NASA/TM—2020-220219

Aerothermodynamics Technical Working Group: 2008 Turbomachinery Technology Assessment and Recommendations

James D. Heidmann
Glenn Research Center, Cleveland, Ohio

Reza S. Abhari
ETH-Zurich, CH-8092 Zürich, Switzerland

Ronald S. Bunker
GE Global Research Center, Niskayuna, New York

Edward M. Greitzer
Massachusetts Institute of Technology, Cambridge, Massachusetts

Howard Hodson
University of Cambridge, Cambridge CB3 0DS, United Kingdom

Malak Malak
Honeywell International, Phoenix, Arizona

Richard B. Rivir
U.S. Air Force Research Laboratory, Wright-Patterson Air Force Base, Dayton, Ohio

Jayant Sabnis
Pratt & Whitney, East Hartford, Connecticut

Om Sharma
United Technologies Research Center, East Hartford, Connecticut

Zoltán S. Spakovszky
Massachusetts Institute of Technology, Cambridge, Massachusetts

Anthony J. Strazisar
Glenn Research Center, Cleveland, Ohio

Karen Thole
The Pennsylvania State University, University Park, Pennsylvania

Aspi R. Wadia
GE Aviation, Evendale, Ohio

Steven R. Wellborn
Rolls-Royce plc, Derby, England, United Kingdom

David E. Ashpis, Editor
Glenn Research Center, Cleveland, Ohio

February 2020

NASA STI Program . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program plays a key part in helping NASA maintain this important role.

The NASA STI Program operates under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA's STI. The NASA STI Program provides access to the NASA Technical Report Server—Registered (NTRS Reg) and NASA Technical Report Server—Public (NTRS) thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counter-part of peer-reviewed formal professional papers, but has less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., "quick-release" reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at http://www.sti.nasa.gov

- E-mail your question to help@sti.nasa.gov

- Fax your question to the NASA STI Information Desk at 757-864-6500

- Telephone the NASA STI Information Desk at 757-864-9658

- Write to:
  NASA STI Program
  Mail Stop 148
  NASA Langley Research Center
  Hampton, VA 23681-2199

NASA/TM—2020-220219

Aerothermodynamics Technical Working Group: 2008 Turbomachinery Technology Assessment and Recommendations

James D. Heidmann
Glenn Research Center, Cleveland, Ohio

Reza S. Abhari
ETH-Zurich, CH-8092 Zürich, Switzerland

Ronald S. Bunker
GE Global Research Center, Niskayuna, New York

Edward M. Greitzer
Massachusetts Institute of Technology, Cambridge, Massachusetts

Howard Hodson
University of Cambridge, Cambridge CB3 0DS, United Kingdom

Malak Malak
Honeywell International, Phoenix, Arizona

Richard B. Rivir
U.S. Air Force Research Laboratory, Wright-Patterson Air Force Base, Dayton, Ohio

Jayant Sabnis
Pratt & Whitney, East Hartford, Connecticut

Om Sharma
United Technologies Research Center, East Hartford, Connecticut

Zoltán S. Spakovszky
Massachusetts Institute of Technology, Cambridge, Massachusetts

Anthony J. Strazisar
Glenn Research Center, Cleveland, Ohio

Karen Thole
The Pennsylvania State University, University Park, Pennsylvania

Aspi R. Wadia
GE Aviation, Evendale, Ohio

Steven R. Wellborn
Rolls-Royce plc, Derby, England, United Kingdom

David E. Ashpis, Editor
Glenn Research Center, Cleveland, Ohio

National Aeronautics and
Space Administration

Glenn Research Center
Cleveland, Ohio 44135

February 2020

Acknowledgments

Supported by the Advanced Air Transport Technology Project (formerly Subsonic Fixed Wing Project) of the NASA Aeronautics Research Mission Directorate.

This work was sponsored by the Advanced Air Vehicle Program
at the NASA Glenn Research Center

Level of Review: This material has been technically reviewed by technical management.

Available from

NASA STI Program
Mail Stop 148
NASA Langley Research Center
Hampton, VA 23681-2199

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22161
703-605-6000

This report is available in electronic form at http://www.sti.nasa.gov/ and http://ntrs.nasa.gov/

Executive Summary

This report documents the 2008 Turbomachinery Technology Assessment and Recommendations of the NASA-led Aerothermodynamics Technical Working Group (TWG). It includes technology review, assessment, and recommendation for future research and development. The areas covered are summarized as follows.

Inlet Flow Distortion Sensitivity and Stability

Current aircraft engine design trends tend towards higher bypass ratio, low-pressure-ratio fan designs for improved fuel burn. Lower fan pressure ratios lead to increased propulsive efficiency, and besides enabling thermodynamic cycle changes for improved fuel efficiency, significant noise reduction can be achieved. Unfortunately, as the fan pressure ratio and fan speed are reduced, the fan design becomes more sensitive to inlet flow distortion and installation stagnation pressure losses. Casing treatments have been extensively used in aeroengine compressors to enhance operability. However, a rigorous assessment of this technology in advanced low-pressure-ratio fan designs has not yet been carried out, and the effectiveness and efficiency penalties are unknown. The proposed effort focuses on the rigorous investigation of the underlying mechanisms and the necessary technologies to reduce inlet distortion sensitivity and stability issues in low-pressure-ratio aeroengine fan systems.

Tip Leakage Flows in High-Pressure Cores

It is well known through a large amount of experimental, analytical, and numerical investigations that the flow in the tip and near-casing region of axial compressors and fans has a strong influence on both the aerodynamic efficiency and the operating range. The demand for smaller cores has increased with new engine architectures. With the anticipated smaller cores and the resulting larger blade tip clearance of 2 to 4 percent of blade height, the resulting compressor efficiency loss due to the tip gap would be on the order of 2 to 5 percent, corresponding to a ~3-percent specific fuel consumption (SFC) impact, relative to current designs. A 50-percent reduction in tip clearance loss in the next 3 years would result in a 1- to 2-percent improvement in SFC and fuel burn.

Endwall Contouring

The three-dimensional (3D) contouring of turbine airfoils has become commonplace since about 1990. Much of this art has now become reality in recent turbine products, driven primarily by gains in aerodynamic efficiency. As an example of the potential for aerodynamic gain, a nearly 0.6-percent increase in high-pressure turbine (HPT) aerodynamic efficiency has been demonstrated through nonaxisymmetric contouring of the vane and blade endwalls. Projection of such gains through the fan, compressor, and turbine stages, leads to potential aerodynamic efficiency increases of several points. Combined with consequent advantages in cooling flow reductions, the overall entitlement for engine cycle efficiency could be as much as +2 percent.

Turbine Tip Flows

For current large commercial engines, tip clearance gaps contribute to about 25 percent of the loss in unshrouded turbine efficiency. It should be pointed out that the HPTs in current operational engines are designed to operate at very tight clearances. Next generation engines, with smaller cores, need to operate at higher normalized clearance levels. New technologies will therefore need to be developed to maintain performance and durability of the turbines at higher normalized clearance levels. Every 1 percent increase in blade gap height results in 1.5 to 2 percent of aerodynamic efficiency reduction. With the anticipated smaller cores and the resulting larger blade tip gaps of 2 to 3 percent of blade height, the resulting turbine efficiency loss due to the tip gap would be on the order of 3 to 6 percent, corresponding to a 2- to 4-percent SFC impact.

Combustor and Cooled-Turbine Interaction

Increased turbine inlet temperatures have been enabled by advances in both materials and turbine cooling technologies. Cooling technologies have historically been responsible for about two-thirds of the improvement. It is difficult to predict—and thereby improve—current designs with existing computational fluid dynamics (CFD) tools because of the combination of complex geometries, near-wall modeling required, and potential unsteady flow effects on the heat transfer and resultant material temperature prediction.

The problem is complicated by the fact that the turbine durability is governed by the turbine material temperature field rather than the gas temperature, so the thermal conduction problem must also be considered. It is imperative for improved future cooled-turbine designs and engine performance that better methods and models be developed that not only accurately describe the complex heat transfer physics, but also have reasonable turnaround time to impact the design cycle. Such a research effort would result in a better understanding of the complex flow and mixing processes in a combustor-turbine flow field, which would lead to validation of both CFD tools and design tools used for cooled-turbine design.

Highly Loaded Low-Pressure Turbines

The efficiency of the low-pressure turbine (LPT) strongly influences the SFC of an engine, where a 1-percent increase in LPT polytropic efficiency improves the fuel consumption by 0.5 to 1 percent. With efficiency levels already much greater than 90 percent, there will be little scope for improving this aspect of performance without a step change in technology. The LPT of the NASA Subsonic Fixed Wing Project is likely to have a Reynolds number (Re) in the range 70,000 to 200,000 at cruise conditions. Understanding and, crucially, being able to predict the unsteady separated and transitional suction-side boundary layer is essential in developing airfoils with increased lift that retain the already high levels of efficiency. Unfortunately, increasing the lift beyond today’s levels represents an even greater challenge, especially as a reduction in core size means that the Re is also reducing. The 3D design of LPT airfoils also holds tremendous promise for achieving improved performance. Although this has some elements in common with the endwall contouring topic, the LPT presents unique challenges, both in the blade and endwall designs.

Contents

Aerothermodynamics Technical Working Group: 2008 Turbomachinery Technology Assessment and Recommendations

James D. Heidmann*
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Reza S. Abhari
ETH-Zurich
CH-8092 Zürich, Switzerland

Ronald S. Bunker†
GE Global Research Center
Niskayuna, New York 12309

Edward M. Greitzer
Massachusetts Institute of Technology
Cambridge, Massachusetts 02139

Howard Hodson†
University of Cambridge
Cambridge CB3 0DS, United Kingdom

Malak Malak†
Honeywell International
Phoenix, Arizona 85072

Richard B. Rivir‡
U.S. Air Force Research Laboratory
Wright-Patterson Air Force Base
Dayton, Ohio 45433

Jayant Sabnis§
Pratt & Whitney
East Hartford, Connecticut 06118

Om Sharma
United Technologies Research Center
East Hartford, Connecticut 06108

Zoltán S. Spakovszky
Massachusetts Institute of Technology
Cambridge, Massachusetts 02139

Anthony J. Strazisar†
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Karen Thole
The Pennsylvania State University
University Park, Pennsylvania 16802

Aspi R. Wadia
GE Aviation
Evendale, Ohio 45246

Steven R. Wellborn
Rolls-Royce plc
Derby, England, United Kingdom

David E. Ashpis**
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

*Technical Working Group leader.
†Retired.
‡Deceased.
§Current affiliation: Massachusetts Institute of Technology.
**Editor.

1.0 Introduction

In March 2007, a NASA-led Aerothermodynamics Technical Working Group (TWG) was assembled, consisting of technical experts from industry, university, and Government agencies. The formation of this group was prompted by the desire of the NASA Subsonic Fixed Wing (SFW) Project to establish working groups focused on the key technologies relevant to the SFW Project goals in each technical discipline. It was anticipated that the TWG would be a forum for exchange of the latest ideas in the Aerothermodynamics Discipline and a method to inform the SFW Project of the critical needs in this area. It was decided to focus the group on turbomachinery, although the purview of the Aerothermodynamics Discipline is slightly more comprehensive to include all engine flows.

