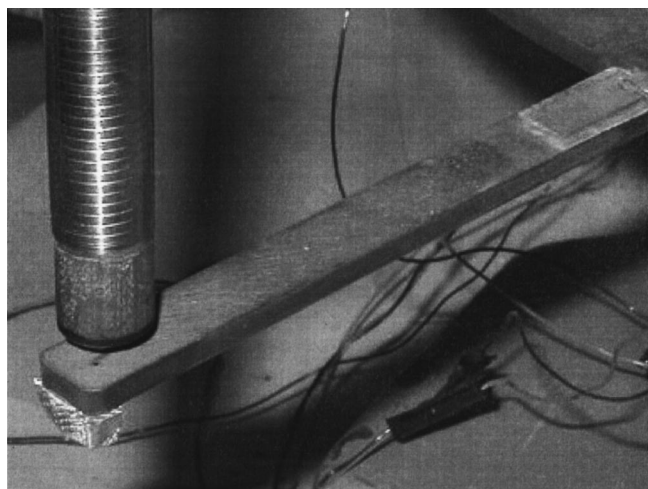


Fig. 2 Closeup of a blade, showing proximity probe above blade tip, piezoelectric actuator at blade root, and added mass at blade tip

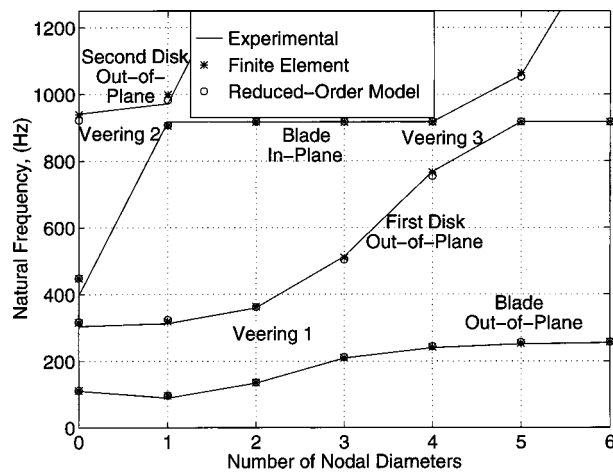


Fig. 3 Natural frequencies versus number of nodal diameters for the tuned blisk

excitation to form traveling waves corresponding to various engine orders, simulating the effect of a rotating bladed disk passing through various numbers of stationary sources of excitation. A number of phase-synchronized signal generators were programmed to provide the desired patterns. Each signal was then passed through an amplifier, the gain of which was adjusted to “tune” the signal to account for variations in the strength of the bonding of the actuators to the blade roots. This tuning process was carried out by considering the resonant peak amplitude response to single blade excitation. In this manner, each actuator delivered a consistent amount of energy to each blade, so as to provide the same excitation amplitude for all blades.

2.2 Computational Models. Numerical simulations predicting the theoretical response of the system were carried out on a detailed finite element model with 9078 degrees-of-freedom. The model included both the bladed disk and the spindles used to hold it in place. Nodes at the ends of the spindles away from the disk were constrained in all directions to simulate the effect of the bearing press.

In addition, the more computationally intensive simulations needed to determine an appropriate mistuning pattern were carried out using a reduced order model generated from the finite element model using component mode techniques ([17,26]). The reduced order model has only 120 degrees-of-freedom. The accuracy of the reduced order model is illustrated in Fig. 3.

3 Response Measurements and Numerical Simulations for the Tuned Blisk

The free-vibration mode shapes of cyclic structures are typically characterized by their number of nodal diameters and nodal circles. Figure 3 summarizes the nature of the modes of vibration of the tuned system under consideration by characterizing them according to the frequencies at which they occur and the number of nodal diameters present in the mode shapes. It should be noted that all modes except those with either 0 or 6 nodal diameters are double modes, in which two orthogonal mode shapes occur at the same frequency.

The previous work reported by Kruse and Pierre [27,28] concentrated on the modes with natural frequency between 900 and 1000 Hz. These modes, most of which occur at very closely spaced frequencies, are characterized by blade motion which is in the plane of the blisk. Two regions of Fig. 3, labeled *veering 2* and *veering 3*, were discussed in particular. In these regions, the modes are mostly a combination of in-plane bending shapes of the blades and out-of-plane motion of the disk, which allows some transfer of energy between the blades and the disk. Thus, these

veering regions exhibit some small amount of *interblade coupling*. Kruse and Pierre showed that in these regions, which combine high modal density with weak interblade coupling, strong mode localization will occur when the system is mistuned.

The present discussion focuses on the area of Fig. 3 labeled *veering 1* and the out-of-plane blade modes in the 200 to 260 Hz range. The wider nature of *veering 1* indicates that the coupling between disk and blade motion is much stronger, and also means that the modes are more widely spread in frequency than in the higher frequency regime previously reported. It will be shown that the strength of the coupling and the wide spacing of the natural frequencies mean that when mistuning is introduced, mode localization does *not* occur to any significant degree within the *veering* region. Instead, when mistuning is introduced, localization occurs only in the modes that had been characterized by five or six nodal diameters before the introduction of mistuning: modes which are more closely spaced and have a smaller amount of interblade coupling.

3.1 Spatially Extended Mode Shapes. A perfectly tuned bladed disk has modes that occur in pairs, at exactly the same frequency. Both modes in each pair are spatially harmonic circumferentially around the disk, and the two are orthogonal to one another. Because they occur at the same frequency, excitation of one mode alone would be difficult for a perfectly tuned assembly. However, the introduction of even a small amount of mistuning causes a phenomenon known as *peak splitting*. The two modes in each pair differ in frequency by an amount that increases with mistuning, roughly in proportion to the amount of stiffness mistuning among the blades ([9,10]). If damping in the system is low enough, the resonant peaks of the two modes will be distinct even for very small amounts of mistuning. In other words, the bladed disk vibrates primarily in only one mode at one frequency, and primarily only in the orthogonal mode at a slightly different frequency.

The experimental blisk was manufactured to tight tolerances, but nevertheless features a small amount of mistuning. By considering the resonant peak splitting, this residual mistuning was estimated as a 0.5 percent standard deviation in the blade stiffness. In addition, the only significant sources of damping in the experimental system are the material damping inherent in steel and the damping added by the ambient air. The so-called “tuned” experimental blisk was thus mistuned enough that peak splitting was observed. Table 1 shows a comparison of the finite element predictions of natural frequencies for a perfectly tuned system with the experimentally observed frequencies for the nominally tuned experimental blisk. In situations with very low damping, the resonant peaks are extremely narrow, and the forced response at each resonance is essentially the response of only one mode. The bladed disk was subjected to excitation one blade at a time, and the resonant forced response shapes were then taken to be the free mode shapes of the system. The agreement between finite element and experimental frequencies is excellent for almost all modes; even the lowest few modes (which are most sensitive to boundary conditions at the center of the blisk) show very good agreement.

The pairs of orthogonal modes (and single modes for zero and six nodal diameters) predicted by the finite element model were compared with the single-blade excitation response at the resonant peaks. Figure 4 shows a typical comparison. The response shapes at the two resonant frequencies with two nodal diameters are compared with sinusoids: a perfectly tuned system would have perfectly harmonic modes, so that displacement of the measured point on each blade would lie on the sinusoid exactly. Note that although the system was mistuned enough to exhibit peak splitting, the observed shapes match the theoretical predictions quite well: The small amount of residual mistuning present in the “tuned” case was not enough to cause the formation of localized modes. This same agreement was observed for all 12 of the modes investigated.

Table 1 Experimental natural frequencies for the nominally tuned case, compared with finite element predictions for a perfectly tuned blisk

Mode #	Nodal Diameters	Finite Element Prediction	Experimental Data	Percent Difference
1	1	96.38 Hz	94.0 Hz	2.47 %
2	1		94.4 Hz	2.05 %
3	0	111.50 Hz	113.3 Hz	1.61 %
4	2	135.11 Hz	135.2 Hz	0.07 %
5	2		135.8 Hz	0.51 %
6	3	210.26 Hz	211.2 Hz	0.45 %
7	3		211.5 Hz	0.59 %
8	4	243.40 Hz	243.3 Hz	0.04 %
9	4		243.7 Hz	0.12 %
10	5	255.54 Hz	255.3 Hz	0.09 %
11	5		255.6 Hz	0.02 %
12	6	257.17 Hz	259.0 Hz	0.71 %

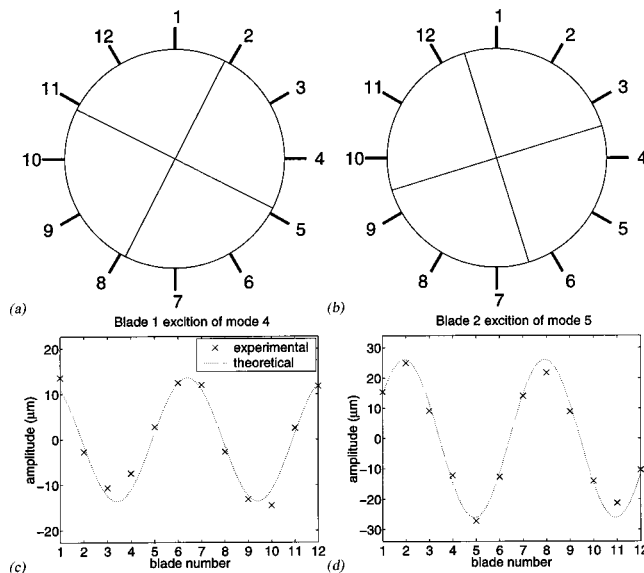


Fig. 4 2-nodal-diameter "free" response shapes: (a) and (b) show schematics of the blisk indicating the location of the nodal diameters, (c) and (d) compare response amplitudes to the sinusoids the data would fall on if the modes were perfectly harmonic (negative amplitudes indicate vibration 180 deg out of phase from excitation, positive amplitudes are in phase with excitation)

3.2 Forced Response. The bladed disk was subjected to traveling wave excitation by driving the piezoelectric actuators at equivalent amplitudes but at different phases. The excitation of a particular blade can be written as

$$F_n = F_o \cos(\omega t + (n-1)\phi) \quad (1)$$

where F_o is the force amplitude, ω is the frequency of excitation, n is the blade number (from 1 to 12), and ϕ is the phase difference between adjacent blades: $\phi = 2\pi C/N$ where C is the engine order of excitation and N is the number of blades. For example, for the 12 bladed disk under consideration, ϕ is $\pi/6$ radians or 30 deg for engine order one, 60 deg for engine order two, etc.

An important observation is that the above form of the excitation can be rewritten as

$$F_n = F_o \cos((n-1)\phi) \cos(\omega t) - F_o \sin((n-1)\phi) \sin(\omega t). \quad (2)$$

Each of these two terms can be interpreted as a harmonic excitation at the same phase on all blades, but with the amplitude dependent on the blade number in a sinusoidal pattern. Thus, traveling wave excitation of the form of Eq. (1) is equivalent to the excitation of two standing waves, orthogonal to each other spatially and out of phase temporally by 90 deg. It should also be noted that changing the sign of the phase shift in Eq. (1) results only in a change of sign of the second term in Eq. (2), in other words, changing the direction of the traveling wave excitation merely reverses the relative phase of the two standing waves. Because the engine order of the excitation determines ϕ , an engine order m excitation can be regarded as the sum of two orthogonal standing waves having m nodal diameters.

Two special cases are worth mentioning: the excitation of engine orders zero and six. In the engine order zero case, $F_n = F_o \cos(\omega t)$; there is no phase difference between the excitation of adjacent blades, and the excitation is a standing wave. For a blisk with 12 blades, engine order six excitation means a 180-deg phase shift between adjacent blades, so that $F_n = (-1)^{n-1} F_o \cos(\omega t)$. Therefore, engine order six is also a standing wave excitation.

Numerical Predictions. For a perfectly tuned bladed disk, the traveling wave excitation, equivalent to two standing waves, excites only the paired mode shapes having the same number of nodal diameters as the engine order of the excitation. However, since these two modes occur at exactly the same frequency, they are forced equally by the traveling wave. By the relations in Eqs. (1) and (2), the responses in these stationary mode shapes can be reassembled into a traveling wave response. Thus, all blades vibrate at the same amplitude, but with a phase difference between each blade. Numerical simulations of the forced response of a perfectly tuned disk confirm this well-known result.

Engine orders zero and six are only one standing wave each rather than traveling waves equivalent to two standing waves. Likewise, there is only one zero-nodal diameter mode and one six-nodal diameter mode, and for the perfectly tuned case, these engine order excitations excite the corresponding nodal diameter mode alone.

