TECHNICAL QUARTERLY PROGRESS REPORT

Prepared By

*Advanced Gas Turbine Systems Research*
Clemson University Research Foundation

For

U. S. Department of Energy
National Energy Technology Laboratory

Contract No. DE-FC21-92MC29061

Reporting Period: January 1, 2002—March 31, 2002

*Advanced Gas Turbine Systems Research*
South Carolina Institute for Energy Studies
Clemson, South Carolina
SUMMARY

The activities of the Advanced Gas Turbine Systems Research (AGTSR) program for this reporting period are described in this quarterly report. The report is divided into discussions of Membership, Administration, Technology Transfer (Workshop/Education), Research and Miscellaneous Related Activity. Items worthy of note are presented in extended bullet format following the appropriate heading.

MEMBERSHIP

- At the close of the reporting period, the AGTSR Performing Membership had grown to 101 universities in 38 states. Colorado State University became a Performing Member in February 2002. No additional inquiries were received this reporting period.
- Invoices have been submitted to all IRB members for CY2002 membership. Membership dues have been received (as indicated with a *) from the following firms:
  
<table>
<thead>
<tr>
<th>Full Member</th>
<th>Associate Member</th>
</tr>
</thead>
<tbody>
<tr>
<td>General Electric</td>
<td>*Parker Hannifin</td>
</tr>
<tr>
<td>Pratt &amp; Whitney</td>
<td>*Ramgen</td>
</tr>
<tr>
<td>*Rolls Royce</td>
<td>*Southern Company Services</td>
</tr>
<tr>
<td>*Siemens Westinghouse</td>
<td>Woodward FST</td>
</tr>
<tr>
<td>Solar Turbines</td>
<td>EPRI</td>
</tr>
</tbody>
</table>

Follow up with the firms not responding to the initial invoice will continue in the following reporting period.
- EPRI provided payment for their 2001 IRB dues in February 2002. With receipt of EPRI’s dues, all IRB firms have submitted their payment for 2001.
- Due to recent retirements and company restructuring, the AGTSR Points of Contacts for member firms has changed. The latest listing is shown in Attachment I. The most notable retirements were Dr.’s Day and Ali from P&W and RRA. Both Dr. Day and Ali had been members of the IRB since inception. Dr. Day had served as Chair of the Board. EPRI and Woodward Focal Points have also changed.
• A new chairman for the IRB will be elected at the next IRB meeting.
• No other membership activities occurred during this reporting period.

ADMINISTRATION

• The AGTSR Quarterly Report for the reporting period, October to December 2001, was submitted to NETL on January 31, 2002. The report was submitted via hard copy and disk.
• The monthly status reports for December 2001, January 2002 and February 2002 were submitted as requested to the NETL COR. The status report for March is now in preparation.
• SCIES submitted an application to the NETL-DOE for an award for a new university research initiative—AGTSR/HEET. The requested federal budget for the first budget period was $3.119 million. The NETL requested a revision to the initial application including a budget reduction to $3.0 million. The revised application was submitted on February 22, 2002. SCIES was notified on March 20, 2002, that the revised application had been accepted by NETL. The new assistance number is DE-FG26-02NT41431. The NETL award document is now being reviewed by the Clemson University Research Foundation. In this application SCIES committed a cost share of $150,000 from Industrial Review Board Membership fees.
• SCIES provided input to DOE turbine technology planning by participating in the HEET Roadmapping Workshop in Reston, Virginia, on February 7-8 and the HEET Roadmapping Recommendations and Conclusions Meeting which immediately followed the DOE Turbine Power Systems Conference and Condition Monitoring Workshop in Galveston, Texas, on February 25-27. SCIES also participated in the Materials R&D Roadmap Meeting in the morning before the Conference. A presentation, “AGTSR Transitions from ATS to HEET” was prepared and delivered at the Turbine Power Systems Conference and university representatives for seventeen AGTSR projects participated in the conference poster session.
• Abbie Layne, NETL, has been active in broadening DOE interactions with European organizations and university research is one area of collaboration that is being pursued. Under the direction of the DOE, SCIES has been interacting with a consulting organization, Global Tech Inc (GTI), to facilitate collaboration with the European Union (EU). One of the possible initial activities that are being explored further is coordination of the next AGTSR RFP with the next EU university RFP. The list of candidate research topics for the next university RFP was sent to GTI to be forwarded to the EU to facilitate possible RFP coordination.
• At DOE request (Kate Lessing, NETL), writeups were provided on five AGTSR university projects for the congressional report on the HEET Program.

• The DOE has requested Fact Sheets for each active university project. These Fact Sheets describe the project goals, activities, and benefits and provide contract information. Fact Sheets for eight university projects were developed and sent to the DOE in December. Texts for Fact Sheets on all AGTSR projects have now been provided, but adequate figures were not available for all projects. Figures have been requested from principal investigators and will be provided to the DOE after receipt.

• NETL has approved SCIES request for a no increase in cost time extension for cooperative agreement DE-FC21-92MC29061. The extension period was approved to June 30, 2003.

• Normal administrative functions continued throughout the reporting period.

TECHNOLOGY TRANSFER (Workshops and Education)

• Proceedings of Materials Workshop II co-hosted with Professor Maurice Gell of the University of Connecticut were released in March. The release of the proceedings had been delayed due to an inability to get all speakers to submit their material for publication.

• Dates for the AGTSR Workshops for CY2002 have been finalized (see Attachment II). The co-hosts of these workshops are Combustion IX—Penn State University, Professor Santavicca Materials III—University of Connecticut, Professor Gell Aero-Heat Transfer Workshop IV—Louisiana State University, Professor Acharya

• AGTSR has received 16 applications for the summer Internship program. The student applications are now being reviewed by the IRB companies. Two of the applications were from students that do not have completed PRA status. SCIES is waiting clarification from DOE if these students may be included in the program or not. Once clarification is received, the interns will be placed.

• Attachment III contains a listing of the progress reports received and distributed during this reporting period. A total of 13 reports have been released.

• Principal investigators for all active AGTSR projects were requested to prepare posters for presentation on February 26 at the Turbine Power Systems Conference and Condition Monitoring Workshop in Galveston, sponsored by the DOE and other organizations.

• Additional normal contact was maintained with University and industry AGTSR members.
RESEARCH

- The university principal investigators of the ten short listed AGTSR 2001 proposals were contacted in October and asked to examine their budgets for possible reductions of 5 to 10% to potentially enable more awards.

To date, eight of the ten short listed proposals have decreased their budgets for a total reduction of $241,608. The principal investigator for one of the two remaining proposals recently indicated that he intends to also reduce his budget.

Once the SCIES continuation request has been approved by NETL and the Clemson University Research Foundation, contract negotiations will begin with the 10 short listed PI's/universities.

- An initial list of candidate research topics was completed for the year 2002 RFP. These topics were compiled from the most recent AGTSR RFP, presentations by technical representatives from the IRB companies at the most recent AGTSR workshops, and DOE input. Although some of the topics are being addressed in current AGTSR projects, they were included to determine if additional research is needed. Topics are in major areas of Combustion, Aero-Heat Transfer, Materials, and Reliability, Availability, and Maintainability (RAM). These topics were sent to the DOE for information purposes and then to the IRB for ranking. Recent DOE emphasis for the developing High Efficiency Engines and Turbine (HEET) program has been directed to coal-based turbine systems. Since coal-fueled turbine applications have not previously been a focus of the university research program, the list sent to the IRB had only limited topics related to those applications. Consequently, the IRB companies were asked to recommend additional university research topics related to coal-based turbine systems and rank them along with those that were provided. The topic rankings were requested from the IRB by February 20.

- **Attachment IV** contains the Success Stories released this reporting period.

MISCELLANEOUS

- At the request of Dr. N. Holcombe of NETL, SCIES is arranging a panel session at the Amsterdam IGTI meeting. The subject for the panel is Issues Facing Widespread Adoption of DG Systems. This panel is part of the Distributed Generation Task Force Effort.
As a pilot session, SCIES is organizing a Regional Workshop focused on the DOE HEET program. This pilot session will be held at Georgia Tech with the cooperation of the DOE Atlanta Research Office. More details will be provided next quarter.
### AGTSR Points of Contact (POC)

**Voting Members:**

**GE - Power Generation**

Focal Point: Kathryn Rominger  518-385-0753

<table>
<thead>
<tr>
<th>Sub Areas</th>
<th>POC</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Combustion (Instability)</td>
<td>*Tony Dean</td>
<td>518-387-6478</td>
</tr>
<tr>
<td></td>
<td>Hukam Mongia</td>
<td>513-243-2552 (GEAE)</td>
</tr>
<tr>
<td>Aero-Optimizaton</td>
<td>Phil Beauchamp</td>
<td>518-385-4933</td>
</tr>
<tr>
<td></td>
<td>Ramani Mani</td>
<td>518-387-6341</td>
</tr>
<tr>
<td></td>
<td>*Kevin Kirtley</td>
<td>518-387-5848</td>
</tr>
<tr>
<td></td>
<td>Phil Andrew</td>
<td>518-385-9328</td>
</tr>
<tr>
<td>Materials</td>
<td>Curt Johnson (TBC’s)</td>
<td>518-387-6421</td>
</tr>
<tr>
<td></td>
<td>Warren Nelson</td>
<td>518-385-3660</td>
</tr>
<tr>
<td></td>
<td>Mike Henry</td>
<td>518-387-5799</td>
</tr>
<tr>
<td></td>
<td>*Steve Balsone</td>
<td>518-387-4141</td>
</tr>
<tr>
<td></td>
<td>Bob Orenstein</td>
<td>518-387-4103</td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>*Ron Bunker</td>
<td>518-387-5086</td>
</tr>
<tr>
<td></td>
<td>Richard Kehl</td>
<td></td>
</tr>
<tr>
<td>GE - CRD Lab Research</td>
<td>Gene Kimura (mgr.)</td>
<td>518-387-6653</td>
</tr>
<tr>
<td></td>
<td>(oversees all R&amp;D)</td>
<td></td>
</tr>
<tr>
<td>Dynamics (Seals, Bearings)</td>
<td>Imdad Imam</td>
<td>518-387-5043</td>
</tr>
</tbody>
</table>

**Pratt & Whitney**

Focal Point: Richard Tuthill  860-610-7877

<table>
<thead>
<tr>
<th>Sub Areas</th>
<th>POC</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heat Transfer</td>
<td>*Mike Blair</td>
<td>860-557-4407</td>
</tr>
<tr>
<td>Materials</td>
<td>*Kevin Schlichting</td>
<td>860-557-2442</td>
</tr>
<tr>
<td>Coatings</td>
<td>Jeanine Marcin</td>
<td>860-565-4784</td>
</tr>
<tr>
<td></td>
<td>Mike Maloney</td>
<td>561-796-6178</td>
</tr>
<tr>
<td></td>
<td>Mladen Trubelja</td>
<td>860-565-0249</td>
</tr>
</tbody>
</table>
### Rolls Royce Corporation

**Focal Point:** Robert Delaney  
**Phone:** 317-230-4624

<table>
<thead>
<tr>
<th>Sub Areas</th>
<th>POC</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Combustion</td>
<td>Duane Smith</td>
<td>317-230-3683</td>
</tr>
<tr>
<td></td>
<td>*Mohan Razdan</td>
<td>317-230-6404</td>
</tr>
<tr>
<td></td>
<td>M. Anand</td>
<td>317-230-2828</td>
</tr>
<tr>
<td></td>
<td>Sunil James</td>
<td>317-230-6195</td>
</tr>
<tr>
<td></td>
<td>Kumar Abhilek</td>
<td>317-230-3174</td>
</tr>
<tr>
<td></td>
<td>André Marshall</td>
<td>317-230-5931</td>
</tr>
<tr>
<td>Aero-Optimization</td>
<td>*Bob Delaney</td>
<td>317-230-4624</td>
</tr>
<tr>
<td></td>
<td>Ed Hall</td>
<td>317-230-3722</td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>*John Weaver</td>
<td>317-230-5398</td>
</tr>
<tr>
<td>Materials</td>
<td>*Malcolm Thomas</td>
<td>317-230-4545</td>
</tr>
<tr>
<td></td>
<td>Ron Cloyd (single crystals)</td>
<td>317-230-6302</td>
</tr>
<tr>
<td></td>
<td>Subhash K. Naik (TBC’s)</td>
<td>317-230-5756</td>
</tr>
<tr>
<td></td>
<td>George Creech</td>
<td>317-230-2269</td>
</tr>
<tr>
<td></td>
<td>Ed Hodge</td>
<td>317-230-2111</td>
</tr>
<tr>
<td></td>
<td>Ray Sinatra</td>
<td>317-230-5152</td>
</tr>
<tr>
<td>Dynamics</td>
<td>Don Burns</td>
<td>317-230-2368</td>
</tr>
<tr>
<td></td>
<td>Kuk Frey</td>
<td>317-230-6524</td>
</tr>
<tr>
<td>Sensors &amp; Controls</td>
<td>Tom Bonsett</td>
<td>317-230-3448</td>
</tr>
<tr>
<td></td>
<td>Andy Brewington</td>
<td>317-230-2246</td>
</tr>
</tbody>
</table>

### Siemens-Westinghouse

**Focal Point:** *Ihor Diakunchak  
**Phone:** 407-736-5115

<table>
<thead>
<tr>
<th>Sub Areas</th>
<th>POC</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Combustion, Instability</td>
<td>Anil Gulati</td>
<td>407-736-2346</td>
</tr>
<tr>
<td>Sub Areas</td>
<td>POC</td>
<td>Phone</td>
</tr>
<tr>
<td>---------------------------</td>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>Sensors &amp; Controls</td>
<td>Tom Burke</td>
<td>619-544-2510</td>
</tr>
<tr>
<td>Heat Transfer &amp; Aero</td>
<td>*Hee Koo Moon</td>
<td>619-544-5226</td>
</tr>
<tr>
<td></td>
<td>Eli Razinsky</td>
<td>619-544-5198</td>
</tr>
<tr>
<td></td>
<td>Jerry Stringham</td>
<td>619-544-2854</td>
</tr>
<tr>
<td></td>
<td>Ulrich Stang</td>
<td>619-595-7573</td>
</tr>
<tr>
<td>Combustion/Alternate Fuels</td>
<td>*Ken Smith</td>
<td>619-544-5539</td>
</tr>
<tr>
<td></td>
<td>Mohan Sood</td>
<td>619-544-5508</td>
</tr>
<tr>
<td></td>
<td>Luke Cowell</td>
<td>619-544-5916</td>
</tr>
<tr>
<td>Materials</td>
<td>*Zaher Mutasim</td>
<td>619-544-5889</td>
</tr>
</tbody>
</table>

**Associate Members:**

**EPRI**

Focal Point:  *David Gandy  
704-547-6198
**Parker Hannifin Corporation**

Focal Point:  *Michael Benjamin  440-954-8105

<table>
<thead>
<tr>
<th>Sub Areas</th>
<th>POC</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Combustion</td>
<td>Adel Mansour</td>
<td>440-954-8171</td>
</tr>
<tr>
<td>Aero-Optimization</td>
<td>Erlendur Steinthorsson</td>
<td></td>
</tr>
<tr>
<td>Materials</td>
<td>Brad Hartley</td>
<td></td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>Adel Mansour</td>
<td>440-954-8171</td>
</tr>
</tbody>
</table>

**Ramgen Power Systems**

Focal Point:  Rob Steele  425-828-4919, x288

<table>
<thead>
<tr>
<th>Sub-Areas</th>
<th>POC</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Blake Chenevert</td>
<td>425-828-4919, x224</td>
</tr>
</tbody>
</table>

**Southern Company**

Focal Point:  *Charles Boohaker  205-257-7537

<table>
<thead>
<tr>
<th>Sub Areas</th>
<th>POC</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Combustion/Emissions</td>
<td>Charles Boohaker</td>
<td>205-257-7537</td>
</tr>
</tbody>
</table>

**Woodward Governor Company**

Focal Point:  *Kelly Benson  970-498-3921

<table>
<thead>
<tr>
<th>Sub-Areas</th>
<th>POC</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Kelly Benson</td>
<td>970-498-3921</td>
</tr>
</tbody>
</table>

**Advisors:**

**DOE**

<table>
<thead>
<tr>
<th>Name</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tom George</td>
<td>304-285-4825</td>
</tr>
<tr>
<td>Abbie Layne</td>
<td>304-285-4603</td>
</tr>
<tr>
<td>Bruce Utz</td>
<td>412-386-5706</td>
</tr>
</tbody>
</table>

**GRI**

<table>
<thead>
<tr>
<th>Name</th>
<th>Phone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Michael Romanco</td>
<td>773-399-5460</td>
</tr>
</tbody>
</table>

*Technical Contacts identified by Focal Points to send AGTSR Proposals*
ANNOUNCEMENT OF THE AGTSR WORKSHOPS FOR YEAR 2002

COMBUSTION WORKSHOP IX
August 26-28
The Atherton Hotel, State College, Pennsylvania
Hosted by Penn State University

MATERIALS WORKSHOP III
October 14-16
Nathan Hale Inn, Storrs, Connecticut
Hosted by University of Connecticut

AERO-HEAT TRANSFER WORKSHOP V
November 11-13
Cook Conference Center Hotel, Baton Rouge, Louisiana
Hosted by Louisiana State University

These workshops are co-sponsored by the U. S. Department of Energy and the South Carolina Institute for Energy Studies (SCIES) under the Advanced Gas Turbine Systems Research/High Efficiency Engines and Turbines (AGTSR/HEET) Program. The workshops will bring together experts from academia, industry, and government to discuss the results of ongoing AGTSR university research projects and identify needed gas turbine research to support the goals of U. S. Department of Energy programs and advance the technology of the U. S. gas turbine industry. Information packets with registration materials will be mailed out approximately ten weeks prior to the workshops. Please put these meetings on your calendar. For questions, please contact Donna Partain (donnak@clemson.edu). If you have not received an information packet within eight weeks prior to a workshop and you have an interest in attending, please contact our office for assistance.
<table>
<thead>
<tr>
<th><strong>Aero-Heat Transfer</strong></th>
<th><strong>Mississippi State University</strong></th>
<th><strong>“Real Surface Effects on Turbine Heat Transfer and Aerodynamic Performance”</strong>&lt;br&gt;<strong>PI – B.K. Hodge</strong>&lt;br&gt;<strong>Semi-Annual Report 3/1/01-9/1/01</strong>&lt;br&gt;<strong>Subcontract #99-01-SR076</strong>&lt;br&gt;<strong>LINK TO REPORT</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Aero-Heat Transfer</strong></td>
<td><strong>Texas Engineering Experiment Station</strong></td>
<td><strong>“Rotating and Stationary Rectangular Cooling Passage Heat Transfer and Friction with Ribs, Pins and Dimples”</strong>&lt;br&gt;<strong>PI – J.C. Han</strong></td>
</tr>
<tr>
<td><strong>Aero-Heat Transfer</strong></td>
<td><strong>Texas Engineering Experiment Station</strong></td>
<td><strong>“Rotating Heat Transfer in High Aspect Ratio Rectangular Cooling Passages with Shaped Turbulators”</strong>&lt;br&gt;<strong>PI - J.C. Han</strong></td>
</tr>
<tr>
<td><strong>Aero-Heat Transfer</strong></td>
<td><strong>Louisiana State University</strong></td>
<td><strong>“Internal Cooling in Leading- and Trailing Edge Passages with Rotation and Buoyancy”</strong>&lt;br&gt;<strong>PI - Sumanta Acharya</strong></td>
</tr>
<tr>
<td><strong>Aero-Heat Transfer</strong></td>
<td><strong>Purdue University</strong></td>
<td><strong>“Turbine Blade Tip, Endwall and Platform Heat Transfer Including Rotation Effects”</strong>&lt;br&gt;<strong>PI - Sanford Fleeter</strong></td>
</tr>
<tr>
<td><strong>Aero-Heat Transfer</strong></td>
<td><strong>University of Minnesota</strong></td>
<td><strong>“Edge Cooling Heat Transfer on Turbine Blades”</strong>&lt;br&gt;<strong>PI - R. J. Goldstein</strong></td>
</tr>
<tr>
<td><strong>Aero-Heat Transfer</strong></td>
<td><strong>University of Texas, Austin</strong></td>
<td><strong>“Attenuation of Hot Streaks and Interaction of Hot Streaks with the Nozzle Guide Vane and Endwall”</strong>&lt;br&gt;<strong>PI - Dave Bogard</strong></td>
</tr>
<tr>
<td><strong>Combustion</strong></td>
<td><strong>University of California, Irvine</strong></td>
<td><strong>“Mechanistic Study of Fuel State and Composition Effects in Natural-Gas Fired Gas Turbine Combustion”</strong>&lt;br&gt;<strong>PI - Scott Samuelsen</strong></td>
</tr>
<tr>
<td><strong>Combustion</strong></td>
<td><strong>Pennsylvania State University</strong></td>
<td><strong>“Optimization of the Injector Fuel Distribution for Stable, Low Emissions Combustion in Lean Premixed Gas Turbine Combustors”</strong>&lt;br&gt;<strong>PI - Dom Santavicca</strong></td>
</tr>
<tr>
<td><strong>Combustion</strong></td>
<td><strong>University of California, Berkeley</strong></td>
<td><strong>“Fuel-Air Mixing Explored With Optical Probes, Tomography, and Large Eddy Simulations”</strong>&lt;br&gt;<strong>PI - Bob Dibble</strong></td>
</tr>
<tr>
<td><strong>Materials</strong></td>
<td><strong>University of California, Santa Barbara</strong></td>
<td><strong>“A Science Based Approach to Enhanced Zirconia-Based Thermal Barrier Coatings For Advanced Gas Turbine Applications”</strong>&lt;br&gt;<strong>PI - David Clarke</strong></td>
</tr>
<tr>
<td><strong>Materials</strong></td>
<td><strong>Northwestern University</strong></td>
<td><strong>“SPPS for Advanced Thermal Barrier Coatings”</strong>&lt;br&gt;<strong>PI - Katherine Faber</strong></td>
</tr>
<tr>
<td><strong>Materials</strong></td>
<td><strong>University of Connecticut</strong></td>
<td><strong>“Thermal Barrier Coatings and Metallic Coatings with Improved Durability”</strong>&lt;br&gt;<strong>PI - Fred Pettit</strong></td>
</tr>
</tbody>
</table>
Quarterly Success Stories

**Advancement of a New Application Technique for Improved Thermal Barrier Coatings**

An important cause of failure for thermal barrier coatings (TBC) in turbines has been the growth of internal aluminum oxide scales within the TBS at the bond coat. This produces internal stresses and ultimate cracking because of the volumetric growth of the internal oxide layer. The detrimental growth of the alumina scales results from oxygen penetration through the porous outer layer of the TBS to the underlying bond coat.

Under the Advanced Gas Turbine Systems Research (AGTSR) program, Northwestern University (NU) has evaluated a new Small Particle Plasma Spray (SPPS) process to apply a thin YAG (yttrium aluminum garnet) layer on TBC bond coats to block oxygen penetration and thereby alleviate TBC failures. This process also offers the potential for applying graded layers of coating materials to facilitate engineered TBC coatings with other specialized properties. Research in this recently completed project showed that SPPS is a viable process for applying the YAG layer on a bond coat. Experiments on specimens of coated turbine materials indicated that a TBC with the YAG layer on the bond coat experienced internal oxidation rates nearly a factor of two lower than rates for specimens with conventional TBCs. The project has also shown the SPPS process can produce TBC coatings that experience lower fatigue damage (through graded porosity). Lower fatigue damage in addition to lower internal oxidation rates demonstrated in this project indicates that the SPPS process offers the potential for enabling longer TBC lifetime in turbines.

Two patents (5,744,777 and 5,858,470) were issued concerning the SPPS process during the course of this AGTSR project.

**University of Connecticut Determines Effects of Turbine Cycles on TBC Life**

There is no accurate measurement technique to predict the expected remaining coating life on turbine parts. Consequently, the great variability of TBC coating lifetimes has resulted in turbine coating failures in the field or parts prematurely taken out of service if removed based on a conservative lower bound of expected coating lifetime.

Under the Advanced Gas Turbine Systems Research (AGTSR) program, the University of Connecticut (UCONN) has been evaluating the use of laser fluorescence (LF) to measure average internal stresses for non-destructive evaluation (NDE) of thermal barrier coatings (TBCs). Experiments in the project subjected TBC coated specimens to thermal
cycles up to a temperature of 1121 C (2050 F). The LF technique was able to predict remaining life of coatings to within 5% for specimens that had been exposed to 1 hour thermal cycles and to within 7% for specimens that had been exposed to 24 hour thermal cycles. Useful engineering predictions of remaining TBC lifetimes were consequently shown for 1 hour and 24 hour thermal cycles. However, except for LF measurements taken near the end of coating life, the correlation of LF data with remaining TBC life differed for the two different cycle times. Additional work using LF measurements at times closer to end of life will evaluate whether the prediction method can be used without requiring knowledge of cycle times.

**Improved Roughness Information for Turbine Airfoil Design**

Roughness characteristics of turbine vane and blade airfoil surfaces change with operation time due to erosion, corrosion, deposition, and spallation of coatings from the parts. These surface changes can degrade the airfoil aerodynamics and cooling from their initial finely tuned, as manufactured, levels.

Under the Advanced Gas Turbine Systems Research (AGTSR) program, the Air Force Institute of Technology (AFIT) and Mississippi State University (MSU) are characterizing the effects of service conditions on turbine vane and blade heat transfer and aerodynamic performance. Over 100 turbine parts have been obtained from Allied-Signal, GE, Siemens-Westinghouse, and Solar Turbines. These components had experienced service in turbines under a wide range of conditions. Scaled models of measured surfaces from turbine parts were produced for wind tunnel experiments in which heat transfer and aerodynamic data were obtained for those surfaces. Comparison of data representing real turbine surfaces and data representing ordered arrays (cones or hemispheres) and equivalent sand grain roughness, which are traditionally used to simulate real surfaces, showed limitations in the traditional methods to characterize roughness of turbine surfaces. A more accurate method was identified to represent turbine surfaces for turbine aerodynamic and heat transfer analyses and design.

The surface roughness measurement database for turbine parts and data from the wind tunnel tests representing the observed roughness are being provided to turbine manufacturers to improve their tools for design and analyses of airfoils.
Real Surface Effects on Turbine Heat Transfer and Aerodynamic Performance

1 March 2001 to 1 September 2001
Semi-Annual Report

B.K. Hodge (Mississippi State University), Principal Investigator
Jeffrey Bons (Air Force Institute of Technology), Co-investigator
Rolf Sondergaard (Air Force Research Lab), Co-investigator
Richard Rivir (Air Force Research Lab), Co-investigator

Subcontract No. 99-01-SR076
Clemson University Research Foundation
South Carolina Institute for Energy Studies
Clemson, South Carolina 29634-5180
EXECUTIVE SUMMARY

AGTSR ANNUAL REPORT

MISSISSIPPI STATE UNIVERSITY
Mississippi State, MS 39762

Project Title: Real Surface Effects on Turbine Heat Transfer and Aerodynamic Performance
AGTSR Subcontract No: 99-01-SR076

Principal Investigator:
B.K. Hodge, (601) 325-7315, (FAX) 325-7223, email: hodge@engr.mssstate.edu

Co-Investigators:
Jeffrey Bons, (937) 255-6565 ext 4327, (FAX) 656-7621, email: jeffrey.bons@afit.edu
Richard Rivir, (937) 255-2744, (FAX) 656-4531, email: rivirrb@wl.wpafb.af.mil
Rolf Sondersgaard, (937) 255-6768, (FAX) 656-4531, email: rolf.sondersgaard@wl.wpafb.af.mil

Project Start Date: 1 March 1999
Projected Completion Date: 28 February 2002

First Year Contract Value: $113,079
Second Year Contract Value: $104,000
Third Year Contract Value: $104,295

SUMMARY OF PROGRESS

Program Developments: This research program is divided into four discrete phases: Surface Cataloguing, Flat-plate Model Testing, Computational Modeling and Additional Experimental Parameters, and Model Validation in a Full-scale Test Facility. This semi-annual report covers results from the Phase II Flat-plate Model Testing as well as progress made with the Phase III Computational Modeling and Additional Experimental Parameters. Progress made toward Phase IV Model Validation in a Full-scale Test Facility is addressed briefly.