The membership of the group at the time was

James D. Heidmann (NASA)
Anthony J. Strazisar (NASA)
Richard B. Rivir (Air Force Research Laboratory, AFRL)
Aspi R. Wadia (GE Aviation)
Ronald S. Bunker (GE Global Research Center)
Jayant Sabnis (Pratt & Whitney)
Om Sharma (United Technologies Research Center, UTRC)
Steven R. Wellborn (Rolls-Royce)
Malak Malak (Honeywell)
Edward M. Greitzer (Massachusetts Institute of Technology, MIT)
Zoltán S. Spakovszky (Massachusetts Institute of Technology, MIT)
Karen Thole (Pennsylvania State University)
Reza S. Abhari (ETH Zurich)
Howard Hodson (University of Cambridge)

The group met for the first time in May 2007 in a face-to-face meeting at the 2007 ASME Turbo Expo Meeting in Montreal, Canada. Subsequently, the group has convened monthly by telecom in addition to a subsequent face-to-face meeting at NASA Glenn Research Center in October 2007.

One of the recent tasks of the group has been to undertake a technology survey of the turbomachinery area. This exercise was meant to address the need for a systematic method to identify high-payoff research activities in the turbomachinery area aimed at addressing the SFW noise and fuel efficiency goals for future-generation aircraft. At the October 2007 NASA Glenn meeting, a fairly comprehensive list of research areas was generated by the group. Subsequently, the group was asked to rank each research area on the basis of "interest and impact." The results were compiled, and upon consolidation of related topics, several key technology areas emerged:

(1) Inlet Flow Distortion Sensitivity and Stability
(2) Tip Leakage Flows In High-Pressure-Ratio Cores
(3) Endwall Contouring
(4) Turbine Tip Flows
(5) Combustor and Cooled-Turbine Interaction
(6) Highly Loaded Low-Pressure Turbines

The group recognized the need to describe the importance and payoff of each technical area to the NASA SFW mission and undertook creation of a white paper on each topic. A technical expert was assigned lead authorship of each topic, but the entire working group was given the opportunity to revise and augment these drafts. The resulting white papers for each of these six topics are compiled in this report.

The turbomachinery research summarized here primarily benefits the SFW Project goals by allowing the Project to meet its N+1 and N+2 reduced fuel burn goals.[1] The Project charts emphasize the need for propulsion improvements to reach the overall aircraft fuel burn goals. In the N+1 configuration, propulsion technology improvements contribute the majority of the expected fuel burn reduction, and they are expected to be required in even greater measure to accomplish the more aggressive N+2 fuel burn goals. In addition, constraints imposed on the propulsion system by noise and emissions technologies require technology improvements even to maintain current fuel burn for a given configuration.

It is hoped that this compilation of white papers will provide insight into the potential research opportunities available to the SFW Project in the turbomachinery field as well as describe the benefit of each research area toward the Project goals and metrics.

[1]N+1 is current year plus 10 years technology, and N+2 is current year plus 20 years technology.

2.0 Inlet Flow Distortion Sensitivity and Stability of Advanced Low-Pressure-Ratio Fan Designs

2.1 Background

Current aircraft engine design trends tend towards higher bypass ratio, low-pressure-ratio fan designs for improved fuel burn, and reduced emissions and noise (Ref. 2.1). The envisaged reduction of the fan pressure ratio can be achieved, for example, through a low-speed geared fan leading to novel engine architectures. Lower fan pressure ratios lead to increased propulsive efficiency, and besides enabling thermodynamic cycle changes for improved fuel efficiency, significant noise reduction can be achieved. Low-speed fan designs can potentially (1) avoid buzz-saw noise, (2) reduce fan broadband and blade-row interaction noise, (3) allow for a cutoff design for the fundamental and first harmonic blade-passing frequency (BPF) tones through a reduced blade number count, (4) reduce cabin noise, and (5) enable steeper takeoff profiles for far-field noise reduction through excess thrust capability at takeoff (Refs. 2.2 to 2.4).

Unfortunately, as the fan pressure ratio and fan speed are reduced, the fan design becomes more sensitive to inlet flow distortion and installation stagnation pressure losses. To date the stall margin assessment for cases with inlet distortion are mostly empirical and based on relations established for high-pressure-ratio fans. This poses a major challenge in quantifying and characterizing the stability boundary of low-pressure-ratio fans. More specifically, the key unknowns are

(1) The performance and stability sensitivity of low-pressure-ratio fan designs to inlet flow distortion
(2) The effectiveness of casing treatment to enhance low-pressure-ratio fan stability
(3) Strategies to reduce the sensitivity of low-pressure-ratio fans to inlet flow nonuniformity and stall onset by design

Casing treatments have been extensively used in aeroengine compressors to enhance operability. In light of the challenges associated with low-speed, low-pressure-ratio fan designs, there might be a real opportunity for casing treatments to be an effective means to enhance both the performance and the operability of such fan systems. A large number of studies have been conducted in the past that indicate casing treatment is effective at high inlet-relative Mach numbers (e.g., Ref. 2.5) and also at low Mach numbers (e.g., Ref. 2.6). However, a rigorous assessment of this technology in advanced low-pressure-ratio fan designs has not yet been carried out, and the effectiveness and efficiency penalties are unknown. In essence, casing treatments could potentially be a very effective means to improve stability and inlet distortion tolerance in low-pressure-ratio fan designs. The proposed effort focuses on the rigorous investigation of the underlying mechanism and the necessary technologies to reduce inlet distortion sensitivity and stability issues in low-pressure-ratio aeroengine fan systems.

2.2 Vision and Value Proposition

The value proposition is as follows. The proposed research is envisioned to exploit a new methodology and existing models to

(1) Identify and determine the boundaries and sensitivities of surge and stall in low-speed fan designs with inlet distortion

(2) Characterize the necessary design changes and casing treatment technologies to reduce inlet distortion sensitivity and enhance fan operability and performance

The central piece in the proposed technical approach is the use of an unsteady Euler method involving body force descriptions previously established by Gong et al. (Ref. 2.7) for multistage low-speed axial compressors.

2.3 Proposed Technical Roadmap

The proposed technical approach combines high-fidelity simulations of a low-pressure-ratio fan stage with a newly developed framework involving three-dimensional (3D) unsteady Euler calculations coupled with body force descriptions in response to inlet flow nonuniformity and casing treatments. It is suggested that all steps be carried out on an existing, advanced low-pressure-ratio fan stage where experimental data and flow-field simulations are available. The following tasks are proposed:

(1) Define body force model and extract body force distributions from computational fluid dynamics (CFD) calculations.

Implement body forces in 3D Euler method with the intermediate goal to replicate the steady-state performance characteristics of the low-pressure-ratio fan.

Besides the 3D CFD-based flow field, it is suggested to also define the body force distribution based on a streamline curvature code combined with loss bucket descriptions and deviation correlations. If experimental data are available, the body forces could also be established based on this information.

All of these calculations would be based on clean inlet flow with the goal to establish a baseline case.

(2) Conduct unsteady flow investigation using 3D unsteady Euler code to determine the onset of flow instability and to estimate the surge line in the compressor map. If available, a comparison against experimental data for validation purposes should be conducted.

(3) Assess the fan sensitivity to inlet flow distortion. Implement a number of inlet flow distortion scenarios, both circumferential and radial flow nonuniformities, representative of current inflow conditions in high-bypass-ratio aeroengines. It is suggested that this be done in consultation with an industry partner to ensure appropriate distortion indices.

Conduct 3D CFD calculations with inlet flow distortion, extract body force distributions and determine steady-state performance characteristics. Compare results against experimental data if available.

If possible, a steady CFD calculation of a full wheel should be conducted to assess the fidelity of the body-force-based approach with inlet distortion.

Carry out unsteady Euler calculations to determine stall onset with inlet distortion. Quantify performance and operability sensitivity of low-speed fan stage design.

(4) Introduce appropriate forcing on the flow in the tip region simulating the effects of casing treatment. It is suggested to carry out a parametric study by implementing various force distributions consistent with current casing treatment geometries. The idea of this study is to

implement body forces in the cells near the casing endwall instead of gridding up the grooves of the casing treatment. This would allow for a rapid and effective turnaround time.

The quest is to define the required forces on the endwall flow field to enhance the stall margin at minimal penalties in efficiency. If successful, the next step would be to determine the necessary casing treatment geometry that achieves such forcing on the tip leakage and endwall flows.

(5) Validate casing treatment method by conducting full CFD simulation with casing treatment grooves grid for inflow and outflow calculation, coupled with blade passage flow.

(6) Extract design guidelines and strategies to reduce the sensitivity of low-speed fans to inlet flow nonuniformity and stall onset by design. More specifically, determine iteratively what body force distribution is required to yield a less sensitive fan design. Based on this, carry out fan redesign and assess performance and operability using established methodology.

References

2.1 Air Travel—Greener by Design Science and Technology Sub-Group: Mitigating the Environmental Impact of Aviation: Opportunities and Priorities. Aeronaut. J., vol. 109, no. 1099, 2005, pp. 361–416.

2.2 Neise, Wolfgang; and Enghardt, Lars: Technology Approach to Aero Engine Noise Reduction. Aerosp. Sci. Technol., vol. 7, 2003, pp. 352–363.

2.3 Hall, Cesare A.; and Crichton, Daniel: Engine Design Studies for a Silent Aircraft. ASME GT–2006–90559, 2006.

2.4 de la Rosa Blanco, E.; Hall, C.A.; and Crichton, D.: Challenges in the Silent Aircraft Engine Design. AIAA 2007–454, 2007.

2.5 Mikolajczak, A.; and Pfeffer, A.: Methods to Increase Engine Stability and Tolerance to Distortions. Distortion Induced Engine Instability, AGARD Lecture Series No. 72, 1974.

2.6 Takata, H.; and Tsukuda, Y.: Stall Margin Improvement of Casing Treatment—Its Mechanism and Effectiveness. J. Eng. Power, vol. 99, 1977, pp. 121–133.

2.7 Gong, Y., et al.: A Computational Model for Short-Wavelength Stall Inception and Development in Multistage Compressors. J. Turbomach., vol. 121, no. 4, 1999, pp. 726–734.

3.0 Controlling and Mitigating Tip Leakage Flows in High-Pressure-Ratio Core Compressors

3.1 Problem Description

It is well known, through a large amount of experimental, analytical, and numerical investigations, that the flow in the tip and near-casing region of axial compressors and fans has a strong influence on both the aerodynamic efficiency and the operating range (Refs. 3.1 and 3.2). The major phenomenon responsible is the tip clearance flow and its effects. The tip clearance (or tip leakage) flow is set up by the pressure difference across the blade, from pressure side to suction side. The leakage flow has low stagnation pressure in the blade-fixed system, and thus low axial momentum, with the result that there is a decrease, or blockage, in available flow area, lowering the peak pressure rise capability and the work done by the rotor.