Experimental Results. As discussed above, the paired modes of the experimental system are not at exactly the same frequencies, and, as shown in Fig. 5, the peaks are narrow enough that only one mode shape exhibits significant response at a given frequency. Thus, although the traveling wave excitation can be decomposed into components aligned with both mode shapes, it drives a response in only one mode or the other. The forced response shapes are therefore essentially the same as the free response mode shapes, as seen in Fig. 6. Although this is qualitatively different from the traveling wave response predicted numerically for the tuned system, the decomposition of the traveling wave into stationary components allows the realization that,

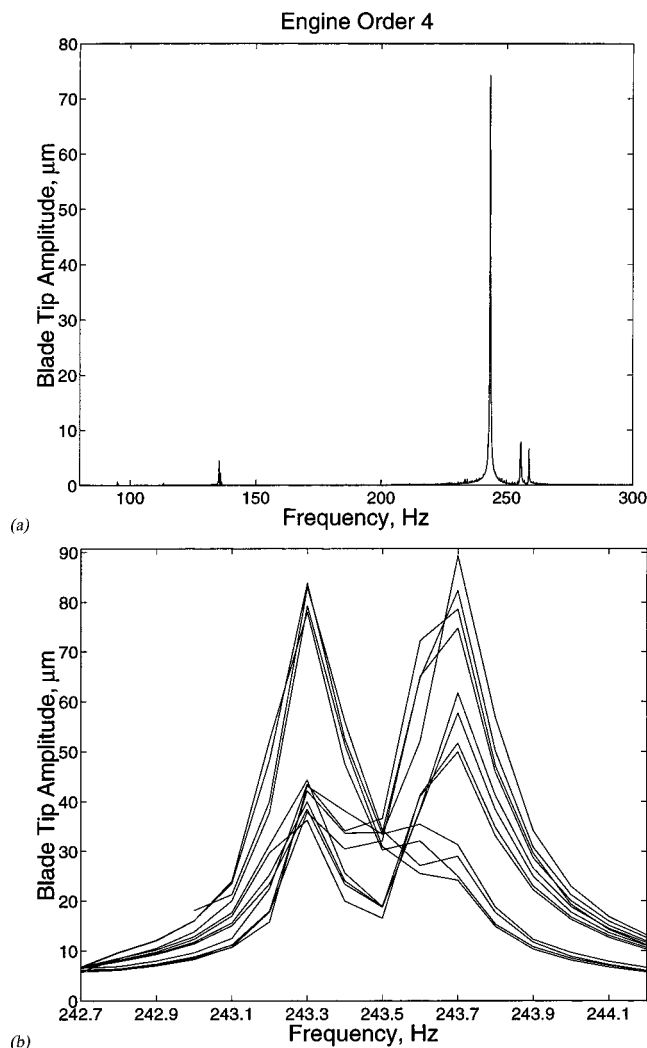


Fig. 5 Experimental frequency response of “tuned” blisk to engine order 4 excitation: (a) full frequency spectrum, blade 1 response, (b) response of all blades in the 4-nodal-diameter modes

aside from the peak splitting which is not present in the perfectly tuned numerical model, the numerical and experimental results actually match quite well.

Because of the extremely low damping inherent in bladed disks, this phenomenon can be expected to occur in real blisks as well. If

the resonant peaks are narrow enough, even a very small amount of mistuning will cause peak splitting, and the response to traveling-wave excitation will be standing waves. As damping is increased, the two peaks widen and thus merge, so that both modes are responding significantly at either resonant frequency. If this is the case, the standing response of both modes combines to form a traveling wave. Thus, the forced response of blisks is extremely sensitive to both mistuning and damping. Extremely lightly damped blisks can be close enough to tuned to not exhibit any mistuning phenomena such as localization, but still have qualitatively different behavior from a perfectly tuned model.

4 Response Measurements and Numerical Simulations for the Mistuned Blisk

As discussed in Section 2.1, mistuning was added to the experimental specimen by gluing small lead weights at the tip of each blade. This method of mistuning is different from that used by Kruse and Pierre [27,28], who manufactured a second, mistuned disk that featured blades of varying length. Even with the precise tolerances maintained in manufacturing, each specimen has some small amount of unknown, random mistuning, in addition to any mistuning purposefully added. This is believed to be an important cause of the poor correlation of the forced response results obtained by Kruse and Pierre [28]. By using the same blisk for both “tuned” and mistuned cases, this undesired yet unavoidable mistuning is kept consistent between the two cases, whereas manufacturing multiple disks introduced differing amounts and patterns of accidental mistuning. In addition, it was found that the forced response measurements were somewhat sensitive to the way the blisk was mounted at its hub. The addition of weights to the “tuned” specimen allowed mistuning to be included without removing the blisk from the apparatus, thus eliminating inconsistencies in boundary conditions.

The drawback of this method of mistuning is that weight was only added, never removed. Thus, the mean natural frequency of the mistuned blades was not the same as the original, tuned frequency, and this had to be accounted for in the analysis. It is also important to note that, although using the same bladed disk for tuned and mistuned cases insures consistency of random, unintended mistuning, it still does not provide any means for identification or removal of such mistuning, nor does it allow it to be accounted for in the numerical models.

4.1 Numerical Predictions

Monte Carlo Simulations. It was desired to mistune the bladed disk in such a way as to best demonstrate the phenomena being investigated. Therefore, extensive numerical predictions were performed to find a mistuning pattern yielding high-amplitude magnification. For excitation in each engine order,

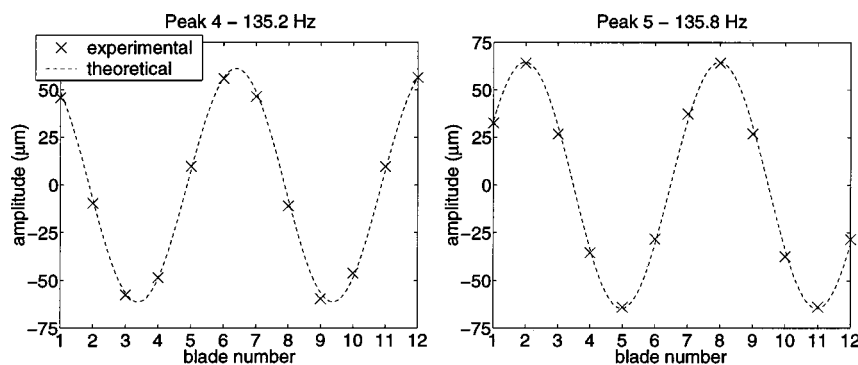


Fig. 6 Engine order 2 forced response shapes of “tuned” system (compare with Fig. 4)

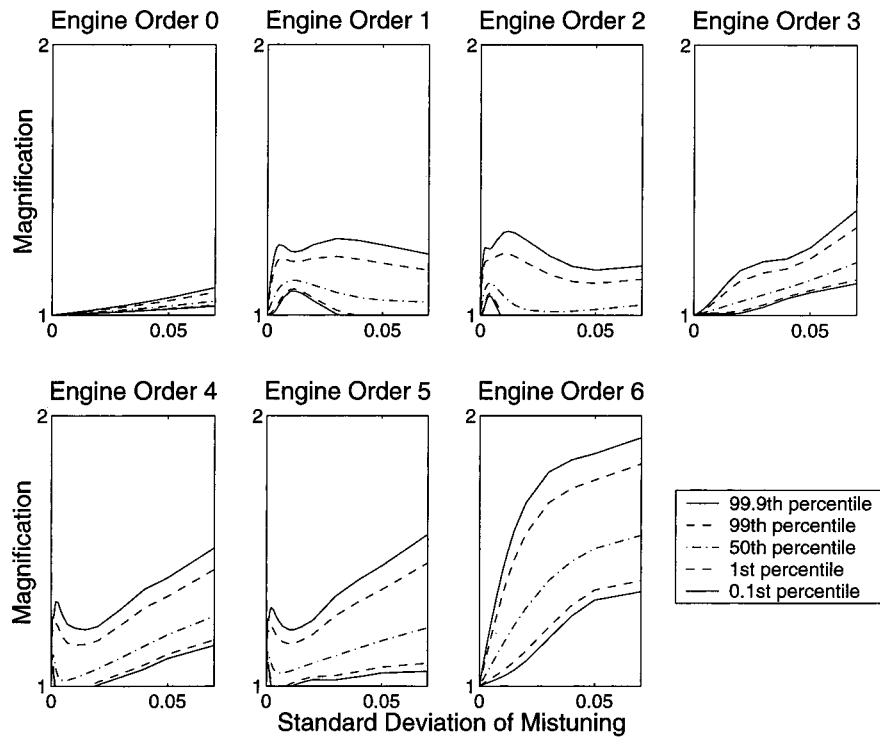


Fig. 7 Predicted amplitude increase (largest blade tip resonant response, scaled by tuned resonant response) versus mistuning, as obtained from Monte Carlo simulations of the reduced order model

2000 random mistuning patterns for various degrees of mistuning were run using the computationally efficient reduced order model ([17,26]).

Figure 7 shows results for each engine order, including engine order 6, for which the potential amplitude magnification was highest. The reduced order modeling code models mistuning as a change in blade stiffness; “standard deviation of mistuning” refers to that of stiffness for a population of blades, from which 12 blades were chosen at random for each pattern. In other words, mistuning with standard deviation 0.01 means that the blades were randomly picked from a population with a one percent standard deviation about the mean. The mean itself for each population was adjusted to be two standard deviations less stiff than the tuned blade, so that all but a very few blades in each population would have lower rather than higher natural frequency. This was done in order to produce a mistuning pattern that could be created experimentally by only the addition, not removal, of mass.

A mistuning pattern was selected that featured blades chosen from a population having five percent standard deviation in stiffness. The pattern is given in Table 2, which also lists the change in blade natural frequency relative to the blades of the perfectly tuned blisk, and the amount of mass added in the experiment to produce this change in natural frequency. This pattern exhibited a forced response amplitude increase of over 75 percent in a simulation with engine order 6 excitation, placing it near the 99th percentile of all such possible patterns. Examination of the predicted forced response shapes revealed strong mode localization in the modes excited by both engine orders 5 and 6.

Comparison of Predictions Based on Stiffness Mistuning and Mass Mistuning. The reduced order model used to carry out the above simulations incorporated mistuning as a change in the stiffness of each blade. Although it was too computationally expensive to run Monte Carlo simulations using the full finite element model, a finite element model that incorporated mistuning as an additional mass at the tip of each blade was run for the pattern chosen from the reduced order model simulations. The predicted

forced response shapes were very similar to those obtained with stiffness mistuning, but the amplitudes predicted differed by as much as 40 percent. This is shown in Fig. 8, which compares the predictions for resonant response to engine order six excitation. The amplitude of each blade is normalized by its amplitude in the 12th tuned mode (the only mode of the tuned model which is excited by engine order six). The stiffness-mistuned model pre-

Table 2 Mistuning pattern: change in blade stiffness of theoretical model, effective change in blade natural frequency, and mass added to experiment to achieve same frequency shift

blade #	change in stiffness	change in frequency	added mass
1	13.40 %	6.53 %	1.802 g
2	3.22 %	1.60 %	0.397 g
3	15.80 %	7.61 %	2.173 g
4	17.39 %	8.34 %	2.429 g
5	15.33 %	7.39 %	2.099 g
6	16.69 %	8.02 %	2.315 g
7	1.80 %	0.90 %	0.220 g
8	8.60 %	4.21 %	1.108 g
9	4.82 %	2.38 %	0.602 g
10	18.59 %	8.90 %	2.627 g
11	14.77 %	7.13 %	2.012 g
12	2.56 %	1.27 %	0.314 g

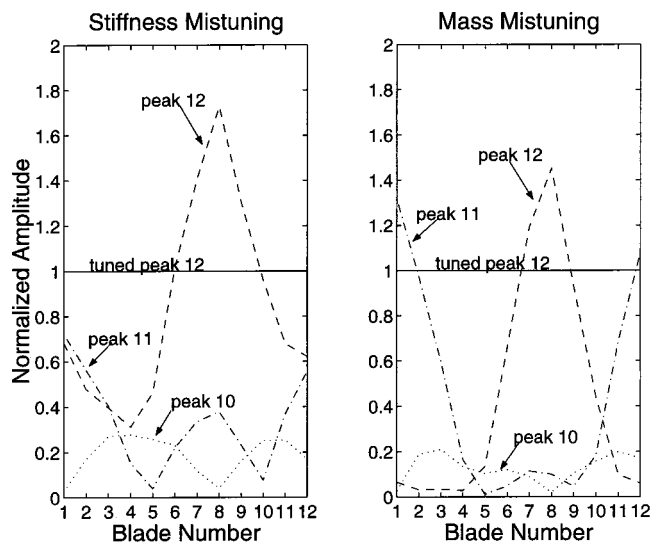


Fig. 8 Comparison of resonant response pattern predictions based on stiffness and mass mistuning, for engine order 6 excitation

dicted a maximum amplitude increase due to mistuning of over 75 percent in the 12th mode, and also predicted smaller yet significant responses at two other peaks: those corresponding to modes 10 and 11. The mass-mistuned model predicted a maximum amplitude increase of only 45 percent but also predicted a response in mode 11 which was 32 percent greater than the tuned mode 12 response. While the response shapes predicted by both models are qualitatively quite similar, the relative amplitudes of the responses differ considerably. It was assumed that predictions based on mass mistuning—a more accurate representation of the mistuning done to the experiment—would be valid for comparison with the experimental data.