Phase II:

Experimental measurements of skin friction ($c_f$) and heat transfer ($St$) augmentation were made for low-speed flows over several scaled turbine roughness surface models. These tests were conducted in the AFRL Aero-thermal Research Laboratory. The models were scaled from surface measurements taken on actual, in-service land-based turbine hardware during Phase I of this effort. Model scaling factors
ranged from 25 to 63, preserving the roughness height to boundary-layer momentum thickness ratio for each case. The roughness models include one pitted surface, two coated/spalled surfaces, one fuel deposit surface, and two erosion/deposit surfaces.

Measurements were made in a zero-pressure gradient turbulent boundary layers at two Reynolds numbers ($Re_x = 500,000$ and $900,000$) at low freestream turbulence levels ($Tu = 1\%$). Measurements indicate augmentation factors ranging from 1.1-1.5 for $St/St_o$ and from 1.3-3.0 for $c_f/c_{f_0}$ ($St_o$ and $c_{f_0}$ are the smooth plate values). For the range of roughness studied (average roughness height, $k$, less than one-third the boundary-layer thickness) the level of $c_f$ augmentation agrees well with accepted equivalent sandgrain ($k_s$) correlations when $k_s$ is determined from a roughness shape/density parameter. This finding is not repeated with heat transfer, in which case the $k_s$-based $St$ correlations overpredict the measurements. Both $c_f$ and $St$ correlations severely underpredict the effect of roughness for $k^+ < 70$ (when $k_s$, as determined by the roughness shape/density parameter, is small). This is the region defined by Nikuradse as being “transitionally rough” (and aerodynamically smooth for $k^+ < 5$) where $c_f$ is a function of both Reynolds number and roughness height. A new $k_s$ correlation based on the rms surface slope angle overcomes this low $k^+$ limitation. This parameter was first reported by Acharya et al. in 1986 and has since received only limited attention in the literature.

Comparison of data from real roughness and deterministic (ordered arrays of cones or hemispheres) roughness suggests that deterministic roughness is fundamentally different from real roughness. Specifically, $k_s$ values that correlate $c_f$ for both deterministic and real roughness are found to correlate $St$ for deterministic simulated roughness but overpredict $St$ for real roughness. A proposed physical explanation for this finding is advanced in the attached full report (IGTI 2002 paper submitted by Bons). These observations expose limitations in the traditional equivalent sandgrain roughness model and the common use of ordered arrays of roughness elements to simulate real roughness surfaces.

Portions of this data set will be presented at the 40th AIAA Aerospace Sciences Meeting and Exhibit in Reno (Jan 2002) by Cpt Jess Drab, one of the AFIT masters students who completed his work last year. A more comprehensive summary of findings from this phase of the effort has been submitted for presentation at the 2002 IGTI Conference in Amsterdam. A copy of this draft manuscript is included in this report. As always, great care is being exerted to maintain complete manufacturer anonymity in all published documents.

Phase III Experimental Effort:

Measurements were made at the same two Reynolds numbers ($Re_x = 500,000$ and $900,000$) with three freestream turbulence levels ($Tu = 1\%, 5\%, \text{ and } 11\%$). Freestream turbulence of turbine flowfields is one of the significant features that can have a synergistic effect with roughness, so freestream turbulence was selected as the first additional parameter to be studied experimentally. Exploring this and other parameters such as pressure gradient are necessary in order to establish the relevance of the
laboratory findings to actual turbine operating environments. These elevated freestream turbulence levels produce augmentation ratios of 1.2 and 1.5 (St/St0) and 1.2 and 1.3 (c_f/c_{f0}) compared to the Tu = 1% flow over the smooth reference plate. The combined effects of roughness and elevated freestream turbulence are greater than their added effects suggesting that some synergy occurs between the two mechanisms. Specifically, skin friction augmentation for combined turbulence and roughness is 20% greater than that estimated by adding their separate effects and 9% greater than compounding (multiplying) their separate effects. For heat transfer augmentation, the combined effect of turbulence and roughness is 6% higher than that estimated by compounding their separate effects at high freestream turbulence (Tu = 11%). At low turbulence (Tu = 5%), there is a negative synergy between the two augmentation mechanisms as the combined effect is 12% lower than that estimated by compounding their separate effects. These Phase III data are included in the paper that has been submitted for presentation at the 2002 IGTI Conference in Amsterdam.

**Phase III Computational Modeling Effort:**

The computational modeling efforts centered about developing procedures for capturing surface characteristics of real surface roughness into the discrete-element roughness model (validated originally for deterministic roughness) and extending the validation of the discrete-element roughness model to real surface roughness. Significant progress has been made in both of these efforts. Details of the successes in using real surface characteristics in the discrete-element roughness model and in extending the validation of the discrete-element roughness model to real surfaces are in the attached progress report by McClain and Hodge.

Previous progress reports have discussed the use of a layered-ellipse model to describe real surface roughness characteristics. A Mathcad program has been developed to take profilometer surface roughness data and extract input information for the discrete-element roughness model. The Mathcad program generates, at discrete elevations, the distributions of ellipses, eccentricities (major and minor axes), and orientations (parallel or perpendicular to the flow) for a given measured surface roughness. These distributions are the geometry input required for the discrete-element roughness model.

Using the surface roughness descriptions and the measurements of skin friction (c_f) and heat transfer (St) augmentation made in the AFRL Aero-thermal Research Laboratory, the discrete-element model validation is being extended to real surfaces. As delineated in the attached report by McClain and Hodge, modifications to the form drag expressions, to account for non-circular cross-section roughness elements, used in the discrete-element roughness model resulted in adequate agreement with the skin friction experimental data from Phase II. Additional skin friction data over a more extended Reynolds range and for additional surface roughness configurations as well as a detailed estimate of skin friction data uncertainties are needed for more extensive validation of the skin friction capability of the discrete-element model for real surface roughness. Additional “tweaking” of the form drag correlations is underway in conjunction with the validation of the Stanton number prediction validation. The primary discrete-element
modeling effort currently underway is the validation of the discrete-element roughness Stanton number (heat transfer) prediction capability for real surface roughness using the Stanton number data from Phase II.

**Phase IV Model Validation in a Full-scale Test Facility:**

Tragedy struck the AFRL Turbine Research Facility (TRF) this summer when a turbine rotor was over-torqued during shake-down tests, breaking the shaft and damaging the turbine casing. Repairs are underway to bring the facility back on line early next year. This will undoubtedly delay the testing planned for the final phase of this project. Dr. Rivir is working directly with the TRF team to ensure that we can get into the TRF as soon as possible after it is operational again. This will most likely be after April 2002. The plan is to continue with these tests beyond the final date of this contract (28 Feb 2002) and include the results in the final report (or as an addendum). As such, two possible methods for constructing rough-surface turbine vanes are being explored at present: casting of roughened vanes using stereolithography or sand-blasting an existing turbine vane set. The results of preliminary studies of these two options are expected to be completed by years’ end so that necessary preparations can be made in the early part of 2002.

**Interactions with Industry:** Phase I results from this contract were well received during a presentation at the 2001 IGTI conference in New Orleans (June 2001). Discussions with numerous industry and academic representatives have continued from that time. Roughness measurements from Phase I have been shared with University of Idaho researchers who are performing AFOSR funded roughness research in the DOE INEL facility.

**GOALS FOR NEXT REPORTING PERIOD**

1) Conduct Phase III experiments with non-zero pressure gradients.

2) Prepare all required parts and instrumentation for Phase IV testing in the AFRL Turbine Research Facility.

3) Extend the validation of the discrete-element roughness model for heat transfer capability (Stanton number) using the results of Phase II and Phase III.

**FULL PROGRESS REPORT:**

During the past six months, two documents have been generated (one by AFIT and one by MSU) which give a comprehensive summary of the experimental and computational results to date. Rather than attempt to “resummarize” these reports, they are attached and constitute the full progress report for this period.
Introduction to the Discrete-Element Model

The discrete-element model is formulated for roughness elements with three-dimensional shapes for which the element cross-section can be defined at every height, y. This form of the discrete-element roughness model has its origin in the work of Finson (1976). The differential equations including roughness effects are derived by applying the basic conservation statements for mass, momentum, and energy transfer to a control volume such as that shown in Figure 1. Basic to this approach is the idea that the two-dimensional, time-averaged turbulent boundary-layer equations can be applied in the flow region below the crests of the roughness elements. The flow variables have been spatially averaged over the transverse (z) direction and the streamwise (x) direction. The physical effects of the roughness elements on the fluid in the control volume are modeled by considering the flow blockage, the local element heat transfer, and the local element form drag. The blockage factors, $\hat{a}$, are defined as the fraction of the area open to flow. The form drag force on the control volume is due to the portion of a roughness element penetrating the control volume and is expressed using a local drag coefficient as

$$F_D = \frac{1}{2} \rho U^2 C_D d(y) \delta y$$  (1)
Likewise, the rate of heat transfer between the portion of the element penetrating the control volume and the fluid is expressed using a local Nusselt number as

$$Q = kNu_d \pi \delta_y (T_e - T)$$  \hspace{1cm} (2)

Figure 1. Discrete-Element Roughness Model Control Volume Schematic

Using the above ideas, the continuity, momentum, and energy equations for a steady, Reynolds averaged, two-dimensional turbulent boundary layer with uniform roughness become

$$\frac{\partial}{\partial x} (\rho \beta_x U) + \frac{\partial}{\partial y} (\rho \beta_y V) = 0$$  \hspace{1cm} (3)

$$\beta_x \rho U \frac{\partial U}{\partial x} + \beta_y \rho V \frac{\partial U}{\partial y} = -\frac{\partial}{\partial x} (\beta_y P) + \frac{\partial}{\partial y} [\beta_x (\mu \frac{\partial U}{\partial y} - \rho U V)] - \frac{1}{2} \rho C_v d \frac{U^2}{L_l}$$  \hspace{1cm} (4)

and

$$\beta_x \rho U \frac{\partial H}{\partial x} + \beta_y \rho V \frac{\partial H}{\partial y} = \frac{\partial}{\partial y} \left[ \beta_x \left( \frac{k}{c_p} \frac{\partial H}{\partial y} - \rho U \dot{h} \right) \right] + U \frac{\partial}{\partial x} (\beta_y P) + \beta_x \frac{\partial U}{\partial y} \left( \mu \frac{\partial U}{\partial y} - \rho U V \right) + \frac{1}{2} \rho C_v d \frac{d}{L_l} U^3 + \frac{kNu_d}{L_l} (T_e - T)$$  \hspace{1cm} (5)
where $L$ and $l$ are the spacing parameters for uniform roughness and are the results of spatially averaging the roughness effects. In addition to the usual turbulence modeling closure requirements for $-\overline{\rho u'v'}$, $\overline{u'^2}$, and $-\overline{\rho v'h'}$, the roughness model has closure requirements for $C_D$ and $Nu_d$.

The turbulence model is not modified to include roughness effects since the physical effects of the roughness on the flow are explicitly included in the differential equations. The Prandtl mixing length with van Driest damping is used for turbulence closure. Hence

$$-\overline{\rho u'v'} = \rho l_m^2 \left( \frac{\partial U}{\partial y} \right) \left( \frac{\partial U}{\partial y} \right)$$

where

$$l_m = \begin{cases} 0.40y\left[1 - \exp(-y^+ / 26)\right] & \text{for } l_m \leq 0.09\delta \\ l_m = 0.09\delta & \text{otherwise} \end{cases}$$

Closure in the energy equation is achieved using a turbulent Prandtl number, $Pr_t$, of 0.9.

$C_D$ and $Nu_d$ for the roughness models are formulated as functions of the local roughness Reynolds number

$$Re_d = U(y)d(y)/\nu$$

thus directly including information on the roughness element size and shape. The functional forms for $C_D$ and $Nu_d$ have been determined by extensive calibrations using a number of deterministic surface roughness data sets. The current functional forms yield good skin friction and Stanton number agreement with a wide range of uniform surface roughness shapes and distributions from a number of data sets. These forms are as follows (Taylor and Hodge 1993):
\[ C_D = \begin{cases} \left( \frac{\text{Re}_d}{1000} \right)^{0.125} & \text{Re}_d < 60,000 \\ 0.6 & \text{Re}_d > 60,000 \end{cases} \]  \tag{8}

and

\[ \text{Nu}_d = \begin{cases} 1.7 \text{Re}_d^{0.49} \Pr^{0.4} & \text{for } \text{Re}_d \leq 13,776 \\ 0.0605 \text{Re}_d^{0.84} \Pr^{0.4} & \text{for } \text{Re}_d > 13,776 \end{cases} \]  \tag{9}

The modeling effort was intended to answer questions about how to model turbulent flow over a randomly-rough surface. One of the most important questions concerned the location of the reference height for a randomly-rough surface. To analyze a deterministic rough surface using the discrete-element model, height is referenced to the “flat plate” surface. For a randomly-rough surface, the first, and most obvious, choice for the zero-height datum is the minimum valley of the surface. The choice of the minimum valley as the datum complicates the discrete-element analysis because of the very high blockage fraction at the minimum valley. If any height other than the minimum valley surface is used as the datum, the effect of the flow “below” the datum on the skin friction and heat transfer must be explored.

A second question was related to the skin friction over a randomly-rough surface. The discrete-element roughness model was implemented in the finite difference code BLACOMP (Boundary LAyer COMPutation), which was used in this study to predict the skin friction and heat transfer results for the rough surfaces (Gatlin and Hodge 1990). The discrete-element roughness model was validated for regularly but sparsely placed cones and hemispheres on a flat surface. The question is “will the element drag relationships be sufficient for modeling a surface with closely-spaced roughness elements of many different diameters, shapes and spacings?” This project is the first at Mississippi
State University to attempt to model a real surface with all surface features determined from 3-D profilometer traces.

The third question concerned the heat transfer. Like the element drag relationships, the usefulness of the heat transfer relations have not been evaluated for randomly-spaced roughness elements of random diameters. Another question regarding heat transfer is the “fin effect,” the temperature change along the height of the roughness. The current discrete-element model assumes the roughness elements to be at a constant temperature. If the roughness material has a low thermal conductivity, the temperature change along the roughness elements can be significant. The “fin effect” of temperature change along the height of the roughness must be evaluated.

**Roughness Reference Line**

The discrete-element roughness model in BLACOMP was validated for a series of sparsely-spaced cones, spheres, or hemispheres placed on a flat surface. For these deterministic type surfaces, the computational field began at the flat surface and extended into the freestream region. A randomly-rough surface does not have an obvious reference surface. A reference surface at the minimum elevation makes sense for the blockage-fraction evaluation, but the roughness model needs both blockage fraction and blockage diameter distributions. For random roughness with closely-spaced roughness features below the mean elevation, the concept of a blockage diameter does not make sense. Because all of the blockage elements are connected together, the effective diameter of the blockages becomes the length of the profilometer trace. In fact, below the mean elevation, most of the flow is blocked, and an “openage” diameter makes more sense.
Figure 2 shows the blockage of a randomly-rough surface at an elevation of 25% of the minimum valley-to-maximum peak. Areas blocked from flow are shown in black, while areas open to flow are shown in white. Figure 2 demonstrates that although a blockage fraction can be calculated (14.1%), determining an appropriate blockage diameter does not make much sense. The discrete-element model assumes that the fluid flows around the blockages, but from Figure 2, the only way that fluid can get from one open region to another is to flow over the blockages. That motion results in secondary-flow phenomena not considered in the discrete element model.

![Figure 2. Randomly-Rough Surface Blockage at 25% Elevation](image)

Schlichting suggested an alternative for a reference surface. In his early work on roughness, Schlichting used wall-variable plots to determine equivalent sandgrain roughness values as compared to Nikuradse’s earlier experiments (Schlichting 1936, Nikuradse 1933). To evaluate the sandgrain roughness values, the correct reference surface had to be used. Schlichting chose to use what he called the “melt down” surface as his reference surface. The “melt down” surface is the resulting surface if all of the
material were melted down and allowed to solidify in the same area with a constant height. The “melt down” surface is equivalent to a surface placed at the mean height of the surface, as evaluated using equation (10) for a continuous rectangular surface.

\[
h_m = \frac{\int_0^{L_x} \int_0^{L_y} y(x, z) dz dx}{L_x L_y}
\]

This reference surface worked well, but Schlichting did not present a justification of his choice.

Taylor ran the discrete element model for a series of closely-packed hemispheres with differing reference heights and compared the results to the experimental results. Taylor found that the discrete element predictions agreed best when the “melt down” height of the closely-packed hemispheres was used as the reference surface. Taylor did not speculate on a physical justification for this choice of reference height (Taylor 1983).

Pinson and Wang (2000) measured Stanton numbers over a rough surface that experienced a decrease in roughness height scale. The measurements were performed to examine the effect of an abrupt decrease in roughness height scale on transition from laminar to turbulent flow. They noticed that the flow sometimes separated then reattached to the smooth surface as if it were a step flow from one elevation to a lower elevation. This provides some physical justification for using the “melt down” height as a reference height. The “melt down” height can be thought of as the mean hydraulic elevation of the rough surface.

Figure 3 depicts channel flow between parallel plates. On the bottom plate, cone roughness has been placed over a small region. The height of the cones is exaggerated to show the decrease in flow area caused by the placement of the cones on the flat surface.
As the fluid flows around an individual roughness element, the flow accelerates because of conservation of mass and the reduced local flow area. Over the roughness region, there is an increase in the average freestream velocity. By conservation of mass and assuming constant density, the increase in the average velocity is proportional to the decrease in the average flow area between the parallel plates. This decrease in flow area for the flow between parallel plates is proportional to the decrease in height of the duct, which is equal to the mean hydraulic elevation of the roughness.

Figure 3. Acceleration Due to Cones Placed on Flat Surface

Using the concept of the mean hydraulic elevation does not explain all the phenomenon seen by Pinson and Wang related to the recovery length of the separation region because of the effects of the roughness elements, but the mean hydraulic elevation does give a reference scale to the equivalent “step height” change in roughness scales. The mean hydraulic elevation provides a way to evaluate the reduction in flow area caused by rough surfaces. The mean hydraulic elevation is also the average height of the “no-slip” location along the surface. Consequently, the mean hydraulic elevation acts as the reference height for spatially-averaged analysis of flow over a rough surface.
Effect of Flow Area Below the Mean Hydraulic Elevation

Below the mean hydraulic elevation, there is flow which contributes to friction drag as well as to heat transfer, but solving the boundary-layer equations requires the application of the “no-slip” condition at the mean hydraulic elevation. Because of the requirement of “no-slip” at the mean hydraulic elevation, accounting for the effect of the flow below the mean hydraulic elevation, on friction is performed after the determination of the velocity profile and during the determination of the skin friction coefficient.

Equation (11) shows the discrete-element expression used to evaluate the skin friction coefficient for a flow over a rough surface.

\[
C_f = \frac{(1 - \alpha) \mu \frac{du}{dy} \bigg|_{y=0} + \frac{1}{2} \frac{1}{L_i L_p} \int_{0}^{\infty} \alpha u^2 C_d \, dy}{\frac{1}{2} \rho U_e^2}
\]  

(11)

In equation (11), \( y = 0 \) is the “no-slip” location. The two terms in the numerator of equation (11) represent the contributions of skin friction on the smooth part of the wall and roughness-element drag. The method used for this study treats any point below the mean hydraulic elevation as flat surface at the mean hydraulic elevation and adds the skin friction associated with the flat surface of the mean hydraulic elevation. The blockage fraction, \( \alpha \), at the mean hydraulic elevation is used in equation (11) to calculate the drag associated with the skin friction.

Since there is little difference between the flat surface of a sparsely-spaced cone or spherical segment rough surface and its mean hydraulic elevation, Taylor essentially used this option to validate the cone and spherical segment model used in BLACOMP. The model worked very well but was mainly validated for sparsely-spaced roughness
elements. Most of the rough surfaces validated by Taylor (1983) had spacings of more than twice the base diameters of the blockages.

The drawback to the above option concerns the “no-slip” condition at the mean hydraulic elevation. The “no slip” condition implies that the fluid velocity at the surface or reference elevation must be equal to the surface velocity and the temperature of the fluid at the surface must equal the temperature of the surface. While solution of the boundary-layer equations on a surface requires applying the “no slip” condition at a reference surface, there is flow beneath the mean hydraulic elevation for randomly-rough surfaces. While there is some type of flow beneath the mean elevation, the fact that the discrete-element model is a spatially-averaged model means that these secondary flows cannot be resolved with the discrete-element model.

As shown in Figure 2 and discussed above, the open regions below the mean hydraulic elevation are small and isolated. The flow in these open regions is likely local cavity flow or likely resembles “skimming” flow as described by Morris (1972). If the flow below the mean hydraulic elevation is cavity flow or “skimming” flow, then specifying the correct boundary condition at the mean hydraulic elevation other than the “no slip” condition would be difficult. The skin friction contribution from the flow in the randomly-sized-and-spaced valleys would be difficult to analyze, but is different from the skin friction and heat transfer from a flat surface. Until more information is obtained on the flow in the valleys, the effects of the flow below mean hydraulic elevation are best treated as skin friction on a “flat surface” placed at the mean hydraulic elevation.

Like the skin friction calculations, applying the “no-slip” condition at the mean hydraulic elevation was also used in the heat transfer predictions. The velocity and
temperature profiles were found by applying the “no-slip” condition at the mean hydraulic elevation. The Stanton number, as presented in equation (12), was then evaluated by adding the heat convected from the roughness elements and the heat convected from the flat surface area on the mean hydraulic elevation.

\[
St = \frac{-(1 - \alpha)k_f \frac{dT}{dy} \bigg|_{y=0} + \frac{1}{L_y L_p} \int_{0}^{\infty} \pi \delta_f \xi \frac{\Delta T}{T_f} dy}{\rho U_c c_p (T_w - T_e)}
\]  

(12)

Surface Descriptions

Two randomly-rough surfaces, the Deposit surface and the Erosion 2 surface, were used in the modeling study. The Deposit surface contains relatively large, anisotropic roughness elements aligned in the direction of the flow. The Deposit surface has a centerline average roughness, \(Ra\), of 1.183 mm. A bitmap image of a section of the Deposit surface is shown in Figure 4. The Erosion 2 surface contains smaller, isotropic roughness elements. The Erosion 2 surface has a centerline average roughness of 0.504 mm. A bitmap image of a section of the Erosion 2 surface is shown in Figure 5.
Along with the randomly-rough surfaces, two layered-analog surfaces were studied. The surfaces were derived from the Deposit surface and the Erosion 2 surface. The spacings and heights of the roughness elements are random as measured from the original randomly-rough surface, but the roughness elements were generated by placing ellipsoidal blockages of equivalent area and eccentricity at the height of the original random blockage element. Because the blockage elements were created at distinct heights between the mean hydraulic elevation and the maximum elevation of the surface, the layered-analog surfaces have a “layered” appearance. The layered representation of the Deposit surface, the Deposit Layered surface, is shown in Figure 6. The layered representation of the Erosion 2 surface, the Erosion 2 Layered surface is shown in Figure 7.
Figure 6. Bitmap Image of the Deposit Layered Surface
Two cone surfaces, the Deposit Cone surface and the Erosion Cone surface, were also studied. These surfaces were used to validate the model developed by Taylor for non-sparse cone surfaces. The Deposit Cone surface consisted of 4.420-mm tall cones with base diameters of 5.969 mm. The cones were evenly spaced 12.065-mm apart. The spacing-to-base-diameter ratio was 2.730. The Erosion Cone surface consisted of 1.905-mm tall cones with base diameters of 4.445 mm. The cones were evenly spaced 5.207-mm apart. The spacing-to-base-diameter ratio was 1.171. The cone surfaces were also created based on the randomly-rough surfaces. The cone surfaces were created with equivalent centerline averaged roughness, $Ra = \frac{1}{N} \sum_{i=1}^{N} |y_i|$, values as their respective randomly-rough surfaces.

**Cone Surface Results**

The method presented concerning the reference surface and flow characteristics below the datum was made concerning a randomly-rough surface, but if the method presented can be used for randomly-rough surfaces, the method should also hold for surfaces with cone roughness-elements. The skin friction coefficients of the two cone surfaces were evaluated using the entire height of the roughness element from the solid...
surface following the model developed by Taylor (1983) and using the mean hydraulic height of the roughness elements and adding wall friction on the unblocked area at the mean hydraulic elevation. Figure 8 demonstrates the variables involved in the determining the characteristics of cone surfaces above the mean hydraulic elevation. The height of the cone above the mean hydraulic elevation and the diameter of the cone at the mean hydraulic elevation are calculated using equations (13) and (14), respectively.

\[ h' = h - \frac{\pi d_0^2 h}{12L_p L_t} \]  

(13)

\[ d_0' = d_0 \frac{h'}{h} \]  

(14)

The heights and diameters of the Deposit Cone surface cones above the mean hydraulic elevation are 4.137 mm and 5.587 mm, respectively. The heights and diameters of the Erosion Cone surface cones above the mean hydraulic elevation are 1.541 mm and 3.596 mm, respectively. The discrete-element model was run for the cone surfaces using each of the following options:

1) the full cone placed on the surface floor with shear acting on the surface floor,

2) the partial cone referenced to the mean hydraulic elevation with shear acting on the area not blocked at the mean hydraulic elevation,

The results are presented in Table 1.
Table 1 shows that modeling the cone surfaces referenced to the surface floor overestimates the drag for both cone surfaces. As expected, this model is adequate for the Deposit Cone surface because it is sparse. For the closely-packed cones of the Erosion Cone surface, this method greatly overestimates the drag.

If a 10% uncertainty in the experimental data is assumed, then setting a datum at the mean hydraulic surface and adding shear on the area at the datum not blocked by flow models the surfaces well. Modeling the surfaces referenced to the mean hydraulic elevation and adding the wall shear to the skin friction coefficient on the area of the
reference surface not blocked by flow has a very low percentage difference for the Deposit Cone surface and barely exceeds the assumed uncertainty bands for the Erosion Cone surface.

**Random Surface Skin Friction Results**

Acknowledging the cone surface results, the random surfaces were modeled as roughness elements placed at the mean hydraulic elevation of the real surface for the skin friction coefficient predictions. After the boundary-layer equations were solved using the “no-slip” condition at the surface, surface shear was added to the skin friction coefficient on the area of the mean hydraulic elevation not blocked by roughness elements. For each blockage element at a given height above the mean hydraulic surface, the maximum width of the blockage element was extracted and used as the diameter. The total drag at a given height in the boundary layer was then calculated with the following expression for circular blockages as suggested by Taylor (1983).

\[
F_D = \frac{2 \mu^2}{2} \frac{\partial \sigma}{\partial x} \sum_{i=1}^{N} C_{D_i} d_i
\]

The above model was applied to each of the randomly-rough surfaces and to the layered-analog surfaces of the randomly-rough surfaces. The results are presented in Table 2. Table 2 shows that the surface that is most like the model, which assumes circular blockages on a flat surface with large spacings relative to the diameter of the blockages, is accurately predicted. The Erosion 2 Layered surface is a very sparse surface with circular blockages and a flat surface beneath the mean hydraulic elevation. For the Erosion 2 Layered surface, the model prediction is within 2% of the experimentally measured value of the average skin friction coefficient.