The tip leakage flow takes the form of an identifiable vortex structure, the leakage (or clearance) vortex, which is evident in experiments and simulations. Figure 3.1, which shows computed streamlines and stagnation pressure contours, gives a representative picture of the leakage flow in an isolated rotor. It can be seen that the vortex represents the region with the lowest stagnation pressure and hence the most susceptibility to the adverse pressure gradients inherent in compressors.

Although these features have been known for a long time, there are notable differences between transonic fans and core compressors: namely, the relative impact that tip clearance flows can have on performance and stall margin. Many military and commercial fans operate with rotor tip clearances that are a small percentage of annulus height (1 percent or less). Core compressor blades, however, generally have larger clearances in relation to their size, especially in the subsonic rear stages of the compressor. This leads to tip clearance flows being a larger source of loss and blockage. Further, core compressor

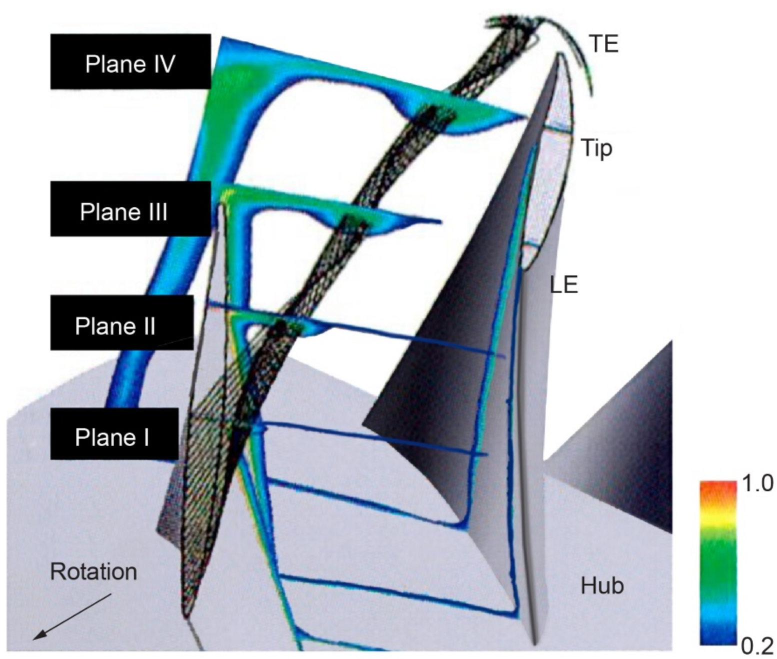

Figure 3.1.—Normalized stagnation pressure contours and streamlines in tip leakage (tip clearance) vortex. LE is leading edge and TE is trailing edge. Reproduced with permission from Reference 3.3.

clearances can open substantially above their design values during "real-world" operation because of deterioration, casing asymmetry, and thermal mismatch of rotating and stationary structures during engine transients. How well the compressor tolerates open clearances can thus determine how robustly it operates in service.

The demand for smaller size cores has increased with new engine architectures, with compressor exit corrected flows dropping from ~6 to 3 lb/s, resulting in decreased blade heights by almost a factor of two (from roughly 1 to 0.5 in. or less in the rear stages). As radial and axial clearances do not scale geometrically, there can be an increase in clearance-to-blade-height ratios from ~2 to 4 percent with consequent adverse impact on compressor operability and efficiency.

As implied above, the extent to which the tip flow influences the overall flow structure within the compressor depends on the size of the tip clearance, but another important factor is blade loading. An increase in (nondimensional) tip clearance impacts the structure of the passage flow: It allows for larger mass flow to pass over the tip, creating a tip vortex that is less amenable to mixing processes and thus more affected by the passage pressure rise. The region of high loss due to the inlet vortex is therefore more severe for large clearances. The mass flow through the clearance is also increased as the blade loading increases. Further, in the core front stages, when a shock may be present, it interacts with the endwall flow and leakage vortex, increasing the possibility of casing region separation on the suction side of the blade tip.

At conditions below design flow, the leakage flow generally exits the clearance gap into the blade passage with a negative axial velocity component and a consequent detriment to aerodynamic stability. Another feature is the amount of low-velocity tip leakage fluid that intersects the pressure surface of the adjacent blade. This flow has leaked across the previous blade and thus already has a lower stagnation pressure in the rotor system as well as lower streamwise velocity. This "double leakage" phenomenon leads to increased loss, because of the larger velocity difference between the leakage and free-stream flows, and increased blockage, compared to the classical picture of tip leakage flow. Design parameters of modern core compressors are such that they can suffer from double leakage, an effect that is exacerbated when the leakage vortex is large (in a nondimensional sense), as in the rear stages, and there is less mixing of the leakage flow with the free stream as it travels across the passage.

3.2 Overview of Proposed Research

The proposed program will consist of a coordinated computational and experimental research effort with the ultimate goal of demonstrating different novel approaches to the blading and endwall casing design to mitigate the effect of the large clearances described above. This includes, but is not limited to, (1) unique spanwise loading distributions, (2) airfoil tangential lean and dihedral (e.g., Refs. 3.4 to 3.7), (3) active control (e.g., tip injection and plasma flow control), (4) passive control such as casing treatment over the blade tips in a high-pressure compressor (Ref. 3.2), and (5) design of the upstream blading to mitigate the effects of leakages.

Some of these aspects (e.g., tailoring of the spanwise loading distribution) can work primarily on decreasing the mass flow over the tip. Others (casing treatment, tailoring of upstream blade rows, and the stagnation pressure and velocity triangle distribution presented to the rotor) are seen as affecting the stagnation pressure of the leakage flow, and thus the vortex blockage. It is likely that the approaches to be applied in practice will necessitate coupled (integrated) blading and flow control strategies. Using steady and unsteady computations, however, there is now capability for the phenomena to be interrogated and assessed computationally (e.g., Ref. 3.8) as a way to help determine (quantitatively as well as qualitatively) mechanisms, influences, and possible remedies in a way that did not exist a decade ago.

In the thrust described, there are two differences with much of the previous work on tip clearances: First, the present efforts are targeted at the large clearance regime, in which there is a major potential for improvement. Second, the emphasis is on the multistage environment, in which the possible effect of upstream blade rows, both positive and negative, needs to be taken into account.

The process envisioned is seen as having two different conceptual steps:

(1) What are the fluid dynamic effects (forces and constraints) that can give rise to tip clearance desensitization (i.e., What would we need to do to the flow to achieve the desired effect?)?
(2) Given that these fluid dynamic effects are known, what geometry or actuation system is needed to implement tip desensitization approaches in an engine (i.e., How do we do it?)?

The emphasis would be on practical innovations, which include realistic engine-level boundary conditions, thus aligning it to make substantial contribution to the specific fuel consumption (SFC) goals of the NASA SFW Project.

It is envisioned that the program will include research organization as well as industry to perform computational and experimental technology development as well as the final validation of novel approaches in high-speed rig tests or demonstrator engines.

3.3 Desired Outcome

With the anticipated smaller axial cores and the resulting larger blade tip clearance of 2 to 4 percent of blade height, the resulting compressor efficiency loss due to the tip gap could be on the order of 2 to 5 percent, corresponding to up to a 3-percent SFC impact relative to the baseline design. A 50-percent reduction in tip clearance loss in the next 3 years could result in a 1- to 2-percent improvement in SFC and fuel burn.

References

3.1 Smith, L.H.: Casing Boundary Layers in Multistage Axial Compressors. Flow Research on Blading, L.S. Dzung, ed., Elsevier Publishing, New York, NY, 1970.
3.2 Cumpsty, N.A.: Compressor Aerodynamics. Krieger Publishing Company, Malabar, FL, 2004.
3.3 Furukawa, M., et al.: The Role of Tip Leakage Vortex Breakdown in Compressor Rotor Aerodynamics. J. Turbomach., vol. 121, no. 3, 1999, pp. 469–480.
3.4 Denton, J.D.; and Xu, L.: The Effects of Lean and Sweep on Transonic Fan Performance. ASME GT2002–30327, 2002.
3.5 Gallimore, Simon J., et al.: The Use of Sweep and Dihedral in Multistage Axial Flow Compressor Blading: Part II—Low and High Speed Designs and Test Verification. ASME GT2002–30329, 2002.
3.6 Inoue M., et al.: Controlled-Endwall-Flow Blading for Multistage Axial Compressor Rotor. ASME 97–GT–248, 1997.
3.7 Wadia, Aspi R.; Hah, Chunill; and Rabe, Douglas: The Impact of Forward Sweep on Tip Clearance Flows in Transonic Compressors. ICAS 2004–6.2.1, 2004.
3.8 Lee, D., et al.: Numerical Investigation on the Effects of Self-Excited Tip Flow Unsteadiness and Blade Row Interactions on the Performance of a Low Speed Compressor Rotor. ASME GT2008–51385, 2008.

4.0 Endwall Contouring

4.1 Description of Problem

The 3D contouring of turbine airfoils has become commonplace since about 1990. Much of this art has now become reality in recent turbine products, driven primarily by gains in aerodynamic efficiency. As firing temperatures have risen to increase engine efficiency, and gas temperature profiles have become flatter to improve emissions, there has been an increasing need for solutions that provide both aerodynamic efficiency gains and lower heat loads. A general rule of hot gas path (HGP) loading is that local heat loads and aerodynamic efficiency vary in proportion to the secondary flows predominantly associated with the endwall regions of the hub and casing. Since a very large portion of the total HGP is influenced by these secondary flows, the resulting sensitivity of the overall heat load and the local part life can be a strong function of the secondary flows. Current HGP technology is restricted to cylindrical annulus endwalls having contouring only in the engine axial direction. As an example of the potential for aerodynamic gain, the study of References 4.1 and 4.2 demonstrated a nearly 0.6-percent increase in high-pressure turbine (HPT) aerodynamic efficiency through nonaxisymmetric contouring of the vane and blade endwalls. Projection of such gains through the fan, compressor, and turbine stages, leads to potential aerodynamic efficiency increases of several points. Combined with consequent advantages in cooling flow reductions due to better aerodynamics, the overall entitlement for engine cycle efficiency could be as much as +2 percent.

4.2 Overview of Proposed Research

Research into nonaxisymmetric flow-path endwalls began in the late 1990s with basic feasibility investigations such as the airfoil-endwall contouring study of Reference 4.3. Several individual studies have followed since then, leading to some initial patented endwall contour designs for vanes and blades (Refs. 4.4 to 4.7). So far, these studies have focused only on a few select or generic geometries aimed at affecting passage endwall secondary flows, shock strength, and horseshoe vortex formation. Furthermore, the majority of studies have considered only the HPT stage, with a heavy focus on aerodynamics and lesser activity on cooling. Although these are good initial efforts, a much larger breadth and depth of research into aspects of endwall contouring is required to provide a substantial contribution to the efficiency and SFC goals of the NASA SFW Project mission. Validated design solutions are sought that reduce the secondary-flow losses for compressor and turbine systems and/or reduce the heat loads or local sensitivities for the turbine HGP. Research and design activities must encompass inner and outer endwalls, HPTs, low-pressure turbines (LPTs), compressor sections, leakage flow interaction regions, blade-to-blade interactions, flow extractions, and multistage effects. This research topic is analogous in both engine scope and potential gains to the previous change from strictly linear to fully 3D airfoils.