Localized Versus Spatially Extended Modes. Both methods of mistuning modeling predicted localization in the 10th, 11th, and 12th mode shapes, which in the tuned case were the shapes characterized by five (10th and 11th) and six (12th) nodal diameters. These modes are blade-dominated and are all quite close to one another in frequency. Lower modes, which occur in and around veering 1 and are thus comprised of both blade and disk motion, retain their spatially extended nature even in the presence of strong mistuning. Not only is the modal density lower in the veering, but the width of the veering implies strong coupling between disk and blade motion. This is consistent with the theory of localization.

4.2 Experimental Results. The resonant frequencies and response shapes measured experimentally were in excellent agreement with those predicted by numerical simulations. Table 3 compares the resonant peak frequencies observed experimentally with those predicted numerically. By comparing Table 3 with Table 1, one can see that the mistuned peaks have spread further apart, but for the most part still occur in pairs: modes 1 and 2, 4 and 5, and 6 and 7 are all paired quite closely (Mode 3 is a zero-nodal-diameter mode and thus is single). These paired modes are essentially the same as the paired orthogonal-nodal diameter modes of the perfectly tuned case: The two modes in each pair are shapes characterized by the same number of nodal diameters and excited most directly by engine order excitation of that same harmonic. For instance, Fig. 9 illustrates that modes 6 and 7 are excited strongly by engine order 3 excitation, which causes minimal response at peaks 1 through 5 and only small response at peaks 8 through 12. Although the increase in mistuning has caused the peaks to move further apart in frequency (to 1.1 Hz apart from an

Table 3 Experimental resonant frequencies for the mistuned case compared with theoretical predictions for a blisk mistuned by added masses

Mode #	Theoretical Prediction	Experimental Data	Percent Difference
1	93.02 Hz	91.0 Hz	2.17 %
2	94.40 Hz	92.0 Hz	2.54 %
3	107.83 Hz	109.4 Hz	1.46 %
4	129.69 Hz	129.1 Hz	0.45 %
5	130.51 Hz	130.2 Hz	0.24 %
6	194.98 Hz	194.0 Hz	0.50 %
7	195.81 Hz	195.1 Hz	0.36 %
8	219.57 Hz	217.8 Hz	0.81 %
9	224.92 Hz	224.2 Hz	0.32 %
10	229.60 Hz	228.7 Hz	0.39 %
11	237.74 Hz	237.3 Hz	0.19 %
12	240.49 Hz	239.8 Hz	0.29 %

original 0.3 Hz), it has not significantly changed the nature of the mode shapes: the patterns for peaks 6 and 7 are spatially extended and characterized by three nodal diameters, as shown in Fig. 10. Note the excellent agreement between experimental measurements and the shapes predicted by the finite element simulation. Similarly, engine order one excites modes 1 and 2 primarily, engine order zero excites mode 3, and engine order 2 excites modes 4 and 5. Agreement between theory and experiment was also excellent for the shapes of these modes.

Localized Modes. The picture is dramatically different for the highest few modes, however. In the tuned case, modes 10 and 11 were the paired five-nodal-diameter modes, while mode 12 was the single six-nodal-diameter mode. However, Table 3 shows that the splitting between modes 10 and 11 is now greater than the

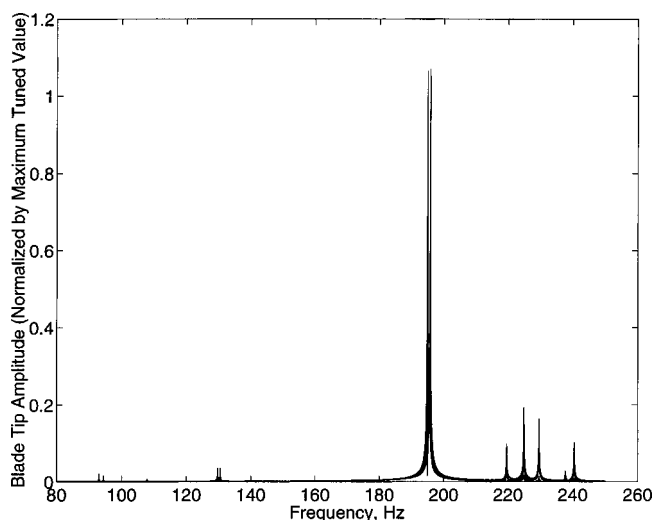


Fig. 9 Finite element prediction of engine order 3 frequency response of all blades for mistuned case (observe strong response for modes 6 and 7)

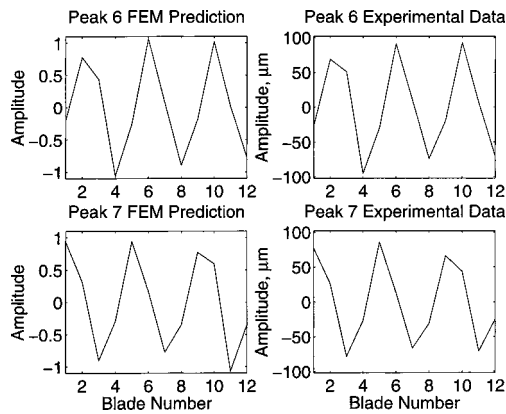


Fig. 10 Engine order 3 response shapes for mistuned case (finite element method predictions are normalized by maximum tuned amplitude)

distance between modes 11 and 12 and between modes 9 and 10. Examination of the response patterns (or mode shapes) for peaks 10, 11, and 12 reveals that they are no longer spatially extended, but instead exhibit localization. Engine orders 5 and 6 both excite strong responses in these three mode shapes. Figure 11 shows the frequency response of the mistuned blisk subjected to engine order 5 excitation, as predicted by finite element analysis. The frequency response for engine order 6 excitation is similar in nature: strong responses in multiple modes instead of only in the single mode which would be excited in a tuned blisk. The shapes of the response at peaks 10, 11, and 12 are shown in Fig. 12. Once again, there is excellent agreement between experimental measurements and theoretical predictions. For these higher modes, however, certain blades have response amplitudes much higher than others, and the shapes cannot be categorized by a single nodal diameter.

The measured mode shapes were analyzed by Fourier transformation to determine their composition in terms of nodal diameter shapes. Figure 13 shows a few examples of lower modes comprised overwhelmingly of a single nodal diameter harmonic. The higher modes, those exhibiting localization, have significant contributions from multiple nodal diameter shapes, as depicted in Fig. 14. In particular, modes 11 and 12 result mostly from combina-

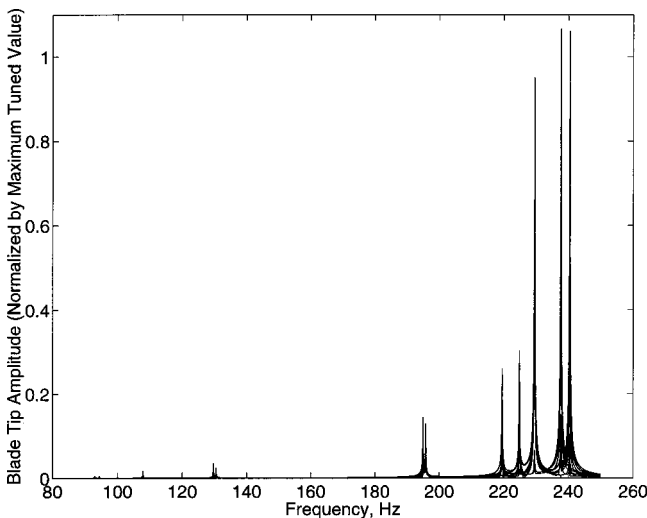


Fig. 11 Finite element prediction of engine order 5 frequency response of all blades for mistuned disk (observe strong response in modes 10, 11, and 12)

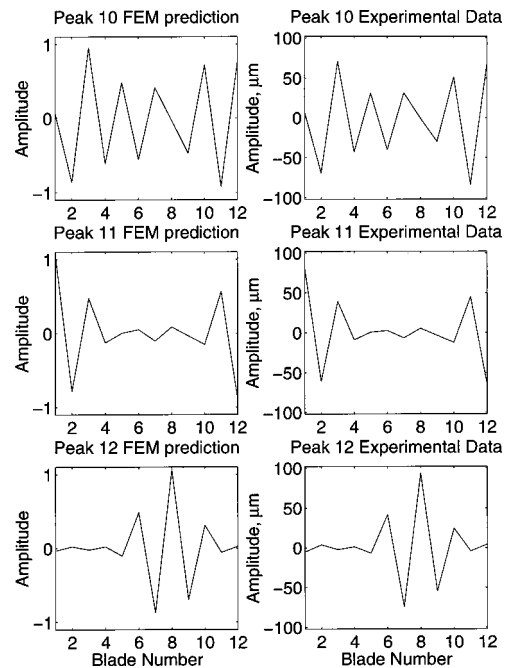


Fig. 12 Engine order 5 response shapes for mistuned blisk (finite element method predictions are normalized by maximum tuned amplitude)

tions of 5 and 6 nodal diameter shapes, while mode 10 has strong components in both of these shapes as well as 4 nodal diameters.

The modes characterized by five and six (and to a lesser extent four) nodal diameters in the tuned case were those closest together in the original tuned system, i.e., modal density was highest for these modes. In addition, they were not directly in the strong disk-blade coupling region of veering 1. In the mistuned case, it can be seen that these are the modes which exhibit localization.

Amplitude Magnification. As discussed in Section 4.1, numerical models predict the possibility of an increase in the maximum amplitude of vibration (the largest resonant amplitude over all blades) when mistuning is introduced. In particular, the responses to engine orders 4, 5, and 6 showed the highest increases. The particular mistuning pattern chosen for the mistuned experimental case was one for which simulations predicted an increase in the maximum amplitude of the response to engine order 6.

Engine order 6 excites the strongest response in the 11th and 12th resonant peaks. Figure 15 depicts the amplitudes of vibration of each blade for the 11th and 12th peak, as well as for the 12th peak in the tuned case (the only peak with a strong response to engine order 6 excitation). In the perfectly tuned case, the 12th mode consists of all blades vibrating with equal magnitude (albeit alternating phase) and thus the tuned case in Fig. 15(a) is a straight line. Figure 15(b) shows experimental data for the same peaks. Note that although the amplitude of peak 12 is somewhat less than expected, the response patterns of both peaks agree quite well with the finite element predictions. Because the “tuned” experimental case did feature a small amount of mistuning, the amplitudes of vibration of the tuned 12th peak are not all identical, but instead fall in a band between 0.25 and 0.36 μm . The horizontal line in Fig. 15(b) represents an estimation of what the amplitudes would be for a perfectly tuned experiment, based on the knowledge that the actual “tuned” case has mistuning of approximately 0.5 percent standard deviation of blade stiffness. The response shapes indicate that the response is highest for blade 1 in the case of peak 11 and blade 8 in the case of peak 12. Figure 16

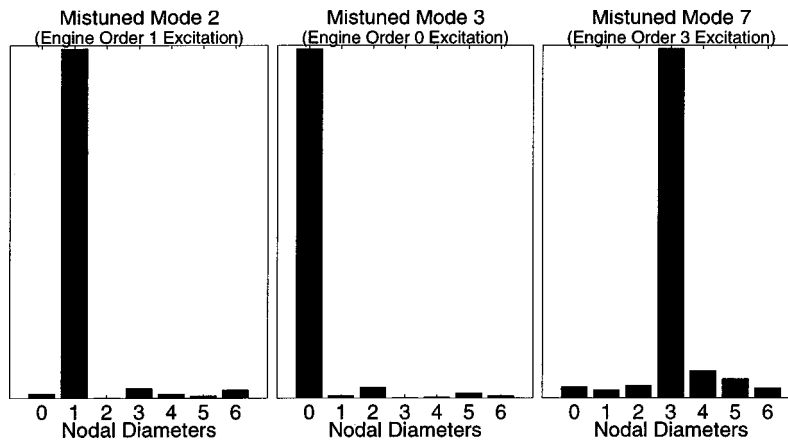


Fig. 13 Nodal diameter composition of experimental mistuned modes remaining spatially extended

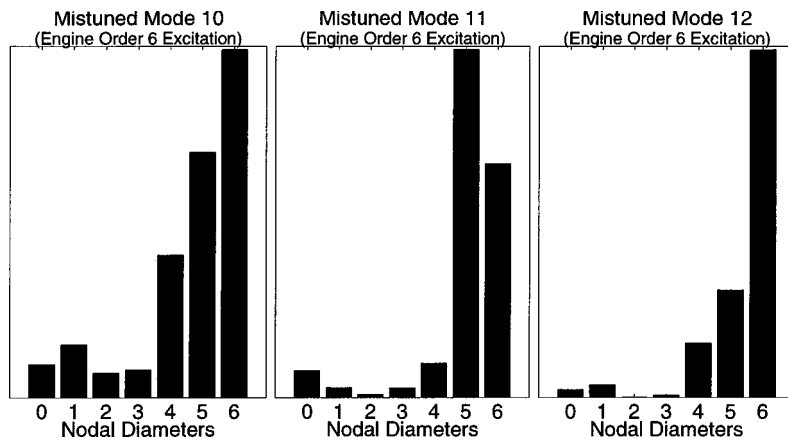


Fig. 14 Nodal diameter composition of experimental mistuned modes which exhibit localization