Table 2. Random Roughness and Layered Ellipsoidal Surface Results
<table>
<thead>
<tr>
<th>Surface</th>
<th>Average Re&lt;sub&gt;x&lt;/sub&gt;</th>
<th>Measured ( \overline{C_f} )</th>
<th>Predicted ( \overline{C_f} )</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Erosion 2</td>
<td>550000</td>
<td>0.010663</td>
<td>0.011522</td>
<td>8.056</td>
</tr>
<tr>
<td>Erosion 2</td>
<td>700000</td>
<td>0.009728</td>
<td>0.011512</td>
<td>18.339</td>
</tr>
<tr>
<td>Erosion 2 Layered</td>
<td>550000</td>
<td>0.008514</td>
<td>0.008680</td>
<td>1.950</td>
</tr>
<tr>
<td>Deposit</td>
<td>700000</td>
<td>0.008578</td>
<td>0.011157</td>
<td>30.065</td>
</tr>
<tr>
<td>Deposit Layered</td>
<td>700000</td>
<td>0.006545</td>
<td>0.007832</td>
<td>19.664</td>
</tr>
</tbody>
</table>

While the average friction coefficient should decrease slightly for the Erosion 2 surface as the Reynolds number increases slightly, the drop of 11.58% of the experimentally measured average friction coefficient from an Re<sub>x</sub> of 5.5×10<sup>5</sup> to an Re<sub>x</sub> of 7.0×10<sup>5</sup> shown in Table 2 is surprising. The EXCEL spreadsheet containing the experimental results indicates that the experimental data were taken with a Pitot probe for Re<sub>x</sub> of 5.5×10<sup>5</sup> while a hot wire anemometer was used for Re<sub>x</sub> of 7.0×10<sup>5</sup>. The difference in experimental techniques may have exaggerated the difference between the two skin friction coefficients. *The skin friction measurements should be repeated at Re<sub>x</sub> = 5.5×10<sup>5</sup> and at Re<sub>x</sub> of 7.0×10<sup>5</sup> with the hot wire anemometer to verify the results.*

Table 2 demonstrates that while applying the circular drag to the roughness elements above the mean hydraulic elevation does overestimate the skin friction on the surface, it does yield the same trends between the surfaces. Each of the layered models was created by placing a flat surface at the mean line of the original surfaces and then building ellipsoidal representations of the blockages at each height. Because the floors were placed at the mean line of the original surfaces, the layered models act as if their roughness elements are shorter than the roughness elements on the original surfaces. The results in Table 2 demonstrate that the experimental results and the model output reflect this. The average skin friction coefficients for the Erosion Layered and the Deposit
Layered surfaces are significantly less than the Erosion and the Deposit surfaces, respectively.

The circular blockage model significantly over estimates the friction coefficient on the Deposit surface. Figure 4 shows a bitmap image of a section of the Deposit surface. Figure 4 demonstrates that the Deposit surface is anisotropic. The blockages are elongated in the flow direction.

This elongation is expected to provide some relieving effect on the drag. To account for this, an ellipsoidal blockage drag model was developed. Figure 9 shows an ellipsoidal blockage aligned with the major axis parallel to the flow direction. The eccentricity, $e$, of the blockage is defined as the ratio of the maximum width perpendicular to the flow direction to the maximum length of the blockage parallel to the flow direction.

$$e = \frac{a}{b}$$ (16)

Figure 9. Ellipsoidal Blockage
Drag coefficient data for ellipsoidal cylinders of varying eccentricities with transverse thickness Reynolds numbers around $1 \times 10^5$ from Lindsey (1938) and Delany and Sorensen (1953) are plotted in Figure 10. A function of eccentricity which best represented the data was found by determining the log-log regression curve of the form shown below.

$$C_{d,Re=10^5} = a \varepsilon^b$$

Equation (8) was multiplied by the eccentricity function to yield a drag coefficient function accounting for the eccentricity of the roughness elements.

![Figure 10. Drag Coefficients of Elliptical Cylinders with Varying Eccentricities [Data from Lindsey (1938) and Delany and Sorensen (1953)]](image-url)

The constants $a$ and $b$ were found to be 0.97379 and 0.73456, respectively. Since the constant $a$ is close to unity and since the eccentricity function should equal unity when the blockage element is circular, the constant $a$ was set to unity. The eccentricity
function with \( a \) set to unity is not a true regression curve, but the eccentricity function represents the trend of the data. The resulting eccentricity function, \( \varepsilon^{0.73456} \), is also shown in Figure 10.

The circular-blockage-element drag function, equation (8), was then multiplied by the eccentricity function to yield an ellipsoidal-blockage-element drag function, shown in equation (17).

\[
C_D = \begin{cases} 
\left( \frac{\text{Re}_d}{1000} \right)^{-0.125} \varepsilon^{0.73456}, & \text{Re}_d < 60,000 \text{ and } \varepsilon < 4.46 \\
0.6\varepsilon^{0.73456}, & \text{Re}_d > 60,000 \text{ and } \varepsilon < 4.46
\end{cases}
\]  

(17)

The above function increases without bound as the eccentricity increases. As the eccentricity increases, the roughness elements are expected to experience drag similar to and limited by the drag of a flat plate perpendicular to the flow. Janna (1993) reports that the drag coefficient of a flat plate perpendicular to the flow will have a drag coefficient of 1.8 for Reynolds numbers above 60,000. The eccentricity of an ellipse that results in a drag coefficient of 1.8 for a Reynolds number of 60,000 is 4.46. For eccentricities greater than 4.46, the blockages are expected to act like flat plates perpendicular to the flow as described by equation (18).

\[
C_D = \begin{cases} 
3.0 \left( \frac{\text{Re}_d}{1000} \right)^{-0.125}, & \text{Re}_d < 60,000 \text{ and } \varepsilon > 4.46 \\
1.8, & \text{Re}_d > 60,000 \text{ and } \varepsilon > 4.46
\end{cases}
\]  

(18)

The Deposit surface and the Deposit Layered surface were then modeled using the ellipsoidal-blockage-element drag function. There are two options available to employ the ellipsoidal-blockage-element drag function in the discrete-element model: 1) use an average eccentricity at each height or 2) build an input file that contains the eccentricity.
for every blockage element at each height. Figure 11 shows the mean eccentricity and the maximum-transverse-width weighted eccentricity for the Deposit surface. The maximum-transverse-width weighted eccentricity weighs the eccentricity of the eccentricities of the larger blockage elements more. The maximum-transverse-width weighted eccentricity is important because the larger blockage elements contribute more drag. The maximum-transverse-width weighted eccentricity was calculated using equation (19).

\[
\mathcal{E}_{TW} = \sum_{i=1}^{N_b} a_i \mathcal{E}_i
\]

Figure 11. Deposit Surface Blockage Eccentricity versus Height

Based on Figure 11, an eccentricity of 0.8 was used for the average eccentricity model. The discrete-element model was also run taking in the eccentricity for each blockage element at a given height. The results of the ellipsoidal-blockage-element
model are presented in Table 3 and compared to experimental results and the results of the circular-blockage-element model. Table 3 shows that the results of the models with the average eccentricity and with the eccentricity for each blockage are comparable and are much closer to the measured results than the circular blockage model. The prediction for the Deposit surface is 21% high, and the prediction for the Deposit Layered surface is 12% high. Without uncertainty information, this suggests other factors influencing the drag on the rough surfaces may not have been considered.

Table 3. Computer Results Incorporating Ellipsoidal-Blockage-Element Eccentricity

<table>
<thead>
<tr>
<th></th>
<th>Deposit Surface</th>
<th>% Difference</th>
<th>Deposit Layered Surface</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean Reynolds Number</td>
<td>700000</td>
<td></td>
<td>700000</td>
<td></td>
</tr>
<tr>
<td>Experimentally Measured $C_f$ (Hodge 2000)</td>
<td>0.008578</td>
<td>—</td>
<td>0.006545</td>
<td>—</td>
</tr>
<tr>
<td>Predicted $C_f$ with Circular Blockage Model</td>
<td>0.011157</td>
<td>30.065</td>
<td>0.007832</td>
<td>19.664</td>
</tr>
<tr>
<td>Predicted $C_f$ with Ellipsoidal Blockage Model (Average Eccentricity of 0.8)</td>
<td>0.010427</td>
<td>21.555</td>
<td>0.007314</td>
<td>11.749</td>
</tr>
<tr>
<td>Predicted $C_f$ with Ellipsoidal Blockage Model (Eccentricity Evaluated for Each Blockage)</td>
<td>0.010393</td>
<td>21.167</td>
<td>0.007317</td>
<td>11.795</td>
</tr>
</tbody>
</table>
Preliminary Heat Transfer Results

The surfaces were also evaluated for the average Stanton number using the modeling choices stated previously. The surfaces were modeled with a reference surface at the mean hydraulic elevation. The Deposit Cone surface, the Erosion Cone surface, the Erosion surface, and the Erosion Layered surface were analyzed using the circular-blockage-element function. The Deposit and the Deposit Layered surfaces were analyzed using the ellipsoidal-blockage-element function for elemental drag calculations, but the circular element Nusselt relations were used for predicting the heat transfer. The fluid was evaluated as a constant property gas. Using a constant property gas code produces some error, but the assumption is adequate for initial estimates of the average surface Stanton numbers. The computational results for all of the surfaces, including the flat plate surface, are compared to the experimental data from Hodge (2000) in Table 4.

<table>
<thead>
<tr>
<th>Surface</th>
<th>Average Re&lt;sub&gt;x&lt;/sub&gt;</th>
<th>Measured $\bar{St}_x$</th>
<th>Predicted $\bar{St}_x$</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Erosion Cone</td>
<td>550000</td>
<td>0.003343</td>
<td>0.005663</td>
<td>69.399</td>
</tr>
<tr>
<td>Erosion 2</td>
<td>550000</td>
<td>0.003291</td>
<td>0.005013</td>
<td>52.325</td>
</tr>
<tr>
<td>Erosion 2</td>
<td>700000</td>
<td>0.003118</td>
<td>0.004729</td>
<td>51.668</td>
</tr>
<tr>
<td>Erosion 2 Layered</td>
<td>550000</td>
<td>0.003256</td>
<td>0.004103</td>
<td>26.014</td>
</tr>
<tr>
<td>Deposit</td>
<td>700000</td>
<td>0.003078</td>
<td>0.003882</td>
<td>28.164</td>
</tr>
<tr>
<td>Deposit Layered</td>
<td>700000</td>
<td>0.002706</td>
<td>0.003101</td>
<td>15.848</td>
</tr>
<tr>
<td>Flat Plate</td>
<td>550000</td>
<td>0.002332</td>
<td>0.002899</td>
<td>24.291</td>
</tr>
<tr>
<td>Flat Plate</td>
<td>700000</td>
<td>0.002275</td>
<td>0.002822</td>
<td>24.077</td>
</tr>
</tbody>
</table>

Table 4 demonstrates that the model overestimates the average Stanton number for all of the surfaces. The most intriguing and disturbing result shown in Table 4 is the disagreement between the experimental data and the BLACOMP predictions for the flat plate Stanton number results. The BLACOMP predictions of Stanton number for the flat
plate are 24% higher than the experimentally measured Stanton numbers at the two Reynolds numbers explored. These results are disturbing because the BLACOMP results are best validated for the case of turbulent flow over a rough surface.

Reasons for the disagreement between the measured Stanton numbers and the BLACOMP predictions were explored. Again the EXCEL spreadsheet containing the experimental data indicated that the data had been corrected for material properties. When the experimental data were first recorded, the correct values of thermal conductivity and thermal diffusivity of the roughness panels were not known. The data were corrected once the values of thermal conductivity and thermal diffusivity had been measured. Interestingly, the discrete element model output compared much better to the raw experimental data. A comparison to the raw experimental data is presented in Table 5.

Table 5. Surface Heat Transfer Results with Raw Experimental Data

<table>
<thead>
<tr>
<th>Surface</th>
<th>Average Re&lt;sub&gt;x&lt;/sub&gt;</th>
<th>Measured $\overline{St}_x$</th>
<th>Predicted $\overline{St}_x$</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Erosion Cone</td>
<td>550000</td>
<td>0.004270</td>
<td>0.005663</td>
<td>32.623</td>
</tr>
<tr>
<td>Erosion 2</td>
<td>550000</td>
<td>0.004204</td>
<td>0.005013</td>
<td>19.244</td>
</tr>
<tr>
<td>Erosion 2 Layered</td>
<td>700000</td>
<td>0.003982</td>
<td>0.004729</td>
<td>18.759</td>
</tr>
<tr>
<td>Erosion 2 Layered</td>
<td>550000</td>
<td>0.004159</td>
<td>0.004103</td>
<td>-1.346</td>
</tr>
<tr>
<td>Deposit</td>
<td>700000</td>
<td>0.003931</td>
<td>0.003882</td>
<td>-1.247</td>
</tr>
<tr>
<td>Deposit Layered</td>
<td>700000</td>
<td>0.003456</td>
<td>0.003101</td>
<td>-10.272</td>
</tr>
<tr>
<td>Flat Plate</td>
<td>550000</td>
<td>0.002979</td>
<td>0.002899</td>
<td>-2.685</td>
</tr>
<tr>
<td>Flat Plate</td>
<td>700000</td>
<td>0.002905</td>
<td>0.002822</td>
<td>-2.857</td>
</tr>
</tbody>
</table>

Table 5 shows that the discrete element predictions are much closer to the raw data for 5 out of the 8 surface conditions presented. The Erosion Cone surface and the Erosion 2 surface predictions are still considerably higher than the raw experimental measurements. One of the possible reasons the computer code is still predicting high is because of temperature change along the height of the roughness elements. This
temperature change is analogous to the temperature change experienced along the length of an extended surface or fin. Temperature change along the blockage elements would decrease the average Stanton number predictions.

The Deposit surface and the Deposit Layered surface predictions are close to the raw experimental data, however, another aspect of ellipsoidal surfaces has not been considered. While the drag on the ellipsoidal blockages has been evaluated, the change in heat transfer coefficient and the increase in surface area caused by the change from circular to ellipsoidal blockages has not yet been considered.

The Erosion 2 Layered surface is also in agreement with the raw experimental data, and would not be changed much by considering temperature drop along the blockages. The Erosion 2 Layered surface is sparse and has very small diameter roughness elements that look like pin fins. While the blockage elements may have a moderately low “pin fin” efficiencies, the surface would have a high surface efficiency because it is sparse. Thus, considering temperature loss along the Erosion 2 Layered surface would not significantly change its average Stanton number prediction.

**Extended Surface Corrections**

To further investigate whether temperature decrease along the height of the blockage elements could have caused the over prediction of the average Stanton number for the Erosion Cone surface and the Erosion 2 surface, the Erosion Cone surface blockage elements were modeled as extended surfaces to determine the temperature change along the cones and the fin efficiency of the cones. The fluid temperature profile and velocity profile generated by BLACOMP at the center of the Erosion Cone surface were used in the investigation. The current heat transfer model was used to calculate heat
transfer coefficients along the cones. The general extended surface equation, equation (20), was numerically integrated to determine the temperature profile along the height of the cones.

\[
\frac{d^2 T_b}{dy^2} + \left( \frac{1}{A_c} \frac{dA_c}{dy} \right) \frac{dT_b}{dy} - \left( \frac{1}{A_c} \frac{h}{k} \frac{dA_c}{dy} \right) (T_b - T(y)) = 0
\]  

(20)

The resulting normalized temperature profile is shown in Figure 12. The normalized temperatures are defined as shown in equations (21 a, b).

\[
\Theta = \frac{T_b - T_e}{T_w - T_e}, \quad \Theta_w = \frac{T(y) - T_e}{T_w - T_e}
\]  

(21 a, b)

The solid line in Figure 12 is the normalized temperature of the cones at the given height, the dashed line is the normalized temperature of the fluid at the given height, and the dotted line is the normalized velocity \((u/U_e)\) of the fluid at the given height.

The fin efficiency of the cone element was determined to be 44.97%. The Erosion Cone surface results were analyzed to determine that the heat transfer from the cones accounted for 69.4% of the total Stanton number. If the contribution from the cones is decreased by the efficiency of the cone treated as an extended surface, the result is an average surface Stanton number of 0.0034. Thus, temperature change along the roughness element is responsible for some of the difference between the measured and the predicted Stanton number for the Erosion Cone surface and the Erosion 2 surface. Since the temperature profile is also changed depending on heat transfer from the cones, the full effect of temperature loss along the roughness elements cannot be evaluated unless the complete conjugate heat transfer problem is analyzed.
Figure 12. Temperature Profile of Erosion Cone Element from General Pin Expression

**Requested Experimental Action**

As mentioned earlier, the experimental skin friction results for the Erosion 2 surface are surprising. The experimental results suggest that the skin friction coefficient decreases 11.58% from an $Re_x$ of $5.5 \times 10^5$ to an $Re_x$ of $7.0 \times 10^5$, while the code prediction decreases negligibly from an $Re_x$ of $5.5 \times 10^5$ to an $Re_x$ of $7.0 \times 10^5$. The code predictions suggest that the flow has reached completely rough flow. Since the two experimental data points were acquired using two different velocity measurement devices, the experimental measurements should be repeated using the hot wire anemometer.
For many of the surfaces, there are heat transfer and skin friction results for only two or three average Reynolds numbers. To validate the model, the surfaces need to be evaluated over a greater Reynolds number range. The annual report for 1 March 2000 to 28 February 2001 reports that the maximum Reynolds number for the wind tunnel is $4 \times 10^6$. Six or seven measurements of skin friction coefficient and Stanton Number are needed for each surface over the Reynolds number range of $5.5 \times 10^5$ to $4 \times 10^6$.

**Future Computational Action**

While some of the questions posed at the beginning of the report have been answered, some questions have not been answered, and some questions have been added. Adding the ellipsoidal blockage model to the BLACOMP predictions of the Deposit surface and the Deposit Layered surface lowered the skin friction predictions. The predicted skin friction coefficients are 21% and 12% higher than the measured coefficients for the Deposit surface and the Deposit Layered surface, respectively. The cause(s) of this over predictions will be investigated.

Several mechanisms are present in causing the over predictions of the Stanton numbers of each of the surfaces. The increased surface area caused by elliptical blockages as compared to circular blockages will be added to the discrete element code. The “fin effect” of temperature change along the height of the roughness elements will also be further investigated. A subroutine could be added to BLACOMP to integrate the fin equation at every x location for the cone surfaces, but creating this subroutine would require considerable effort. Adapting the subroutine for a randomly-rough surface, will require even more time because of the range of shapes, spacings, and heights of random
blockage elements. An average element diameter and spacing technique needs to be developed for randomly-rough surfaces.

Conclusions

The modified discrete-element model for predicting skin friction and heat transfer over randomly-rough surfaces involves placing a reference surface at the mean hydraulic elevation of the rough surface. Drag and heat transfer are calculated on the roughness elements depending on the local flow velocity and the equivalent ellipsoidal dimensions of the roughness.

The friction coefficient predictions using the model are higher than the experimental measurements for most of the surfaces. While the predictions are higher than the experimental measurements, the model does predict trends that are similar to trends in the experimental data. For example, the model predicts that the Deposit Cone surface will have a higher friction coefficient than the Deposit surface, which will have a higher friction coefficient than the Deposit Layered surface. The experimental data show the same trend with the deposit surfaces. The Erosion Layered surface is most like the circular-blockage, discrete-element model in that it has sparse roughness with circular blockages and flat surface below the mean elevation. For the Erosion Layered surface, the discrete-element prediction is within 2% of the experimentally measured friction coefficient.

The discrete-element model did fail to predict one trend shown in the experimental results. For an increase in Reynolds number over the Erosion 2 surface, the experimentally measured friction coefficient decreased over 10%. The discrete-element predictions showed a negligible decrease over the same increase in Reynolds number.
The Stanton number predictions using the new model are considerably higher than the experimentally measured values for all of the surfaces. The Stanton number predictions agree much better with the raw experimental measurements, but after the experimental measurements are corrected for roughness material properties, the predictions are higher. The “fin” effect, by which the temperature of the roughness elements changes along the height of the blockage into the flow, was investigated as a possible cause for the over predictions of Stanton number. The Erosion Cone surface was found to exhibit some “fin” effect, but the total “fin” effect cannot be evaluated without the solution of the full conjugate heat transfer problem.

More experimental data are needed to refine the model for both skin friction and heat transfer predictions. At least four more Reynolds numbers over the region of $9 \times 10^5$ to $4 \times 10^6$ are needed for each rough surface. This should help validate the trend in measured friction coefficient exhibited by the Erosion 2 surface. A better estimate of the experimental uncertainty is also needed to help refine and validate the model.

References


Abstract

Experimental measurements of skin friction ($c_f$) and heat transfer (St) augmentation are reported for low speed flow over scaled turbine roughness models. The models were scaled from surface measurements taken on actual, in-service land-based turbine hardware. Model scaling factors ranged from 25 to 63, preserving the roughness height to boundary layer momentum thickness ratio for each case. The roughness models include samples of deposits, TBC spallation, erosion, and pitting. Measurements were made in a zero pressure gradient turbulent boundary layer at two Reynolds numbers ($Re_x = 500,000$ and $900,000$) and three freestream turbulence levels ($Tu = 1\%, 5\%$, and $11\%$). Measurements at low freestream turbulence indicate augmentation factors ranging from 1.1-1.5 for $St/St_o$ and from 1.3-3.0 for $c_f/c_{fo}$ ($St_o$ and $c_{fo}$ are smooth plate values). For the range of roughness studied (average roughness height, $k$, less than $1/3^{rd}$ the boundary layer thickness) the level of $c_f$ augmentation agrees well with accepted equivalent sandgrain ($k_s$) correlations when $k_s$ is determined from a roughness shape/density parameter. This finding is not repeated with heat transfer, in which case the $k_s$-based $St$ correlations overpredict the measurements. Both $c_f$ and $St$ correlations severely underpredict the effect of roughness for $k^+ < 70$ (when $k_s$, as determined by the roughness shape/density parameter, is small). A new $k_s$ correlation based on the rms surface slope angle overcomes this limitation.

Comparison of data from real roughness and simulated (ordered cones or hemispheres) roughness suggests that simulated roughness is fundamentally different from real roughness. Specifically, $k_s$ values that correlate $c_f$ for both simulated and real roughness are found to correlate $St$ for simulated roughness but overpredict $St$ for real roughness. These findings expose limitations in the traditional equivalent sandgrain roughness model and the common use of ordered arrays of roughness elements to simulate real roughness surfaces. The elevated freestream turbulence levels produce augmentation ratios of 1.2 & 1.5 ($St/St_o$) and 1.2 & 1.3 ($c_f/c_{fo}$) compared to the $Tu=1\%$ flow over the smooth reference plate. The combined effects of roughness and elevated freestream turbulence are greater than their added effects suggesting that some synergy occurs between the two mechanisms. Specifically, skin friction augmentation for combined turbulence and roughness is $20\%$ greater than that estimated by adding their separate effects and $9\%$ greater than compounding (multiplying) their separate effects. For heat transfer augmentation, the combined effect of turbulence and roughness is $6\%$ higher than that estimated by compounding their separate effects at high freestream turbulence ($Tu = 11\%$). At low turbulence ($Tu = 5\%$), there is a negative synergy between the two augmentation mechanisms as the combined effect is now $12\%$ lower than that estimated by compounding their separate effects.
A_l – windward frontal surface area of roughness elements on sample
A_s – windward wetted surface area of roughness elements on sample
c_f – skin friction coefficient, \( \tau_w/(0.5\rho U_\infty^2) \)
c – blade/vane chord
c_p – specific heat at constant pressure
h – convective heat transfer coefficient
k – average roughness height (= Rz)
(also thermal conductivity),
k_s – equivalent sandgrain roughness
k^+ = \kappa u_t/\nu \equiv Re_k
Ku – kurtosis of height distribution (Eqn 5)
N – number of points in profile record
Pr – Prandtl number (\( \nu/\alpha \)) (= 0.71)
Pr_t – turbulent Prandtl number (\( \approx 0.9 \))
Ra – centerline average roughness (Eqn 1)
Re_x – Reynolds number (\( U_\infty x/\nu \))
Re_c – Reynolds number based on blade chord and exit conditions
Rq – rms roughness (Eqn 2)
Rt – maximum peak to valley roughness (Eqn 3)
Rz – average peak to valley roughness
S – surface area of sample without roughness
S_f – total frontal surface area of sample
S_w – total wetted surface area of sample
Sk – skewness of height distribution (Eqn 4)
St – Stanton Number, \( h/(\rho c_p U_\infty) \)
T – temperature [°C]
t – time [s]
Tu – freestream turbulence, \( u'/U_\infty [%] \)
U_\infty – freestream velocity
u’ – fluctuating velocity (rms)
\( u_t \) – friction or shear velocity \( \sqrt{\tau_w/\rho} \)
x – streamwise distance from tunnel floor leading edge
y – surface height coordinate after removal of polynomial fit to surface curvature
\alpha – thermal diffusivity (k/\( \rho c_p \))
\alpha_{rms} – rms deviation of surface slope angles
\delta – boundary layer thickness
\eta – efficiency factor (St/St_o)/(c_f/c_f_o)
\lambda_c – correlation length
\Lambda_s – roughness shape/density parameter (Eqn 6)
\nu – kinematic viscosity
\theta – boundary layer momentum thickness
\rho – density
\tau_w – wall shear

subscripts
adj – adjusted k_s value required to match Schlichting c_f correlation to data
max – maximum height in profile record
min – minimum height in profile record
o – smooth plate reference at low freestream turbulence
s – surface value
\infty – freestream value

Introduction/Background

Modern land-based turbine airfoils operate in severe environments with high temperatures and near critical stresses. Highly turbulent combustor exit flows spew hot combustion products and other airborne particulates at the turbine surfaces for more than 20,000 hours before regularly scheduled maintenance. Due to this harsh operating environment, turbine surfaces experience significant degradation with service. Measurements reported previously by this author [1] and others [2,3,4] indicate an order of magnitude or greater increase in rms roughness is typical for a first stage high pressure turbine vane or blade.
For over twenty years, the effects of these elevated levels of surface roughness on turbomachinery performance have been studied at all practical levels; from fundamental flat-plate wind tunnel research, to multi-blade cascade facilities, to full-up system level tests. These studies all support the expected result that roughness increases surface drag and heat transfer (though to varying degrees). For turbomachinery, this translates to higher heat loads, accelerated part degradation, and lower stage efficiencies.

At the system level, Blair [5] was perhaps the first to report roughness-related increases in St on a rotating turbine facility. In his study, premature boundary layer transition combined with other roughness-induced effects to produce a nearly 100% increase in St for some cases. Guo et al. [6] also reported a two-fold increase in heat transfer for a factor of 25 increase in roughness height (Rz) on their fully scaled nozzle guide vane facility. In their studies with compressors and pumps, Boynton et al. [7], Suder et al. [8], and Ghenaiet et al. [9] all observed 3-5 points loss in efficiency with roughened blades. With the exception of Ghenaiet et al., who actually simulated metal erosion due to sand ingestion, all of these system-level tests have been conducted using uniformly distributed roughness (e.g. sand or painted-on particulates).

Sand and powders have also been used to simulate rough surfaces in turbine cascades, where more detailed blade surface measurements can be made [10,11,12]. These studies have each documented effects similar to those of their system-level counterparts; accelerated boundary layer transition, increased heat transfer, and increased blade losses. A fourth cascade study by Abuaf et al. [13] explored the benefits of metal polishing processes and found that a factor of three reduction in centerline-averaged roughness produced up to a 15% reduction in blade surface integrated heat transfer at some Reynolds numbers.

Unlike system and cascade level tests, flat-plate wind tunnel testing has explored a broader spectrum of surface roughness charaterizations. In addition to sand roughness experiments [2,14], researchers have used distributed cylinders [15,16], spherical segments [17,18], cones [19,20], and pedestals [21]. In each case, the roughness element size and spacing was selected to match a predetermined set of roughness statistical parameters. This set could include traditional parameters such as Ra or Rz (Rz ≈ k, the mean roughness height) or more sophisticated characterizations such as correlation length [17], rms deviation of surface slope angles [2], or a roughness shape/density parameter [20] in combination with other parameters. The majority of these researchers have also correlated their findings in terms of the equivalent sandgrain roughness, ks, in an effort to translate their characterizations of turbine roughness into the much wider array of roughness encountered in pipe flows and external aerodynamics. A notable exception to this is the work of Taylor et al. [17] which has pursued a discrete-element model (DEM) to evaluate distributed roughness elements. Both methods (ks and DEM) have met with varying degrees of success when brought to bear at the cascade and system level. Consequently, designers and operators continue to make significant allowances (safety margins) for uncertainty when calculating heat transfer or losses in operating rough turbines.