4.3 Research Activities

Research tasks of higher priority in this effort include

(1) Design space mapping for the HPT that determines the most beneficial endwall and airfoil-endwall contouring for aerodynamic performance under cruise conditions. Individual airfoil and multistage analyses are required. Unsteady analyses will probably also be necessary.

(2) Analysis of turbine cooling with endwall contouring, including film cooling, gas temperature effects, and leakage flow interactions. Again, both individual airfoil and multistage analyses are required.
(3) Experimental validation of both aerodynamics and cooling with appropriate endwall contouring for HPTs of various designs and loading. Some low-speed testing is valuable to discern fundamental behaviors, but high-speed engine-matched testing will be required for design utilization in engines.
(4) Design space mapping for the high-pressure compressor that determines the most beneficial endwall and airfoil-endwall contouring for aerodynamic performance under cruise conditions. Individual airfoil and multistage analyses are required, including effects of stage flow extractions or injections.
(5) Analysis of compressor endwall contouring effects on surge and stall.
(6) Experimental validation of aerodynamics with appropriate endwall contouring for high-pressure compressors.

4.4 Outcomes

Research results are expected to verify that significant efficiency gains can be obtained in both the HPT and compressor sections of engines, with lesser but additive gains in the low-pressure stages. In some cases, relatively simple geometry changes can have a great impact on reducing losses. Aggregate design upgrades will result in a very high overall gain in engine efficiency, up to 2 percent. Given the typical sensitivity of aerodynamics and cooling to changes, the final proof will lie in turbine rig testing and core durability testing. The primary risks in endwall contouring will be either the lack of validation data or the inability of computational methods to fully capture unsteady phenomena in multistage turbomachinery.

References

4.1 Brennan, G., et al.: Improving the Efficiency of the Trent 500 HP Turbine Using Non-Axisymmetric End Walls: Part 1—Turbine Design. ASME 2001–GT–0444, 2001.
4.2 Rose, M.G., et al.: Improving the Efficiency of the Trent 500 HP Turbine Using Non-Axisymmetric Endwalls: Part II—Experimental Validation. ASME 2001–GT–0505, 2001.
4.3 Sauer, H.; Mueller, R.; and Vogeler, K.: Reduction of Secondary Flow Losses in Turbine Cascades by Leading Edge Modifications at the Endwall. ASME 2000–GT–0473, 2000.
4.4 Harvey, Neil; and Rose, Martin G.: Bladed Ducting for Turbomachinery. U.S. Patent 6,283,713 B1, Sept. 4, 2001.
4.5 Kvasnak, William A., et al.: Apparatus and Method for Inhibiting Radial Transfer of Core Gas Flow Within a Core Gas Flow Path of a Gas Turbine Engine. U.S. Patent 6,419,446 B1, July 16, 2002.
4.6 Hoeger, Martin; and Schmidt-Eisenlohr, Uwe: Rotary Turbomachine Having a Transonic Compressor Stage. U.S. Patent 6,017,186, Jan. 25, 2000.
4.7 Tam, Anna, et al.: Scalloped Surface Turbine Stage. U.S. Patent 7,134,842 B2, Nov. 14, 2006.

5.0 Turbine Tip Flows

5.1 Description of Problem

Tip clearance between the blade tip of a rotor and the casing is necessary for the free rotation of the rotor blade row in an unshrouded configuration. However, this clearance gap allows the fluid to migrate from the inlet to the exit of the rotor without contributing to the extraction of work in an efficient manner. The gap allows fluid to cross the blade tip from the pressure side of the blade to the suction side because of the pressure difference between them. For current large commercial engines, tip clearance gaps contribute to about 25 percent to the loss in unshrouded turbine efficiency.

This flow yields in at least three adverse impacts on the operation of the turbine:

(1) While the fluid particles are in the clearance gap region they cannot produce work because of the absence of the blade to extract the work.

(2) The blade does not turn the fluid crossing the gap, and thus no work is extracted from it. It is therefore interpreted as lost work extraction. The phenomenon of the absence of fluid particles turning in the clearance gap region and turning in the blade passage region results in the formation of a vortex in the blade passage region that contains most of the leakage fluid from the tip clearance region. The formation and mixing of the fluid from this tip clearance vortex results in loss of performance of the turbine. The loss in efficiency of the turbine due to clearance is typically 1.5 to 2.0 times the ratio of the clearance area to the flow area (~1.5 to 2.0 × (clearance gap)/(span of rotor airfoil passage)).

(3) Hot streaks entering and exiting the first vanes of the turbine tend to migrate towards the rotor tip region because of the natural radial upwash of lower density fluid. These cause loss in durability of rotor tips and higher-than-expected temperatures at the inlet to the second vanes. If the prediction models do not account for the physics of the migration of hot air in rotor passages and tip clearance regions properly, then the durability of both blade and second-vane airfoils are compromised.

It should be pointed out that the HPTs in current operational engines are designed to operate at very tight clearances, and active clearance control is also utilized where appropriate to maintain high performance. This reduced gap height, however, increases the risk of rubbing of the rotor blade on the casing due to transients, a rotor dynamic excursion, or through-casing thermal distortions. The next generation of engines, with smaller cores, need to operate at higher normalized clearance levels. New technologies would, therefore, need to be developed to maintain performance and durability of the turbines at higher normalized clearance levels.

Roughly one-third of all the aerodynamic losses in a rotor row are related to the tip leakage vortex, which forms when the tip leakage over the blade tip enters the passage flow again on the blade suction side. It creates mixing loss when it mixes out with the main flow and perturbs the pressure field on the blade tip wall that is responsible for the blade lift.

The blade material in the tip area is relatively thin. In this region, the structure of the flow causes high heat load gradients at the blade surface. In addition, hot streaks tend to migrate towards this region because of the natural radial upwash of lower density fluid. In fact, blade tips experience the highest constraint in terms of heat load, and this is a limiting factor in the blade lifetime. As such, it is desirable to minimize to less than 1 percent of the blade height the tip clearance gap height in order to improve the performance through reduction of the tip leakage mass flow. This reduced gap height, however, increases the risk of rubbing of the rotor blade on the casing due to transients, a rotor dynamic excursion, or through casing thermal distortions. In the case of a severe blade rub with a flat tip, catastrophic coolant loss could occur if the tip wears off. Even in the case of a relatively minor rub for a flat tip, any cooling holes located on the tip may be damaged, resulting in inadequate cooling and eventually leading to blade tip burnout. In addition, with the trend in the commercial aircraft engines towards smaller engine core configurations for a given fan size, the relative size of the gap as defined by the operational conditions of the engine increases. This can occur, for example, if the rotor expands further than the casing due to transients, a rotor dynamic excursion, or through casing thermal distortions.

Although the impact of tip clearance on the performance and durability of HPTs is fairly well known, the basic mechanisms governing the leakage flow is not well understood because of the complex nature of the flow in the tip region. It is, therefore, desirable to initiate a research program to develop design methodology and concepts to allow optimization of the performance and durability of HPT rotors. One solution to leverage these challenges in the blade tip design optimization for this problem is to use a recessed blade tip instead of a simple flat tip. This is especially true for the first stage of a HPT, in which shroud burnout is too likely to happen. The recess cavity inside the blade tip has several beneficial features. In the case of a tip rub, only the thin cavity rim is damaged. Wear damage to the casing is also limited, and since the purge holes for the blade tip cooling are located inside the cavity, the rubbing does not damage the outlet of the holes. Adequate cooling can thus be assured even if rubbing occurs. Alternatively, the recess cavity may act as a labyrinth seal, which could be beneficial in reducing tip clearance mass flow. However, unlike the case of a flat blade tip, the more complex flow physics for a recessed blade tip is less understood. A partial list of recent publications in this area is included at the end of this section.

5.2 Overview of Proposed Research

It is proposed here that a coordinated computational and experimental research program be initiated with the goal of demonstrating different novel approaches to the design and passive control of the blade tips in a HPT with an emphasis on comprehensive aerothermal optimization. The emphasis would be on practical developments, taking account of unsteady multistage effects, including realistic engine-level boundary conditions, which would include appropriate flow and thermal boundary conditions (i.e., inlet temperature distortions) into the turbine.

Possible research approaches could include the development of better optimization of the 3D blade design and it impact on the tip flows, novel tip geometry design methodology, blade tip cooling strategy to minimize both heat load and improve performance, outer air-band coolant management and its use to minimize blade tip losses and heat load, and casing treatment to minimize heat load. It would be beneficial to consider both unshrouded as well as shrouded, including partially shrouded, configurations.

It is envisioned that the program will include research organizations and industry to perform computational and experimental technology development as well as the final validation of novel approaches in engine-level demonstrators.

5.3 Outcomes

Blade tips are often the life-limiting parts in an aircraft engine, resulting in the expenditure of a significant amount of the cooling air to keep the temperature within an acceptable level. In addition, every 1 percent in blade gap height increase would result in a 1.5- to 2.0-percent decrease of the aerodynamic efficiency. With the anticipated smaller cores and the resulting larger blade tip gaps of 2 to 3 percent of blade height, the resulting turbine efficiency loss due to the tip gap would be on the order of 3 to 6 percent, corresponding to a 2- to 4-percent SFC impact relative to the baseline design. A 50-percent reduction in tip clearance loss in the next 3 years would have a 1- to 2-percent improvement in SFC and direct fuel consumption.

Bibliography

Mischo, Bob; Behr, Thomas; and Abhari, Reza S.: Flow Physics and Profiling of Recessed Blade Tips: Impact on Performance and Heat Load. J. Turbomach., vol. 130, no. 2, 2008, pp. 021008-1–021008-8.

Behr, T.; Kalfas, A.I.; and Abhari, R.S.: Unsteady Flow Physics and Performance of a One-and-1/2-Stage Unshrouded High Work Turbine. J. Turbomach., vol. 129, no. 2, 2007, pp. 348–359.

Mischo, Bob; Abhari, Reza; and Behr, Thomas: Turbine Blade With Recessed Tip. World International Property Organization, International Publication Number WO 2007/080189 A1, July 2007.

Molter, S.M., et al.: Heat-Flux Measurements and Predictions for the Blade Tip Region of a High-Pressure Turbine. ASME Paper GT2006–90048, 2006.

Camci, Cengiz; Dey, Debashis; and Kavurmacioglu, Levent: Aerodynamics of Tip Leakage Flows Near Partial Squealer Rims in an Axial Flow Turbine Stage. J. Turbomach., vol. 127, 2005, pp. 14–24.

Green, Brian R., et al.: Time-Averaged and Time-Accurate Aero-Dynamics for the Recessed Tip Cavity of a High-Pressure Turbine Blade and the Outer Stationary Shroud: Comparison of Computational and Experimental Results. ASME GT2004–53443, 2004.

Kwak, Jae S.; Ahn, Jaeyong; and Han, Je-Chin: Effects of Rim Location, Rim Height, and Tip Clearance on the Tip and Near Tip Region Heat Transfer of a Gas Turbine Blade. Int. J. Heat Mass Tran., vol. 47, no. 26, 2004, pp. 5651–5663.