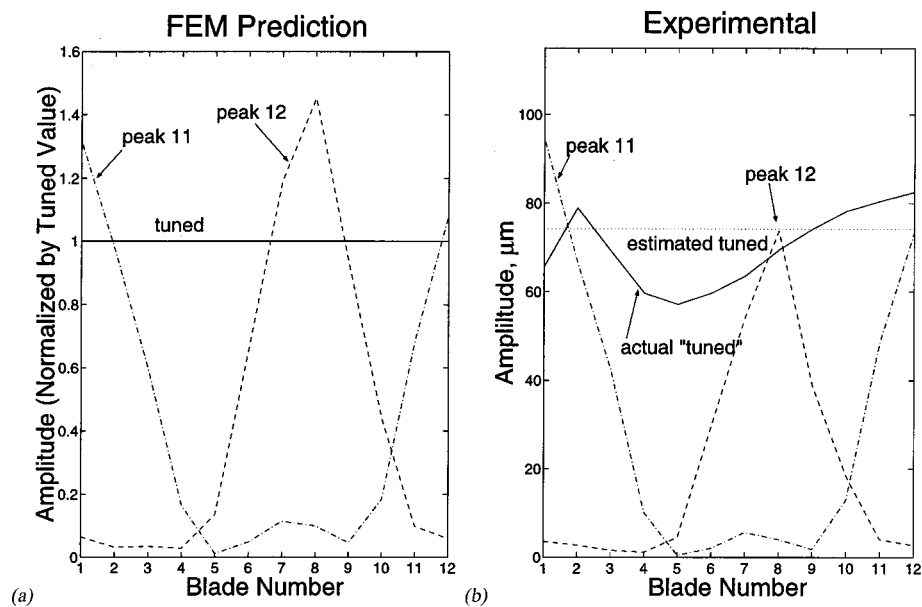


Fig. 15 Engine order 6 response amplitudes of mistuned peaks 11 and 12 and tuned peak 12: (a) finite element, and (b) experimental

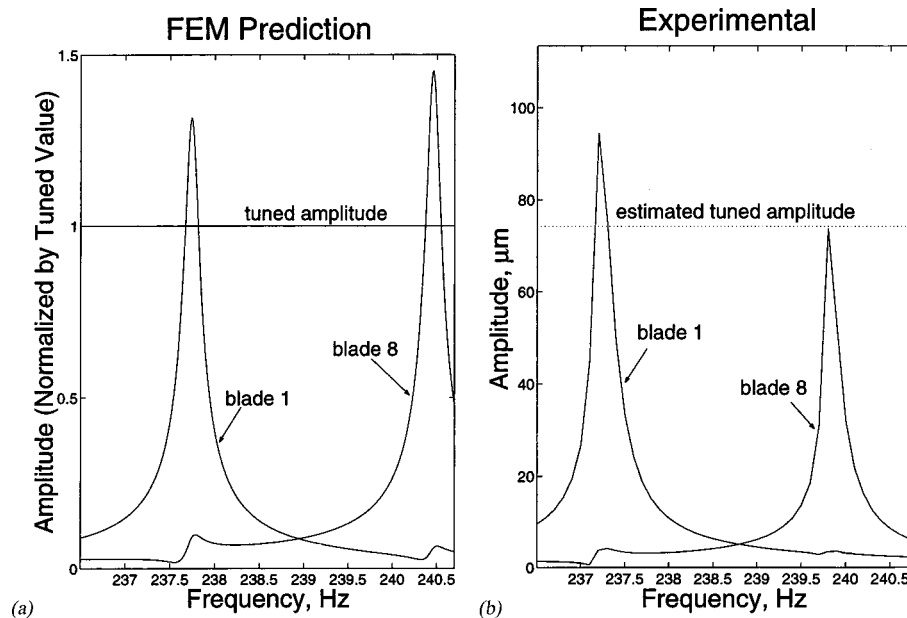


Fig. 16 Engine order 6 frequency response of blades with highest amplitude: (a) finite element, and (b) experimental

shows the frequency response of these blades compared to the amplitude of vibration in the tuned case (or the estimation of the tuned case, in Fig. 16(b)).

Peak 11 demonstrates a reasonably accurate prediction by finite element modeling of the actual amount of magnification occurring: a 32 percent increase was predicted, and the experimental data shows an increase of just over 27 percent. The experimental data for peak 12, however, did not show any evidence of amplitude increase, despite a 45 percent increase predicted theoretically. An important consideration here is that in addition to the “tuned” experimental case having an inherent amount of unknown mistuning, the mistuned experimental case has this same unknown mistuning, and this could not be accounted for in the finite element model. The same sensitivity to mistuning that brings about the amplitude magnification makes the discrepancy between finite element predictions and experimental results not completely unexpected. If this sensitivity is indeed the cause of the difference, it is then interesting to note that the occurrence of amplitude magnification appears to be more sensitive to mistuning than the occurrence of localized modes, the shapes of which were predicted quite accurately. The difference between theoretical and experimental amplitudes may also be due in part to simplifications made in modeling the added mass, or differences in damping between the finite element model and the physical experiment.

5 Conclusions

Mode localization and increases in maximum vibratory amplitude due to mistuning are common phenomena that cause difficulty in the design and analysis of bladed disk assemblies. The experimental investigation presented here documents the occurrence of these phenomena for the forced response of a 12-bladed test blisk, for a frequency regime containing modes that feature strong interaction between disk and blade motion. The localization occurs not in the veering region, as was the case in previous studies focusing on a different frequency regime, but for the more closely spaced modes dominated by blade motion. This finding validates the theory that weak interblade coupling allows a mechanism for mistuning to cause mode localization, while stronger coupling allows the energy of vibration to remain distributed throughout the structure. In the experiment by Kruse and Pierre [27,28] the veerings were narrow and thus coupling was weak. In

the present work, the stronger coupling present in the wider *veering 1* prevents localization from occurring there, while outside the veering, coupling is weaker and localization occurs. The experimental results correlate very well with theoretical results predicted by finite element analysis and the corresponding reduced-order model, for both single blade and engine order excitation, giving further confidence in the ability of these theoretical methods to predict mistuning phenomena.

Modest increases in maximum amplitude due to mistuning were also documented. Correlation with theoretical predictions of amplitude increase was excellent in the case of the 11th peak, but somewhat less accurate for the 12th peak, despite excellent agreement for the blade response patterns at all peaks. The discrepancy observed is likely due to the residual mistuning, to which the blisk's response is extremely sensitive at certain resonant peaks and for certain engine order excitations. Because of this extreme sensitivity of blisks with very low damping to minute amounts of mistuning, accurate prediction of forced response amplitude increase may well depend on the ability to either manufacture a bladed disk to tolerances precise enough to avoid any significant accidental mistuning, or to identify the mistuning characteristics of individual blades in order to account for unintentional mistuning in the theoretical model. Intentional mistuning strategies that reduce this high sensitivity should also be explored.

Acknowledgments

This work was supported by NASA Glenn Research Center, Grant NAG3-1880. The authors also wish to thank Prof. Steven Ceccio for his help with the design of the traveling wave excitation system, and Marlin Kruse, whose previous work with the same experimental blisk provided the foundation upon which this work was built.

References

- [1] Tobias, S. A., and Arnold, R. N., 1957, “The Influence of Dynamical Imperfection on the Vibration of Rotating Disks,” *Proc. Inst. Mech. Eng.*, **171**, pp. 669–690.
- [2] Whitehead, D. S., 1966, “Effect of Mistuning on the Vibration of Turbomachine Blades Induced by Wakes,” *J. Mech. Eng. Sci.*, **8**, No. 1, pp. 15–21.
- [3] Dye, R. C. F., and Henry, T. A., 1969, “Vibration Amplitudes of Compressor Blades Resulting From Scatter in Blade Natural Frequencies,” *ASME J. Eng. Power*, **91**, pp. 182–188.

- [4] Ewins, D. J., 1969, "The Effects of Detuning Upon the Forced Vibrations of Bladed Disks," *J. Sound Vib.*, **9**, No. 1, pp. 65–79.
- [5] Ewins, D. J., 1973, "Vibration Characteristics of Bladed Disc Assemblies," *J. Mech. Eng. Sci.*, **15**, No. 3, pp. 165–186.
- [6] Ewins, D. J., 1976, "Vibration Modes of Mistuned Bladed Disks," *ASME J. Eng. Power*, **98**, pp. 349–355.
- [7] Fabunmi, J., 1980, "Forced Vibration of a Single Stage Axial Compressor Rotor," *ASME J. Eng. Power*, **102**, pp. 322–329.
- [8] Irrerier, H., 1983, "Spectral Analysis of Mistuned Bladed Disk Assemblies by Component Mode Synthesis," *Vibrations of Bladed Disk Assemblies, Proceedings of the ASME 9th Biennial Conference on Mechanical Vibration and Noise*, Dearborn, MI, ASME, New York, pp. 115–125.
- [9] Wei, S. T., and Pierre, C., 1988, "Localization Phenomena in Mistuned Assemblies with Cyclic Symmetry, Part I: Free Vibrations," *ASME J. Vibr., Acoust., Stress, Reliab. Des.*, **110**, No. 4, pp. 429–438.
- [10] Wei, S. T., and Pierre, C., 1988, "Localization Phenomena in Mistuned Assemblies with Cyclic Symmetry, Part II: Forced Vibrations," *ASME J. Vibr., Acoust., Stress, Reliab. Des.*, **110**, pp. 439–449.
- [11] Stange, W. A., and MacBain, J. C., 1983, "An Investigation of Dual Mode Phenomena in a Mistuned Bladed Disk," *ASME J. Vibr., Acoust., Stress, Reliab. Des.*, **105**, pp. 402–407.
- [12] Jay, R. L., MacBain, J. C., and Burns, D. W., 1984, "Structural Response Due to Blade Vane Interaction," *ASME J. Eng. Gas Turbines Power*, **106**, pp. 50–56.
- [13] Jay, R. L., and Burns, D. W., 1986, "Characteristics of the Diametral Resonant Response of a Shrouded Fan Under Prescribed Distortion," *ASME J. Vibr., Acoust., Stress, Reliab. Des.*, **108**, pp. 125–131.
- [14] MacBain, J. C., and Whaley, P. W., 1984, "Maximum Resonant Response of Mistuned Bladed Disks," *ASME J. Vibr., Acoust., Stress, Reliab. Des.*, **106**, pp. 218–223.
- [15] Wei, S. T., and Pierre, C., 1990, "Statistical Analysis of the Forced Response of Mistuned Cyclic Assemblies," *ASME Journal*, **28**, pp. 861–868.
- [16] Ottarsson, G. S., and Pierre, C., 1995, "On the Effects of Interblade Coupling on the Statistics of Maximum Forced Response Amplitudes in Mistuned Bladed Disks," *Proceedings of the 36th AIAA/ASME Structures, Structural Dynamics, and Materials Conference*, Vol. 5, AIAA, New York, pp. 3070–3078.
- [17] Kruse, M. J., and Pierre, C., 1996, "Forced Response of Mistuned Bladed Disks Using Reduced-Order Modeling," *Proceedings of the 37th AIAA/ASME Structures, Structural Dynamics, and Materials Conference*, Vol. 4, AIAA, New York, pp. 1938–1950.
- [18] Kruse, M. J., and Pierre, C., 1996, "Dynamic Response of an Industrial Turbomachinery Rotor," *Proceedings of the 32nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*, Lake Buena Vista, FL.
- [19] Mehmed, O., and Murthy, D. V., 1988, "Experimental Investigation of Propfan Aeroelastic Response in Off-Axis Flow With Mistuning," *Proceedings of the 24th AIAA/SAE/ASEE Joint Propulsion Conference*, Boston, MA.
- [20] Ewins, D. J., 1980, "Further Studies of Bladed Disc Vibration: Effects of Packeting," *Vibrations in Rotating Machinery*, Mechanical Engineering Publications Ltd., London, pp. 97–102.
- [21] Griffin, J. H., and Hoosac, T. M., 1984, "Model Development and Statistical Investigation of Turbine Blade Mistuning," *ASME J. Vibr., Acoust., Stress, Reliab. Des.*, **106**, pp. 204–210.
- [22] Crawley, E. F., and Hall, K. C., 1985, "Optimization and Mechanisms of Mistuning in Cascades," *ASME J. Eng. Gas Turbines Power*, **107**, pp. 418–426.
- [23] Yiu, H., and Ewins, D. J., 1986, "Dependence on Blade Arrangements of Mistuned Bladed Disc of the Optimal and Critical Resonant Responses," *Rotating Machinery, Proceedings of the 6th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery*, Vol. 1, Hemisphere, Washington, DC, pp. 237–251.
- [24] Castanier, M. P., and Pierre, C., 1997, "Consideration on the Benefits of Intentional Blade Mistuning for the Forced Response of Turbomachinery Rotors," *Proceedings of The 1997 International Mechanical Engineering Congress and Exposition, The Winter Annual Meeting of the ASME*, Vol. AD-55, pp. 419–425.
- [25] Castanier, M. P., and Pierre, C., 1998, "Investigation of the Combined Effects of Intentional and Random Mistuning on the Forced Response of Bladed Disks," *Proceedings of the 34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*, Cleveland, OH.
- [26] Castanier, M. P., Ottarsson, G., and Pierre, C., 1997, "A Reduced-Order Modeling Technique for Mistuned Bladed Disks," *ASME J. Vibr. Acoust.*, **119**, No. 3, pp. 439–447.
- [27] Kruse, M. J., and Pierre, C., 1997a, "An Experimental Investigation of Vibration Localization in Bladed Disks, Part I: Free Response," *Proceedings of the 42nd ASME Gas Turbine & Aeroengine Congress, User's Symposium & Exposition*, Orlando, FL.
- [28] Kruse, M. J., and Pierre, C., 1997b, "An Experimental Investigation of Vibration Localization in Bladed Disks, Part II: Forced Response," *Proceedings of the 42nd ASME Gas Turbine & Aeroengine Congress, User's Symposium & Exposition*, Orlando, FL.