It is possible that part of this difficulty in modeling lies with the fact that different roughness-producing mechanisms (deposits, erosion, pitting, and coating spallation) have unique surface features. For example, as shown in [1], pitting and spallation have large roughness recesses below the surface mean line while deposits are characterized by peaks
above the mean line. Fundamental fluid dynamics research (most recently by Kithcart
and Klett [22]) has shown that recesses have a more marked effect on St than on $c_f$
when compared with hills of equivalent dimensions. This is attributed to the reduced form
drag and enhanced three-dimensional flows (vortices) associated with recesses vs. peaks.
Nuances such as this can easily be lost when 1 or 2 roughness parameters are used to
characterize a wide variety of rough surfaces with discrete, uniform elements. This then
creates biases in the final roughness correlation which accentuate one type of roughness
while de-emphasizing another.

One way to avoid this bias generated by distributed roughness characterizations of
“real” roughness is to employ scaled replicas of actual turbine surfaces in fundamental
wind tunnel testing. Results can then be evaluated without bias to a particular dominant
shape or spacing. The objective of the present study is to begin building such a database.
Accordingly, a diverse (though certainly not comprehensive) collection of six actual
surface roughness samples (from [1]) were scaled and tested in a flat plate wind tunnel.
Both $c_f$ and St were measured and comparisons were made with contemporary roughness
correlations found in the open literature. Correlations were also attempted with a number
of the roughness parameters cited in previous studies and the most promising candidates
were identified.

In addition, it is natural for the turbine designer to be suspicious of data taken in
the often pristine laboratory environment. Thus, the relevance of laboratory findings to
the actual turbine operating environment is of critical importance. One of the significant
features of turbine flowfields that could have a synergistic effect with roughness is
freestream turbulence. There have been very few studies which have considered the
coupled effects of freestream turbulence and surface roughness. Turner et al. [11] found
the effect of grid-generated (~7%) turbulence to be similar to a two order of magnitude
increase in Ra (as determined by the blade mean heat transfer coefficient). The combined
effect of turbulence and roughness was approximately additive. A similar finding was
reported by Bogard et al. [20] for turbulence levels up to 17%. Finally, the results
reported by Hoffs et al. [12] using 5% and 10% turbulence in a cascade facility with
rough blades lack sufficient detail to determine whether roughness and turbulence effects
are complementary. However, the effects of the two different mechanisms on heat
transfer are clearly shown to be distinct and significant. The present study expands the
existing database to include both St and $c_f$ measurement at 5% and 11% freestream
turbulence. Again, these measurements are taken using scaled “real” roughness models
versus the simulated roughness characterizations (e.g. sand or cones) employed in
previous studies.

Surface Roughness Measurement and Fabrication

In preparation for the current study, nearly 100 land-based turbine components
were assembled from four manufacturers: General Electric, Solar Turbines, Siemens-
Westinghouse, and Honeywell (formerly Allied-Signal) Corporation. The articles were
selected by each manufacturer to be representative of surface conditions generally found
in the land-based gas turbine inventory. Chord dimensions on the assembled blades and
vanes ranged from 2 to 20cm and included samples with thermal barrier coatings (TBC).
In order to respect proprietary concerns of the manufacturers, strict source anonymity has been maintained for all data presented in this publication.

Extensive 2-D and 3-D surface measurements were made on the assembled hardware [1] using a Taylor-Hobson Form Talysurf Series 2 contact stylus measurement system. This device uses a 1.5\(\mu\)m radius diamond-tipped conical stylus to follow the surface features for a given part. The instrument has a maximum horizontal stroke of 50mm and can measure a total vertical range of 2mm with a precision of 32nm. 3D measurements were made by indexing the part by specified increments in the horizontal direction perpendicular to the stroke of the contact stylus. Increments from 5 to 40 microns were used to map out regions from 1x1mm square to 40x40mm square for various components. Once a 3D map was taken, the Talymap\textsuperscript{TM} software was used to remove the part’s form with a polynomial least squares surface fit. With the form removed, the relevant statistics could be extracted from the roughness data. Evaluations were conducted to compute the centerline averaged roughness, Ra, the rms roughness, Rq, the maximum peak-to-valley roughness, Rt, the skewness, Sk, and the Kurtosis, Ku, as defined below (\(y_{\text{mean}} = 0\)):

\[
Ra = \frac{1}{N} \sum_{i=1}^{N} |y_i|, \quad Rq = \sqrt{\frac{1}{N} \sum_{i=1}^{N} y_i^2}, \quad Rt = y_{\text{max}} - y_{\text{min}} \quad \text{[Eqns. 1-3]}
\]

\[
Sk = \left\{ \frac{1}{N} \sum_{i=1}^{N} y_i^3 \right\} \frac{1}{Rq^3}, \quad Ku = \left\{ \frac{1}{N} \sum_{i=1}^{N} y_i^4 \right\} \frac{1}{Rq^4} \quad \text{[Eqns. 4-5]}
\]

Each 3D map was also evaluated in smaller subsets (or cutoffs) to estimate Rz, the average of the local Rt values over the entire map. Rz is commonly used as an estimate of the average roughness height, k. The 3D maps were then evaluated using 2D auto-correlations in both the streamwise and cross-stream directions. The correlation lengths, \(\lambda_c\), were calculated as the distance at which the autocorrelation functions fell to a value of 0.1. In addition, by dissecting the 3D surface map into its 200 to 1000 individual 2D traces, the local surface slope angles between each measurement point could be calculated. The sum of these angles over the entire 3D surface was then used to compute the rms deviation of surface slope angles, \(\alpha_{\text{rms}}\), considered by some [2,20] to be an important roughness parameter. \(\alpha_{\text{rms}}\) was calculated for both the streamwise and cross-stream directions.

One final measurement made from the 3D surface maps was the roughness shape/density parameter, \(\Lambda_s\). This parameter was developed by Sigal and Danberg [23] to correlate the \(k_s\) estimation process to both the spacing and shape of roughness elements. The parameter was derived for use with two and three dimensional roughness elements (e.g. ribs or cones) mounted to a smooth surface. It is defined as,

\[
\Lambda_s = \frac{S}{S_f} \left( \frac{A_f}{A_s} \right)^{-1.6} \quad \text{[Eqn. 6]}
\]

where S is the reference area of the sample surface (without roughness), \(S_f\) is the total frontal surface area of the roughness elements on the sample, \(A_s\) is the windward wetted surface area of a roughness element, and \(A_f\) is the frontal surface area of a roughness element. Since the surfaces being evaluated in this study were real roughness surfaces
rather than ordered cones or hemispheres, the calculation of \( A_f, S_f, \) and \( A_s \) was adapted accordingly. To do so, each cell of the 3D surface height matrix was evaluated independently to determine its windward frontal area and windward wetted area. These were then summed to obtain \( A_f \) and \( A_s \) for the entire surface. Since there are no discrete roughness elements, \( A_f = S_f \) following this procedure. Performing the calculation cell by cell in this manner removed any subjectivity that might have been introduced by selecting only conically shaped peaks above some critical height in the surface height matrix. The values of \( \Lambda_s \) were then used to estimate \( k_s \) for each surface using a curve fit to data assembled by Sigal and Danberg and repeated in [20].

\[
\log \left( \frac{k_s}{\kappa} \right) = -1.31 \log(\Lambda_s) + 2.15 \quad \text{[Eqn. 7]}
\]

This same cell-by-cell computation yielded yet another area ratio found to be of some importance in this study. This is the total wetted surface area (both windward and leeward) to smooth surface area ratio \( (S_w/S) \).

Of the 25 3D maps reported in [1], six were selected for this study. These include one pitted surface, two coated/spalled surfaces, one fuel deposit surface, and two erosion/deposit surfaces. The statistics for the scaled models of each of the six surfaces are contained in Table 1. Representative 2D traces from each surface are shown in Figure 1. [Note: the vertical dimension has been magnified to show roughness features.] The first is a surface that exhibited severe pitting. It is pockmarked with 40-80\( \mu \)m deep craters, each with a width-to-depth ratio of 10 to 20. The second surface is a TBC coated surface that exhibited intermittent debond without spallation. This created an undulating surface with a regular pattern of peaks and valleys not unlike closely-packed spherical segments. Surface #3 is a TBC surface which experienced extensive spallation. Surface #4 is an example of fuel deposits that are elliptical in shape and aligned with the streamwise flow direction. The last two surfaces are representative of combined erosion and deposits with smaller, more jagged roughness elements than surface #4.

To properly scale these six surfaces for wind tunnel testing, two parameters were monitored: the roughness height to boundary layer momentum thickness ratio \( (Rz/\theta) \) and the roughness regime as defined by \( k^+ \) \( (Re_k \text{ in some texts}) \). To properly compute these two parameters for the turbine hardware used in this study would have required a detailed knowledge of the operating environment (pressures, temperatures, etc…) for each of the measured blades. Using this operational data, a boundary layer calculation could be performed to compute \( \theta \) (for \( Rz/\theta \)) and \( u_\tau \) (for \( k^+ \)). Since operational data was not supplied by the manufacturers, estimates of these parameters were made as follows. For all cases, the blade Reynolds number, \( Re_c \), was approximated at \( 2 \times 10^6 \). This is in the range of values considered typical for high pressure turbine vanes/blades as reported by other researchers [4,13]. The local momentum thickness at the given 3D map’s chordwise location was then estimated using a zero pressure gradient, incompressible, turbulent boundary layer correlation from the leading edge to that point on the blade \( [\delta/x = 0.16Re_x^{-1/7} \text{ or } \delta/c = 0.16(x/c)^{0.97}Re_c^{-0.17}] \) and \( \theta \cong 0.097\delta \) from a power-law boundary layer profile. Admittedly, this estimate is an oversimplification. Turbine blade boundary layers are not turbulent from the leading edge and they are subject to pressure gradients, transonic Mach numbers, freestream turbulence, and (of course) roughness. Yet, the
range of $R_z/\theta$ estimates obtained by this method (0.5 to 3) are comparable to those obtained by more sophisticated means [20,21] (see Table 1). Moreover, even a more rigorous, fully-scaled computational estimate on a smooth blade geometry would not account for the boundary layer altering effects of the surface roughness and may not be more accurate. Finally, the goal of this study is not to assess engine-specific roughness effects but rather to use realistic turbine roughness to develop an improved physical understanding that will benefit the entire turbine industry. So, the simple estimates were deemed appropriate and adequate.

From this estimate for $\theta$, the ratio $R_z/\theta$ was computed and the necessary scaling was determined for each 3D map. At the nominal $Re_x$ of $9 \times 10^5$ used in this study, the boundary layer momentum thickness in the wind tunnel is approximately 2mm ($Re_\theta = 1500$). This mandated scalings between 25 and 63 depending on the model (Table 1).

Because of the inherent difficulty with trying to match 2 parameters simultaneously for a wide variety of surfaces, the $k^+$ roughness scaling was monitored to insure operation in the same roughness regime for the actual part and the scaled model. Nikuradse [24] classified roughness into three regimes: aerodynamically smooth ($k^+ < 5$), transitionally rough ($5 < k^+ < 70$), and completely rough ($k^+ > 70$). In the smooth regime, roughness is not a factor and $c_f$ is only a function of $Re$. In the completely rough regime, roughness dominates $c_f$ which becomes essentially independent of $Re$. Finally, in the transitionally rough regime $c_f$ is a function of both roughness and $Re$. Again, for the present study a standard, zero-pressure gradient boundary layer correlation for $c_f$ ($c_f = 0.026 Re^{-1/7}$) was used to estimate $k^+$ ($k^+ = Re_c (k_s/c) \sqrt{c_f / 2}$) for each surface. Use of the Sigal-Danberg $k_s$ correlation (Eqn. 7) insured that $k_s$ scaled directly with $k$ ($R_z$), the result being that $k^+$ was in the same regime for each surface and its scaled model.

Once properly scaled, plastic roughness models were fabricated using a StrataSys Inc. GeniSys Xi 3D printer. The printer has a maximum part fabrication size of 200x300x200mm and creates models by extruding plastic in 0.3mm thick layers to slowly build up the part. The typical wind tunnel roughness model was composed of six individual roughness panels (140mm x 120mm each) with a mean thickness of 6mm. Arranged three abreast, the panels nearly fill the wind tunnel test section width of 380mm. If after scaling the 3D roughness map there was insufficient area to fill the entire wind tunnel test section (280mm x 360mm), the roughness data were mirrored until the minimum area requirements were exceeded.

**Description of Experimental Facility**

**Wind Tunnel Facility:**

The research facility used for the experiments is shown in Figure 2. The open loop wind tunnel uses a main flow blower to provide a nominal mass flow of 1.2kg/s to the test section. A heat exchanger at the main flow blower discharge can be used to vary the flow temperature from 18 to 54°C. The main flow enters a conditioning plenum of 0.6m diameter before reaching the rectangular test section. This conditioning plenum has one layer of perforated aluminum plate followed by 7.6cm of honeycomb straightener, and three layers of fine screen. A circular-to-rectangular foam nozzle conducts the flow from the plenum cross-section to the 0.24m by 0.38m test section. With this conditioning, 2D flow uniformity of ±3% in velocity is obtained over the center 0.32m of
the test section span. Without employing turbulence generation devices, the freestream turbulence level at the test section was 1%.

At 1.22m from the plenum exit a knife-edge boundary layer bleed with suction removes the bottom 1.27cm of the growing boundary layer, making the aspect ratio (span/height) of the final test section approximately 1.7. The top wall of this final section pivots about its forward end in order to adjust the pressure gradient in the tunnel. For the tests presented here, the wall was adjusted to produce zero freestream acceleration over the roughness test panels. At 2.54cm from the boundary layer suction point, a 1.6mm diameter cylinder spans the test section to trip the boundary layer to turbulent. The leading edge of the roughness panel sections are located 1.04m from the boundary layer suction point. The roughness panels (generally six panels make a single set) are installed in a 0.28m streamwise gap in the lower wall. The tunnel then continues 0.62m beyond the trailing edge of the roughness panels.

Freestream turbulence is generated using two distinct methods. For high turbulence generation, a “Tee” located upstream of the conditioning plenum inlet leads to a bypass blower, which generates a bypass flow in parallel with the main flow. This bypass flow is re-injected from two opposing rows of holes located on the top and bottom of the test section, 1.02m upstream of the boundary layer bleed. A heat exchanger in the bypass line is used to remove the heat of compression from the bypass flow. This jet-injection turbulence generation device produces a turbulence level of 11% at the roughness panels. A standard square-bar grid was used to obtain a lower turbulence level of 5%. The grid is composed of 1.34cm square bars spaced 6cm apart and is located 0.2m upstream of the boundary layer bleed.

Flow velocity is measured using a single-element hot-wire anemometer. A co-located flow thermocouple with 0.3mm bead diameter is used for flow temperature measurement. The two instruments are mounted on a 3-axis traverse system located atop the wind tunnel. A magnetically encoded linear position indicator affixed to the traverse was used to determine the probe position to within 2.5µm. Uncertainty in the velocity measurement stems primarily from the calibration fit accuracy. When compared to a co-located Kiel probe velocity measurement, the error is within ±1.5% at flow rates of interest.

\( c_f \) Measurement:

A number of researchers have struggled with the complexities of making drag measurements over rough surfaces. Two common methods relying solely on velocity measurements are a boundary layer momentum balance and log-region curve fitting. For a zero streamwise pressure gradient flow, the change in boundary layer momentum thickness (\( \theta \)) can be related directly to \( c_f \) by \( c_f = 2d\theta/dx \). This is the momentum balance method. It requires a minimum of two velocity profiles separated by a streamwise distance (\( dx \)) over the rough surface. The second method assumes the presence of a log-linear region in the rough wall turbulent boundary layer velocity profile (when plotted in wall units). This “log-law” region has been shown to be present in rough wall data, though the log-region constant (B) is typically adjusted as a function of \( k_s \). Both of these methods require assumptions to be made about the \( y=0 \) wall elevation, which is problematic for rough surfaces. Considering this and other drawbacks of velocity-based \( c_f \) measurements, Acharya et al. [2] concluded that, “an accurate independent
measurement of $c_f$ is thus of central importance to any experiment on rough-wall boundary layers.” Based on this conclusion, Acharya et al. employed a force-balance to measure the actual skin friction force on a roughness coupon in their flowfield.

Following this reasoning, the present work uses a hanging element balance to obtain $c_f$. The balance configuration is shown in Figure 3. Four 0.25mm diameter, 0.7m long Nichrome wires attached to an apparatus atop the tunnel allow the roughness panels to be freely suspended in the floor of the wind tunnel. The wires are located outside the wind tunnel and are affixed to the four corners of a metal support plate upon which the six plastic roughness panels are mounted. When air is flowing in the tunnel, the plate moves downstream under the applied shear force. This motion was a maximum of 1.9mm for the roughest panel tested (with turbulence). This horizontal plate motion is accompanied by a slight vertical plate motion (2.6µm for the maximum case) of the heavy support plate which produces the necessary restoring force. For small-angle motions such as this, the restoring force is approximately linear with streamwise plate deflection. Using a string-pulley apparatus with fractional gram weights, this restoring force was calibrated over the full range of deflections observed in practice. The plate deflection was measured using a Capacitec Model #4100-S capacitance meter mounted to the side of the test plate, outside the tunnel walls. The meter is stationary, while a parallel metal bracket is mounted to the moving panel. The air gap between the stationary meter face and the parallel bracket face is the capacitor thickness. The required ground loop from the moving bracket to the power supply and meter is formed using the metal suspension wires. In this way, no friction is added to the plate motion due to the meter. The wire-pulley calibration is remarkably linear and repeatable with least squares correlation coefficients of 0.9999 and repeatable slopes within ±1.5%.

The test plate is suspended with a 0.5mm gap at the leading edge and a trailing edge gap which is 0.5mm greater than the maximum expected excursion. These gaps allow unrestrained motion of the plate under the applied shear force. The gaps also permit differential pressure forces to affect the net displacement of the test plate. To mitigate these pressure forces, the leading edge gap was covered with a 0.05mm thick stainless steel sheet with 7mm overlap with the roughness panels. The initial 10mm of each panel was smooth to accommodate this overlap without interference. Despite this precaution, differential pressures still accounted for 5-15% of the net plate motion. To calculate this component of the force, three pressure taps were installed at mid-plate thickness on the adjoining stationary plexiglass pieces, both upstream and downstream of the suspended aluminum support plate with the roughness panels. The three pressure taps were ganged together to produce mean pressures for both the leading and trailing edge of the free-floating test section. A Druck LPM-5481 low pressure transducer was used to monitor this differential pressure and deduct it from the total displacement (force) measured by the Capacitec meter. With these precautionary measures, smooth plate $c_f$ values were found to be within 5% of standard correlations. Repeatability was within ±2% and bias uncertainty was estimated at ±0.0002 for the smooth plate measurement of $c_f = 0.00354$ at $Re_x = 900,000$. The operation of the Druck transducer and Capacitec meter were found to be very sensitive to temperature, greatly affecting the quoted uncertainty. As such, the main flow heat exchanger was employed to maintain constant room temperature to within ±0.5°C during testing.
St Measurement:

For the heat transfer measurements, a FLIR Thermacam SC 3000 infrared camera system is mounted with the lens fit into a hole in the plexiglass ceiling of the tunnel. The camera has a sensitivity of 0.03°C (at 30°C) and allows framing rates of approximately one Hz. At the focal distance of 37 cm, the camera field of view is roughly 70 x 90 mm. This limited field of view is centered at a distance of 1.20m from the leading edge of the tunnel floor. This puts the mean streamwise position of the heat transfer measurement roughly 2cm downstream of the center of the roughness panels (x = 1.18m). The 320 x 240 pixels of the FLIR camera allowed excellent resolution of the roughness features during the transient experiments. For the measurements reported in this study however, the surface temperatures were simply area-averaged to obtain the representative surface temperature history required for St determination.

The Stanton number was determined from this surface temperature history using the method of Schultz and Jones [25]. This technique uses Duhamel’s superposition method to calculate the surface heat flux given the surface temperature history. It assumes the panels are a semi-infinite solid at constant temperature at time t = 0. To accomplish this, the entire test section was soaked at room temperature for several hours before testing. Using the main flow heat exchanger, a hot-gas air flow was then initiated instantaneously while monitoring the freestream velocity and temperature as well as the average surface temperature (with the IR camera). The heat transfer coefficient (h) at the \(i^{th}\) time step was then calculated using the expression from Schultz and Jones.

\[
h_i = \frac{1}{T_{so} - T_{si}} \left[ \frac{2k}{\sqrt{\pi \alpha}} \sum_{j=1}^{i} \frac{T_{sj} - T_{sj-1}}{\sqrt{t_j} - \sqrt{t_{j-1}}} \right]
\]

In this expression, the summation is made over all steps prior to the \(i^{th}\) time step. A typical h history calculation using this method is shown in Figure 4. As shown, after some initial instability due to random temperature fluctuations over the initial time steps, the h history settles down to a near constant value. The figure also shows the transient flow and surface temperatures over the same time history. The thermophysical properties, thermal conductivity (k) and thermal diffusivity (\(\alpha = \frac{k}{\rho c_p}\)), for the plastic panels were determined using a National Standards conductivity meter and calorimeter. The measurements yielded the following values: k = 0.226 W/mK ±5% and \(c_p = 1913\) J/kgK ±3%. The plastic density is 1207 kg/m³ ±2%.

For heat transfer measurements, the hanging \(c_f\) balance was removed and the six roughness panels were mounted on a 12-mm thick plexiglass sheet. This plexiglass sheet has approximately the same thermophysical properties as the plastic panels to avoid thermal wave reflections at the contact surface. A thermocouple sandwiched between the panels and the plexiglass sheet indicated a slight rise in temperature after approximately 30 seconds for the typical test case. Thermocouples mounted to the underside of the plexiglass support sheet showed no significant change within the total test time of approximately 90 seconds. This confirmed the use of the semi-infinite conduction assumption in the data processing. Uncertainties due to surface radiation were eliminated by performing an in-place calibration with the panels soaked at various temperatures over the range of tunnel operation. Thermocouples mounted in the test panel assured uniform
temperatures to within ±0.5°C during this calibration. The average of these thermocouple readings was then correlated to the average surface temperature as recorded by the infrared camera. This measured difference between the actual temperature and the recorded infrared temperature was used to adjust the recorded temperature histories during transient testing. In this way, radiation heat transfer losses are accounted for in the final temperature history. The infrared measurement was also sensitive to the ambient temperature of the air between the roughness panels and the receiving optics. This too was accounted for in the data processing.

By meticulously accounting for various losses as outlined above, smooth plate St values were found to be within 2% of a standard correlation. Repeatability was within ±5% and bias uncertainty was estimated at ±0.00015 for the smooth plate measurement of $St = 0.00216$ at $Re_x = 900,000$.

**Results and Discussion**

The results are presented in order of increasing complexity. First, the smooth plate data are presented to verify that the wind tunnel facility meets the accepted standards for a zero-pressure gradient, turbulent boundary layer. Following this, the rough plate data are presented in detail. A smaller subset of this data is contained in [26] with limited discussion. Finally, the elevated turbulence data are presented for the smooth and rough panels in that order.

**Smooth Baseline:**

The smooth plate St and $c_f$ data are presented in Figures 5 & 6 for two Reynolds numbers. Standard flat plate correlations for $c_f$ and St [27] are also indicated on the plots as follows:

$$c_f = \frac{0.026}{Re_x^{1/7}} \quad \text{and} \quad St = \frac{0.5c_f}{Pr_f + \sqrt{0.5c_f \left[ 5Pr + 5 \ln(5Pr + 1) - 14Pr \right]}}$$

(Eqns 8 & 9)

The smooth plate data show agreement to within 5% and 2% of the $c_f$ and St correlations respectively.

**Effect of “Real” Roughness:**

The same figures also show the $c_f$ and St values obtained for the six rough panels at the same two $Re_x$ values. St augmentation ratios (St/Sto) vary from 1.1-1.5 and $c_f/c_{fo}$ varies from 1.3-3.0 (a factor of 3-4 times greater augmentation). As indicated previously, empirical correlations for $c_f$ and St of rough surfaces have universally been developed based on experimental data with either sand roughness or uniform arrays of roughness elements. Even the uniform roughness array data are generally converted to equivalent sandgrain roughness, $k_s$, before correlation. So, in order to assess how well these correlations apply to the “real” roughness in this study, the $k_s$ from each roughness type must be determined. These values were computed based on $\Lambda_s$ (Eqn. 7) and are tabulated in Table 1.

A comparison with four $c_f$ roughness correlations is shown in Figure 7 for the data at $Re_x = 900,000$. These correlations are respectively:
\[ c_f = [1.4 + 3.7 \log(x/k_s)]^{-2} \] from White [28]

\[ c_f = 0.168 \left[ \ln(84\delta/k_s) \right]^{-2} \] from Kays and Crawford [29]

\[ c_f = [2.87 + 1.58 \log(x/k_s)]^{-2.5} \] from Schlichting [30]

\[ c_f = [3.476 + 0.707 \log(x/k_s)]^{-2.46} \] from Mills [27]

The smooth \( c_f \) value is indicated as a dashed line in Figure 7 for reference. As shown, the correlations bound the data for all but the first three surfaces. It should be noted that these correlations were all developed using data in the fully rough regime \((k^+ > 70)\). Referencing the data in Table 1, it is clear that the first 3 surfaces in the plot do not meet this criterion. Hence, the correlations significantly underpredict \( c_f \) for these panels (even falling below the smooth reference at very low \( k^+ \)). The fact that the \( c_f \) correlations nicely bracket the experimental data in the fully rough regime suggests that \( k_s \) (as estimated using \( \Lambda_s \) in Eqn. 7) might be an appropriate parameter for \( c_f \) prediction of “real” rough surfaces in this regime. In the other roughness regimes, however, the \( k_s \) values obtained using Eqn. 7 are clearly inappropriate. Most notable are the two “aerodynamically smooth” plates \((k^+ < 5)\) which are clearly not smooth. In this regime, an equivalent roughness parameter directly related to the characteristic roughness height, \((e.g. k_s = k/2)\), is perhaps more appropriate. If this simple relation is substituted for the Eqn. 7 \( k_s \) formulation in cases where it yields \( k_s < k/2 \), the results are closer to reality. Predictions with each of the four correlations using this minimum threshold for \( k_s \) are also included in the figure with dashed lines. Similar results are found for the \( c_f \) data at \( Re_x = 500,000 \), where the Rz/\( \theta \) ratios are slightly smaller and the \( k^+ \) values are about half of their level at \( Re_x = 900,000 \).

The \( St \) data are presented in the same manner in Figure 8. In this case, three correlations are used for comparison:

\[ St = \frac{0.5c_f}{Pr_{+} + \sqrt{0.5c_f k^{+0.44} Pr^{0.44}/C}} \] from Kays and Crawford [29]

\[ St = \frac{0.5c_f}{1 + \sqrt{0.5c_f [5.19k^{+0.44} Pr^{0.44} - 8.5]}} \] from Dipprey and Sabersky [31]

\[ St = \frac{0.5c_f}{Pr_{+} + \sqrt{0.5c_f [4.8k^{+0.44} Pr^{0.44} - 7.65]}} \] from Wassel and Mills [32]

For the reference \( c_f \) value required in each correlation, one could use either experimental data (since it is available in this study) or one of the empirical correlations. Since the \( St \) correlations are intended for use as a predictive tool, it was deemed most useful to base \( St \) on a \( c_f \) correlation rather than on data. For this reason, the Schlichting \( c_f \) correlation was employed since it had the best match to the experimental data in the \( k_s > k/2 \) regime of
Figure 7 (average 3% difference with data). Kays and Crawford suggest $C = 1$ in their correlation based on a fit to data obtained with closely-packed spheres. Since this value gave extremely large $St$ predictions, the constant $C$ was arbitrarily dropped to a value of 0.35 to better match the present “real” roughness models (both cases are shown on the plot). The Kays and Crawford correlation predictions thus adjusted are not a significant improvement over the Dipprey and Sabersky results which are obtained without any special tailoring. Both predict values of $St$ which are on average 8-11% higher than the data in the $k^+ > 70$ regime. Again, in the region $k_s < k/2$, the $k_c$-based correlations are clearly inappropriate when using Eqn. 7 to determine $k_s$ for these “real” roughness models. The more reasonable results using the minimum threshold of $k_s \geq k/2$ in this region are indicated in Figure 8 as they were in Figure 7.