Azad, Gm Salem, et al.: Effect of Squealer Geometry Arrangement on a Gas Turbine Blade Tip Heat Transfer. J. Heat Transfer, vol. 124, 2002, pp. 452–459.

Ameri, A.A.: Heat Transfer and Flow on the Blade Tip of a Gas Turbine Equipped With a Mean-Camberline Strip. J. Turbomach., vol. 123, no. 4, 2001, pp. 704–708.

Tallman J.; and Lakshminarayana, B.: Numerical Simulation of Tip Leakage Flows in Axial Flow Turbines, With Emphasis on Flow Physics: Part I—Effect of Tip Clearance Height. J. Turbomach., vol. 123, no. 2, 2001, pp. 314–323.

Tallman J.; and Lakshminarayana, B.: Numerical Simulation of Tip Leakage Flows in Axial Flow Turbines, With Emphasis on Flow Physics: Part II—Effect of Outer Casing Relative Motion. J. Turbomach., vol. 123, no. 2, 2001, pp. 324–333.

Ameri, A.A.; and Bunker, R.S.: Heat Transfer and Flow on the First Stage Blade Tip of a Power Generation Gas Turbine: Part 2—Simulation Results. J. Turbomach., vol. 122, no. 2, 2000, pp. 272–277.

Azad, Gm. S.; Han, Je-Chin; and Boyle, Robert J.: Heat Transfer and Flow on the Squealer Tip of a Gas Turbine Blade. J. Turbomach., vol. 122, no. 4, 2000, pp. 725–732.

Bunker, Ronald S.; Bailey, Jeremy C.; and Ameri, Ali A.: Heat Transfer and Flow on the First-Stage Blade Tip of a Power Generation Gas Turbine: Part 1—Experimental Results. J. Turbomach., vol. 122, no. 2, 2000, pp. 263–271.

Dunn, M.G.; and Haldeman, C.W: Time-Averaged Heat Flux for a Recessed Blade Tip, Lip, and Platform of a Transonic Turbine Blade. ASME GT2000–0197, 2000.

Ameri, A.A.; Steinthorsson, E.; and Rigby, D.L.: Effect of Squealer Tip on Rotor Heat Transfer and Efficiency. J. Turbomach., vol. 120, no. 4, 1998, pp. 753–759.

Staubach, J. Brent, et al.: Reduction of Tip Clearance Losses Through 3–D Airfoil Designs. ASME 96–TA–13, 1996.

Bindon, J.P.; and Morphis, G.: The Development of Axial Turbine Leakage Loss for Two Profiled Tip Geometries Using Linear Cascade Data. J. Turbomach., vol. 114, no. 1, 1992, pp. 198–203.

Heyes, F.J.G.; Hodson, H.P.; and Dailey, G.M.: The Effect of Blade Tip Geometry on the Tip Leakage Flow in Axial Turbine Cascades. J. Turbomach., vol. 114, no. 3, 1992, pp. 643–651.

Yaras, M.I.; and Sjolander, S.A.: Effects of Simulated Rotation on Tip Leakage in a Planar Cascade of Turbine Blades: Part I—Tip Gap Flow. J. Turbomach., vol. 114, no. 3, 1992, pp. 652–659.

Yaras, M.I.; and Sjolander, S.A.: Effects of Simulated Rotation on Tip Leakage in a Planar Cascade of Turbine Blades: Part II—Downstream Flow Field and Blade Loading. J. Turbomach., vol. 114, no. 3, 1992, pp. 660–667.

Bindon, J.P.: The Measurement and Formation of Tip Leakage Loss. J. Turbomach., vol. 111, no. 3, 1989, pp. 257–263.

Yaras, M.; Zhu, Yingkang; and Sjolander, S.A.: Flow Field in the Tip Gap of a Planar Cascade of Turbine Blades. J. Turbomach., vol. 111, no. 3, 1989, pp. 276–283.

Sjolander, S.A.; and Amrud, K.K.: Effects of Tip Clearance on Blade Loading in a Planar Cascade of Turbine Blades. J. Turbomach., vol. 109, no. 2, 1987, pp. 237–244.

6.0 Combustor and Cooled-Turbine Interaction

6.1 Problem Description

As gas turbine engine cycle temperatures increase for improved engine thermodynamic performance, HPT components are increasingly subject to failure caused by elevated temperatures and thermal stress. Increased turbine inlet temperatures have been enabled by advances in both materials and turbine-cooling technologies. Cooling technologies have historically been responsible for about two-thirds of the improvement, as can be inferred from Figure 6.1 and Figure 6.2. Additional 100 K temperature capabilities were enabled by thermal barrier coatings (TBCs) starting in the 1980 timeframe (see Figure 1 in Ref. 6.1). Cooling advances have progressed from plain internal cooling passages—via augmentation of the internal cooling with flow turbulators in the channels and the addition of film cooling holes on the vanes, blades, endwalls, and blade tips—to increasingly complex configurations of the above technologies. However, it is difficult to predict, and thereby improve, these designs with existing CFD tools. This is because of the combination of complex geometries, near-wall modeling required, and potential unsteady flow effects on the heat transfer and resultant material temperature prediction. One factor that is difficult to predict is the effect of a circumferentially nonuniform temperature field entering the turbine from the combustor and how that thermal field propagates through the turbine in the unsteady rotor frame of reference. This problem is complicated by the existence of leakage flows between the combustor and turbine, which impose a thermal gradient near the endwall that is also difficult

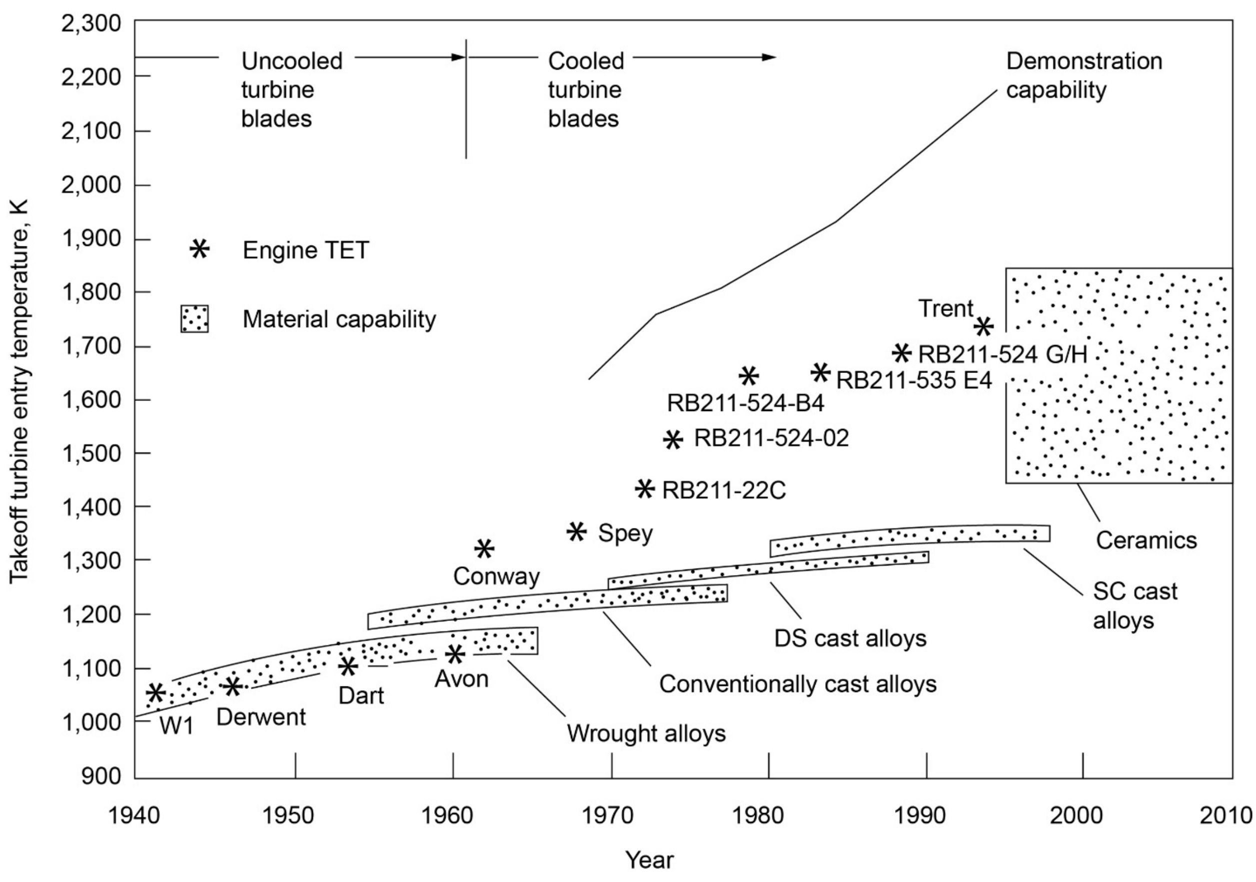

Figure 6.1.—Rolls-Royce high-temperature technology advances since 1940. TET is turbine entrance temperature, DS is directionally solidified, and SC is single crystal (from Reference 6.2; reproduced with permission of the Licensor through PLSclear).