A Nonlinear, Transient, Single-Cylinder Diesel Engine Simulation for Predictions of Instantaneous Engine Speed and Torque

Z. S. Filipi

D. N. Assanis

W. E. Lay Automotive Laboratory
Automotive Research Center,
Department of Mechanical Engineering and
Applied Mechanics,
University of Michigan,
Ann Arbor, MI 48109-2121

A non-linear, transient, single-cylinder diesel engine simulation has been developed for predictions of instantaneous engine speed and torque. The foundation of our model is a physically based, thermodynamic, steady-state diesel engine simulation (Assanis, D. N., and Heywood, J. B., 1986, "Development and Use of a Computer Simulation of the Turbocompounded Diesel System for Engine Performance and Component Heat Transfer Studies," SAE Paper 860329), which has been comprehensively validated for various engine designs. The transient extension of the parent model represents the diesel engine as a non-linear, dynamic system. The instantaneous crank-shaft speed is determined from the solution of the engine-external load dynamics equation, where the engine torque is tracked on a crank-angle basis. Validation of the transient model during rapid engine acceleration shows that both the cyclic fluctuations in the instantaneous crank-shaft speed line and the overall engine response are in good agreement with experimental measurements. Predictions of single-cylinder engine starting reveals the importance of selecting the proper value of the engine moment of inertia in order to control the amplitude of angular velocity fluctuations and ensure stable engine operation. It is further shown that the variation in the inertial forces on the reciprocating components with speed has a dramatic impact on the instantaneous torque profile, and consequently on angular velocity fluctuations. [DOI: 10.1115/1.1365122]

Introduction

The direct-injection (DI) diesel engine has been widely adopted as the heavy-duty automotive powerplant of choice, primarily because of its high-energy conversion efficiency and reliability. Present initiatives such as the Partnership for the New Generation Vehicle (PNGV) also focus on DI diesels for passenger-car applications. With the modern emphasis on analytical design and virtual prototyping, high fidelity, predictive transient engine simulations are needed to model performance and fuel economy of diesel propulsion systems that frequently experience rapid changes of speed and load.

Previous efforts in the area of modeling transient engine performance can be grouped in three major categories, i.e., linear, quasi-linear, and non-linear models. Examples of the linear model based on the sampled data theory are the works of Bowns [1], Hazell and Flower [2], and Windett and Flower [3]. Such linear models rely on the analogy between the periodic and discrete nature of fuel injection and a data sampling process. However, although the sampled data models are capturing some aspects of the discontinuous nature of a reciprocating engine, their major handicap comes from the assumption of a constant sampling interval.

As a further step towards dynamic simulation, quasi-linear models link steady-state, measured thermal and gas flow data with dynamic models of engine components ([4–7]). The idea is to use the experimental relations that link engine torque, mass flow, and temperature rise of the working fluid with the pressure in the

intake manifold, fuel flow, trapped air-fuel ratio, and engine speed in addition to the steady-state compressor and turbine characteristics. The most important disadvantage of all quasi-linear models is their heavy dependence on experimental data which need to be generated prior to the simulation study. Also, complex phenomena such as combustion and valve flow are represented by very simplified equations, a fact that limits simulation accuracy, especially at off-design conditions.

Since the diesel engine is a highly non-linear mechanical system, truly predictive capabilities can be attained only if purely non-linear simulations are developed. Thermodynamic engine simulations have been refined with phenomenological process descriptions over the years (e.g., ([8,9])) and have been used very successfully for predictions of steady-state engine operation. Consequently, the natural approach to the development of the non-linear diesel engine simulation is by building onto and extending such steady-state thermodynamic models to consider engine dynamics. However, the path followed by Winterbone et al. [10] was to somewhat simplify the calculation of the closed part of the cycle in order to speed up calculations, and use a mean torque value in the engine dynamic equation; the latter was integrated only at the end of each computed cycle. Watson and Marzouk [11] applied a set of differential equations to all thermodynamic processes and included the engine dynamics equation on a crank-angle basis; however, the instantaneous engine torque was tied to its mean indicated pressure to avoid modeling of the crank-slider kinematics. Both groups presented successful studies of aspects related to system control and response, e.g., turbocharger lag, but due to the inherent simplifications, simulation fidelity on a cycle-resolved basis was compromised.

An important aspect of engine dynamic operation is that instantaneous torque, and hence crank-shaft speed, fluctuate in response to the cyclic nature of the gas pressure force and the slider-crank

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 Jan. 1997; final revision received by the ASME Headquarters Oct. 2000. Editor: H. D. Nelson.

kinematics. Such cyclic fluctuations are accentuated for single-cylinder engine operation. Therefore, the objective of this work is to develop a non-linear single-cylinder diesel engine simulation with full transient capability, on a crank-angle basis. The foundation for the transient model is the physically based, thermodynamic, steady-state diesel engine simulation introduced by Assanis and Heywood [9] in the context of the turbocharged and turbocompound system. In order to extend the steady-state model, the instantaneous crank-shaft speed must be determined from the dynamic balance of the engine and external load torques. Hence, an instantaneous torque model is a necessary prerequisite for yielding accurate predictions of the cyclic fluctuations of the crank-shaft angular velocity.

This paper is arranged as follows. First, the assumptions involved in the parent thermodynamic model are described, along with a brief summary of its physical submodels. Next, the engine dynamics model and the instantaneous single-cylinder engine torque model are formulated on a crank-angle basis, and are implemented in the parent model to form its transient extension. Subsequently, the dynamic engine operation during a single cycle is illustrated and the predicted engine response is validated through comparison with measured data. Next, the sensitivity of the transient simulation to changes of some input parameters likely to vary significantly during engine transient operation is examined. Predictions of transient engine behavior during starting and during free acceleration from low speed to rated speed are presented in the final section. The effect of the moment of inertia and the mean speed of the engine on the amplitude of fluctuations in crank-shaft angular velocity is also examined. The results demonstrate the importance of simulating the single-cylinder, diesel engine as a non-linear system. The extension of the non-linear single-cylinder model to multi-cylinder simulation can be readily accomplished with the aid of a graphical programming environment ([12]).

Thermodynamic Model of the Engine Cylinder and Physical Submodels

The thermodynamic, zero-dimensional, steady-state diesel engine simulation, which lay the foundation for the current work, is described by Assanis and Heywood [9]. In the parent model, the cyclic processes in the cylinder are represented by a blend of more fundamental and phenomenological models of turbulence, combustion, and heat transfer. The parent simulation has been validated against test results from diesel engines of various sizes, ranging from highway truck engines ([9]) to large locomotive engines ([13]). This section briefly summarizes the main assumptions of the diesel engine model and its phenomenological submodels. Additional details on the parent code are given by Assanis [14].

The diesel four-stroke cycle is treated as a sequence of continuous processes: intake, compression, combustion (including expansion), and exhaust. The durations of the individual processes are as follows. The intake process begins when the intake valve opens and ends when the intake valve closes. The compression process begins at the intake valve closing and ends at the time of ignition. The combustion process begins when ignition occurs and ends when the exhaust valve opens. The exhaust process begins upon exhaust valve opening and ends when the intake valve opens (and not when the exhaust valve closes).

The system of interest is the instantaneous contents of the cylinder, i.e., air, fuel, and combustion products. In general, this system is open to the transfer of mass, enthalpy, and energy in the form of work and heat; see Fig. 1. Throughout the cycle, the cylinder is treated as a variable volume plenum, spatially uniform in pressure. Furthermore, the cylinder contents are represented as one continuous medium by defining an average equivalence ratio and temperature at all times.

Gas properties are calculated assuming ideal gas behavior. At low temperatures (below 1000 K), the cylinder contents are

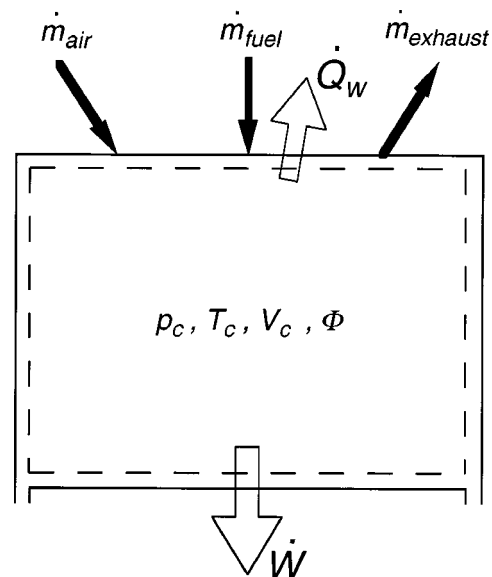


Fig. 1 Diesel engine cylinder as a thermodynamic control volume

treated as a homogeneous mixture of non-reacting ideal gases. At high temperatures (above 1000 K), the properties of the cylinder contents are calculated with allowance for chemical dissociation by assuming that the burned gases are in equilibrium, using an approximate calculation method based on hydrocarbon-air combustion.

Quasi-steady, adiabatic, one-dimensional flow equations are used to predict mass flows past the intake and exhaust valves. The intake manifold and the exhaust port are treated as plenums whose pressure and temperature history are specified as input.

The compression process is defined so as to include the ignition delay period, i.e., the time interval between the start of the injection process (the point at which the injector needle starts to lift) and the ignition point (the start of positive heat release due to combustion). The total length of the ignition delay is related to the mean cylinder gas temperature and pressure during the delay period by an empirical Arrhenius expression ([9]).

Combustion is modeled as a uniformly distributed heat release process. The rate of heat release is assumed to be proportional to the rate of fuel burning, which is modeled empirically. Since the diesel combustion process is comprised of a pre-mixed and a diffusion-controlled combustion mechanism, Watson's fuel burning rate correlation ([15]), consisting of the sum of two algebraic functions, one for each combustion mechanism, is used. The fraction of the total fuel injected that is burnt by either mechanism depends on the length of the ignition delay period and the engine load and speed.

Heat transfer is included in all the engine processes. Convective heat transfer is modeled using available engine correlations based on the Nusselt number for the turbulent flow in pipes. The characteristic velocity and length scales required to evaluate the Reynolds number in the correlation are obtained from a mean and turbulent kinetic energy model ([9]), hence any changes in the flow field inside the cylinder directly affect the heat transfer process. Radiative heat transfer is added during combustion. The steady-state inside wall surface temperatures of the piston, cylinder head, and liner can be either specified or calculated from a specification of the component wall structure.

Engine Dynamics

Figure 2 shows a schematic representation of the single-cylinder engine dynamic system. At a given instant in time, the operator command, the fueling rate, and the ambient conditions

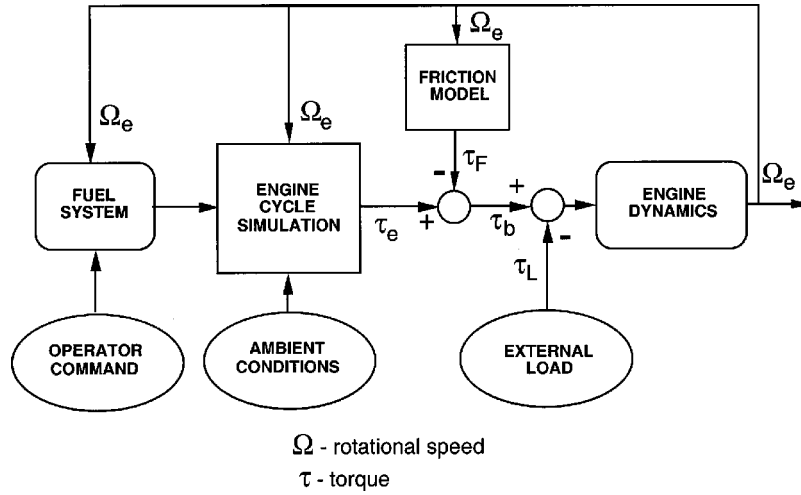


Fig. 2 Block diagram for the single-cylinder engine dynamic system

are key inputs to the diesel engine cycle simulation. The indicated engine torque is the output required for the engine dynamics equation. The friction torque is subtracted from the instantaneous indicated torque value to produce the brake torque at the shaft. Subsequently, the resistance torque, which is the result of external loading imposed on the engine by the vehicle or the dynamometer, is subtracted from the brake torque and the net value is passed on to the engine dynamics submodel. If the net torque value is positive, the engine will accelerate, and vice versa, a deficit in brake torque will trigger a decrease in shaft speed. Solution of the engine dynamics equation supplies the instantaneous value of the angular shaft velocity, which is a necessary input for computing friction and overall cycle parameters at the next instant in time.