The fact that using the $k_s(\Lambda_s)$ correlation appears to be inadequate to span the entire range of real roughness in this study led to a closer examination of this formulation for $k_s$. In order to assess the appropriateness of the Sigal-Danberg correlation, $k_s$ was adjusted to the value required to exactly match the Schlichting $c_f$ correlation with the experimental data for each of the six panels. This $k_{sadj}$ (non-dimensionalized by $k$) is plotted vs. $\Lambda_s$ in Figure 9 (data for both Reynolds numbers). Also shown are the log fit (Eqn. 7) and the Schlichting [30] and Hosni et al. [18] simulated roughness data compiled in Bogard et al. [20]. The data in the figure show that in the fully rough regime, the real roughness models follow the Eqn. 7 fit to the simulated roughness data. Whereas, for the models with $\Lambda_s > 100$, there is a large discrepancy between “real” roughness and simulated roughness. An alternate log fit to the “real” roughness data is also indicated on the figure:

$$\log \left( \frac{k_{sadj}}{k} \right) = -0.4 \log (\Lambda_s) + 0.83 \quad \text{[Eqn. 10]}$$

This substantial difference between real and simulated roughness $k_s$ estimates led to the consideration of alternate geometrical dependencies for $k_s$ that might unify the entire data set. As such, each of the statistical parameters in Table 1 was correlated with the $k_{sadj}$ values for the six panels. Table 2 contains the correlation coefficient of a least squares fit to the resultant data. Different fitting functions were attempted in each case (linear, logarithmic, and polynomial) and the most successful attempt is included in the table. The results clearly show that $Rz$ ($\approx k$), $Ra$, $\lambda_c$, $Sk$, and $K$ do not correlate well with the $k_{sadj}$ parameter. This may explain why Eqn. 7 (which estimates $k_s$ as a function of $k$ and $\Lambda_s$) has difficulty for some of the roughness samples. When $\Lambda_s$ alone is correlated directly against $k_{sadj}$, the result is quite good (see Table 2). Surprisingly, $\alpha_{rms}$ and $S_w/S$ outperform $\Lambda_s$ as parameters of choice for this data set. Of these three, $\alpha_{rms}$ is the easiest to obtain for real turbine roughness, requiring only a handful of 2D traces. While determining $S_w/S$ and $\Lambda_s$ require full 3D surface maps. Figure 10 contains the $k_{sadj}$ vs. $\alpha_{rms}$ data and the associated fit:

$$k_{sadj} = 0.0192 \alpha_{rms} + 0.0135 (\alpha_{rms})^2 \quad \text{[Eqn. 11]}$$

This parameter was first reported by Acharya et al. [2] and has received only limited attention in the literature. The reader is cautioned to be careful in the application of Eqn. 11 inasmuch as the data set is extremely limited (only six data points thus far).
Also, the surface angles were determined from data acquired using a contact stylus, sampled every 5-40\(\mu\)m. Both of these factors will influence the resultant value of \(\alpha_{\text{rms}}\). Finally, one can readily construct surfaces with widely disparate roughness heights that could register identical \(\alpha_{\text{rms}}\) on a single 2D trace (e.g. right-angle cones of 1mm height separated by 2mm vs. right angle cones of 10mm height separated by 20mm). Clearly this cannot mean that the two would have the same \(k_s\) if both were in a 10mm thick boundary layer. So, this correlation must be considered applicable only in the range of \(0.5 < \text{Rz}/\theta < 3\) used in this study. It would also be limited to roughness that is locally fairly uniform (such as the panels studies) as opposed to widely-spaced, isolated roughness elements which would register widely varying \(\alpha_{\text{rms}}\) levels from trace to trace. Despite these limitations, it is encouraging to note that the correlation has the correct physical behavior as \(\alpha_{\text{rms}}\) approaches zero \((k_s \rightarrow 0)\). Until further validation can be obtained over a wider array of surfaces, it is only presented as a candidate for calculating \(k_s\) for real roughness.

The foregoing discussion has been focused exclusively on matching \(c_f\). Of course, it is of interest to see if the same selection of \(k_{\text{adj}}\) (matching the Schlichting \(c_f\) correlation) also brings the experimental St data in line with the corresponding St correlations. Figure 11 shows the St correlation values of the six panels where both \(c_f\) and \(k^*\) correspond to the \(k_{\text{adj}}\) values. Of the three St correlations considered in Figure 8, only Dipprey and Sabersky’s is shown in the figure since it showed the most promise in Figure 8 (without tailoring). It is also the oldest and most comprehensive data set of the three. (Both Kays and Crawford and Wassel and Mills based their correlations at least partially on the Dipprey and Sabersky findings.) This correlation still overpredicts five of the six panels by an average of 11%, while the #2 surface is matched to within 1%. This curious result prompted a closer inspection of the model surfaces. Each of the six surfaces is quite unique (as shown by the traces in Figure 1), but surface #2 has an order or regularity that makes it clearly distinct. As mentioned earlier, this intermittently debonded TBC surface has an undulating surface character not unlike that of closely-packed spherical segments. As such, it is most like the simulated roughness surfaces using cones or spheres or the close-packed sandgrain surfaces on which the correlations (like Dipprey-Sabersky) are based. Perhaps these correlations consistently overpredict the “real” roughness St precisely because they were developed with simulated (vs. real) surface roughness. So, even if the \(k_s\) of the “real” roughness surface is adjusted to match the Schlichting \(c_f\) correlation value, the Dipprey-Sabersky St correlation based on the same \(k_s\) will always be high for “real” roughness. Whereas if \(k_s\) is determined for a simulated rough surface with ordered roughness elements, both the \(c_f\) and St correlations would be accurate.

To test this hypothesis, a 7\(^{th}\) model surface was fabricated consisting of densely packed cones with the following dimensions: height = 2mm, base diameter = 5mm, spacing = 5mm. The \(\Lambda_s\) and \(\alpha_{\text{rms}}\) values for this surface are 20.2 and 24.6\(^\circ\) respectively, a close match to surface #6 (22.1 and 25.3\(^\circ\)). When the \(k_s\) for this new cone surface is adjusted to match the Schlichting \(c_f\) correlation, the St correlation of Dipprey-Sabersky matches the data to within 3% (Figure 12). This startling result provides at least a preliminary indication that the hypothesis proposed above is worthy of further investigation. If proven to be more generally true, it would suggest that simulated rough surfaces with ordered roughness elements can be used to model either the heat transfer
behavior or the skin friction behavior of “real” turbine roughness, but not both simultaneously. A physical explanation for this may be related to the 3D vortical secondary flow patterns that are generated by regular roughness elements. These secondary flows are known to enhance $St$ with little or no effect on $c_f$ [33]. The randomness and wide range of scales present in “real” roughness may serve to break-up these secondary flows and reduce $St$ for the same $c_f$. A possibly related mechanism was reported by Pinson and Wang [34] regarding roughness-induced boundary layer transition. They noted that when large roughness elements were followed by small roughness elements, boundary layer transition was actually suppressed compared to the case of large roughness elements followed by a smooth surface. Their explanation was that the smaller roughness elements break up flow disturbances generated by the larger roughness elements, thus suppressing their amplification and subsequent transition. This same mechanism may partially inhibit the heat transfer augmentation of secondary flows induced by large roughness peaks on real roughness surfaces.

If the first constant in the denominator of the Dipprey-Sabersky $St$ correlation is increased from 5.19 to 6.08, the match with the “real” roughness panels is within ±2% (also shown on Fig. 12). The new correlation would then be,

\[ St = \frac{0.5c_f}{1 + \sqrt{0.5c_f \left( 6.08k^{0.44}Pr^{0.44} - 8.5 \right)}} \]  

[Eqn. 12]

Of course, this correlation now underpredicts the $St$ for the two ordered surfaces (#2 and #7) by approximately 8%.

Incidentally, the $k_{adj}$ for surface #7 does fall in line with the $k_s$ vs. $\alpha_{rms}$ correlation shown in Figure 10 (Eqn. 11), further suggesting that $\alpha_{rms}$ may be a suitable candidate for unifying simulated and real roughness correlations for $k_s$ insofar as $c_f$ is concerned. Given the above findings, even if such a unifying correlation were derived and verified, there would still be a discrepancy with regard to $St$.

Though the above findings are presented for $Re_x = 900,000$ only, similar results were obtained at the lower Reynolds value. Only minor modifications to the $k_{adj}$ values were required (Figure 9) and the $St$ overprediction for the “real” roughness surfaces was nearly identical. The $Re_x = 500,000$ data also fall along the Eqn. 11 curve fit for $k_s$ vs. $\alpha_{rms}$.

**Effect of Freestream Turbulence:**

Boundary layer velocity profiles taken at midspan near the test section leading edge are shown in Figure 13 for three levels of freestream turbulence: 1%, 5%, and 11%. Also shown in the figure are the turbulence levels in the boundary layer. Note that in the highest turbulence case, the turbulence level in the freestream approaches the level of turbulence associated with the near wall peak. This feature has been determined by other researchers to significantly alter the momentum and energy transport of the turbulent boundary layer.

The levels of $c_f$ and $St$ augmentation produced on the smooth test plate due to the two levels of freestream turbulence are shown in Figures 14 & 15 and tabulated in Table 3. The table also contains the efficiency factor, $\eta$, defined as the ratio of $St$ augmentation to $c_f$ augmentation, $(St/St_0)/(c_f/c_{f0})$. The $St$ augmentation range with turbulence is comparable to that generated by the “real” roughness in the previous portion of this
The $c_f$ augmentation, on the other hand, is 50\% less than the St augmentation in this case. This is in stark contrast to the augmentation results with roughness where $c_f/c_{f0}$ was up to four times $St/St_o$. The physical reason for this is that increases in $c_f$ due to roughness are primarily due to form drag on the individual roughness elements. There is no heat transfer analog to this form drag component of $c_f$ augmentation, thus $c_f/c_{f0}$ is 2 to 4 times greater than $St/St_o$ for rough surfaces.

In the case of freestream turbulence, augmentation occurs due to increased momentum and energy exchange with the freestream. There are no alternate mechanisms for surface drag in this case. The finding that freestream turbulence favors heat transfer augmentation over $c_f$ augmentation is consistent with data presented by other researchers. Pedisius et al. [35] reported efficiency factors ($\eta$) from 1.10 to 1.15 for grid generated turbulence up to 8\% over a smooth plate. Maciejewski and Moffat [36] reported $St/St_o$ values up to 1.80 for their free jet turbulence with over 15\% Tu. The associated $c_f/c_{f0}$ was estimated at 1.10, yielding $\eta = 1.64$. Blair [37] proposed a correlation to capture this preference for $St$ over $c_f$ using a correction to the popular Reynolds analogy ($2St/c_f \equiv 1$). His correlation (developed for grid turbulence with up to 7\% Tu) is:

\[
\frac{2St}{c_f} = 1.18 + 0.013Tu
\]

where the freestream turbulence level ($Tu$) is in percent. This correlation underestimates the present results by approximately 50\%. A more appropriate second constant would be 0.031. Baskaran et al. [38] likewise found an up to 100\% increase in this turbulence coefficient in their grid-generated turbulence data.

In addition to the increase in Tu that accompanies the change in turbulence generation mechanism from the square-bar grid to the opposing jets in the current study, the integral lengthscale also more than doubles (from 3.5cm to 8cm). Blair [37] and Simonich and Bradshaw [39] have both studied the role of turbulence scale in heat transfer augmentation. The physical explanation offered is that larger scale turbulence has difficulty influencing the momentum and energy exchange near the wall due to damping at the solid surface. Thus, Simonich and Bradshaw reported reductions in $St$ augmentation as lengthscale increased at constant Tu. Blair tried to capture this effect by plotting $\Delta St/St_o$ and $\Delta c_f/c_{f0}$ vs. $Tu/(\alpha\beta)$ where $\alpha$ and $\beta$ are defined as:

\[
\alpha = \frac{L_u}{\delta} + 2 \quad \beta = 3\exp(-Re_\theta/400) + 1
\]

$L_u$ is the dissipation lengthscale which is typically 50\% greater than the integral lengthscale quoted above for this facility. Using these correlations, the results in Table 3 show good agreement (within 8\%) with Blair’s data for $\Delta c_f/c_{f0}$, but the heat transfer results are again significantly higher than those reported by Blair (50\% higher at 5\% Tu and 85\% higher for 11\% Tu). This difference may be attributed to the high level of Tu in this study (Blair reported on grid data up to 8\% Tu only) and the mechanism for its generation (jets vs. grids). The results of Maciejewski and Moffat ($\eta \equiv 1.64$) give the indication that the effect of freestream turbulence on heat transfer is not linear with Tu. Different mechanisms may come into play at the higher Tu levels and longer lengthscales associated with non-grid generated turbulence that may explain the sometimes 50-100\% differences between various experiments. As such, the smooth plate data with freestream
turbulence reported in this study appear to be within the range of results reported in previous studies.

There are a number of different methods to assess the combined effects of roughness and freestream turbulence on \( c_f \) and \( \text{St} \). Of critical importance to the designer is whether results with freestream turbulence alone can be simply added to results with roughness alone to approximate the effect when both are present. If true, this would imply a lack of synergy between the two augmentation mechanisms. This is attractive to the designer because it allows correction factors to be simply superposed without additional parametric testing. One way to determine the degree to which these two mechanisms are synergistic is to compare the augmentation results obtained with both mechanisms present to that achieved by either adding or compounding their individual effects. For example, if the smooth plate \( c_f \) augmentation due to freestream turbulence was 20\% and the \( c_f \) augmentation of a rough plate (without turbulence) was 30\%, the additive prediction method would result in a combined effect of 50\%. For the same case, the compound method would predict a combined effect of 56\%. These three methods of comparison are outlined algebraically for \( c_f \) as follows:

Synergistic:

\[ \frac{c_{RTu}}{c_{fo}} - 1 \equiv \frac{\Delta c_f}{c_{fo}} \]

Additive:

\[ \left( \frac{c_R}{c_{fo}} - 1 \right) + \left( \frac{c_{Tu}}{c_{fo}} - 1 \right) \]

Compound:

\[ \frac{c_R c_{Tu}}{c_{fo}^2} - 1 \]

The subscripts "R", "Tu", and "RTu" represent \( c_f \) measurements with roughness only, turbulence only, and the two effects combined respectively. Identical expressions can be written for \( \text{St} \) as well. Figures 16a-d contain these measures (for \( c_f \) and \( \text{St} \)) for all 6 samples in Table 1 at the two elevated \( Tu \) levels.

In all four figures, the additive predicted effects are less than the synergistic (or actual combined) effects. The average discrepancy is 2\% for the 5\% \( Tu \) \( \Delta \text{St}/\text{St}_o \) results (Figure 16c) and roughly 20\% for the other three figures. Even when the compound prediction is used, the results are on average 6\% low for \( \Delta \text{St}/\text{St}_o \) (at 11\% \( Tu \)) and 9\% low for \( \Delta c_f/c_{fo} \) (at both 5\% and 11\% \( Tu \)). This suggests that there is indeed some physical coupling mechanism between the two effects that is responsible for the added enhancement when they are combined.

In the case of skin friction, freestream turbulence and roughness have opposing effects on the near wall momentum distribution. When plotted in wall units, the log region of the smooth-wall, low turbulence velocity plot is suppressed by roughness and enhanced by turbulence. As such, it would be unlikely that a synergy could be occurring in the near wall shear itself. The likely candidate for coupling is the form drag associated with the roughness elements themselves. For the panels in this study, all roughness peaks are smaller than 1/3\(^{rd}\) of the boundary layer thickness. The majority of the peaks are situated around the momentum thickness, approximately 1/10\(^{th}\) of the boundary layer thickness. In this region of a rough-wall turbulent boundary layer, the momentum is relatively depressed. Accordingly, peaks in this region of the boundary layer have a
reduced drag signature due to the lower dynamic pressure available. Since freestream turbulence has a tendency to enhance the near-wall momentum (Figure 13), the drag on roughness elements would increase synergistically. Since this form drag is such a large component of the overall c\textsubscript{f} augmentation due to roughness, this is a probable explanation for the observed synergy.

The results with heat transfer are less tractable. At 5\% Tu, the additive effect is slightly less than the synergistic effect on St (2\% less on average). This is consistent with results reported by Turner et al. [11] when 7-8\% Tu was added to their rough-surfaced cascade facility. They found the combined effect to be approximately additive, though boundary layer transition was a complicating factor that could not be isolated and removed from the results. At the higher turbulence level of 11\% in this study, however, the additive result undershoots the synergistic effect by an average of 21\%. This does not follow the stated conclusion of Bogard et al. [20] that at high turbulence levels the effects are essentially additive as well. In their flat-plate roughness study, Bogard et al. reported average turbulence levels of 13\% and the individual effects on heat transfer were ΔSt/St\textsubscript{o} ≡ 30\% for turbulence and ΔSt/St\textsubscript{o} ≡ 60\% for roughness. The additive effect from these two is 90\% while the compound effect is 108\%. Their reported synergistic effect was 100\%, essentially midway between the two computations. So, their data may show some synergy after all. Perhaps there is some threshold turbulence level at which synergy begins to be apparent.

This conjecture is supported by the ΔSt/St\textsubscript{o} results for the compound predictions in figures 16 c&d. In this case, there is actually a negative synergy (average of −12\%) between roughness and turbulence at 5\%Tu. This means that if the net effect of the two individual augmentation mechanisms is compounded, the result is greater than the effect in “real life”. Since secondary (vortical) flows have been identified as a critical feature of St augmentation in roughness, it may be that low freestream turbulence levels disrupt the natural formation of shed vorticity from roughness peaks and valleys. Whatever the explanation, the effect is clearly limited since at the higher turbulence level of 11\%, synergy is once again positive (average +6\%, as noted above).

This non-linear behavior in St augmentation between 1\%, 5\%, and 11\% freestream turbulence is evidenced when comparing efficiency factors as well. As mentioned in the introduction, previous work with regular arrays of dimples and hemispheres by Kithcart and Klett [22] showed that η increases as roughness elements become more recessed below the mean surface height. In their study, dimple arrays measured an η of 0.75 or greater whereas hemispheres with identical spacing and radius measured η ≡ 0.5. For a “real” rough surface the statistical skewness (Sk) provides a quantitative measure of the relative concentration and size of peaks and valleys. As reported in [1], a large positive skewness denotes large protrusions above the mean and is typical of surfaces with deposits or erosion. A large negative skewness denotes recesses or cavities typical of pitting or spallation.

Figure 17 shows the efficiency factors vs. skewness for all 6 roughness panels in this study. Data for each of the three levels of Tu (1\%, 5\%, and 11\%) is presented. Polynomial curve fits to each set of data are superposed on the plot. These fits exclude the data points near η ≈ 0.9 with Sk ≡ 0 as these correspond to surface #2. This surface was identified earlier as having an augmented heat transfer more typical of simulated roughness rather than “real” roughness. The curves highlight a noticeable trend to higher
η with decreasing Sk, as expected based on the Kithcart and Klett results. As anticipated, surface #2 does not follow this trend. The curves also underscore a distinct difference between the two levels of freestream turbulence. 5% Tu has virtually no effect on η while 11% Tu results in an across the board increase of 0.1 in η. Clearly, between the grid-generated low turbulence and jet-generated high turbulence, some mechanism comes to bear to alter the energy and momentum exchange in this turbulent boundary layer. More work is needed to determine if the observed behavior is a function of turbulence level, roughness type, turbulence lengthscale, or some combination of these parameters.

**Summary and Conclusions**

Heat transfer and skin friction measurements have been made on roughness panels in a low-speed, zero pressure gradient wind tunnel. The roughness panels are scaled models of actual turbine surfaces rather than the traditional simulated roughness using sand or ordered arrays of cones or spherical segments. Results indicate that this “real roughness” is distinctly different from simulated roughness. Standard roughness correlations for both St and c_f have been evaluated and new correlations are proposed in some cases. The combined effect of freestream turbulence and roughness is also evaluated in detail for these “real roughness” models. Based on the findings, the following conclusions are made:

1) Roughness effects on skin friction are 2-4 times as significant as those on heat transfer.

2) Standard correlations (e.g. from Schlichting) provide a good estimate for c_f augmentation due to roughness when the roughness is in the fully rough regime (k^+ > 70). This conclusion is based on the use of the Sigal and Danberg correlation for k_s as a function of k and Λ_s (a shape/density roughness parameter).

3) Standard correlations (e.g. from Dipprey and Sabersky) overpredict rough surface St by 10% when the roughness is in the fully rough regime (k^+ > 70). Again, k_s is calculated as a function of k and Λ_s.

4) Existing St and c_f correlations severely underpredict the effect of roughness when k^+ < 70. This discrepancy is related to the dependency of k_s on k. An alternative formulation for k_s as an exclusive function of the rms surface slope angle (α_{rms}) is proposed to replace the Sigal Danberg formulation over the range of roughness included in this study (0.5 < k/θ < 3).

5) Even when k_s is adjusted to match the Schlichting c_f correlation with the experimentally measured values, St correlations based on this k_s are still too large by 10%. This observation is true for the “real” roughness models but not for “simulated” (cone) roughness models, in which case this k_s gives an excellent St match between data and correlations. This observation highlights a distinct difference between “real” and “simulated” roughness. If shown to be more generally true, it would suggest that simulated rough surfaces with ordered roughness elements can be used to model either the heat transfer behavior or the skin friction behavior of “real” turbine roughness, but not both simultaneously.
6) Freestream turbulence effects on heat transfer are 2-3 times as significant as those on skin friction, the opposite of roughness effects.

7) When turbulence and roughness are both present, synergies are generated which create larger effects on $c_f$ and $St$ than those obtained by adding (or compounding) their individual effects. This difference can reach 20% when compared to a simply additive approach to account for both effects. An exception to this is the combined effect on $St$ at low turbulence levels. In this case, negative synergies appear to be present when both roughness and turbulence are present.

Acknowledgements

The author is indebted to numerous personnel at the four industrial partners for providing the turbine hardware. Primary among these are: Dr. Boris Glezer formerly of Solar Turbines, Dr. Ron Bunker and Mr. Paul Suttmann at General Electric, Mr. Mohan Hebbar at Siemens-Westinghouse, and Dr. William Troha and Mr. Shawn Pollock of Honeywell Corporation. The author would also like to acknowledge those who assisted in the collection of this data: Captain Jess Drab and 2Lt Christine Ellering at the Air Force Institute of Technology and Cadets Nathan Loucks and Dick Janssen of the United States Air Force Academy. In addition, collaborations with Dr. Keith Hodge and Mr. Steve McClain at Mississippi State University and Drs. Richard Rivir and Rolf Sondergaard of the Air Force Research Lab are appreciated. The testing was conducted at the Air Force Research Lab Aero-thermal research laboratory with technical support from Mr. William Nilson, Mr. Jay Anderson, and Mr. Andy Pitts. The assistance of Ms. Nikki Widmor at the University of Dayton Research Institute in determining the plastic properties is gratefully acknowledged. This work was sponsored by the US Department of Energy – National Energy Technology Laboratory through a cooperative agreement with the South Carolina Institute for Energy Studies at Clemson University. The views expressed in this article are those of the author and do not reflect the official policy or position of the United States Air Force, Department of Defense, or U.S. Government.

References


<table>
<thead>
<tr>
<th>Surface #1</th>
<th>Type</th>
<th>Scale</th>
<th>Ra (mm)</th>
<th>Rz (mm) ≅</th>
<th>Rt (mm)</th>
<th>Rq (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface #2</td>
<td>TBC Debond</td>
<td>25</td>
<td>0.30</td>
<td>1.67</td>
<td>2.01</td>
<td>0.36</td>
</tr>
<tr>
<td>Surface #3</td>
<td>TBC Spalled</td>
<td>28.1</td>
<td>0.84</td>
<td>4.51</td>
<td>6.43</td>
<td>1.04</td>
</tr>
<tr>
<td>Surface #4</td>
<td>Deposit</td>
<td>34.25</td>
<td>1.17</td>
<td>5.96</td>
<td>7.30</td>
<td>1.44</td>
</tr>
<tr>
<td>Surface #5</td>
<td>Dep/Erosion</td>
<td>62.7</td>
<td>0.46</td>
<td>3.10</td>
<td>4.05</td>
<td>0.58</td>
</tr>
<tr>
<td>Surface #6</td>
<td>Erosion</td>
<td>57.7</td>
<td>0.52</td>
<td>3.83</td>
<td>4.23</td>
<td>0.65</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Surface #1</th>
<th>Sk</th>
<th>K</th>
<th>λc (mm)</th>
<th>αrms (deg)</th>
<th>Sw/S</th>
<th>Λs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface #2</td>
<td>-2.54</td>
<td>14.56</td>
<td>6.62</td>
<td>7.23</td>
<td>1.016</td>
<td>1957</td>
</tr>
<tr>
<td>Surface #3</td>
<td>0.01</td>
<td>2.45</td>
<td>8.1</td>
<td>5.48</td>
<td>1.009</td>
<td>111</td>
</tr>
<tr>
<td>Surface #4</td>
<td>-0.65</td>
<td>3.12</td>
<td>28.6</td>
<td>16.4</td>
<td>1.08</td>
<td>111</td>
</tr>
<tr>
<td>Surface #5</td>
<td>0.11</td>
<td>2.2</td>
<td>14.5</td>
<td>20.6</td>
<td>1.224</td>
<td>35.1</td>
</tr>
<tr>
<td>Surface #6</td>
<td>0.33</td>
<td>3.00</td>
<td>4.1</td>
<td>24.1</td>
<td>1.197</td>
<td>29.3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Surface #1</th>
<th>Rz/θ</th>
<th>k⁺ (Eqn 7)</th>
<th>k⁺</th>
<th>ksa</th>
<th>ksa⁺</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface #2</td>
<td>0.57</td>
<td>0.017</td>
<td>0.68</td>
<td>0.68</td>
<td>27</td>
</tr>
<tr>
<td>Surface #3</td>
<td>0.84</td>
<td>0.011</td>
<td>0.44</td>
<td>0.47</td>
<td>18</td>
</tr>
<tr>
<td>Surface #4</td>
<td>2.26</td>
<td>1.32</td>
<td>64.2</td>
<td>3.4</td>
<td>164</td>
</tr>
<tr>
<td>Surface #5</td>
<td>2.98</td>
<td>7.95</td>
<td>427</td>
<td>7.6</td>
<td>408</td>
</tr>
<tr>
<td>Surface #6</td>
<td>1.55</td>
<td>5.24</td>
<td>279</td>
<td>7</td>
<td>372</td>
</tr>
</tbody>
</table>

Table 1: Statistics and measurements for 6 rough surfaces. Last set for Reₓ = 900,000.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Fit Type</th>
<th>Correlation Coefficient, R²</th>
</tr>
</thead>
<tbody>
<tr>
<td>αₐrms</td>
<td>polynomial</td>
<td>0.94</td>
</tr>
<tr>
<td>Sw/S</td>
<td>Linear</td>
<td>0.929</td>
</tr>
<tr>
<td>Λs</td>
<td>Logarithmic</td>
<td>0.917</td>
</tr>
<tr>
<td>Rz ≅ k</td>
<td>Linear</td>
<td>0.48</td>
</tr>
<tr>
<td>Sk</td>
<td>Linear</td>
<td>0.34</td>
</tr>
<tr>
<td>K</td>
<td>Logarithmic</td>
<td>0.24</td>
</tr>
<tr>
<td>Ra</td>
<td>Linear</td>
<td>0.21</td>
</tr>
<tr>
<td>λc</td>
<td>Logarithmic</td>
<td>0.08</td>
</tr>
</tbody>
</table>

Table 2: Correlation coefficient values for least squares fit to ksa vs. various parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Tu = 5%</th>
<th>Tu = 11%</th>
</tr>
</thead>
<tbody>
<tr>
<td>St/St₀</td>
<td>1.28</td>
<td>1.48</td>
</tr>
<tr>
<td>c%cf₀</td>
<td>1.13</td>
<td>1.20</td>
</tr>
<tr>
<td>η = (St/St₀)/(c%cf₀)</td>
<td>1.13</td>
<td>1.23</td>
</tr>
</tbody>
</table>

Table 3: St and cf augmentation ratios for smooth plate with freestream turbulence.
Figure 1: Sample 2D traces from each of the 6 roughness surfaces. Surfaces #1 to #6 ordered from top to bottom. Traces are offset vertically to eliminate overlap.