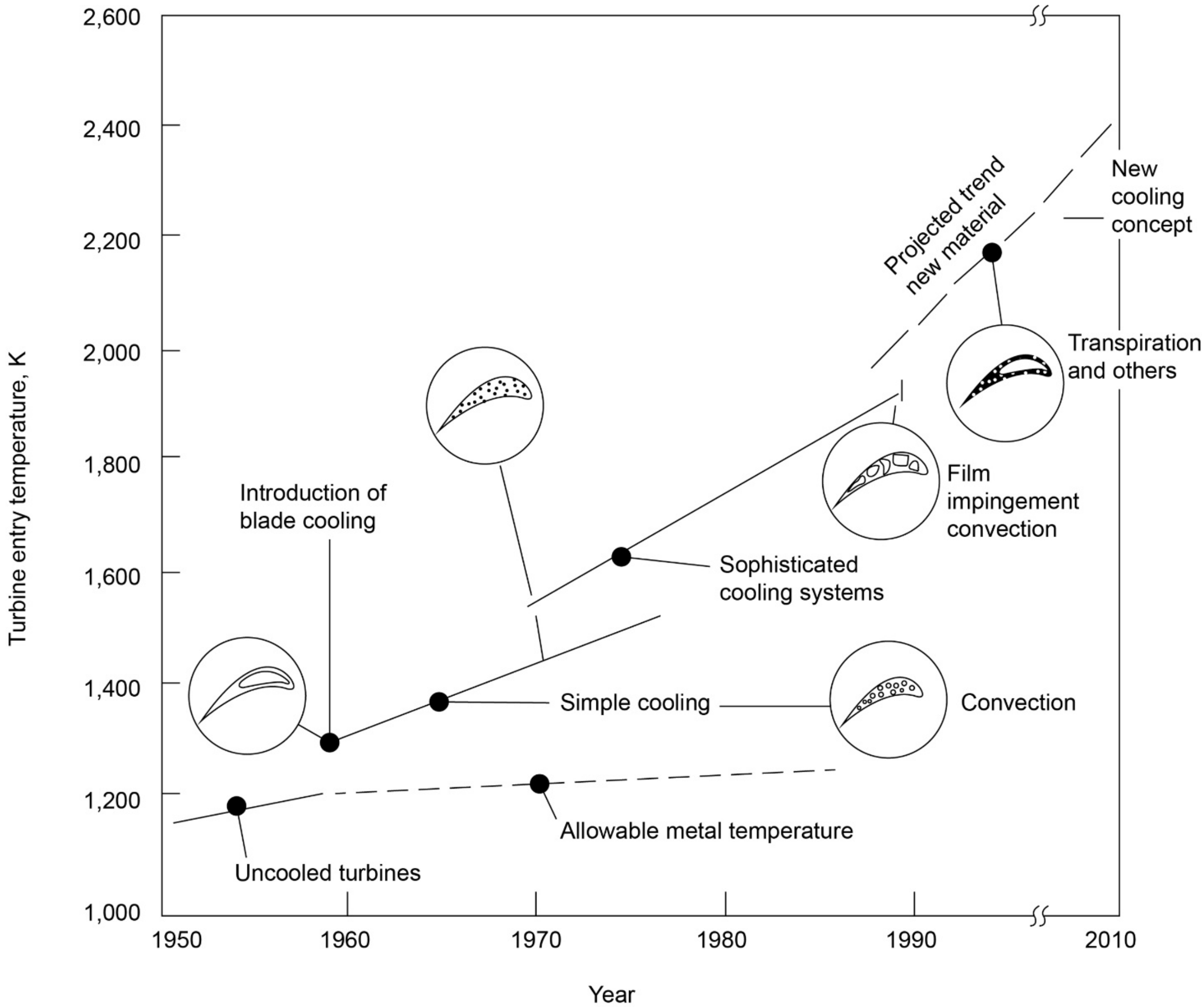

Figure 6.2.—Variation of turbine entry temperature over recent years (from Ref. 6.3; reproduced with permission from the North Atlantic Treaty Organization (NATO)).

to predict by CFD. This is due to a general inability of the codes to accurately model thermal mixing, especially in an inherently unsteady flow field. The problem is further complicated by the fact that the turbine durability is governed by the turbine material temperature field rather than the gas temperature, so the thermal conduction problem must also be considered. The turbine components in high-performance engines are generally coated with TBCs, so nonuniform material properties must also be considered.

It is imperative for improved future cooled-turbine designs and engine performance that improved methods and models be developed to not only accurately describe the complex heat transfer physics, but also to have reasonable turnaround time to impact the design cycle. Faster computers may help with some of this issue, but it will also be necessary to incorporate, for example, unsteady thermal field interactions into steady heat transfer codes so that the unsteady problem does not need to be solved for all cases. In addition, well-described benchmark heat transfer datasets are required to assess and improve the heat transfer models in the CFD codes. For example, the thermal mixing issue is not well understood in the turbine environment. This problem must be isolated in a CFD-quality dataset in order to understand the shortcomings of the existing steady CFD models.

6.2 Overview of Proposed Research

A comprehensive study is required accounting for the nonuniform temperature field exiting the combustor and first turbine vane, which results in an unsteady temperature field entering the first-stage HPT rotor. This study should include engine-relevant turbine vane and blade geometries as well as film and internal cooling in order to assess the impact of this imposed unsteady thermal field on the cooling effectiveness. Likewise, the thermal field exiting the combustor and first vane should be engine relevant in terms of spanwise and circumferential variation including the effects of hub and shroud leakage in order to adequately assess the spanwise thermal mixing. This thermal mixing process has a dramatic effect on endwall temperature and life. Conjugate heat transfer is also a consideration, both in terms of data reduction in a thermally conductive test article as well as in the assessment of the true effectiveness of the cooling design on turbine temperature reduction.

The proposed study should produce experimental data in a rotating environment and make use of the latest instrumentation to generate data that can be used by the turbine heat transfer community to assess CFD codes and design philosophies, and to understand the flow physics involved. It is anticipated that there will be an associated computational study conducted in concert with these experiments that can account for the unsteady flow inherent to blade rows in relative motion, the thermal field in the solid and fluid, and the complex geometries associated with cooled-turbine components.

6.3 Outcomes

The expected outcome of such a research effort would be a better understanding of the complex flow and mixing processes in a combustor-turbine flow field, which would lead to validation of both CFD tools and design tools used for cooled-turbine design. The dataset would be available to the heat transfer community to independently validate tools and cooling designs. Improved cooling designs could be compared to benchmark data associated with this geometry, and the industry as a whole would benefit.

References

6.1 Schulz, Uwe, et al.: Some Recent Trends in Research and Technology of Advanced Thermal Barrier Coatings. Aero. Sci. Techn., vol. 7, no. 1, 2003, pp. 73–80.

6.2 Cumpsty, Nicholas; and Heyes, Andrew: Jet Propulsion. Cambridge University Press, New York, NY, 2015.

6.3 Clifford, R.J.: Rotating Heat Transfer Investigations on a Multi-Pass Cooling Geometry. Presented at Conference on Heat Transfer and Cooling in Gas Turbines, AGARD CP–390, 1985.

7.0 Highly Loaded Low-Pressure Turbines[2]

7.1 Part I: Profile Design

7.1.1 Description of Problem

The low-pressure turbine (LPT) powers the fan and, in some cases, the booster stages. It is constrained to rotate at the same speed as the fan unless a gearbox is provided. However, a gearbox is not usually used in an aeroengine because of the weight penalty and cooling required for such a large structure. The outer diameter of the LPT is also constrained by the presence of the bypass duct and stress limitations on the materials used. The combination of a low rotational speed and limited diameter means that LPTs usually operate at blade-relative exit-flow Mach numbers in the range 0.6 to 0.9. Transonic LPTs are rarely seen in aeroengines. The work output for a given stage is limited because the blade speeds and flow Mach numbers are relatively low. Because the majority of the thrust is produced by the fan in high-bypass-ratio engines, there are many stages in the LPT. This means that the LPT is heavy, accounting for about 20 to 30 percent of the engine's weight, and is expensive to manufacture. The engine weight and cost can be reduced if each airfoil can be made to carry a greater aerodynamic load, thereby reducing the number of airfoils. The efficiency of the LPT strongly influences the SFC of an engine, where a 1-percent increase in LP polytropic efficiency improves the fuel consumption by 0.5 to 1 percent. With efficiency levels already much greater than 90 percent, there would appear to be little scope for improving this aspect of performance without a step change in technology.

A typical LPT has blades with large aspect ratios, which are usually in the range of 3:1 to 7:1. Because of this, the profile loss is by far the largest single contributor to the total loss of efficiency in these blades. Because the blades have relatively thin trailing edges, the magnitude of the profile loss depends mainly on the development of the airfoil boundary layers, particularly those on the suction surfaces. The Reynolds number (Re) of an LPT blade ranges from about 0.5×10^5 in the final stage at high altitude in small business jet applications to about 5×10^5 at sea-level takeoff in the first stage of the largest turbofans. Between takeoff and cruise altitude, the Re might fall by a factor of between 3 and 4. Given these Re values, boundary-layer transition and separation play important roles in determining engine performance at different operating conditions.

Low Re values mean that an inability to accurately predict how the boundary layers undergo transition from laminar to turbulent flow limits the degree of certainty associated with a given design. For example, increasing the lift coefficient or decreasing the Re is likely to cause the growth of laminar separation bubbles on the rear part of the suction surface. This is one of the main sources of the suction-side loss. In the past decade, by using incoming wakes from upstream blade rows to periodically promote transition in the suction-side boundary layer, it is possible to increase the lift on each blade and therefore reduce the number of blades and weight of the engine without reducing the efficiency. In some cases, it has actually been possible to raise the efficiency.

The introduction of unsteady wakes into an LPT with separation bubbles has three effects: (1) the wake interacts with the separation bubble to produce a high-loss turbulent flow associated with a rolling up and breakdown of the shear layer (a significant source of increased loss), (2) turbulence-induced attached flow follows this vortex, and (3) the so-called calmed period that follows. During this time, the boundary layer has not yet reseparated from the surface, and the losses remain at typically attached laminar levels. This is a truly unsteady effect. The effect on the overall loss is governed by the

[2]Lead authors: Howard Hodson, University of Cambridge, and Om Sharma, United Technologies Research Center.

balance between the reduction in the bubble-generated losses in the calmed period and the high-loss turbulent regions generated before this.

Below a Re of 100,000, the profile losses of airfoils tend to follow a laminar trend (loss scales with $\mathrm{Re}_C^{-0.5}$, where Re_C is Reynolds number based on axial chord and exit velocity). This indicates that in the low-Re regime, the large suction-side separation bubble dominates the overall loss production. Thus, the introduction of unsteady wakes beneficially suppresses the separation, causing a reduction in the boundary layer loss. Increasing the reduced frequency tends to reduce the loss further. At Re values above 200,000, they are more likely to follow a turbulent trend (loss scales with $\mathrm{Re}_C^{-0.2}$). This indicates that at high Re values, the bubble is small or nonexistent and the loss generated in the turbulent boundary layer downstream of reattachment dominates.

For a given amount of deceleration over the rear part of the suction surface (usually quantified by a diffusion factor), moving the peak velocity forward on the surface reduces the rate of deceleration. This tends to reduce the bubble-generated losses, at the cost of increasing the extent of the turbulent boundary layer. Front-loading therefore improves the low-Re performance, where bubble-generated losses are more significant, at the expense of the high-Re performance, where turbulence-generated losses are more dominant. Conversely, moving the peak velocity further aft has the opposite impact, reducing loss at high Re values at the expense of low-Re performance. This also helps in reducing the development of secondary flows and improving the tolerance to incidence, which is useful because increasing the lift tends to have the opposite effect.

The LPT of the NASA Subsonic Fixed Wing (SFW) Project is likely to have Re values in the range 70,000 to 200,000 at cruise conditions. Understanding and, crucially, being able to predict the unsteady separated and transitional suction-side boundary layer is essential in developing airfoils with increased lift that retain the already high levels of efficiency. Unfortunately, increasing the lift beyond today's levels represents an even greater challenge, especially as a reducing core size means that the Re is also reducing. There is also evidence that without a step change in technology, the lift cannot be increased without a reduction in efficiency. Recently, boundary-layer flow control has shown some promise in this direction, particularly when it is used in conjunction with the incoming wakes.

Today, the majority of advanced LPTs are still designed using two-dimensional (2D) steady codes. Yet, it is well known that unsteady flow is necessary for the efficient operation of many LPTs. Indeed, the MISES code of Drela and Giles seems to be almost universally employed. MISES is a 2D solver that couples an integral boundary-layer solver to an inviscid free stream. The correlations used for modeling transition onset in attached and separated flows are often adapted by its users. Though these correlations are simple, they are the mainstay of many transition predictions, and their exact details are closely guarded. In some cases, these have been adapted to allow some consideration of the effects of the unsteady flow. It should also be noted that these correlations are only applicable to 2D, often incompressible, boundary layers.