The instantaneous rotational speed of the crank shaft is unsteady during any engine cycle, even if the mean speed in rev/min is constant, due to rapid changes in cylinder pressure and consequent changes in forces acting on the crank during the cycle. Therefore, the engine dynamics equation should be applied on a crank-angle basis. The engine rotational velocity is

$$\omega_e = \frac{d\Theta}{dt}, \quad (1)$$

where ω_e is the rotational velocity (rad/s), Θ is the angular position of the crank shaft (rad), and t is time (s). As shown in Fig. 3, the engine/load system can be modeled as two disks connected by a weightless shaft, one disk having the equivalent polar moment of inertia of the engine moving parts (I_e), and the other having the equivalent polar moment of inertia of the load (I_L , vehicle or dynamometer in the test cell). The active torque is the engine indicated torque (τ_e), and it is opposed by the engine friction torque (τ_F) and the external load (τ_L). The following two equations describe the dynamic system in the case of an elastic shaft:

$$I_e \dot{\omega}_e + \tau_F + k(\Theta_e - \Theta_L) = \tau_e, \quad (2)$$

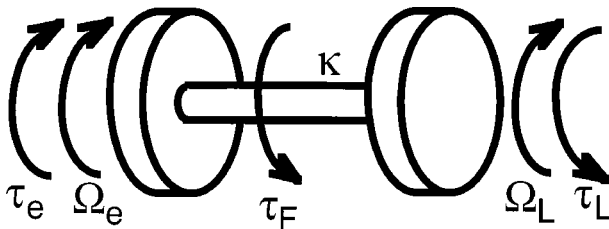


Fig. 3 Equivalent system for engine-external load dynamics

$$I_L \dot{\omega}_L + \tau_L = k(\Theta_e - \Theta_L), \quad (3)$$

where k is the rigidity of the shaft. The power transmission shaft can usually be considered sufficiently rigid, and consequently the twist between the two disks can be neglected. Hence, ω_e is equal to ω_L , and thus there is no phase shift, i.e., $\Theta_e = \Theta_L$. In that case, the system dynamic equation becomes

$$\dot{\omega}_e = \frac{\tau_e - \tau_F - \tau_L}{I_e + I_L}, \quad (4)$$

where τ is the torque in N m, I is the polar moment of inertia in kg m^2 , and ω is the shaft rotational velocity in rad/s. Sudden changes in either operator command or external load during transients can result in dramatic changes in the values of angular crank-shaft acceleration, according to Eq. (4). Since the engine cycle simulation is using the angular position of the crank shaft in degrees crank angle to time different events, the units of Eq. (4) need to be changed from radians to degrees. Hence, the engine dynamics equation is transformed into

$$\frac{d\Omega_e}{dt} (^{\circ}\text{CA/s}^2) = \dot{\omega}_e \frac{180}{\pi} = \frac{180}{\pi} \frac{\tau_e - \tau_F - \tau_L}{I_e + I_L}. \quad (5)$$

This derivative becomes an additional equation in the system of first order differential equations that form the diesel engine simulation. Typically, all time derivatives are converted to derivatives with respect to crank angle before they are passed on to the integrator routine. The integrator used is the standardized FORTRAN routine ODERT developed by Shampine and Gordon [16]. ODERT is based on a predictor-corrector technique that uses a modified divided difference form of the Adams Pece formulas. It has proved to be fully capable of handling the additional differential equation for cycle-resolved engine dynamics without any stability problems. ODERT returns the new value of Ω_e , from which the instantaneous engine speed (rev/min) can be computed:

$$N_e = \frac{\Omega_e}{6}. \quad (6)$$

This dynamic analysis requires a high fidelity, instantaneous engine torque model, such as the one presented in the next section. In addition, a friction submodel is needed to provide the instantaneous value of the friction resistance torque. Our approach is based on calculating the latter at each integration step for the current value of engine speed using the Millington and Hartles [17] correlation for the mean friction torque.

Instantaneous Engine Torque

A schematic of the piston (slider)-crank mechanism is shown in Fig. 4. Important geometrical parameters are the crank-shaft angular position Θ , the angle of swing β of the connecting rod from the cylinder axis, the crank radius R equal to half of the stroke, the distance L between the axes of the two connecting rod pins/journals, and $\lambda = R/L$.

There are two forces acting on the piston, i.e., the pressure force

$$\vec{F}_{\text{pres}} = (p_{\text{cyl}} - p_{\text{atm}}) \cdot \frac{B^2 \Pi}{4} \quad (7)$$

and the inertial force

$$\vec{F}_{\text{iner}} = -(m_{\text{pg}} + m_{\text{rcr}}) \cdot a, \quad (8)$$

where p_{cyl} is the instantaneous cylinder pressure (Pa), p_{atm} is the ambient pressure (Pa), m_{pg} is the mass of the piston group, m_{rcr} is the reciprocating part of the connecting rod mass, and a is the linear acceleration of reciprocating components. The acceleration can be calculated using the following exact expression:

$$a = R\omega^2 \left[\cos \Theta + \frac{\lambda (\cos 2\Theta + \lambda^2 \sin^4 \Theta)}{(1 - \lambda^2 \sin^2 \Theta)^{3/2}} \right] + R\dot{\omega} \sin \Theta \left[1 + \frac{\lambda \cos \Theta}{(1 - \lambda^2 \sin^2 \Theta)^{1/2}} \right], \quad (9)$$

where ω is the shaft rotational velocity. The resultant force on the piston is

$$\vec{F}_{\text{pstn}} = \vec{F}_{\text{pres}} + \vec{F}_{\text{iner}}. \quad (10)$$

The component force in the direction of the connecting rod axis is

$$\vec{C} = \vec{F}_{\text{pstn}} \left(\frac{1}{\cos \beta} \right) \quad (11)$$

and the component of the force on the crank journal in the tangential direction is

$$\vec{T} = \vec{F}_{\text{pstn}} \left(\frac{\sin(\Theta + \beta)}{\cos \beta} \right). \quad (12)$$

Finally, the torque on the shaft can be calculated directly if the value of the tangential force is multiplied by the radius R , i.e.,

$$\text{TORQUE} = T \cdot R. \quad (13)$$

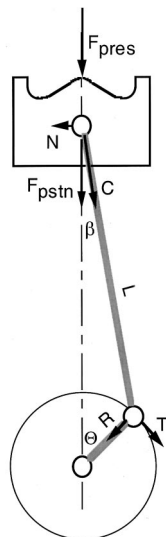


Fig. 4 Forces on the slider-crank mechanism

Implementation of the instantaneous torque model obviously requires accurate masses of the reciprocating components in addition to the detailed engine geometry. In smaller engines, the piston pin is often slightly offset in order to reduce engine noise and wear during the change of the direction of normal force on the piston at the end of compression. In that case, the correct expression for the piston acceleration becomes somewhat more complex, although the actual correction does not significantly affect torque calculations.

Engine Dynamics Model Behavior and Experimental Validation

In order to explore the behavior of the engine dynamics model in the light of experimental results, an air-cooled, direct-injection, single-cylinder Deutz engine, specified in Table 1, was simulated. This engine was selected because it is fully instrumented and set up for cold start and friction investigations at Wayne State University, a member of the Automotive Research Center. Detailed transient experimental data for this engine are presented by Liu et al. [18].

The calculated instantaneous torque that the single-cylinder diesel engine develops at a nominal, "steady-state" speed of 980 rpm is shown versus crank angle in Fig. 5. The peak torque value corresponds to the peak pressure in the cylinder during combustion. As a consequence of huge fluctuations in single-cylinder engine torque during the cycle, the cycle-periodic variations of the instantaneous crank-shaft rotational speed are also very pronounced, as shown in Fig. 6.

The sudden drop of speed and its subsequent increase can be associated with the negative and positive peaks of the engine torque, shown in Fig. 5. Very rapid acceleration or deceleration of the crank shaft during a cycle can have a significant impact on engine friction. The observed trends agree very well with experimental results published by Marek et al. [19].

A comparison of measured and predicted values of the instantaneous angular velocity of the crank shaft during an elementary engine transient is presented in Fig. 7. The engine accelerates from idle under full load, defined as 100 percent fueling rate, and goes through the whole speed range until it reaches its maximum speed. From that point on, the governor cuts off fuel in order to prevent the engine from overspeeding. At that point, the engine essentially operates with minimum fuel and decelerates towards the rated speed. The amplitudes of cyclic speed fluctuations tend to decrease as the mean engine speed increases. The agreement between calculated and measured cyclic profiles that make up the characteristic line of the crank-shaft angular velocity is excellent, especially between 500 and 2000 rpm. This effectively validates the model for instantaneous torque predictions. Even beyond 2000 rpm, the overall agreement between the measured and predicted traces is acceptable. Note that it takes the model only one extra cycle to reach the maximum engine speed, which slightly exceeds 2500 rpm. The minor discrepancies at very high speed can be attributed to transient phenomena not accounted for in the process submodels, and also in model inputs that are difficult to measure

Table 1 Deutz MAG F1L 210 D engine specification

Engine type		Four-stroke, single-cylinder, DI
Bore	m	0.095
Stroke	m	0.095
Connecting rod length	m	0.16
Compression ratio		17
Inlet valve opens	° BTDC	23
Inlet valve closes	° ABDC	63
Exhaust valve opens	° BBDC	63
Exhaust valve closes	° ATDC	23
Maximum fuel per cycle	mm ³	40

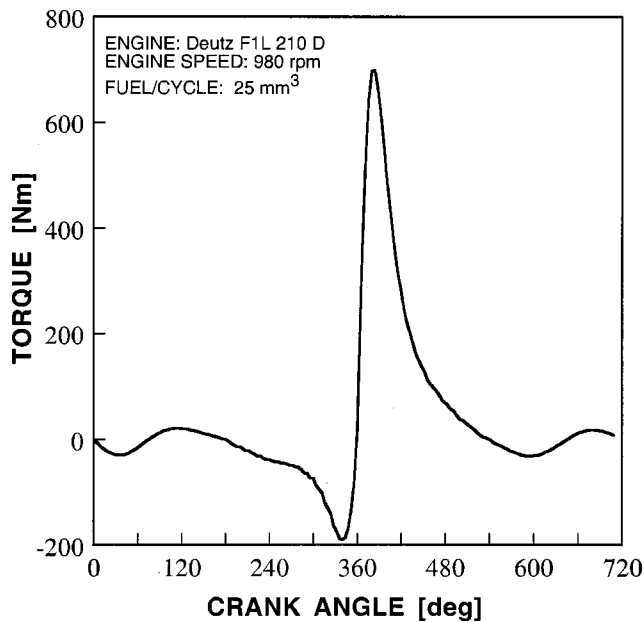


Fig. 5 Calculated fluctuations of torque on the crank shaft of the single-cylinder engine during a "steady-state" cycle

during a transient, such as the change of oil temperature and friction, the effect of engine speed on the cyclic amount of fuel injected, etc.