Figure 2: Schematic of Flat Plate Wind Tunnel at AFRL
Figure 3: Schematic of floating panel $c_f$ measurement apparatus on AFRL wind tunnel.

Figure 4: Typical temperature and $h$ histories during transient test.
Figure 5: Skin friction vs. Reynolds number for smooth and rough panels.

Figure 6: Stanton number vs. Reynolds number for smooth and rough panels.
Figure 7: Skin friction for rough panels compared to standard roughness correlations. Each correlation shown using $k_s$ from Eqn. 7 over all surfaces. Surfaces 1-3 also shown for $k_s = k/2$ (dashed lines). Data for $Re_s = 900,000$ only.

Figure 8: Stanton number for rough panels compared to standard roughness correlations. Each correlation shown using $k_s$ from Eqn. 7 over all surfaces. Surfaces 1-3 also shown for $k_s = k/2$ (dashed lines). Data for $Re_s = 900,000$ only.
Figure 9: Equivalent sandgrain correlation data vs. fit. Data from Bogard et al. [20] with accompanying fit (Eqn. 7). Data from “real” roughness surfaces at both Reynolds numbers with accompanying fit (Eqn. 10).

Figure 10: Equivalent sandgrain (k_{adj} as determined by matching Schlichting c_f prediction with “real” roughness data) vs. rms surface slope angle (\alpha_{rms}) with accompanying polynomial fit (Eqn. 11).
Figure 11: Stanton number for “real” roughness panels compared to Dipprey and Sabersky correlation using $k_{sadj}$ and $c_f$ from Schlichting. Data for $Re_x = 900,000$ only.

Figure 12: Stanton number for 6 “real” rough panels + cone surface (#7) compared to Dipprey and Sabersky correlation using $k_{sadj}$ and $c_f$ from Schlichting. Also, adjusted Dipprey and Sabersky correlation (Eqn. 12). Data for $Re_x = 900,000$ only.
Figure 13: Mean and fluctuating velocity profiles at roughness surface leading edge. Three levels of freestream turbulence: 1%, 5%, and 11%. Data for $Re_x = 900,000$ only.

Figure 14: Skin friction vs. Reynolds number for smooth surface at $Tu = 1\%, 5\%, & 11\%$. 
Figure 15: Stanton number vs. Reynolds number for smooth surface at Tu = 1%, 5%, & 11%. Elevated turbulence data for Re=900,000 only.

Figure 16a: Comparison of combined (synergistic) roughness/turbulence effects on $c_f$ with estimates using added and compounded individual effects of roughness alone and turbulence alone. Data for 6 rough surfaces at Tu = 5% for Re=900,000 only.
Figure 16b: Comparison of combined (synergistic) roughness/turbulence effects on $c_f$ with estimates using added and compounded individual effects of roughness alone and turbulence alone. Data for 6 rough surfaces at $Tu = 11\%$ for $Re=900,000$ only.

Figure 16c: Comparison of combined (synergistic) roughness/turbulence effects on $St$ with estimates using added and compounded individual effects of roughness alone and turbulence alone. Data for 6 rough surfaces at $Tu = 5\%$ for $Re=900,000$ only.
Figure 16d: Comparison of combined (synergistic) roughness/turbulence effects on St with estimates using added and compounded individual effects of roughness alone and turbulence alone. Data for 6 rough surfaces at Tu = 11% for Re=900,000 only.

Figure 17: Efficiency factor, $\eta = (St/St_0)/(cf/cf_0)$, vs. statistical skewness of roughness. Data for 6 rough surfaces at Tu = 1%, 5%, & 11% for Re=900,000 only. Polynomial fits to all but surface #2 data.
Rotating and Stationary Rectangular Cooling Passage Heat Transfer and Friction with Ribs, Pins and Dimples:

J.C. Han  
(Principal Investigator)  
Department of Mechanical Engineering, Texas A&M University  
Ph: (979) 845-3738 **Fax: (979) 862-2418 **E-mail: JCHan@mengr.tamu.edu

Phil Ligrani  
(Principal Investigator)  
Department of Mechanical Engineering, University of Utah  
Ph: (801) 581-4240 **Fax: (801) 585-9826 **Email: ligrani@mech.tamu.edu

and

H.C. Chen  
(Co-Principal Investigator)  
Department of Civil Engineering, Texas A&M University

December 1, 2002
# TABLE OF CONTENTS

**PROJECT SUMMARY** .................................................................................................................. 4

**Part I - Rotating Heat Transfer:** ................................................................................................... 4
**Part II - Stationary Heat Transfer and Flow Field:** ........................................................................ 5
**Part III - Computational Study:** ................................................................................................... 6

**PART I: ROTATING HEAT TRANSFER** .................................................................................... 7

**ABSTRACT** ................................................................................................................................ 8
**NOMENCLATURE** .................................................................................................................... 8
**INTRODUCTION** ..................................................................................................................... 10
**EXPERIMENTAL FACILITY** ................................................................................................. 14
**DATA REDUCTION** ................................................................................................................ 16
**RESULTS AND DISCUSSION** ................................................................................................ 18
  - Secondary Flow Behavior ............................................................................................................. 19
  - Smooth Channel Results ............................................................................................................. 20
  - Dimpled Channel Results .......................................................................................................... 22
  - Streamwise Averaged Nusselt Number Ratio ............................................................................. 24
  - Comparison with Ribbed Channel ............................................................................................. 25
**CONCLUSIONS** ....................................................................................................................... Error! Bookmark not defined.
**REFERENCES** .......................................................................................................................... 27

**PART II: STATIONARY HEAT TRANSFER AND FLOW FIELD** ............................................ 8

**ABSTRACT** .............................................................................................................................. 30
**NOMENCLATURE** .................................................................................................................. 31
**INTRODUCTION** ................................................................................................................... 33
**EXPERIMENTAL APPARATUS AND PROCEDURES** ................................................................. 38
  - Channel and test surface for heat transfer measurements ......................................................... 38
  - Local Nusselt number measurement ......................................................................................... 41
  - Friction factors .......................................................................................................................... 42
  - Time-averaged flow velocity components and pressure .......................................................... 43
Flow visualization .......................................................... 44

UNCERTAINTY ESTIMATES ................................................. 45

EXPERIMENTAL RESULTS AND DISCUSSION ......................... 46
  Baseline Nusselt numbers .................................................. 46
  Spatially-resolved distributions of local Nusselt numbers ................................. 46
  Local instantaneous and time-averaged flow structure .................................... 49
  Spatially-averaged Nusselt numbers ........................................... 51
  Globally-averaged Nusselt numbers and friction factors .............................. 53
  Performance parameters ..................................................... 55

SUMMARY AND CONCLUSIONS ........................................... 56

REFERENCES ........................................................................... 58

PART III: NUMERICAL PREDICTION ........................................... 64

ABSTRACT ............................................................................... 65

NOMENCLATURE ..................................................................... 65

INTRODUCTION ......................................................................... 66
  Motivation ........................................................................... 66
  Literature Review: Experimental Studies ........................................... 67
  Literature Review: Numerical Studies ............................................ 68

DESCRIPTION OF PROBLEM ................................................ 70

COMPUTATIONAL PROCEDURE .............................................. 71
  Overview: ........................................................................... 71
  Chimera RANS Method: ..................................................... 71
  Boundary conditions: ................................................................ 72
  Computational grid details: .................................................. 72

RESULTS AND DISCUSSION .................................................. 73
  Velocity and Temperature Fields: ........................................... 73
  Detailed Local Heat Transfer Coefficient Distribution: ............................ 77
  Spanwise-Averaged Heat Transfer Coefficients and Comparison with Experimental Data ............................................ 80

CONCLUSIONS ......................................................................... 82

REFERENCES ............................................................................. 84
ONGOING WORK ....................................................................................................................... 89

Part I: Rotating Heat Transfer .................................................................................................... 89
Part II: Stationary Heat Transfer and Flow Field ................................................................. 89
Part III: Numerical Prediction ................................................................................................. 90

APPENDIX I: FIGURES FOR PART 1 ....................................................................................... 92

APPENDIX II: FIGURES FOR PART 2 ................................................................................... 104

APPENDIX III: FIGURES FOR PART 3 .................................................................................. 117
PROJECT SUMMARY

The objective of this three-part investigation is to provide industry with much needed heat transfer data to assist the designers in improving the cooling performance and thermal efficiency of power generation and industrial turbine engines. More specifically, this investigation ventures into the heat transfer phenomenon of internal cooling channels near the trailing edge of a turbine blade, which has yet to be reported in detail. The research is divided into three parts: Part I - Rotating Heat Transfer; Part II - Stationary Heat Transfer; Part III - Numerical Prediction. This investigation is a collaboration between Dr. J.C. Han and Dr. H.C. Chen of Texas A&M University and Dr. Phil Ligrani of the University of Utah. This report details the second stage of this investigation, namely the channel of aspect ratio \( AR = 4:1 \). This investigation also concerns heat transfer investigations into a channel of \( AR = 8:1 \) to be completed subsequent to the \( 4:1 \) investigation presented in this report. A more detailed breakdown of the investigation of each of the three parts follows.

Part I - Rotating Heat Transfer:

The objectives of part I are to obtain experimental data from rectangular, internal cooling passages with aspect ratios of 4:1 and 8:1. The following parameters will be varied: (1) Surface geometry (2) Reynolds number, (3) rotation number, (4) rotation angle, and (5) channel aspect ratio. Ribs, pins, and dimples will be installed on the leading and trailing sides of a rectangular internal cooling passage with rotation. The ratio of inlet coolant temperature to surface temperature (TR) will be around 0.8 - 0.9. The experiments is designed so that regionally averaged heat transfer coefficients will be measured at different locations along the cooling
passages under rotating conditions. Both the streamwise and spanwise distribution of the heat transfer enhancement will be obtained. The new heat transfer data will be correlated and compared with numerical predictions in part III. The existing rotating facility and instrumentation available in the Turbine Heat Transfer Laboratory of Texas A&M University is used in this study.

Part II - Stationary Heat Transfer and Flow Field:

The objectives of Part II are to obtain experimental data from rectangular, internal cooling passages with aspect ratios of 4:1 and 8:1. The following parameters will be varied: (i) surface geometry, (ii) ratio of absolute inlet temperature to wall temperature, (iii) Reynolds number, and (iv) channel aspect ratio. Ribs, pin fins, dimples and smooth surfaces will be installed on the surface of a rectangular internal flow passage (without rotation). The ratio of inlet coolant temperature to surface temperature (TR) will range from 0.6 to 1.0. The experiments will be designed so that: (a) spatially-resolved surface heat transfer coefficients will be measured at different location along the instrumented surface which contains the concavities, and (b) detailed flow structure above the different surface configurations will be measured using existing five-hole pressure probes, flow visualization apparatus, and pressure transducer equipment. Spatially-averaged data will be deduced from the spatially-resolved data. The flow structure data will aid numerical model development in Part III. Two existing test facilities, available in the Convective Heat Transfer Laboratory of the University of Utah, are employed.
Part III - Computational Study:

The objectives of part III are to predict flow and heat transfer behaviors from rectangular, internal cooling passages with aspect ratios of 4:1 and 8:1. An ongoing Chimera Reynolds-Averaged Navier-Stokes (RANS) code together with an advanced state of the art second-order Reynolds stress (second moment) turbulence model will be used for the prediction of rotating and stationary rectangular cooling channels with ribs, pins or dimples. The present numerical model has been tested to provide much better flow and heat transfer predictions than the standard \( k-\varepsilon \) turbulence model for rotating multi-pass square channels. The numerical predictions will be calibrated/compared with the part I-rotation at \( TR = 0.9 \) and with the part II-stationary at \( TR = 0.6 \). The ultimate goal is to predict flow and heat transfer in rotating rectangular channels with ribs (pins or dimples) at very high Reynolds number and buoyancy parameter conditions.
PART I: HEAT TRANSFER IN ROTATING RECTANGULAR COOLING CHANNELS (AR=4) WITH DIMPLES

Part 1 is conducted at Texas A&M University under the direction of Dr. J.C. Han
ABSTRACT

The world of research seeks ways of improving the efficiency of turbomachinery, attention has recently focused on a relatively new type of internal cooling channel geometry, the dimple. Preliminary investigations have shown that the dimple enhances heat transfer with minimal pressure loss. An investigation into determining the effect of rotation on heat transfer in a rectangular channel (aspect ratio = 4:1) with dimples is detailed in this paper. The range of flow parameters includes Reynolds number (Re = 5000-40000), rotation number (Ro = 0.04-0.3) and coolant to wall density ratio (∆ρ/ρ=0.122). Two different surface configurations are explored, including a smooth duct and dimpled duct with dimple depth-to-print diameter (δ/Dp) ratio of 0.3. A dimple surface density of 10.9 dimples/in² was used for each of the principal surfaces (leading and trailing) with a total of 131 equally spaced hemispherical dimples per surface; the side surfaces are smooth. Two channel orientations of β = 90° and 135° with respect to the plane of rotation are explored to determine channel orientation effect. Results show a definite channel orientation effect, with the trailing-edge channel enhancing heat transfer more than the orthogonal channel. Also, the dimpled channel behaves somewhat like a 45° angled rib channel, but with less spanwise variations in heat transfer.

NOMENCLATURE

- \( A_p \) surface area of copper plate (m²)
- \( AR \) aspect ratio
- \( D \) hydraulic diameter (m)
- \( D_p \) dimple print diameter (m)
- \( e \) rib height (m)
- \( h \) heat transfer coefficient (W/m² K)
\( k \) thermal conductivity of coolant (W/mK)

\( L \) length of duct (m)

\( Nu \) regionally averaged Nusselt number, \( hD/k \)

\( Nu_o \) Nusselt number in fully-developed turbulent

\( Pr \) Prandtl number

\( Q \) heat transfer (W)

\( q_{net} \) net heat flux at wall based on projected area (W/m\( ^2 \))

\( R \) mean rotating radius (m)

\( Re \) Reynolds number, \( \rho VD/\mu \)

\( R_o \) rotation number, \( \Omega D/V \)

\( T_{bi} \) inlet coolant bulk temperature (K)

\( T_{bx} \) local coolant bulk temperature (K)

\( T_w \) wall temperature (K)

\( V \) bulk velocity in streamwise direction (m/s)

\( \beta \) angle of channel orientation

\( \delta \) dimple depth

\( \Omega \) rotational speed (rad/s)

\( \rho \) density of coolant (kg/m\( ^3 \))

\( \Delta \rho /\rho \) coolant-to-wall density ratio, \( (T_w - T_{bi})/T_w \)
INTRODUCTION

Extensive research efforts have recently focused on methods for reducing the consumption of non-renewable energy resources. The turbomachinery industry is one industry committed to improving the efficiency of its equipment. Gas turbines are used for a wide variety of applications including power generation, gas compression, and jet propulsion. The efficiency of a turbine can be increased by increasing the combustion temperature or decreasing the amount of compressor dilution air used to cool the extremely hot gas exiting the combustor. This poses a major problem in the hastened degradation of temperature sensitive components of the turbine, principally the turbine blades and vanes. To counter the high turbine inlet temperatures (1600-1800K), the physics of turbulent heat transfer under rotation are investigated in a cooling model. Turbine blades incorporate internal cooling passages to extract the thermal energy absorbed from the hot combustion gases. This prolongs the life of the blade as well as allowing for increased combustion temperatures, which ultimately increases performance of the turbine.

A small amount of pressurized air is extracted from the compressor and injected into the turbine blades via the cooling air bypass. This relatively low enthalpy gas is forced through the internal cooling passages of the turbine blades, convectively extracting heat from the internal walls. For further thermal protection of the blade, a portion of the internal cooling air is ejected through tiny holes in the walls and tip of the blade, creating a cool film thermal boundary.

When considering the effects of rotation, certain flow phenomena are exhibited that are not observable in the stationary reference frame. Forces are generated under rotation, principally the Coriolis and buoyancy forces. These forces generate secondary flows in the plane orthogonal to the mean flow direction. For radial outward flow, the Coriolis and buoyancy forces combine to shift the velocity profile toward the trailing surface. The coolant flow migrates along with the heat transfer augmentation toward the trailing surface. This rotationally induced migration of the
cooler core flow results in the advantageous enhancement of heat transfer at the trailing surface, but it is typically balanced by the disadvantageous reduction in heat transfer from the leading surface. As with most temperature sensitive components, thermal failure in an isolated region is oftentimes just as problematic as failure of the entire component. This is why it is important to analyze the heat transfer phenomenon segment by segment along the length of the blade.

The aspect ratio of the channel also has a profound impact on the effect of rotation. Moving from the mid-chord to the trailing edge of the blade, the channels must become more rectangular as the blade becomes thinner. The orientation of a 4:1 aspect ratio cooling channel in a gas turbine blade is shown in Figure 1.1. This thinning of the channel changes the effective secondary flow pattern from that of a square duct. For this reason, one cannot simply apply the knowledge of the rotationally induced flow patterns in a square channel to that of a rectangular channel. Therefore, an investigation of the rectangular channel is necessary to further understand the heat transfer characteristics of the internal cooling channels in a gas turbine blade.

To promote heat transfer in the internal cooling passages, various types of turbulators are used to trip the boundary layer. Ribs or “trip-strips” and pin-fins have been commonly used in gas turbines over the past decade. The dimple is a relatively new approach, investigated first by the Russians. When combining the effects of tripping the boundary layer (using ribs, pins, or dimples) and rotational forces (Coriolis and buoyancy), entirely different turbulence and flow phenomenon are achieved. Combining into this equation the various shapes and sizes of internal cooling channels, it is clear that there is no one single solution that can be applied universally in the field of turbine heat transfer. For this reason, an experimental investigation into each combination of the previously mentioned parameters is necessary.
Numerous studies on turbulent flow and heat transfer in the cooling channels of a gas turbine blade have been performed in the past. Han and Park [1] published experimental investigations of the heat transfer phenomenon in a stationary rib roughened rectangular channel. Wagner et al. [2,3] conducted detailed experimental investigation to determine the effects of rotation, or more specifically the effects of Coriolis and buoyancy forces on the regionally averaged heat transfer distribution of a serpentine square channel with smooth walls. This study determined that in the first pass, the effect of rotation created a thinner boundary layer on the trailing surface and a thicker boundary layer on the leading surface.

Parsons et al. [4] studied the effects of channel orientation and wall heating condition on the regionally averaged heat transfer coefficients in a rotating two-pass square channel with ribbed walls. Parsons et al. [4] discovered that the heat transfer enhancement for the constant wall heat flux boundary condition was more pronounced when the duct is twisted 45° to the plane of rotation when compared to a channel oriented orthogonal to the plane of rotation. Johnson et al. [5] determined that the model orientation with respect to the rotation plane greatly affected the heat transfer distribution.

Dutta and Han [6] investigated the regionally averaged heat transfer coefficients in a rotating two-pass square channel with three different model orientations. They found that the orientation of the channel with respect to the plane of rotation affected the heat transfer distribution. More specifically, they determined that orienting the channel at an angle with respect to the plane of rotation reduced the effect of rotation when compared to the orthogonal channel orientation.

Until recently, most of the experimental studies have explored only square ducts. However, it is quite common to find rectangular cooling passages, particularly toward the trailing edge of a gas turbine blade. Since the profile of a turbine blade is curved, the exclusive use of square
channels is not practical. Past research focused mainly on the square channel; therefore, published data for a rectangular cooling channel is rare.

Willett and Bergles [7] performed a detailed investigation of the heat transfer in a narrow, 10:1 smooth rectangular channel oriented at 60° to the r-z plane. Most of their focus dealt with exploring the contribution of buoyancy forces under rotation. They found that the duct orientation induced a significant heat transfer gradient in the spanwise direction. It was also found that the normalized Nusselt number at the far-aft-end of the trailing side (or the trailing-outer equivalent in this paper) is a strong function of rotation number and buoyancy number.

Griffith et al. [8] studied the effects of rotation on a AR=4:1 rectangular channel with β=90° and 135°. They determined that the rib induced secondary flow dominated the rotation-induced vortices, particularly at lower rotation numbers. They also found that significant spanwise heat transfer gradients exist for both channel orientations.

Dimpled channel literature is even less common than rectangular channel publications. Chyu et al. [9] investigated hemispheric and teardrop shaped concavities in a stationary channel using a liquid crystal technique. They found that the overall performance was nearly equal for the geometries, showing enhancement of approximately 2.5 times that of a smooth duct for the stationary case.

Moon et al. [10] analyzed a stationary dimpled channel using a liquid crystal technique. They found that the heat transfer enhancement occurs mostly outside of dimples. They also found that the enhancement is reduced in the upstream portion of dimple and the enhancement is increased in the downstream portion (rim) of dimple. In addition, they determined that enhancement not a function of Reynolds number, and is typically constant at around 2.1.
Mahmood et al. [11] investigated a stationary dimpled channel using infrared thermography and smoke-stream flow visualization. They found that a large upwash region occurs in the center of the dimple, and pairs of vortices are shed at the dimple diagonals. They also determined that the enhancement principally occurs on the plateau (or flat) surface.

Considering that the effect of rotation has shown to significantly influence the heat transfer enhancement of a cooling channel, it is of interest to explore the rotational effects on the heat transfer in a dimpled channel. The following questions arise:

1.) How does the spanwise heat transfer distribution vary within a dimpled, rotating, rectangular channel?
2.) How does the dimpled channel compare to a smooth and ribbed channel?
3.) What is the effect of the channel orientation with respect to the plane of rotation?
4.) How do rotational forces quantitatively influence the heat transfer enhancement in a dimpled rectangular channel?

Answers to these questions are pursued in this paper.

**EXPERIMENTAL FACILITY**

The experimental test rig previously used by Dutta and Han [6] is utilized in this investigation. A variable frequency motor is connected via a gear-and-belt mesh to a hollow, rotating shaft. This shaft runs from the base of the test rig to the work platform and is attached orthogonal to the hollow, rotating arm. The test section is inserted inside the hollow rotating arm, which rotates in a plane orthogonal to the rotating shaft. The mean rotating radius to hydraulic diameter ratio is R/D=33. Thermocouple and heater wires are connected to a 100-channel slip-
ring assembly mounted to the rotating shaft. The output of the thermocouples is transferred to a data logger. Power is applied to the heaters from the variac transformers, which is also transmitted through the slip ring assembly. Cooling air is pumped from a steady flow compressor, through an ASME orifice flow meter, then through the hollow rotating shaft, turning 90° and passing into the rotating arm, then through the test section and is finally expelled into the atmosphere.

The test section is a 0.5 inch by 2 inch (1.27x5.08 cm) one-pass rectangular channel of aspect ratio 4:1. The direction of airflow is radially outward from the axis of rotation. Two rows of copper plates are installed on both the leading and trailing surface to provide a grid for analysis of the spanwise variation in the regionally averaged heat transfer coefficient.

Figure 1.2 shows a detailed top view of the test section. The test section is divided into six cross-sections, each with six copper plates: two for the leading, two for the trailing, one for the outer and one for the inner surface. Moving along the direction of the flow (radially outward), there are six streamwise segments for a total of 36 copper plates in the entire test section. The channel length-to-hydraulic diameter ratio \((L/D)\) is 7.5 with a ratio of 1.25 for each of the six cross-section segments. Each plate is separated by a 0.0626 inch (0.159 cm) thin strip of nylon to prevent heat conduction between plates. This is important since the objective is to study the spatial distribution of heat transfer.

The copper plates are mounted in a nylon substrate, which comprises the bulk of the test section. Acid etched thin foil flexible heaters encased in a high temperature polymer (kapton) are installed beneath the leading and trailing surfaces, two to each surface. The outer and inner walls (or side walls) are smooth and each are heated by a wire-wound resistance heater, which is also installed beneath the copper plates. Sufficient power is supplied in order to maintain a maximum
wall temperature of nearly 340K for the corresponding section. The maximum wall temperature, however, will vary between 338-344K in order to maintain the coolant to wall density ratio at 0.122. Thermal conducting paste is applied between the heater and copper plates to promote heat transfer from the heater to the plate. Each plate has a blind hole drilled on the backside in which the thermocouple is installed with thermal conducting glue.

Two different surface configurations (smooth and dimpled $\delta/D_p = 0.3$) are studied as well as two different channel orientations with respect to the direction of rotation ($\beta = 90^\circ$ and $135^\circ$). Figure 1.2 shows the dimpled surface configuration. The dimples are machined using hemispherical end-mills. The experiments were conducted for Reynolds numbers of 5000, 10000, 20000 and 40000. The test section rotates at a constant speed of 550 rpm, resulting in a range of rotation number ($Ro$) from approximately 0.04-0.3.

**DATA REDUCTION**

This investigation focuses on detailing the regionally averaged heat transfer coefficient at various locations within the internal cooling channel. This heat transfer coefficient is determined by the net heat flux from the heated plate to the cooling air, the surface area of the plate ($A_p$), the regionally averaged temperature of the plate, and the local bulk mean air temperature by the following:

$$h = \frac{q_{net}}{T_w - T_{b,x}}$$

The net heat flux is calculated using the measured voltage and current supplied to the heater multiplied by the area fraction exposed to the respective plate minus the previously determined amount of heat losses due to external conduction, convection, and radiation energy.
escaping from the test section. This heat loss calibration is performed for both stationary and rotation experiments with a piece of insulation inserted inside the test section to inhibit natural convection. For this calibration, by knowing the amount of power supplied to the heater and measuring the temperature of the plate, it is possible to determine how much the heat is being lost into the environment using the conservation of energy principle. Equation 1 is used throughout the experiment, neglecting the change of area effect with the addition of dimples. That is, the heat transfer coefficient is calculated based on the projected area, neglecting the 19.3% increase in area due to the addition of dimples.

The regionally averaged wall temperature \( T_w \) is measured directly by the thermocouple installed in the back of each plate. The local bulk mean air temperature \( T_{b,x} \) is determined by a linear interpolation between the measured bulk air inlet and the average of two outlet temperatures (each installed at the midpoint of the two spanwise sections) due to the applicable constant heat flux assumption. Another method used to check the interpolation values is by performing an energy balance. It is reassuring to note that performing an energy balance to calculate the expected outlet temperature resulted in a close match to that of the average measured exit temperature value, typically to within 5%. Therefore the linear interpolation method is validated and is the method used in the calculation of the results presented in this paper. The energy balance equation is:

\[
T_{b,i} = T_{in} + \sum_i (q - q_{loss})/\dot{m}c_p, \quad i = 1, 2, \ldots, 6
\]
To provide a common reference for each analysis, a correlation is used comparing the Nusselt number for the specific duct case to that of fully developed flow through a smooth stationary pipe at the same Reynolds number. For this investigation, the Dittus-Boelter correlation for heating \((T_w>T_{bx})\) is used:

\[
\frac{Nu}{Nu_o} = \frac{hD}{k_{air}} \frac{1}{(0.23 \text{Re}^{0.8} \text{Pr}^{0.4})}
\]

(3)

All air properties are taken based on the mean bulk air temperature with the Prandtl number \((Pr)\) for air as 0.71.

Overall uncertainty for the regionally averaged heat transfer coefficient is predominantly dependent upon the difference between the wall temperature and the bulk air temperature, the net heat flux input and the ability to maintain a steady mass flow rate. As with most experiments, the uncertainty for this investigation decreases with the increasing magnitude of input parameters. For higher Reynolds numbers, the uncertainty has been determined to be nearly 7%. However, for lower Reynolds numbers (\(Re=5000\)), the uncertainty could be as much as 20%. The uncertainty analysis was performed using the Kline and McClintock [12] uncertainty analysis procedure.