There are many reasons why 2D steady codes are still the mainstay of LPT design. One reason is that no practical alternative currently exists. Many of the better codes rely on using the integral parameters to define the onset location via correlations and then employ a low-Re turbulence model to predict the growth of the transitional flow. But even 2D URANS (unsteady Reynolds-averaged Navier-Stokes) codes are currently unable to predict the entire range of possible transition mechanisms in the LPT with sufficient fidelity, even for a single stage. However, most implementations are technically flawed because they do not acknowledge the fundamental rules governing the growth of unsteady transitional flow. Prediction of multimode transition (i.e., transition which at certain times occurs in a separated shear layer and at other times in attached flow), prediction of the speed at which the transitional flow regions grow and move along the surface, as well as prediction of the size and duration of the calmed period remain a particular difficulty. There are no high-fidelity 2D multistage transitional calculations.

7.1.2 Overview of Proposed Research

The efficiency, cost, and weight of the LPT are critical to the success of the NASA SFW Project. The proposed research should focus on three areas. First, simultaneous data are required concerning the disturbance environment and boundary-layer behavior in multistage machines. Second, methods must be developed that can properly and efficiently model this flow. Third, methods of supplementary flow control are required.

Though a handful of measurements have been made in LPT test rigs, these data are very sparse and inadequate, and the geometry is unavailable to the research community.

7.1.3 Research Activities

Proposed research will include the following areas.

7.1.3.1 Profile Development

A large body of research already exists studying the effects of wakes on suction-side boundary layers of LPT airfoils in the 2D cascade (or single-stage) environment. These data have been successfully used to improve LPT designs in the past. This approach should continue, using low-speed as well as high-speed (transonic) facilities to develop and demonstrate new approaches to designs. These cascades must be fitted with upstream wake generators to simulate the primary source of unsteadiness in the LPT.

In the first instance, cascade facilities should be developed or upgraded to include wake-generating mechanisms. In the case of transonic facilities, this will require considerable effort. The first studies should focus on determining the style of velocity and pressure distributions appropriate to profiles with an increased lift. Since this is likely to lead to a forward loading of the airfoil, the effects of incidence must also be studied in detail. This is a significant omission from many studies. New profiles can then be prepared based on the outcome of this research. The effects of high peak Mach numbers also require special consideration as these may exceed unity. The initial part of this study should take no more than 2 years.

The most successful studies of this type have taken place in universities, and this is likely to continue to be the case.

7.1.3.2 Multistage Test Vehicle

Although there is much information available from cascades, there is almost no information on the behavior of the suction-side boundary layers in multistage LPTs, apart from a small number of surface shear stress measurements. These have shown that although the first few blade rows behave in a manner that is similar to those in cascades fitted with upstream wake generators, the rising level of unsteadiness through succeeding blade rows creates a confusing picture of the behavior of the boundary layers. The next generation of experimental research must include a detailed study of the multistage environment, including measurements of both the boundary layers and the disturbance environment. Without this basic information, it will be impossible to validate the necessary method developments.

An LPT test vehicle should be developed to enable the measurement of the disturbance environment and the development of the blade surface boundary layers. The same experimental test vehicle can also be used to validate new LPT designs and to assess novel methods of flow control. A program of this type is expensive and time consuming, requiring at least 3 if not 5 years to bring to completion. Industry should be encouraged to provide the test vehicle, but universities and other research organizations should be enlisted to support the development of the advanced instrumentation that will be required.

7.1.3.3 Transition and Flow Control Devices

Flow control has not yet been used in LPTs, although some data exist from cascade studies that suggest that low-Re operation can be improved using simple passive systems such as surface roughness or trips that enhance the transition process between the wakes. In addition, active techniques, such as vortex generator jets, could be advantageous at very high lift coefficients and/or very low Re values. A two-part program of work is envisaged, which is aimed at exploiting flow control in very high lift situations. A combination of experimentation and large Eddy simulation (LES) and direct numerical simulation (DNS) calculations is envisaged:

(1) In the first part, fundamental studies of the effects of passive and active systems should be studied in depth to provide information on how these systems work and for the development of suitable prediction methods. These studies must include an accurate assessment of the impact of the flow control on the SFC.

(2) In the second part, passive and active systems should be developed that can be applied in the LPT in practice. It should be sufficient, on the first instance, to demonstrate these technologies using a representative 2D cascade that is fitted with upstream wake generators. Should this prove practical and worthwhile, the system can then be demonstrated in the multistage LPT described above.

Universities and other research organizations are best suited to the first part. The second part requires a close collaboration between these groups and industry. This program is expected to last for up to 10 years.

7.1.3.4 Model Development

Calculating transition in 3D unsteady flows presents a severe challenge today. Very little data and even fewer suitable models exist for relevant 3D flows. Compromised methods do exist for 3D URANS equations, and these are used, but with limited success. LES and DNS do offer a greater possibility of success, but not within the next 5 years. These methods can, however, be used to enhance experimental databases. No methods exist that have been successfully applied to the calculation of LPT boundary layers when using supplementary methods of control.

The prediction of the transitional behavior of boundary layers in the unsteady multistage 3D environment cannot continue to be based on correlations developed originally for integral boundary-layer solvers. Nor can it ultimately rely on 2D methods: to do so denies the three dimensionality of the unsteady flow in the LPT. Nevertheless, 2D methods will continue to be required for preliminary design. Therefore, the development of an entire hierarchy of computational methods is required, which deliver consistent results of increasing fidelity as the complexity of the model increases. These methods must acknowledge the peculiar aspects of unsteady multimode transition, the importance of the disturbance environment (which may include transition control devices), and be capable of use in 3D simulations. Most existing RANS methods are not capable of the required extension. It is expected that the methods will range from 2D URANS through 3D LES with zonal DNS. The timeframe for the development of such a complete set of tools would be in excess of 5 years. Universities would be best suited to this development role.

The development of a high-fidelity 2D (with streamtube height variation) computational method based on URANS/LES is of the highest priority.

7.1.4 Outcomes

Immediate results include a determination of whether or not it is likely to be able to increase the lift, whether or not control devices are required to achieve this, a description of the 3D unsteady flow field in the LPT, high-fidelity 2D URANS methods, and a roadmap for model development. Long-term results include a demonstration of very high lift technology; successful flow control devices, if viable; methods for use in design; and the development of a hierarchy of transition models for use in LPTs that includes the effects of periodic unsteadiness, turbulence, roughness, and other flow control devices.

A 1-percent reduction in direct operating costs is approximately equivalent to a 1-percent increase in component efficiency, an 8-percent reduction in engine cost, and a 17-percent reduction in engine weight.

The greatest risk is that it will not be possible to develop an LPT with increased lift and a substantially unchanged efficiency even when using flow control.

7.2 Part II: 3D Design

7.2.1 Description of the Problem

The design of modern LPTs is a compromise between efficiency, cost, and weight. Increasing the 2D profile loading of LPTs reduces the cost and weight, but risks compromising the efficiency of each blade section. This is because each blade carries a greater aerodynamic load, and this tends to increase the net aerodynamic loss generated within the blade surface boundary layers, even though there are fewer blades. This aspect of LPT design has received much attention in recent years, and considerable progress has already been made. However, increasing the blade loading also increases the so-called cross-passage pressure gradient as the difference between the suction and pressure side pressures increases. This gradient leads to the development of secondary or 3D flows close to the hubs and tips of the LPT blades.

Several features, including the overturning of the endwall boundary layers and the formation of the passage vortex, identify the secondary flows. The cross-passage pressure gradient that turns the mainstream flow also affects the endwall boundary layer, which contains slower moving fluid and is therefore overturned. In addition to the cross-passage pressure gradient being increased when the profile loading is increased, local streamwise pressure gradients are also increased. As a result, local flow reversal can occur, and this serves to increase the magnitude of the secondary flows and associated losses still further. This problem is exacerbated in low-Re flows because laminar flows, which are then more prevalent, will withstand only small adverse pressure gradients before separating. When solid, thin—as opposed to hollow—blade sections are used to reduce the cost and weight of the LPT, increasing the loading can lead to the formation of a very long (typically 30 to 50 percent of the chord length) separation bubble on the pressure side. The flow within the separation bubble can interact with the secondary flows on the endwalls and lead to a further increase in the strength of the secondary flows and losses. This is a result of the spanwise (radial) pressure gradients.

There is an opportunity to alter the spanwise loading, reaction or vortex design in LPTs. As a consequence, a more optimal balance between the profile and secondary-flow losses may be obtained. This is suggested because the secondary-flow losses are traditionally thought to deteriorate less as the core size, and therefore the Re, is reduced. This may be because, as has recently been shown, the boundary layers on the endwalls of multistage LPTs can begin as laminar and progressively become more turbulent as they develop. This transitional behavior has been observed in the case of the boundary layer upstream of the leading edge and in the new boundary layer that develops inside the blade passage as a result of the cross-passage pressure gradient. Given that some HPT cascade studies have revealed transitional boundary layers at much higher Re values, this behavior in LPTs is likely to be common. Furthermore, it has been shown that the state of the incoming endwall boundary layer has a significant effect on the development of the secondary flows, and it is likely that the effectiveness of 3D designs will also be dependent on this state.

The secondary flows and losses in LPTs have many similarities to those in HPTs. As a result, LPTs can be controlled using features such as endwall profiling, lean, and sweep, which are beginning to be found in HPTs. However, additional care must be taken when using these geometrical features to control the near-endwall static pressure field and therefore, the secondary flows. This is because the endwall flow can change dramatically when steps and leakage flows are introduced into the annulus line. This is partly related to the transitional nature of the flows, as the laminar flows are less robust. These steps and leakage flows occur because stationary and rotating parts are encountered in the turbine, and this implies the existence of gaps. At the hub, hot gas-path flows can enter the disc cavities, causing thermal fatigue. To prevent the ingestion of hot gas, relatively cold air from the compressors is directed to the turbine discs. The effect of this air is to cool the disc cavities and to avoid the ingestion of hot gas due to the pressure difference between the cooling air and the mainstream. At the casing, flow leaks over the shroud of the turbine rotor. Exactly how this leakage of cooling air is introduced into the main gas path leads to the aforementioned steps.

7.2.2 Overview of Proposed Research

The effect of features such as steps, leakage flows, and endwall profiling can be predicted with some success. However, computations of the flow associated with the real LPT geometry are rare, and the transitional nature of the endwall flow has yet to be addressed in experimental or computational studies. Furthermore, in an engine the general pattern of endwall flows will be affected by the unsteadiness associated with the vortices and wakes from upstream blade rows and the potential fields of upstream and downstream blade rows. The presence of tip leakage and hub leakage flows will also influence the endwall flow structures, possibly through several stages. In addition, potential interactions and the presence of radial pressure gradients will modify the structures of the endwall flows and their interactions with the mainstream flow. Whether a flow is laminar, transitional, or turbulent, and whether it is separated or attached, will affect the success of particular design choices.