Behavior of Cycle Process Submodels During Transients

Before exercising the transient model, a series of steady-state calculations were performed for different operating conditions that are likely to occur during a transient sequence. The intent of this exercise was to examine the behavior of the various quasi-steady physical submodels (ignition delay, rate of heat release, turbulence, heat transfer) built into the simulation, and to assess their

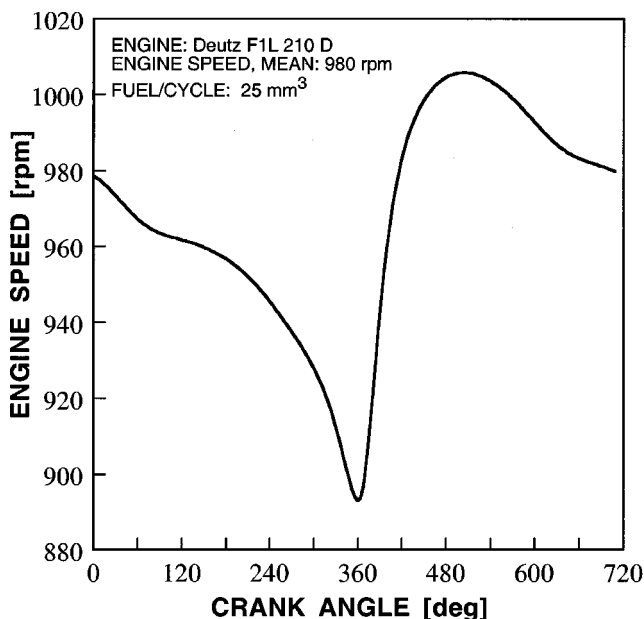


Fig. 6 Calculated fluctuations of the instantaneous crankshaft angular velocity in the single-cylinder engine during a "steady-state" cycle

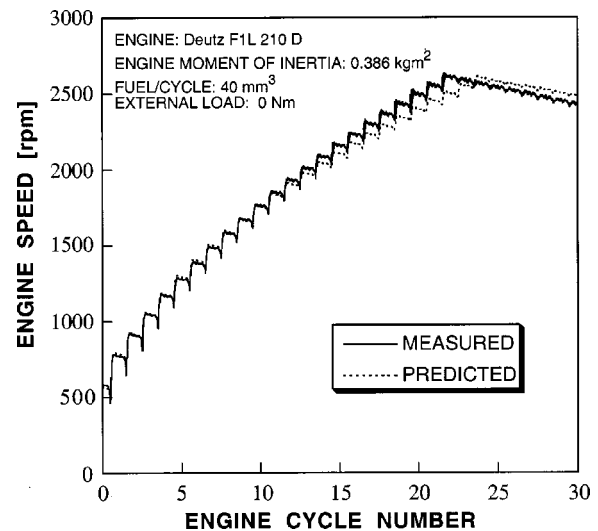


Fig. 7 Comparison of predicted and measured instantaneous single-cylinder engine speed during free acceleration with 100 percent fueling rate and no external load

ability to capture the effects of extreme changes in various operating parameters on engine processes. Engine speed and boost pressure were selected as the input parameters expected to vary significantly during transients. Computations have been performed for a hypothetical single-cylinder version of a representative, heavy-duty, truck diesel engine; its geometrical specifications are summarized in Table 2.

The results of the parametric study of the effect of the engine speed on the cylinder pressure, temperature, rate of heat release, turbulence intensity, and rate of heat transfer are shown in Fig. 8. All runs were performed with the same amount of fuel injected per cycle, corresponding to full load, and the same injection timing (340°CA). The ignition delay was predicted using an Arrhenius expression ([9]). The effect of speed on the cyclic pressure profile is very significant, primarily due to attendant changes in ignition delay, rate of burning profiles, and heat transfer. The observed behavior is the result of complex interactions between the physics of the in-cylinder processes and the kinematics of the engine mechanism. Clearly, for any given real time interval, the engine crank shaft will rotate more degrees, and thus the piston will move much more quickly towards top dead center (TDC) at higher engine speeds. That will also cause more rapid changes in gas pressure and temperature [see Fig. 8(a)] i.e., the primary factors affecting the ignition delay correlation. The rate of heat release (RHR) profiles change as a result of ignition delay changes, as well as the fact that a given mass of fuel has less real time for injection and burning within the target crank-angle window. The observed behavior stresses the need for electronic control of injection timing if the engine is to operate at its optimum point at any speed.

Figure 8(b) illustrates the direct impact of engine speed on turbulence intensity. Although turbulence intensity profiles are different at the beginning of the intake process, the trends become very regular during the main part of the intake, and differences persist throughout the rest of the cycle. Figure 8(b) shows that the rate of heat transfer (RHT) increases with engine speed as well, but the differences become obvious only during combustion and expansion. One mechanism behind the increased RHT values is the effect of enhanced turbulence and mean flow parameters on the heat transfer coefficient; however, this effect is cushioned by the fact that increasing speed allows less time for thermal equilibrium and heat transfer exchanges to take place. The combined speed effects cause RHT variations that are significant, but certainly non-linear with the change in speed.

Table 2 Single-cylinder engine specification

Engine type		Four-stroke, DI
Bore	m	0.13
Stroke	m	0.16
Connecting rod length	m	0.2693
Compression ratio		15

Figure 9 illustrates even more dramatic changes in cylinder pressure for three simulated levels of turbocharging, i.e., for three values of the boost pressure in the inlet manifold. Engine speed (1200 rpm), injection timing, and the temperature in the inlet manifold (intercooling was assumed) were maintained constant for all three runs. The amount of fuel injected per cycle was adjusted in each run to produce the same air-fuel ratio for the specified level of boost pressure. The variation in the manifold air density has an obvious effect on the mass flow rate through the inlet valves [see Fig. 9(a)]. Higher pressures at the end of induction lead to higher pressures at the end of compression. Consequently, the peak cylinder pressure is much higher (148.4 bars) for the highly boosted condition compared to only 61.4 bars for naturally aspirated operation [see Fig. 9(b)].

The RHR profiles are also changing, with higher compression pressures resulting in shorter ignition delays and smaller premixed spikes. While the predicted change in ignition delay when boost pressure is reduced to 1.65 bars is relatively small, the peak RHR value during the premixed phase of burning is substantially higher; further, a much larger fraction of fuel-air mixture burns in the premixed mode than for the highly boosted condition. With natural aspiration, ignition occurs very late, and the distribution between the two modes of burning is changed towards a further reduction of the diffusion burning fraction; however, since the piston is already close to TDC at ignition and starts to move away from TDC before the rapid burning phase is over, the premixed spike is not as sharp as in the previous two cases. The different

pressure gradients at the beginning of combustion as a function of degree of boost should be taken into consideration in engine noise and vibration studies, and in injection strategy optimization. Studying this type of “snapshot” during a transient can be a very valuable aid for the engine designer when calculating forces acting on engine components, and structural stresses and vibrations resulting from those forces.

Results of cycle calculations discussed in this section indicate that the “quasi-steady” engine process submodels are capable of predicting most of the non-linear effects of changes in operating conditions during transients on engine performance. Note, however, that irregular behavior during transients might occur during engine excursions to off-design points, especially after sudden changes of speed and load. The extensions of quasi-steady models of fuel-air mixing, heat transfer, and friction to include additional terms dependent on shaft angular acceleration values will be the focus of further model refinements.

Overall Transient Model Behavior

Having demonstrated the validity of the engine dynamics model and the consistent behavior of various physical process submodels, the simulation was used to examine the overall behavior of the single-cylinder engine of Table 2 during two elementary transients, one dealing with cold start and the other with free acceleration from idle.

For the cold start test, it is assumed that the starter brings the engine to the starting speed, and as soon as the first cycle fires, the starter disengages, i.e., the only torque acting on the crank shaft is the algebraic sum of the indicated and the friction torque. The baseline value of the polar moment of inertia of the engine is 1.4 kg m^2 . However, in order to examine the effect of inertia on engine response and on the amplitude of fluctuations in crank-shaft speed during starting, the simulation was run for two additional values (2.8 and 4.2 kg m^2) of the moment of inertia. For all three

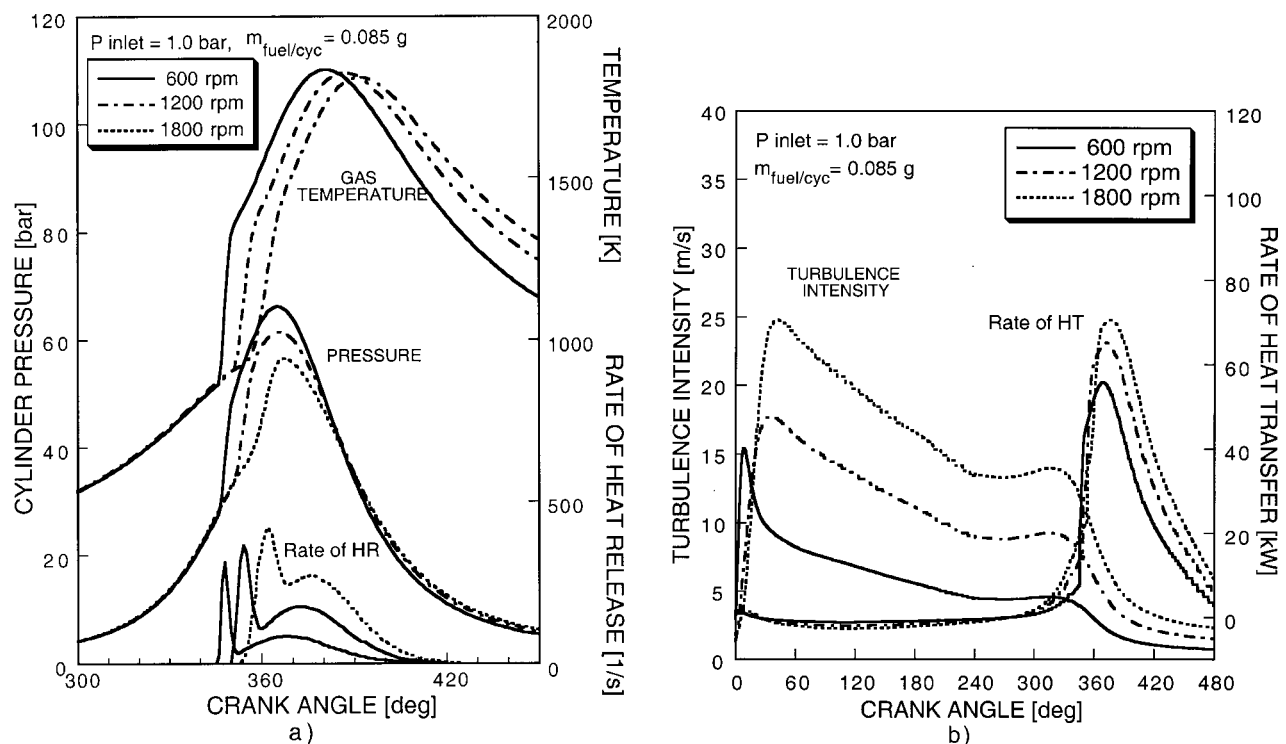


Fig. 8 Predicted effect of engine speed on (a) cylinder pressure, temperature, and rate of heat release; (b) turbulence intensity and rate of heat transfer in the single-cylinder diesel engine

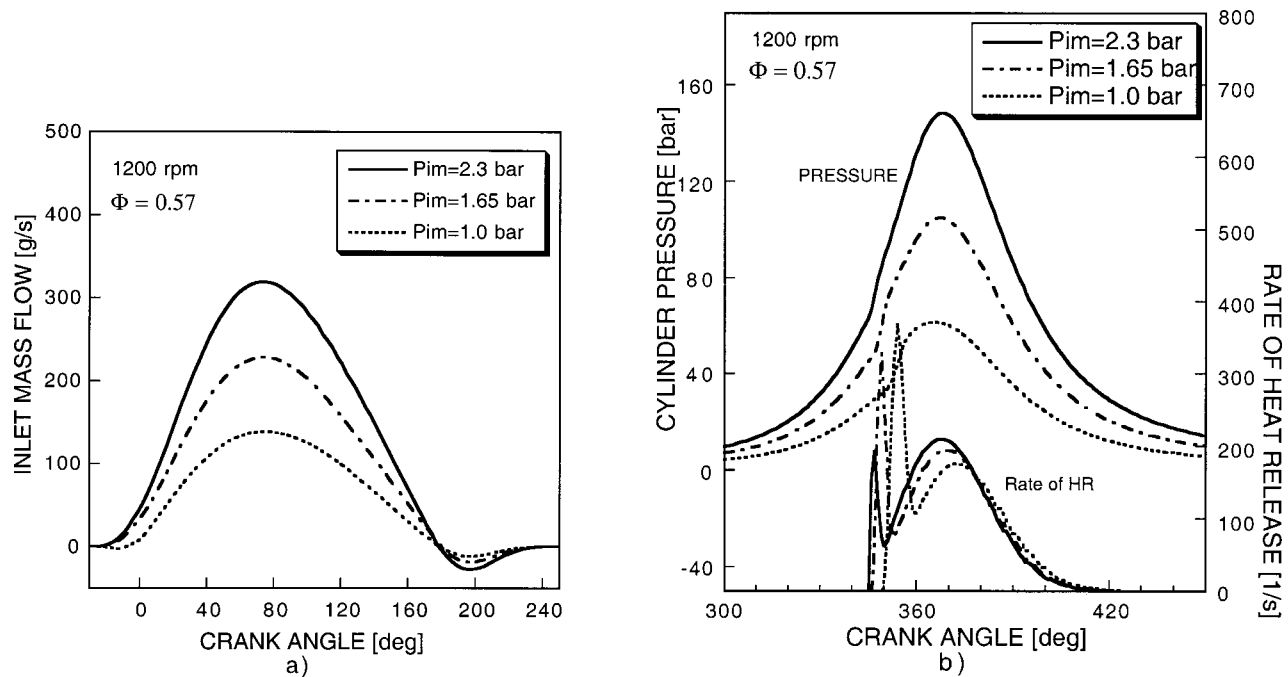


Fig. 9 Predicted effect of the inlet manifold pressure on (a) flow rate through the inlet valve; (b) cylinder pressure and rate of heat release in the single-cylinder diesel engine

runs, the mass of fuel injected per cycle was assumed to be constant and equivalent to 100 percent engine load until the engine reaches its prescribed idle speed; then the amount of fuel is reduced to keep the engine operating at “steady state.”