**RESULTS AND DISCUSSION**

The surface labeling scheme, seen in Figure 1.3, will be used throughout this paper. The inner and outer surface side walls are named according to their location in the turbine blade. That is, the inner surface is closer to the mid-chord position of the blade (a relatively internal position), and the outer surface is closer to the trailing edge of the blade, and thus is closer to an external surface of the blade. The leading and trailing surfaces of the blade follow the
conventional definitions of these surfaces, however each surface is subdivided into two surfaces in order to investigate the span-wise distribution of heat transfer along the major surfaces (leading and trailing). Therefore we have a total of six surfaces: leading-outer, leading-inner, trailing-outer, trailing-inner, outer, and inner. A brief discussion on the secondary flow pattern induced by a dimple is presented below.

**Secondary Flow Behavior**

*Figure 1.4* shows a conceptualization of the secondary flow patterns over a dimpled surface. As the flow approaches the upstream portion of the dimple, flow separation occurs, and a recirculation zone appears in the upstream portion of the dimple resulting in mitigation of the heat transfer. As the flow reattaches at the downstream half of the dimple, an increase in the heat transfer enhancement occurs. Continuing in the streamwise direction, it has been shown that a large upwash region is produced by the dimple. This upward directed flow mixes to some degree with the cold core mainstream flow. Finally, pairs of vortices are shed along the dimple diagonals, enhancing the heat transfer on the flat portion of the surface. Considering the dimple induced secondary flows and superimposing the rotation induced secondary flow upon it, it is apparent that there is no primarily constructive combination of the two, as was shown in the case of a 45° angled-rib rotating channel investigated by Griffith et al. [8]. This complex combination of the dimple induced vortices in various directions with the rotation induced secondary flow does not generate any vision of a primary coherent flow structure, however the heat transfer enhancement is still increased at the trailing surface due to the thinning of the boundary layer under rotation.

*Figure 1.5* shows a conceptualization of the secondary flow induced by rotating a smooth, rectangular channel. The Coriolis force induces two counter rotating vortices, which serve to
push the colder fluid closer to the trailing surface. When the channel is twisted such that \( \beta = 135^\circ \), the linear distance of the Coriolis force main vector increases from a relatively small distance (as in the case of \( \beta = 90^\circ \)) to a much longer distance. The Coriolis vector now traverses the diagonal from the leading most corner to the trailing most corner of the twisted channel. This shifting of the rotation induced vortices serves to mix the flow more effectively.

**Smooth Channel Results**

Figures 1.6-1.8 contain the smooth duct data for three different channel configurations: stationary, rotation with \( \beta = 90^\circ \) and rotation with \( \beta = 135^\circ \). Each case is subdivided into four experiments: (a) \( \text{Re}=5000 \), (b) \( \text{Re}=10000 \), (c) \( \text{Re}=20000 \), and (d) \( \text{Re}=40000 \). The corresponding rotation numbers for these cases are 0.305, 0.151, 0.075 and 0.038 respectively. Please reference Figure 1.3 for the data legend and surface locations within the channel. Figure 1.6 contains data for the stationary cases. The initial decrease in the normalized Nusselt number plots is attributable to the entrance effect in thermally developing flow. The plots all approach a horizontal asymptote as the flow approached the thermally fully developed state.

Figure 1.7 shows the results for the rotation cases where the duct is oriented at \( \beta = 90^\circ \), that is, orthogonal to the plane of rotation. As was expected, the trailing surfaces exhibit higher heat transfer enhancement than the leading surfaces due to the migration of the colder core fluid toward the trailing surface caused by the Coriolis rotational forces. At a duct angle of \( \beta = 90^\circ \), the channel can be assumed to hold symmetry about the plane of rotation. This means that both of the leading surfaces (leading-outer and leading-inner) should have identical Nu plots, the trailing surfaces should exhibit identical behavior, and the two side surfaces should be equal. This is validated relatively well as seen in the figures, with a slight bias between the two trailing surfaces. An increase in the Reynolds number tends to suppress the effect of rotation. All six
surfaces show very little streamwise variation in the Nu number plots. Both of the side surfaces (inner and outer) have a heat transfer enhancement nearly equal to the value of the two trailing surfaces.

**Figure 1.8** presents the results of the smooth rotation case with the channel oriented at $\beta=135^\circ$ with respect to the plane of rotation. Figure 1.8a shows that at a low Reynolds number (high rotation number), there are distinguishable differences in the heat transfer trends among the various surfaces. It can be seen that the trailing-outer and outer surfaces exhibit the highest heat transfer enhancement of all of the surfaces in the duct. This is attributed to the fact that these two surfaces are the primary recipients of the shifting of the cooler core flow under rotation. This phenomenon is illustrated in Figure 1.5 of the preceding section. After the flow impinges on the trailing-outer and outer surfaces, it passes along the leading and trailing surfaces to the inner surface, where the heat transfer coefficient is the lowest, and the secondary flow slows down dramatically. Then the flow cycles again, passing from the leading most corner diagonally across the channel toward the trailing most corner. At a high rotation number, the inner surface heat transfer follows a trend quite similar to the stationary cases. It appears that this inner surface is barely affected by rotation. Both of the trailing surfaces have higher heat transfer coefficients than the leading surfaces. A new and interesting finding is the substantial difference in the heat transfer coefficient between the two trailing surfaces. Furthermore, this span-wise difference does not come into effect until nearly half-way through the channel for high rotation numbers ($\text{Ro}=0.305$). It is also shown that the leading surface heat transfer increased when compared to the orthogonal channel. The overall increase in heat transfer from nearly all surfaces can be attributed to the fact that twisting the channel greatly increased the linear distance along which the main Coriolis force is directed (from leading most to trailing most corner) and provides an
overall better mixing than the $\beta=90^\circ$ case. In the $\beta=90^\circ$ case, the principal Coriolis vector in the core region of the flow acts across only a short distance (the short width of the channel) and does not serve to mix the flow as well as the twisted channel.

One evident contrast of the results of the $\beta=135^\circ$ case (Figure 1.8) compared to the $\beta=90^\circ$ (Figure 1.7) case is apparent in the side surfaces. For the twisted channel, the trend of the outer surface increases while the inner surface trend decreases with $X/D$. Furthermore, the inner surface decreases in a similar way as seen in the stationary case. The outer surface, which trails after the inner surface, experiences a heat transfer enhancement of as much as three times that of the inner surface for the $\beta=135^\circ$ case. This is due to the shift of the primary Coriolis induced flow vector from the center of the trailing surface in the $\beta=90^\circ$ case to the trailing most corner in the $\beta=135^\circ$ case. This trailing most corner is adjacent to the outer surface, and therefore the outer surface benefits greatly in heat transfer enhancement due to the twisting of the duct. This is desirable since the outer surface of the $\beta=135^\circ$ case is closer to the trailing edge of the turbine blade, and thus is likely to experience a higher external heat flux than the inner surface. The inner surface interfaces with the side surface of the adjacent cooling passage, and therefore is less likely to be considered a critical surface.

**Dimpled Channel Results**

The data plots for the dimpled channel cases are presented in Figures 1.9-1.11. Figure 1.9 shows the stationary dimpled channel results. An enhancement of approximately 2.0 is produced by the dimpled surface. This is in close agreement with the results of Moon et al. [10]. The smooth side surfaces (inner and outer) appear to only benefit marginally from the mixing induced by the dimpled surfaces. It is shown that for Reynolds numbers 10000, 20000, and
40000, there appears to be almost no dependence upon Reynolds number. This observation was also made in the past by Moon et al. [10]. However, when the data for Re=5000 is considered, we see the emergence of Reynolds number dependence, with the enhancement decreasing with decreasing Reynolds number, but only at the lowest Re value. Perhaps this is because the Reynolds number is much closer to the laminar-to-turbulent flow transition region of Re≤2300. Whatever the reason, such a low Reynolds number is not encountered in gas turbines, therefore consideration of this Reynolds number effect is only necessary for those wishing to consider the use of dimples for some other application outside of gas turbine heat transfer.

Figure 1.10 presents the data for the dimpled channel under rotation, orthogonal to the plane of rotation (β = 90°). It is apparent that there is a definite enhancement due to rotation for the trailing surfaces, which increases with increasing rotation number. Symmetry is achieved relatively well between the two spanwise segments of each dimpled surface, and symmetry between the two side surfaces. Also, the side surfaces experience enhancement equal to the leading surface. This is completely different than the smooth case, where the side surfaces were more equal to the trailing surface.

Figure 1.11 shows the results for the dimpled channel under rotation, twisted with respect to the plane of rotation (β = 135°). The trailing-outer surface is enhanced more than the other surfaces, as it benefits from both the shifting of the cold flow toward the outer half of the channel, as well as the local mixing induced by the vortices shed by the dimples. In addition, the trailing-inner and leading-outer surfaces are enhanced (although to a lesser degree) by rotation. This occurs as the Coriolis vortex also serves to distribute some of the cold fluid in the core of the mainstream flow to the other surfaces, allowing the smaller scale dimple induced vortices to capture some of this cold fluid and pull it near to the wall. The outer surface is enhanced more
than the inner surface due to the shifting of the majority of the colder flow toward the outer half of the twisted channel under rotation. This behavior was also seen in the smooth duct.

Streamwise Averaged Nusselt Number Ratio

Averaging the streamwise data for each surface provides a method of comparing the surfaces and the effect of rotation. Figures 1.12 and 1.13 present the streamwise averaged data vs. Rotation number for the orthogonal and twisted channel. The solid line plots are for the dimpled channel and the dotted line plots are for the smooth channel.

Figure 1.12 presents the data for the orthogonal ($\beta = 90^\circ$) dimpled and smooth channel. From this plot, it can be seen that the dimpled trailing surfaces show greater dependence on rotation number than all other surfaces, with nearly 100% improvement in enhancement from the stationary to the highest rotation number case. The dimpled leading surfaces show very little dependence on rotation number due to the stable, thick boundary layer on the leading surface. The slight non-symmetry between the two leading surfaces is due to experimental uncertainty. A most interesting issue arises when comparing the side surfaces (inner and outer) of the dimpled channel to those of the smooth channel. It can be shown that while the side surfaces of the dimpled channel initially experience a higher enhancement without rotation, the smooth channel side surfaces show greater dependence on rotation than the dimpled channel side surfaces. This is possibly attributable to the disruption of the Coriolis vortices by the dimples. For the smooth surface, the Coriolis vortex passes from the center of the channel toward the side surfaces (see Figure 1.5), where it enhances the heat transfer from the side surfaces. The secondary flow generated by the dimple has no single principal direction, and likely serves to reduce the
intensity of the Coriolis vortices. Because of this, the effect of rotation is reduced for the side walls of the dimpled channel.

**Figure 1.13** shows the streamwise averaged data vs. Rotation number for the twisted (β =135°) dimpled and smooth channel. The dimpled trailing-outer surface shows the strongest dependence on rotation number and is clearly the primary recipient of enhancement for the twisted channel under rotation. In addition, the dimpled leading-outer and trailing-inner surfaces now show a moderate to high dependence on rotation number. This was also explained in the discussion of figure 1.11 where it was noted that the smaller scale vortices shed by the dimple are able to capture some of the lower enthalpy fluid, which is better distributed by the Coriolis vortices for the twisted channel. Again, it is noticed that the side surfaces of the twisted channel are enhanced less by rotation than those of the smooth duct due to the disruption of the Coriolis vortices by the vortices shed by the dimple.

**Comparison with Ribbed Channel**

When the results of this investigation are compared with the ribbed channel investigated by Griffith et al. [8], we can see that the dimpled channel behaves somewhat similarly to the ribbed channel of the same geometrical and flow parameters. Some exceptions include the absence of spanwise variations for the orthogonal (β =90°) dimpled channel under rotation. Spanwise variations were quite significant in the case of the ribbed rotating channel due to the 45° rib-angle effect. Also, symmetry is achieved between the two side surfaces (inner and outer) for the orthogonal dimpled channel, which again was not the case for the orthogonal 45° ribbed channel. For the twisted channel (β =135°), the dimple induced secondary flow does not dominate the Coriolis vortices quite as much as the ribbed channel, which was shown to dominate the rotation
induced secondary flow. This is evident in the emergence of the dimpled trailing-outer surface as the primarily enhanced surface for the twisted dimpled channel; the ribbed channel exhibited rib-dominance by producing identical behavior for the leading-outer and trailing outer surface of the twisted channel. From this observation, coupled with the overall slightly lower heat transfer enhancement of the dimpled channel, it can be determined that the dimpled channel does not produce as strong of a secondary flow pattern as the 45° angled-rib channel. However some of the geometrical parameters are possibly not of equivalent value, such as $e/D=0.1$ for the 45° angled-rib duct and $\delta/Dp=0.3$, and $\delta/D=0.094$ for the dimpled channel.

**CONCLUSIONS**

1.) Spanwise variations in heat transfer enhancement of the dimpled rotating rectangular channel exist only for the twisted ($\beta=135^\circ$) orientation.

2.) The dimpled channel behaves somewhat similarly to the ribbed channel. The dimpled channel however, does not dominate the rotational effects quite as significantly as in the ribbed channel. It appears that the dimpled channel promotes smaller scale, and more complex mixing vortices than the ribbed channel.

3.) The twisted ($\beta=135^\circ$) dimpled channel experiences greater overall enhancement than the orthogonal dimpled channel ($\beta=90^\circ$).

4.) For ($\beta=90^\circ$), enhancement at the trailing surfaces increases by almost 100% from the stationary to highest rotation number case. Also, the leading surfaces show little dependence on rotation number. The side surfaces show slightly less dependence on rotation than those of a smooth duct.

5.) For ($\beta=135^\circ$), enhancement at the trailing-outer surface increases by more than 100% from the stationary to highest rotation number case. In addition, the trailing inner and leading outer
surfaces experience nearly identical enhancement, increasing by more than 50% from the stationary to highest rotation number case. The outer surface increases by nearly 100% and the inner surface slightly decreases from the stationary to highest rotation number case.

ACKNOWLEDGMENTS

The Advanced Gas Turbine Systems Research (AGTSR) program (project number SR-082) funded this experimental investigation under the supervision of the United States Department of Energy (DOE). The authors greatly appreciate the support of the AGTSR program and the DOE. Dr. H.K. Moon of Solar Turbines, Inc. and Dr. P. Ligrani of the University of Utah provided direction regarding the dimple concept. Without such support, this research would not have been possible.

REFERENCES


PART II: STATIONARY HEAT TRANSFER AND FLOW FIELD

Part 2 is conducted at the University of Utah under the supervision of Prof. Phil Ligrani.
ABSTRACT

Spatially-resolved Nusselt numbers and flow structure are presented for a stationary channel with an aspect ratio of 4 and angled rib turbulators inclined at 45° with perpendicular orientations on two opposite surfaces. The flow structure results include time-averaged distributions of streamwise vorticity, streamwise velocity, total pressure, and static pressure, surveyed over flow cross-sectional planes, as well as flow visualization images and friction factors. Results are given at different Reynolds numbers based on channel height from 270 to 90,000. The ratio of rib height to hydraulic diameter is .078, the rib pitch-to-height ratio is 10, and the blockage provided by the ribs is 25 percent of the channel cross-sectional area. Spatially-resolved local Nusselt numbers are highest on tops of the rib turbulators, with lower magnitudes on flat surfaces between the ribs, where regions of flow separation and shear layer re-attachment have pronounced influences on local surface heat transfer behavior. Also important are intense, highly unsteady secondary flows and vortex pairs, which increase secondary advection and turbulent transport over the entire channel cross-section. The resulting augmented local and spatially-averaged Nusselt number ratios (rib turbulator Nusselt numbers normalized by values measured in a smooth channel) generally increase on the rib tops as Reynolds number increases. Nusselt number ratios decrease on the flat regions away from the ribs, especially at locations just downstream of the ribs, as Reynolds number increases. Globally-averaged Nusselt number ratios vary from 3.36 to 2.82 as Reynolds number increases from 10,000 to 90,000. Thermal performance parameters also decrease somewhat as Reynolds number increases over this range, with values in approximate agreement with, or slightly higher than 60° continuous rib data measured by other investigators in a square channel.
NOMENCLATURE

\( a \) streamwise extent of test surface

\( b \) spanwise extent of test surface

\( D_h \) channel hydraulic diameter

\( e \) rib turbulator width

\( e' \) rib turbulator width in streamwise direction

\( f \) friction factor

\( f_o \) baseline friction factor in a smooth channel with no rib turbulators

\( H \) channel height

\( h \) heat transfer coefficient based on flat projected area, \( \dot{q}'/(T_w - T_m) \)

\( k \) thermal conductivity

\( L \) coordinate in direction parallel to ribs

\( Nu \) local Nusselt number, \( hD_h / k \)

\( Nu_o \) baseline Nusselt number in a smooth channel with no rib turbulators

\( p \) streamwise pitch spacing of rib turbulators

\( P \) time-averaged local static pressure

\( Pr \) Prandtl number

\( \dot{q}' \) surface heat flux

\( Re_H \) Reynolds number based on channel height

\( Re_{Dh} \) Reynolds number based on hydraulic diameter

\( t \) time

\( t^* \) non-dimensional time, \( tU / H \)
\( T \)  local static temperature

\( \bar{U} \)  streamwise bulk velocity averaged over the channel cross section

\( u \)  time-averaged local streamwise velocity

\( w \)  coordinate in direction normal to ribs

\( x \)  streamwise coordinate measured from the test section inlet

\( y \)  normal coordinate measured from the test surface between the ribs

\( z \)  spanwise coordinate measured from the test surface centerline

\( \omega_x \)  streamwise vorticity

**Subscripts**

\( a \)  ambient value

\( m \)  time-averaged, local mixed mean value

\( o \)  total or stagnation value

\( w \)  local wall value

\( \text{max} \)  maximum value

**Superscripts**

\( - \)  spanwise- or streamwise-averaged value

\( = \)  globally-averaged value
INTRODUCTION

A variety of techniques are used for enhancing convective heat transfer rates in gas turbine engine passages used for internal cooling of turbine airfoils and combustion chamber liners. These include rib turbulators, pin fins, jet impingement cooling, dimpled surfaces, surface roughness, surfaces with protrusions or other types of turbulence promoters, and swirl chambers. All of these devices act to increase secondary flows and turbulence levels to enhance mixing, in some cases, to form coherent fluid motions in the form of streamwise oriented vortices. Such vortices and secondary flows not only act to increase secondary advection of heat away from surfaces, but also to increase three-dimensional turbulence production by increasing shear and creating gradients of velocity over significant flow volumes. These then give larger magnitudes of turbulence transport over larger portions of the flow fields. The overall objective of each device is then significant enhancement of turbulence transport and convective heat transfer coefficients with minimal increases in pressure penalties.

To determine the capabilities of these devices in accomplishing these tasks, spatially-resolved, spatially-averaged, and globally-averaged heat transfer coefficient data are needed, along with friction factors. Also helpful are details of the flow field structure because this aids the development of numerical models and prediction schemes, and because this provides important insight into the flow structural characteristics responsible for local heat transfer coefficient augmentations. However, in spite of the value of heat transfer data, obtained together with friction factor and flow structure data, very few papers present such data together for the internal cooling augmentation devices mentioned. The present paper is aimed at remedying this deficiency for internal passages with rib turbulators.
A significant number of experimental and numerical studies address the effects of rib turbulators on heat transfer in internal channels. Considered are single pass and multi-pass channels, square and rectangular channels, channels with and without rotation, and rotating channels with different orientations with respect to the axis of rotation.

The earliest experimental studies consider single pass, stationary channels with no rotation. Of these studies, Han et al. [1] address the effects of rib shape, angle of attack, and pitch-to-height ratio. According to the investigators, ribs with 45° inclinations produce better heat transfer performance than ribs with 90° orientations, when compared at the same friction power. Han and Park [2] vary the channel aspect ratio, and conclude that the best heat transfer performance is obtained using a square channel with a rib turbulator angle of attack from 30° to 45°. This range of angles of attack also yield the best heat transfer performance for rectangular channels. Han et al. [3] indicate that best heat transfer enhancements in square channels are produced by V-shaped ribs with 45° and 60° arrangements, followed by 45° and 60° parallel ribs, which are followed by 45° and 60° crossed ribs. Han et al. [4] investigate wedge-shaped and delta-shaped turbulence promoters in square channels, and compare their performance with existing data for different types of rib turbulators. Delta-shaped ribs generally perform better than the wedge-shaped ribs, especially when the delta-shaped ribs on opposite walls are aligned, and arranged with a backward flow direction. The investigators also indicate that broken configurations of delta-shaped ribs and wedge-shaped ribs both give better performance than full length configurations. Taslim et al. [5] provide additional evidence that 45° ribs produce higher thermal performance factors than 90° ribs. The authors also indicate that, of the configurations examined, the highest heat transfer enhancements and highest friction factors are produced by low-blockage ratio V-shaped ribs. In a later study, Taslim et al. [6] study twelve different
geometries of ribs that are placed on all four walls of channels with both square and trapezoidal cross sections. Compared to channels with ribs on two walls, heat transfer coefficients and thermal performance factors are enhanced.

More recent studies of *stationary channels with no rotation consider single pass and multi-pass channels*. Wang et al. [7] present heat transfer results from square ducts with 45° ribs. Thurman and Poinsatte [8] measure heat transfer and bulk air temperature in a three-pass duct with orthogonal ribs and bleed holes both located on one wall. According to the investigators, changing the locations of the ribs relative to the holes produces large changes to surface heat transfer coefficient distributions. Cho et al. [9] employ continuous and discrete, parallel and cross arrays of ribs in a single-pass square duct. Discrete ribs with gaps in between are found to produce more uniform heat transfer coefficient distributions than continuous ribs.

Other recent experimental studies investigate heat transfer in *channels with rotation and square cross-section*. Of these studies, Johnson et al. [10] and Fan et al. [11] examine 4-pass serpentine channels and indicate that the best performance is produced by ribs with 45° arrangements. Parsons et al. [12], Johnson et al. [13], Zhang et al. [14], Parsons et al. [15], and Dutta and Han [16] consider 2-pass channels with different rib turbulator configurations. Three of these studies [13,15,16] also consider the influences of changing the channel orientation with respect to the axis of rotation. Heat transfer data are presented which show that the influences of Coriolis forces and cross-stream flows decrease as channel orientation changes from normal to angled. Another pair of studies by Park et al. [17,18] consider the effects of rib size on local heat transfer coefficients with radially outward flow and transverse ribs on the trailing and leading walls of an internal passage.
Experimental heat transfer studies using *channels with rotation and rectangular cross-section* are less numerous. Of studies in this area, Taslim et al. [19,20] employ single pass channels with orthogonal rotation, and either staggered transverse ribs [19], or 45° ribs arranged with perpendicular orientations on opposite channel walls [20]. In both cases, more pronounced rotation effects are evidenced as the channel aspect ratio increases, or as the rib blockage ratio decreases. Azad et al. [21] employ a two-pass channel with rib turbulators on leading and trailing sides at angles of 45° with respect to the mainstream flow. Rotating rib wall heat transfer coefficients are 2 to 3 times values measured on rotating smooth walls. Heat transfer coefficients on the first pass trailing side and second pass leading side are enhanced by rotation, whereas the first pass leading side and second pass trailing side values are diminished by rotation. In addition, 45° ribs, which are parallel on opposite channel walls, produce higher augmentations than 45° ribs, which are perpendicular on opposite channel walls.

An experimental and numerical investigation by Dutta et al. [22] examines the effects of rib turbulators on heat transfer behavior in a *rotating, two-pass channel with triangular cross-section*. Two channel orientations with respect to the axis of rotation are considered, along with the effects of channel orientation on secondary flows.

A number of other experimental investigations address *flow behavior (without heat transfer) in channels with rib turbulators*. Of these, Bonhoff et al. [23] and Schabacker et al. [24] consider non-rotating channels, and Tse and Steuber [25], and Prabhu and Vedula [26] consider rotating channels. In one case, different velocity components are measured in a serpentine channel with 45° ribs [25], and in another, surface static pressure variations are measured with different channel aspect ratios, and different rotational speeds in a rectangular channel with transverse ribs on one wall [26].
Computational studies of flows and heat transfer in ducts with rib turbulators consider straight single pass ducts [27,28,30,33,35], two pass ducts [31,32,36], two pass ducts with U-shaped channels in between [29,34], 90° orthogonal ribs [27,30,33,34], 45° angled ribs [28,29,31,32,35,36], and rotation [27,29,31-36]. In one case [28], the angled ribs placed on two opposite walls of the channel are rounded. The results from the most notable of these investigations with rotation show that the secondary flows induced by angled ribs, rotating buoyancy, and Coriolis forces produce strong non-isotropic turbulence stresses and heat fluxes, along with important alterations to flow fields and surface heat transfer coefficients.

The present experimental study is conducted using a large-scale test section, without rotation, so that detailed, spatially-resolved surface heat transfer coefficients, spatially-resolved flow structure (instantaneous and time-averaged), and friction factors can be measured. A single-pass channel with aspect ratio of 4 is employed, which models internal cooling passages employed near the mid-chord and trailing edge regions of turbine airfoils used in gas turbine engines for utility power generation. The ribs are placed so that they are perpendicular to each other on the two widest, opposite walls of the channel with 45° angles with respect to the streamwise flow direction. Reynolds numbers, based on channel height, range from 270 to 90,000. The one other study which uses a stationary, non-rotating test section, similar rib geometry, and same channel aspect ratio [2], does not present spatially-resolved surface heat transfer data or any flow structure data. Two other studies with similar (but not exactly matching) rib turbulator geometry and non-rotating test sections, either use a square channel [5] or a channel with an aspect ratio of 20 [7].

The results given in the present paper are thus new and unique because of these differences, and because new spatially-resolved heat transfer and flow structure data are presented and inter-
related to each other, something which is impossible for experimental rotating ribbed channel studies [10-22,25-26]. Included in the present study for different channel Reynolds numbers are: (i) time-sequences of flow visualization images, illustrating instantaneous flow structure, (ii) time-averaged distributions of local streamwise velocity, local streamwise vorticity, local total pressure, and local static pressure, (iii) spatially-resolved, spatially-averaged, and globally-averaged surface Nusselt number data, and (iv) friction factor data. Also discussed are the effects of thermal boundary layer development on local Nusselt number distributions. These results are valuable because the spatial resolution of the heat transfer and flow measurements is much better than provided by rib turbulator results from other sources.

EXPERIMENTAL APPARATUS AND PROCEDURES

The overall experimental apparatus (but not the test section) is similar to the one described by Mahmood et al. [37]. A brief description of this apparatus is also presented here.

Channel and test surface for heat transfer measurements. A schematic of the facility used for heat transfer measurements is shown in Fig. 2.1a. The air used within the facility is circulated in a closed-loop. One of three circuits is employed, depending upon the Reynolds number and flow rate requirements in the test section. For Reynolds numbers \( Re_H \) less than 10,000, a 102 mm pipe is connected to the intake of an ILG Industries 10P type centrifugal blower. For \( Re_H \) between 10,000 and 20,000, the same pipe is connected to the intake of a Dayton 7C447 1.0 hp centrifugal blower. For higher Reynolds numbers, a 203 mm pipe is employed with a New York Blower Co. 7.5 HP, size 1808 pressure blower. In each case, the air mass flow rate from the test section is measured (upstream of which ever blower is employed)
using an ASME standard orifice plate and Validyne M10 digital pressure manometer. The blower then exits into a series of two plenums (0.9 m square and 0.75 m square). A Bonneville cross-flow heat exchanger is located between two of these plenums, and is cooled with liquid nitrogen at flow rate appropriate to give the desired air temperature at the exit of the heat exchanger. As the air exits the heat exchanger, it enters the second plenum, from which the air passes into a rectangular bell mouth inlet, followed by a honeycomb, two screens, and a two-dimensional nozzle with a contraction ratio of 5.6. This nozzle leads to a rectangular cross-section, 411 mm by 103 mm inlet duct which is 1219 mm in length. This is equivalent to 7.4 hydraulic diameters (where hydraulic diameter is 164.7 mm). Two trips are employed on the top and bottom surfaces of the inlet duct, just upstream of the test section, which follows with the same cross-section dimensions. It exits to a 0.60 m square plenum, which is followed by two pipes, each containing an orifice plate, mentioned earlier.