7.2.2.1 Endwall Flow Studies in Cascades

A limited number of studies of the 3D endwall flows and losses already exist for conventional and high-lift LPTs. This work requires extension to higher lift and needs to cover a wide range of Re values. Fundamental studies, using CFD and experiments, of the individual and combined effects of the style of velocity and pressure distributions appropriate to profiles with an increased lift, annulus line changes, endwall profiling, shroud and hub leakage paths, blade lean, and blade sweep are required. Much of the experimental work can probably be carried out in low-speed cascades, with the extension to high-speed flows being performed using CFD calculations. The effects of high peak Mach numbers require consideration as these may exceed unity.

The most successful studies of this type have taken place in universities, and this is likely to continue to be the case. A timeframe of 3 to 5 years is envisaged.

7.2.2.2 Multistage Environment

A multistage test vehicle has been proposed for the research related to high-lift profile design. It is proposed to use this same vehicle for an in-depth analysis of the 3D flows within the engine environment. Steady as well as unsteady fast response measurements are required. Prior to this study, 3D design of LPTs should be examined computationally. This study should extend to the optimization of the leakage paths, the endwall profiles as well as fundamental studies of blade loading, reaction, and vortex design. A limited number of unsteady analyses should also be performed.

A program of this type is expensive and time consuming, requiring at least 3 if not 5 years to bring to completion. Industry should be encouraged to provide the test vehicle and to perform the design studies but universities and other research organizations should be enlisted to support the development of the advanced instrumentation and computational methods that will be required.

7.2.2.3 Transition, Separation, and Reattachment

The 3D endwall flows in actual LPTs have recently found to be transitional. Separation and reattachment are also features of endwall flows. Unfortunately, there is very little data available concerning transition in highly skewed boundary layers, which are subject to severe favorable and adverse pressure gradients, such as those found on the endwalls of the LPT. A three-part program of work is envisaged that is aimed at improving this situation.

In the first part, fundamental studies of 3D skewed boundary layers in controlled pressure gradients is required. This will assist in the physical understanding of the transition mechanisms and quantify the relative importance of the pressure gradients, skew, and free-stream disturbances.

In the second part, cascade models would be used to verify the findings of the fundamental studies and to provide validation data for CFD codes.

In the third part, models are required to be developed that are capable of predicting these flows. LES and DNS should be used to enhance the experimental database. The required methods must acknowledge the peculiar aspects of the unsteady 3D endwall boundary layers, including the leakage flows themselves. The timeframe for the development of such model is likely to be in excess of 5 years. Universities and other research organizations are best suited to this research.

7.2.3 Outcomes

Immediate results include a determination as to whether or not the secondary-flow losses will limit or enhance the ability to increase the lift of the LPT and whether or not endwall flow control will be successful. Long-term results include a demonstration of very high lift technology; successful flow control techniques, if viable; and methods for use in the design and development of transition models for use in 3D endwall flows such as those found in LPTs.

Bibliography

Part I

Sharma, O.P., et al.: Boundary Layer Development on Turbine Airfoil Suction Surfaces. J. Eng. Power, vol. 104, no. 3, 2009, pp. 698–706.

Coull, John D.; Thomas, Ricard L.; and Hodson, Howard P.: Velocity Distributions for Low Pressure Turbines. ASME GT2008–50589, 2008.

Prakash, C., et al.: Effect of Loading Level and Distribution on LPT Losses. ASME GT2008–50052, 2008.

Opoka, Maciej M.; and Hodson, Howard P.: Transition on the T106 LP Turbine Blade in the Presence of Moving Upstream Wakes and Downstream Potential Fields. ASME GT2007–28077, 2007.

Vera, Maria, et al.: Endwall Boundary Layer Development in an Engine Representative Four-Stage Low Pressure Turbine Rig. ASME GT2007–27842, 2007.

Bons, Jeffrey: Unsteady Transition and Separation in an LPT Cascade. Minnowbrook V 2006 Workshop on Unsteady Flows in Turbomachinery, NASA/CP—2006-214484, 2006, pp. 43–44. http://ntrs.nasa.gov

Clarke, J.P.; and Koch, P.J.: Designing Turbine Airfoils to Answer Research Questions in Unsteady Aerodynamics. Minnowbrook V 2006 Workshop on Unsteady Flows in Turbomachinery, NASA/CP—2006-214484, 2006, pp. 65–67. http://ntrs.nasa.gov

Hourmouziadis, J.; and Hofmann, G.: Response of Separation Bubble to Velocity and Turbulence Wakes. Minnowbrook V 2006 Workshop on Unsteady Flows in Turbomachinery, NASA/CP—2006-214484, 2006, pp. 41–42. http://ntrs.nasa.gov

Vera, Maria, et al.: Separation and Transition Control on an Aft-Loaded Ultra-High-Lift LP Turbine Blade at Low Reynolds Numbers: High-Speed Validation. ASME GT2005–68893, 2005.

Zhang, Xue Feng, et al.: Separation and Transition Control on an Aft-Loaded Ultra-High-Lift LP Turbine Blade at Low Reynolds Numbers: Low-Speed Investigation. ASME GT2005–68892, 2005.

Stieger, R.D.; and Hodson, H.P.: The Transition Mechanism of Highly-Loaded LP Turbine Blades. J. Turbomach., vol. 126, no. 4, 2004, pp. 536–543.

Michelassi, V.; Wissink, J.; and Rodi, W.: DNS, LES, and URANS of Periodic Unsteady Flow in a LP Turbine Cascade: A Comparison. Proceedings of the 5th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, Prague, 2003, pp. 1185–1195.

Haselbach, Frank, et al.: The Application of Ultra High Lift Blading in the BR715 LP Turbine. J. Turbomach., vol. 124, 2002, pp. 45–51.

Howell, R.J., et al.: Boundary Layer Development in the BR710 and BR715 LP Turbines—The Implementation of High-Lift and Ultra-High-Lift Concepts. J. Turbomach., vol. 124, 2002, pp. 385–392.

Howell, R.J., et al.: High Lift and Aft-Loaded Pressure Profiles for Low-Pressure Turbines. J. Turbomach., vol. 123, 2001, pp. 181–188.

Harvey, N.W., et al.: The Role of Research in the Aerodynamic Design of an Advanced Low-Pressure Turbine. Presented at the Third European Conference on Turbomachinery (Proc. Instn. Mech. Engrs., vol. 213, part A, 1999, p. 235), London, 1999.

Wisler, D.C.: The Technical and Economic Relevance of Understanding Blade Row Interactions Effects in Turbomachinery. von Karman Institute for Fluid Dynamics Lecture Series No. 1998–02, 1998.

Curtis, E.M., et al.: Development of Blade Profiles for Low-Pressure Turbine Applications. J. Turbomach., vol. 119, no. 3, 1997, pp. 531–538.

Halstead, D.E., et al.: Boundary Layer Development in Axial Compressors and Turbines: Part 1 of 4—Composite Picture. J. Turbomach., vol. 119, no. 1, 1997, pp. 114–127.

Halstead, D.E., et al.: Boundary Layer Development in Axial Compressors and Turbines: Part 3 of 4—LP Turbines. J. Turbomach., vol. 119, no. 2, 1997, pp. 225–237.

Mayle, Robert Edward: The Role of Laminar-Turbulent Transition in Gas Turbine Engines. ASME 91–GT–261, 1991.

Hourmouziadis, J.: Aerodynamic Design of Low Pressure Turbines. AGARD Lecture Series No. 167, 1989.

Part II

Vera, Maria, et al.: Endwall Boundary Layer Development in an Engine Representative Four-Stage Low Pressure Turbine Rig. ASME GT2007–27842, 2007.

de la Rosa Blanco, E.; Hodson, H.P.; and Vazquez, R.: Effect of the Leakage Flows and the Upstream Platform Geometry on the Endwall Flows of a Turbine Cascade. ASME GT2006–90767, 2006.

Torre, Diego, et al.: A New Alternative for Reduction of Secondary Flows in Low Pressure Turbines. ASME GT2006–91002, 2006.

Cherry David, et al.: Analytical Investigation of a Low Pressure Turbine With and Without Flowpath Endwall Gaps, Seals and Clearance Features. ASME GT2005–68492, 2005.

de la Rosa Blanco, Elena; Hodson, H.P.; and Vazquez, R.: Effect of Upstream Platform Geometry on the Endwall Flows of a Turbine Cascade. ASME GT2005–68938, 2005.

Bohn, Dieter E., et al.: Influence of Open and Closed Shroud Cavities on the Flowfield in a 2-Stage Turbine With Shrouded Blades. ASME GT2003–38436, 2003.

Gier, Jochen, et al.: Interaction of Shroud Leakage Flow and the Main Flow in a Three-Stage LP Turbine. ASME GT2003–38025, 2003.

Schlienger, J., et al.: Effects of Labyrinth Seal Variation on Multistage Axial Turbine Flow. ASME GT2003–38270, 2003.

Zess, G.A.; and Thole, K.A.: Computational Design and Experimental Evaluation of Using a Leading Edge Fillet on a Gas Turbine Vane. J. Turbomach., vol. 124, 2002, pp. 167–175.

Langston, L.S.: Secondary Flows in Axial Turbines—A Review. Ann. NY Acad. Sci., vol. 934, 2001, pp. 11–26.

McLean, Christopher; Camci, Cengiz; and Glezer, Boris: Mainstream Aerodynamic Effects Due To Wheelspace Coolant Injection in a High-Pressure Turbine Stage: Part II—Aerodynamic Measurements in the Rotational Frame. J. Turbomach., vol. 123, 2001, pp. 697–703.

Harvey, Neil W., et al.: Nonaxisymmetric Turbine End Wall Design: Part I—Three-Dimensional Linear Design System. J. Turbomach., vol. 122, 2000, pp. 278–285.

Hunter, Scott D.; and Manwaring, Steven R.: Endwall Cavity Flow Effects on Gaspath Aerodynamics in an Axial Flow Turbine: Part I—Experimental and Numerical Investigation. ASME 2000–GT–0651, 2000.

Sauer, H.; Muller, R.; and Vogeler, K.: Reduction of Secondary Flow Losses in Turbine Cascades by Leading Edge Modifications at the Endwall. ASME 2000–GT–0473, 2000.

Sieverding, C.H.: Recent Progress in the Understanding of Basic Aspects of Secondary Flows in a Turbine Blade Cascade. J. Eng. Gas Turbines Power, vol. 107, no. 2, 1985, pp. 248–257.

Weiss, A.P.; and Fottner, L.: The Influence of Load Distribution on Secondary Flow in Straight Turbine Cascades. J. Turbomach., vol. 117, no. 1, 1995, pp. 133–141.

8.0 Concluding Remarks

This report objective is to archive the “white papers” compiled in 2008 by the Aerothermodynamics Technical Working Group (TWG) in the area of turbomachinery technology to make them widely available. Each section of the report includes an article that addresses one of the high-payoff areas selected by the TWG. The articles provide an assessment of the state of the art, describes the barriers, and provides recommendations for future work. The recommendations are in the form of detailed research objectives that can be converted into research plans. The expectation is that industry, academia, and Government laboratories will be guided by the recommendations in their research and development effort in subsequent years. It is hoped that at a future date, the TWG or a similar group will review and evaluate the progress and impact on advances in turbomachinery technology.