The evolution of the engine speed profile for the three cases under investigation is shown in Fig. 10. The cyclic fluctuations in the rotational speed of the crank shaft for the low inertia case are very large as a consequence of the instantaneous torque behavior. Note that the amplitude and period of fluctuations are changing from cycle to cycle. The amplitude of fluctuations is much higher at low speeds, highlighting the critical phase of engine deceleration at the end of compression, when the engine might come to a stop if the mean starting crank-shaft speed is not high enough. For larger values of the engine moment of inertia, the amplitudes of the angular velocity fluctuation decrease significantly, and the engine response lags behind; however, this trend is less dramatic for higher values of the engine moment of inertia. High values of the crank-shaft angular acceleration can have a significant impact on instantaneous engine friction ([19]), on shock loading of the engine starter, and on the dynamic behavior of the fuel injection system (pump and governor).

For the second elementary transient, the engine first idles steadily at 900 rpm, and after the fifth cycle experiences a step change in its fueling rate to 100 percent; consequently, the engine accelerates rapidly until it reaches its rated speed. After the engine accelerates beyond its rated speed of 2100 rpm, it is kept at the high idle speed due to the fuel cutoff provided by a two-speed governor. Figure 11 shows the predicted instantaneous engine speed versus crank angle, as the single-cylinder engine goes through the above described elementary transient schedule. The change in the shape of the crank-shaft speed profile within each cycle is very obvious at the start of the transient. The positive gradient part of the cyclic profile becomes much higher due to the increased pressure in the cylinder, and thus increased peak torque during the expansion stroke of full load cycles. Both the amplitudes and the periods of cyclic fluctuations of crank-shaft speed gradually decrease as the mean speed increases. The explanation for the amplitude change is provided by Fig. 12.

Figure 12 shows a suite of instantaneous torque curves calcu-

lated during the entire 3-sec engine transient and superimposed on the same plot. The shape of the instantaneous torque lines changes significantly from cycle to cycle due to the direct impact of changing crank-shaft angular velocity on inertial forces on the reciprocating parts. Thus, the peak torque gradually decreases, even though the amount of fuel injected is kept at the maximum value. After the governor cuts off the fuel, there is a sudden decrease in expansion torque. Consequently, the positive gradient part in the cyclic velocity profiles becomes much smaller as peak torque values decrease, as illustrated in Fig. 11. Note also that at higher

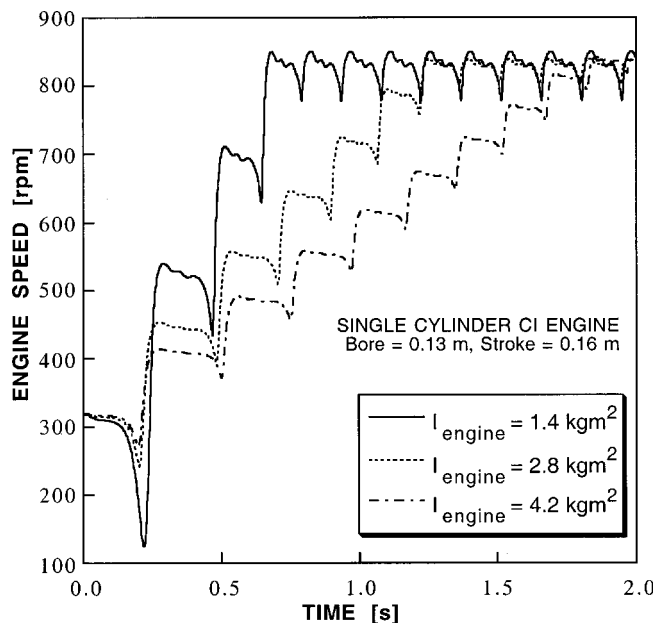


Fig. 10 Predicted crank-shaft speed fluctuations during starting for three values of engine polar moment of inertia

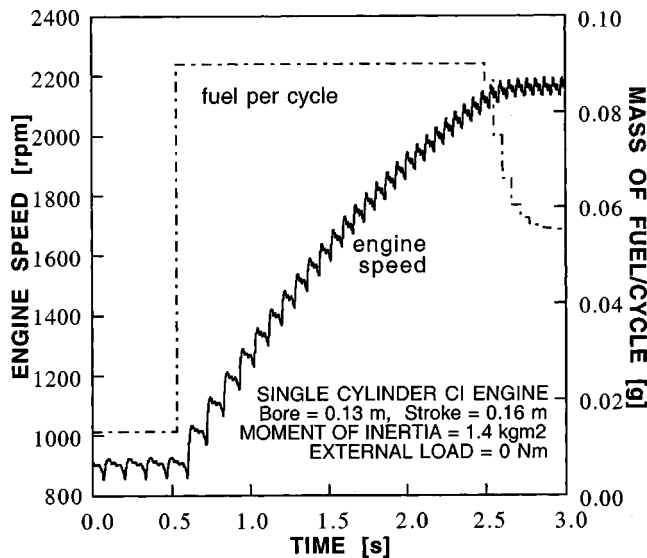


Fig. 11 Simulated engine response during an elementary transient initiated by the step change of the fueling rate

speeds, the amplitude of the instantaneous torque fluctuations increases significantly during intake and exhaust, making it comparable to the amplitude of fluctuations during the “high pressure” part of the cycle.

Summary and Conclusions

A steady-state, phenomenological, zero-dimensional model ([9]) has been used as the foundation for the development of a transient, single-cylinder, engine simulation module. The transient extension has involved the implementation of instantaneous engine torque and engine dynamics models on a crank-angle basis. Subsequently, the transient simulation has been validated against experimental results from a single-cylinder engine, and selected

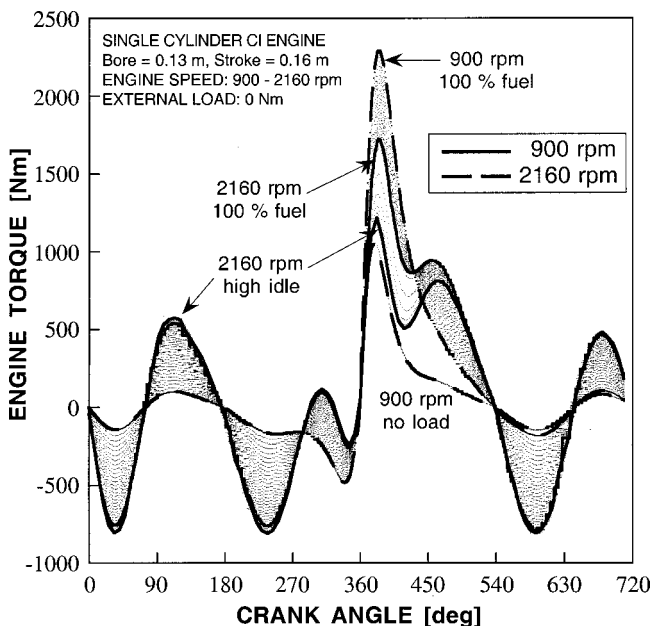


Fig. 12 A suite of the single-cylinder torque fluctuation lines calculated for individual cycles occurring during the transient shown in Fig. 11

parametric studies have been performed to illustrate the model’s capabilities. The major conclusions of the present study are as follows:

- Simulation predictions of the cyclic fluctuations in the instantaneous crank-shaft speed characteristic are in excellent agreement with the experiment. The overall engine response during free acceleration with the maximum amount of fuel injected also agrees well with the measured data.
- The amplitude of the fluctuation in the rotational speed of the crank shaft can be as high as 300 rpm, at low speeds and for low values of the polar moment of inertia. Therefore, inertia is one of the key design parameters to be considered when single-cylinder engine stability during starting is analyzed. The cushioning effect of increasing engine inertia on speed fluctuations and engine response gradually diminishes for very high values of the mean rotational velocity.
- Increased inertial forces on the reciprocating components at higher speeds cause reduction of the peak instantaneous torque value close to TDC during combustion. Hence, instantaneous fluctuations in crank-shaft angular velocity are significantly reduced at higher speeds, where engine operation is much more uniform.
- Overall, the results demonstrate the importance of simulating the single-cylinder, diesel engine as a non-linear system.

Acknowledgments

The authors would like to acknowledge the technical and financial support of the Automotive Research Center (ARC) by the National Automotive Center (NAC) located within the US Army Tank-Automotive Research, Development, and Engineering Center (TARDEC) in Warren, Michigan. The ARC is a consortium of eight universities directed by the University of Michigan. The authors would also like to thank Professor Naeim Henein and other members of the ARC research team at Wayne State University for making experimental results measured on the Deutz single-cylinder engine available for model validation purposes.

References

- [1] Bowns, D. E., 1970–71, “The Dynamic Characteristics of Reciprocating Engines,” *I. Mech. E. Proc.*, **185**, pp. 185–201.
- [2] Hazell, P. A., and Flower, J. O., 1970, “Sample-Data Theory Applied to the Modeling and Control Analysis of Compression Ignition Engines,” Part I and II, *Int. J. Control*, **13**, pp. 549–562.
- [3] Windett, G. P., and Flower, J. O., 1974, “Sample-Data Frequency-Response Measurements of a Large Diesel Engine,” *Int. J. Control*, **19**, pp. 1069–1086.
- [4] Ledger, J. D., Benson, R. S., and Whitehouse, N. D., 1973, “Dynamic Modelling of a Turbocharged Diesel Engine,” *I. Mech. E. Proc.* CP15.
- [5] Benson, R. S., Ledger, J. D., Whitehouse, N. D., and Walmsley, N. D., 1973, “Comparison of Experimental and Simulated Transient Responses of a Turbocharged Diesel Engine,” *SAE Paper 730666*.
- [6] Jensen, J. P., Kristensen, A. F., Sorenson, S. C., Houbak, N., and Hendrics, E., 1990, “Transient Simulation of a Small Turbocharged Diesel Engine,” *SAE Paper 904182*.
- [7] Berglund, S., 1993, “A Model of Turbocharged Engines as Dynamic Drivetrain Members,” *SAE Paper 933050*, *SAE Trans.*, **102**, pp. 1027–1034.
- [8] Benson, R. S., 1971, “A Comprehensive Digital Computer Program to Simulate a Compression Ignition Engine Including Intake and Exhaust Systems,” *SAE Paper 710773*.
- [9] Assanis, D. N., and Heywood, J. B., 1986, “Development and Use of a Computer Simulation of the Turbocompounded Diesel System for Engine Performance and Component Heat Transfer Studies,” *SAE Paper 860329*.
- [10] Winterbone, D. E., Thiruarooran, C., and Wellstead, P. E., 1977, “A Wholly Dynamic Model of a Turbocharged Diesel Engine for Transfer Function Evaluation,” *SAE Paper 770124*.
- [11] Watson, N., and Marzouk, M., 1977, “A Non-Linear Digital Simulation of Turbocharged Diesel Engines Under Transient Conditions,” *SAE Paper 770123*.
- [12] Zhang, G., Filipi, Z. S., and Assanis, D. N., 1997, “A Flexible, Reconfigurable, Transient Multi-Cylinder Diesel Engine Simulation for System Dynamics Studies,” *Mech. Struct. Mach.*, **25** No. 3, pp. 357–378.
- [13] Poola, R. R., Sekar, R., Assanis, D. N., and Cataldi, G. R., 1996, “Study of Oxygen-Enriched Combustion Air for Locomotive Diesel Engines,” *ICE-Vol. 27-4, Proceedings of ASME-ICE Fall Technical Conference*, Fairborn, OH.
- [14] Assanis, D. N., 1985, “A Computer Simulation of the Turbocharged Turbocompounded Diesel Engine System for Studies of Low Heat Rejection Engine Performance,” Ph.D. thesis, M.I.T.

- [15] Watson, N., Pilley, A. D., and Marzouk, M., 1980, "A Combustion Correlation for Diesel Engine Simulation," SAE Paper 800029.
- [16] Shampine, L. F., and Gordon, M. K., 1974, *Computer Solution of Ordinary Differential Equations: The Initial Value Problem*, Freeman, San Francisco.
- [17] Millington, B. W., and Hartles, E. R., 1968, "Frictional Losses in Diesel Engines," SAE Paper 680590, SAE Trans, **77**, pp. 2390–2406.
- [18] Liu, H., Chalhoub, N. G., and Henein, N., 1997, "Simulation of a Single-Cylinder Diesel Engine Under Cold-Start Conditions Using Simulink," *ASME-ICE Spring Technical Conference Proceedings*, Fort Collins, CO.
- [19] Marek, S. L., Henein, N. A., and Bryzik, W., 1991, "Effect of Load and Other Parameters on Instantaneous Frictional Torque in Reciprocating Engines," SAE Paper 910752.