Figure 2.2 gives the geometric details of the test surface, including rib turbulator geometry. A total of 13 ribs or rib segments are used on the top wall and on the bottom wall of the test section. As mentioned, these are arranged with 45° angles with respect to the streamwise flow direction, such that the ribs on opposite walls of the channel are perpendicular to each other. Each rib has 12.8 mm height and square cross-section. The ratio of rib height to hydraulic diameter is .078, the rib pitch-to-height ratio is 10, and the blockage provided by the ribs is 25 percent of the channel cross-sectional area. The top wall of the test section also has two cut-out regions (one at the upstream end and one at the downstream end) where a zinc-selenide window can be installed to allow the infrared camera to view a portion of the test surface on the bottom wall. When this window is not in use, inserts with ribs (which exactly match the adjacent rib turbulators on the top wall) are used in its place. Also identified in Fig. 2.2 is the test section
coordinate system employed for the study. Note that the $\gamma$ coordinate is directed normal to the bottom wall.

All exterior surfaces of the facility (between the heat exchanger and test section) are insulated with Styrofoam ($k=0.024$ W/mK), or 2 to 3 layers of 2.54 cm thick, Elastomer Products black neoprene foam insulation ($k=0.038$ W/mK) to minimize heat losses. Calibrated copper-constantan thermocouples are located between the three layers of insulation located beneath the test section to determine conduction losses. Between the first layer and the 3.2 mm thick acrylic test surfaces are custom-made Electrofilm etched-foil heaters (each encapsulated between two thin layers of Kapton) to provide a constant heat flux boundary condition on the test surface. The acrylic surfaces, which are adjacent to the airstream, contain 35 copper-constantan thermocouples, which are placed within the ribs as well as within the flat portions of the test surface between the ribs. Each of these thermocouples is located 0.0508 cm just below this surface to provide measurements of local surface temperatures, after correction for thermal contact resistance and temperature drop through the 0.0508 cm thickness of acrylic. Acrylic is chosen because of its low thermal conductivity ($k=0.16$ W/mK at 20°C) to minimize streamwise and spanwise conduction along the test surface, and thus, minimize "smearing" of spatially varying temperature gradients along the test surface. The power to the foil heater is controlled and regulated using a variac power supply. Energy balances, performed on the heated test surface, then allow determination of local magnitudes of the convective heat flux.

The mixed-mean stagnation temperature of the air entering the test section is measured using five calibrated copper-constantan thermocouples spread across its cross-section. To determine this temperature, thermocouple-measured temperatures are corrected for thermocouple wire conduction losses, channel velocity variations, as well as for the differences between stagnation
and recovery temperature. All measurements are obtained when the test facility at steady-state, achieved when each of the temperatures from the thermocouples (on the bottom test surface) vary by less than 0.1 °C over a 10 minute period.

**Local Nusselt number measurement.** To determine the surface heat flux (used to calculate heat transfer coefficients and local Nusselt numbers), the convective power levels provided by the etched foil heaters are divided by flat test surface areas. Spatially-resolved temperature distributions along the rib turbulator test surface are determined using infrared imaging in conjunction with thermocouples, energy balances, digital image processing, and *in situ* calibration procedures. To accomplish this, the infrared radiation emitted by the heated interior surface of the channel is captured using a VideoTherm 340 Infrared Imaging Camera, which operates at infrared wave lengths from 8 μm to 14 μm. Temperatures, measured using the calibrated, copper-constantan thermocouples distributed along the test surface adjacent to the flow, are used to perform the *in situ* calibrations simultaneously as the radiation contours from surface temperature variations are recorded.

This is accomplished as the camera views the test surface through a custom-made, zinc-selenide window (which transmits infrared wave lengths between 6 and 17 μm) located on the top wall of the test section. Reflection and radiation from surrounding laboratory sources are minimized using an opaque shield which covers the camera lens and the zinc selenide window. Frost build-up on the outside of the window is eliminated using a small heated air stream. The window is located either just above portions of the second, third, and fourth ribs, or just above portions of the tenth, eleventh, and twelfth ribs downstream from the leading edge of the test surface. Eleven to thirteen thermocouple junction locations are usually present in the infrared
field viewed by the camera. The exact spatial locations and pixel locations of these thermocouple junctions and the coordinates of a 12.7 cm by 12.7 cm field of view are known from calibration maps obtained prior to measurements. During this procedure, the camera is focused, and rigidly mounted and oriented relative to the test surface in the same way as when radiation contours are recorded.

With these data, gray scale values at pixel locations within video taped images from the infrared imaging camera are readily converted to temperatures. Because such calibration data depend strongly on camera adjustment, the same brightness, contrast, and aperture camera settings are used to obtain the experimental data. The in situ calibration approach rigorously and accurately accounts for these variations.

Images from the infrared camera are recorded as 8-bit gray scale images on commercial videotape using a Panasonic AG-1960 video recorder. Images are then digitized using NIH Image v1.60 software, operated on a Power Macintosh 7500 PC computer. Subsequent software is used to convert each of 256 possible gray scale values to local Nusselt number values at each pixel location using calibration data. Thermal conductivity in the Nusselt number is based on the average of the local wall temperature and the temperature of the air at the upstream inlet. Contour plots of local surface temperature and Nusselt number are prepared using DeltaGraph v4.0 software. Each individual image covers a 300 pixel by 300 pixel area. Sargent et al. [38], and Mahmood et al. [37] provide additional details on the infrared imaging and measurement procedures.

**Friction factors.** Wall static pressures are measured along the test section simultaneously as the heat transfer measurements are conducted, using 12 side wall pressure taps, located 25.4
mm apart near the downstream portion of the test section. These measurements are made in the test section with rib turbulators, as well as in a baseline test section with smooth surfaces on all four walls. Friction factors are then determined from streamwise pressure gradient magnitudes. Pressures from the wall pressure taps are measured using Celesco LCVR pressure transducers. Signals from these transducers are processed using Celesco CD10D Carrier-Demodulators. Voltages from the Carrier-Demodulators are acquired using a Hewlett-Packard 44422A data acquisition card installed in a Hewlett-Packard 3497A data acquisition control unit, which is controlled by a Hewlett-Packard A4190A Series computer.

**Time-averaged flow velocity components and pressure.** A separate channel facility (but with the same test section), and the same interior geometry identical to that in the heat transfer facility, is employed for flow visualization as well as quantitative surveys of flow structure. This facility is shown schematically in Fig. 2.1b.

A 1.27 mm diameter miniature five-hole pressure probe, manufactured at the University of Utah especially for these measurements, is used to obtain time-averaged surveys of total pressure, static pressure, and the three mean velocity components. These data are then used to deduce distributions of streamwise vorticity. To obtain the surveys, the probe employed is mounted on an automated two-dimensional traverse, and inserted into the test section through a slot lined with foam to prevent air leakage. The output ports of the probe are connected either to Validyne DP103-06 pressure transducers (to measure differential pressures up to 2.5 mm of water), or Celesco LCVR pressure transducers (to measure differential pressures up to 20.0 mm of water). Signals from the transducers are then processed using Celesco CD10D Carrier-Demodulators. Voltages from the Carrier-Demodulators are acquired using a Hewlett-Packard.
A data acquisition card installed in a Hewlett-Packard 3497A data acquisition control unit. This control unit, the Superior Electric type M092-FD310 Mitas stepping motors on the two-dimensional traverse, a Superior Electric Modulynx Mitas type PMS085-C2AR controller, and a Superior Electric Modulynx Mitas type PMS085-D050 motor drive are controlled by a Hewlett-Packard A4190A Series computer. Contour plots of measured quantities are generated using a polynomial interpolating technique (within DeltaGraph software) between data points. In each survey plane, 1560 data points are spaced 2.54 mm apart. Additional details of the five-hole probe measurement procedures, including calibration details, are given by Ligrani et al. [39,40].

**Flow visualization.** Flow visualization using smoke is used to identify vortex structures and other secondary flow features. Smoke from four horizontally-oriented smoke wires is employed for this purpose. These are located 4.8 mm, 11.9 mm, 88.5 mm, and 95.7 mm from the bottom test surface 25 to 29 mm from the downstream edge of the test section, which is equivalent to $x = 1258$-$1261$ mm. To accomplish this, each wire is first coated with Barts Pneumatics Corp. super smoke fluid and then powered using a Hewlett-Packard 6433B DC power supply. With this arrangement, the smoke forms into single thin lines parallel to the test surface. As the smoke is advected downstream, the secondary flows which accompany vortex and secondary flow development cause the smoke to be rearranged in patterns which show the locations and distributions of these flow phenomena. Smoke patterns are illuminated in a spanwise-normal light plane located at $x = 1462$ mm using a thin sheet of light provided by a Colotran ellipsoidal No. 550, 1000 watt spot light, and a slits machined in two parallel metal plates. Images are recorded using a Panasonic WV-BP330 CCTV video camera, connected to a Panasonic AG-1960 type 4-head, multiplex video cassette recorder. Images recorded on video
tape (taken individually or in sequence) are then digitized using a Sony DCR-TRV900 digital video camera recorder. The resulting images are then further processed using a Power Macintosh 7500 PC computer, and finally printed using a Panasonic PV-PD 2000 digital photo printer. Additional discussion of the procedures used for flow visualization is provided by Ligrani [41].

**UNCERTAINTY ESTIMATES**

Uncertainty estimates are based on 95 percent confidence levels, and determined using procedures described by Kline and McClintock [42] and Moffat [43]. Uncertainty of temperatures measured with thermocouples is 0.15°C. Spatial and temperature resolutions achieved with the infrared imaging are about 0.52 mm and 0.8°C, respectively. This magnitude of temperature resolution is due to uncertainty in determining the exact locations of thermocouples with respect to pixel values used for the *in situ* calibrations. Local Nusselt number ratio uncertainty is then about ±0.13 (for a ratio of 2.00), or about ±6.5 percent. Reynolds number uncertainty is approximately ±1.7 percent for \( Re_H \) of 10,000. The uncertainties of local total pressure (relative to atmospheric pressure), local static pressure (relative to atmospheric pressure), local streamwise velocity, and local streamwise vorticity are ±4.0, ±4.0, ±2.5, and ±8.0 percent, respectively.
EXPERIMENTAL RESULTS AND DISCUSSION

Baseline Nusselt numbers. Baseline Nusselt numbers $\text{Nu}_o$, used to normalize the ribbed channel values, are measured in a smooth rectangular test section with smooth walls replacing the two ribbed test surfaces. Except for the absence of the ribs, all geometric characteristics of the channel are the same as when the ribbed test surfaces are installed. These measurements are made in the downstream portions of the test section where the channel flow is hydraulically and thermally fully developed. All $\text{Nu}_o$ baseline values are obtained using a $T_{oi}/T_w$ temperature ratio of 0.93 to 0.95. In addition, baseline measurements are conducted with all four surfaces wrapped with etched foil heaters to provide a heat flux boundary condition around the entire test section. The variation of baseline Nusselt numbers with Reynolds number $Re_{Dh}$ is shown in Fig. 2.3. Here, values determined from an average of measurements made on the top and bottom walls, are presented. These values are in agreement with the McAdams smooth circular tube correlation [2] for $Re_{Dh}<60,000$, with values which are slightly lower than the correlation at higher $Re_{Dh}$.

Spatially-resolved distributions of local Nusselt numbers. Figure 2.4 presents spatially-resolved Nusselt number ratios, measured over about two periods of ribbed pattern, on the bottom test surface. The results are shown for $Re_H=10,000$ and $T_{oi}/T_w=0.95$. In the figure, flow is directed from bottom to top in the increasing $X/D_h$ direction. The red diagonal regions show $\text{Nu}/\text{Nu}_o$ values measured on the tops of the ribs. The data presented are time-averaged, determined from 25 instantaneous data sets acquired over a period of 25 seconds.

As indicated in Fig. 2.4, $\text{Nu}/\text{Nu}_o$ ratios are relatively very high along the tops of the ribs. When compared along the rib tops, values are then highest near the upstream and downstream
edges. As one moves from the rib in the streamwise or $+X / D_h$ direction, local $Nu / Nu_o$ values initially decrease, and then are low relative to other locations on the test surface. This is due to a re-circulating flow region just downstream of the rib, where flow direction next to the surface is opposite to the bulk flow direction. Note that a shear layer forms between this re-circulating flow region and the bulk flow located just above the rib. A region with relatively higher values of $Nu / Nu_o$ values then follows at slightly higher $X / D_h$ (where $Nu / Nu_o > 3.2$), which is due to reattachment of the shear layer which is initially formed above the re-circulating flow region. With an additional increase in streamwise development, $Nu / Nu_o$ values decrease slightly once again (and then sometimes increase locally) as a second 45° rib is approached. These are due to a smaller region of re-circulating flow which forms just upstream of each rib turbulator. This pattern of flow and surface Nusselt number variations then repeats itself as additional ribs are encountered along the test surface. Other factors which affect the heat transfer augmentations are the skewing and three-dimensional nature of the boundary layer which develops due to the angled orientations of the ribs. Increased levels of three-dimensional turbulence production, turbulence transport, and the large-scale vortex pairs, which form in the channel, also make contributions.

The effects of Reynolds number on spatially-resolved Nusselt number ratios are apparent if the data in Fig. 2.5 for $Re_H=90,000$ are compared to the data in Fig. 2.4 for $Re_H=10,000$. The data presented in both figures are measured on the same part of the test surface for $T_{wi} / T_w$ of 0.92 to 0.95. The results shown in Fig. 2.5 are also time-averaged using 25 instantaneous data sets. As before, the red diagonal regions with relatively high $Nu / Nu_o$ ratios are also present on the tops of the ribs in Fig. 2.5. Over this region, again, as before, the highest Nusselt number ratio values are again present near the edges of the rib tops. Comparison at the two different $Re_H$ then
shows that $Nu / Nu_o$ values on rib tops are higher as $Re_H$ increases, a conclusion also supported by measurements made at $Re_H$ of 18,300, 25,000, and 60,800. On the flat region just downstream of the ribs at slightly larger $x / D_h$, local $Nu / Nu_o$ values at $Re_H=90,000$ are then lower than values measured at the same $x / D_h$ and $z / D_h$ at $Re_H=10,000$. $Nu / Nu_o$ values for $Re_H=90,000$ then increase continually as $x / D_h$ increases, and the next rib is approached (Fig. 2.5). As mentioned, this is different from the results presented in Fig. 2.4 for $Re_H=10,000$, where the initial increase due to shear layer re-attachment is followed by a decrease of local $Nu / Nu_o$ values. These differences with Reynolds number are thus due to re-circulation zones downstream of the ribs, which appear to become larger and stronger as the Reynolds number increases. Thus, the effects of Reynolds number on $Nu / Nu_o$ distributions are most prominent on the ribs, and on regions just downstream and parallel to ribs.

The effects of Reynolds number on local $Nu / Nu_o$ distributions are further illustrated by the results presented in Figs. 2.6 and 2.7. In the first of these figures, local $Nu / Nu_o$ data are given as they vary with $x / D_h$ for constant $z / D_h=0$. In the second of these figures, $Nu / Nu_o$ data are given as they vary with $z / D_h$ for constant $x / D_h=6.90$. These data are obtained from survey results (such as the ones shown in Figs. 2.4 and 2.5) at $Re_H$ ranging from 10,000 to 90,000, and about constant temperature ratio $T_{oi}/T_w$ of 0.92-0.95. In Fig. 2.6, local $Nu / Nu_o$ values at $x / D_h=6.53$ to $x / D_h=6.67$ correspond to locations upstream of the central rib. $x / D_h=6.67$ to $x / D_h=6.75$ are then located on the rib, and $x / D_h=6.75$ to $x / D_h=7.14$ correspond to the flat surface downstream of the rib. In Fig. 2.7, $z / D_h=-0.20$ to $z / D_h=0.16$ and $z / D_h=0.27$ to $z / D_h=0.39$ correspond to spanwise locations on the flat portions of the surface between ribs, and
Figures 2.6 and 2.7 both show that $N_{u}/N_{u_0}$ ratios generally increase somewhat on the rib tops as $Re_H$ increases. In contrast, Nusselt number ratios decrease on the flat regions away from the ribs, especially at locations just downstream of the ribs, as $Re_H$ increases. Because of the normalization employed, this means that the observed Nusselt number $N_{u}$ increases with Reynolds number are slower than baseline $N_{u_0}$ increases with $Re_H$ on flat regions, and more rapid on the ribs themselves. Such changes are partially due to increases in the size and strength of the flow re-circulation region, and the shear layer associated with it, as the Reynolds number increases. In both cases, $N_{u}/N_{u_0}$ values are generally much higher than 1.0 on most of the test surface, including the flat regions between the ribs, irrespective of the value of Reynolds number employed.

**Local instantaneous and time-averaged flow structure.** A time-sequence of flow visualization images are presented in Fig. 2.8 for $Re_H$=480, which are illuminated downstream of the rib turbulator test section at $x$ =1462 mm. These data are obtained at this low Reynolds number because diffusion and increased unsteadiness at higher Reynolds numbers result in smeared and unrecognizable flow patterns. Each image in Fig. 2.8 extends in the vertical direction from the bottom to the top of the channel, and in the horizontal direction over a distance of about 2.0 channel heights. The spanwise center of each image is then located at the spanwise center of the test section at $Z/D_h$ =0. The most important features in each image are two large vortex pairs (indicated by mushroom-shaped smoke patterns), which emanate from the bottom and top channel surfaces. These vortex pairs are formed by the effects of the ribs, which are arranged perpendicular to each other on the top and bottom surfaces, as they force air to and from both of these test surfaces. As a result of this motion, viscous effects, and continuity, pairs
of counter-rotating streamwise vortices form. The secondary flows within and around these vortices are especially intense because of the blockage effects produced by the ribs. Figure 2.8 shows that these vortex pairs change substantially with time, as indicated by different convolutions and distortions of smoke patterns in different images. Each pattern is produced by complex, unsteady secondary flows, which also rearrange distributions of streamwise velocity. As a result, shear gradients are spread over the entire channel cross section.

Figure 2.9 then shows that two dominant vortex pairs, with opposite orientations, are present at Reynolds numbers $Re_H$ from 270 to 800. At some $Re_H$, these primary vortex pairs are more convoluted and distorted than at others. The two vortex pairs are sometimes located on opposite sides of the channel, with upwash regions directed in opposite directions (i.e. at $Re_H=310$), or with the vortex pairs oriented diagonally with respect to each other (i.e. at $Re_H=430$ and $Re_H=480$). The vortices aid convective processes for heat transfer augmentation by: (i) increasing secondary advection of fluid between the central parts of the channel and regions near the wall, and (ii) producing regions with high, three-dimensional shear and high magnitudes of turbulence production over much of the channel cross section, thereby substantially increasing turbulence transport levels in all three coordinate directions.

Time-averaged data also provide evidence of significant mixing and pairs of counter-rotating vortices in the rib turbulated channel. These data are presented in Figs. 2.10a-d, and are measured at $Re_H=10,500$ and $x=1235$ mm, which is located just downstream of the test section. The surveys are made over a spanwise-normal plane, which extends about one channel height in each direction.

Evidence of the primary vortex pairs is provided by regions of positive and negative vorticity, which are adjacent to each other, in the survey of streamwise vorticity in Fig. 2.10d. One of
these primary pairs is positioned near the top surface just to the right of $Z/H = 0$, and one is positioned near the bottom surface also at $Z/H$ values slightly larger than zero. In each case, an upwash region is located between the two vortices in each pair. These upwash regions are evident in the surveys of local streamwise velocity and local total pressure in Figs. 2.10a and 2.10b, respectively. In each case, the upwash region, which is present near each wall, is indicated by a deficit of each quantity because of advection of fluid with low total pressure and low velocity away from the walls by secondary flows contained within the vortices. The static pressure survey in Fig. 2.10c also shows important variations in the upwash regions, however, in this case, local upwash magnitudes are not necessarily lower than surrounding values.

The nature of these distributions, as well as the vorticity distribution in Fig. 2.10d, are consistent with the flow visualization results in Figs. 2.8 and 2.9. In both cases, complicated, spatially-varying distributions are present. For the data in Fig. 2.10d, this is because the results are a time-average of a flow field with highly distorted and convoluted vortices and secondary flows. The complexity of the flow field is also illustrated by the smaller vorticity signatures from other secondary vortices and vortex pairs in Fig. 2.10d (which are located away from the vorticity signatures associated with the primary vortex pairs).

**Spatially-averaged Nusselt numbers.** Spatially-averaged Nusselt number ratios $\bar{\mathcal{N}}_u / \mathcal{N}_{u_o}$, determined from local data (such as that shown in Figs. 2.4 and 2.5), are determined for the diagonal directions shown in Fig. 2.11. These diagonal directions are oriented parallel to and perpendicular to the direction of the ribs located along the bottom test surface. As shown in Figure 2.11, $W / D_h$ is then the diagonal directed normal to the ribs, and $L / D_h$ is the diagonal directed parallel to the ribs. The origin of the $L / D_h$ and $W / D_h$ axes is positioned at $X / D_h = 6.53$. 


\[ \frac{(W/D_h)_{\text{max}}}{h_{DW}} \] then equals the spacing between adjacent ribs, or the rib pitch in the direction normal to the ribs.

The resulting data, for \( Re_H \) from 10,000 to 90,000 at constant \( \frac{T_a}{T_w} \) of 0.93-0.95, are shown in Figs. 2.12 and 2.13. \( \overline{Nu}/Nu_o \) in the first of these figures are averaged in the \( W/D_h \) direction, and shown as they vary with \( L/D_h \). The data are approximately constant with \( L/D_h \) at each Reynolds number, \( Re_H \). This is important because it means that the flow at each Reynolds number considered is thermally fully developed.

The \( \overline{Nu}/Nu_o \) distributions at each \( Re_H \) in Fig. 2.13 (averaged in the \( L/D_h \) direction and shown as they vary with \( W/D_h \)), then show important variations with \( W/D_h/(W/D_h)_{\text{max}} \) because this coordinate is oriented perpendicular to the direction of the ribs. These include \( \overline{Nu}/Nu_o \) increases with \( Re_H \) at the top of the central rib. At larger \( W/D_h > 0.55 \), which correspond to locations downstream of the central rib, \( \overline{Nu}/Nu_o \) values then decrease substantially as \( Re_H \) increases. This is mostly due to increases in the strength and size of the flow re-circulation region downstream of the rib, which occurs as the Reynolds number \( Re_H \) becomes larger.

The \( \overline{Nu}/Nu_o \) data presented in Figs. 2.14 and 2.15 are obtained for \( Re_H \) of about 60,000, and the same surface layout given in Fig. 2.11, except these data are obtained at a location farther upstream where the thermal boundary layers are still developing. In this case, the origin of the \( L/D_h \) and \( W/D_h \) coordinates is located at \( X/D_h = 0.15 \). In Fig. 2.14, the \( \overline{Nu}/Nu_o \) data measured upstream show significant variations with \( L/D_h \), which confirms that the thermal boundary layers at this measurement location are not fully developed.
A similar conclusion is reached if the $\frac{\overline{Nu}}{Nu_o}$ data, given in Fig. 2.15 as dependent upon $\frac{W}{D_h}/(W/D_h)_{\text{max}}$, are examined. For regions upstream of the rib ($\frac{W}{D_h}/(W/D_h)_{\text{max}} < 0.42$), on the rib ($0.42 < \frac{W}{D_h}/(W/D_h)_{\text{max}} < 0.52$), and downstream of the rib ($\frac{W}{D_h}/(W/D_h)_{\text{max}} > 0.52$), the $\frac{\overline{Nu}}{Nu_o}$ values measured at the upstream locations are often higher than values measured at the downstream locations, when compared at the same $\frac{W}{D_h}/(W/D_h)_{\text{max}}$. This is because of thinner, less-than-fully developed thermal boundary layers at the upstream location, which also cause $\frac{\overline{Nu}}{Nu_o}$ to decrease continually with streamwise development over each segment of the test surface.

**Globally-averaged Nusselt numbers and friction factors.** Figures 2.16, 2.18, and 2.19 present globally-averaged Nusselt number ratios for fully-developed flow conditions, which are determined by averaging all the local data in the rectangular area enclosed by the lengths $(L/D_h)_{\text{max}}$ and $(W/D_h)_{\text{max}}$ shown in Fig. 2.11. Globally-averaged Nusselt number ratios are thus determined from the results in Figs. 2.12 and 2.13. The data in Figs. 2.16-2.19 are given for Reynolds numbers $Re_H$ from 10,000 to 90,000 and $T_{wi}/T_w$ of 0.93-0.95. Results from Han and Park [2], Han et al. [4], and Taslim et al. [5] are included for comparison.

Recall that the present 45° square ribs are arranged so that they are perpendicular on opposite channel walls. They are installed in a channel with aspect ratio of 4, ratio of rib height to hydraulic diameter $e/D_h$ of .078, and rib pitch-to-height ratio $p/e$ of 10. The Han and Park [2] data used for comparison are obtained in a channel with the same aspect ratio, same $e/D_h$, and same $p/e$. The continuous ribs on two opposite walls are arranged so that they are parallel to each other at angles of 30°, 45°, 60°, and 90° relative to the bulk flow direction. The data used
for comparison from Han et al. [4] are for square channels with 90° continuous ribs, 60°
continuous ribs, and 60° broken ribs. In all cases, the ribs are parallel on opposite channel walls,
\(e/D_h\) is .063, and \(p/e\) is 10. The Taslim et al. [5] data used for comparison are also obtained in
a square channel with \(p/e=10\). For these data, \(e/D_h=0.083\). In addition, the 45° oriented ribs are
arranged in the same direction on opposite channel walls, and placed at streamwise locations so
that they are staggered with respect to each other. Figure 2.16 shows that globally-averaged
Nusselt number ratios generally decrease somewhat as \(Re_H\) increases for all of the
configurations. For the present rib-turbulator arrangement, globally-averaged ratios vary from
3.36 to 2.82 as \(Re_H\) increases from 10,000 to 90,000. At Reynolds numbers lower than 20,000,
these data are also in approximate agreement with results from Taslim et al. [5]. At higher \(Re_H\),
the present results lie between the 60° broken rib data, and the 60° continuous rib data from Han
et al. [4]. The present data are slightly higher than the Han and Park [2] Nusselt number ratios in
Fig. 2.16. These data are given for \(Re_H=30,000\), and increase continually as the rib angle
increases from 30° to 90°.

Measured friction factor ratios in the rib-turbulator channel for \(Re_H\) of 10,000 to 90,000 at
\(T_{oi}/T_w\) of 0.93-0.95 are shown as they depend upon \(Re_H\) in Fig. 2.17. These \(f/f_o\) data are also
compared with results from Han and Park [2], Han et al. [4], and Taslim et al. [5]. Here, the Han
and Park [2] \(f/f_o\) data again increase continually with rib angle for \(Re_H=30,000\). The present
data are then in rough agreement with the Han and Park 45° oriented rib results. The present
friction factor data are then higher than the other data presented in Fig. 2.17, except for
\(Re_H>40,000\), where the present results show approximate agreement with the 60° continuous rib
data from Han et al. [4].
Globally-averaged Nusselt number ratios and friction factor ratios from Figs. 2.16 and 2.17 are plotted together in Fig. 2.18 for $Re_H$ from 10,000 to 90,000 and $T_0/T_w$ of 0.93-0.95. Here, globally-averaged Nusselt number ratios generally decrease as $f/f_o$ increases in almost all cases as channel and rib geometry are held constant. This is consistent with the Han and Park [2] data in Fig. 2.18, since each data point from this source represents a different rib angle with respect to the oncoming flow direction. The four associated data points are located from left to right in this figure as rib angle increases from 30° to 45° to 60° to 90°. Overall, the best performance in the coordinates of Fig. 2.18 (as characterized by the highest Nusselt number ratios at the lowest $f/f_o$ values) is produced by the 45° oriented ribs from Taslim et al. [5], followed by the 60° broken ribs from Han et al. [4], and then by the present 45° rib turbulators. The differences between the present data and the Taslim et al. [4] results are partially due to the different channel aspect ratios employed.

**Performance parameters.** These comparisons are further illustrated by the plot of performance parameters $\frac{Nu}{Nu_o} / (f/f_o)^{1/3}$ as dependent upon $Re_H$ in Fig. 2.19. This performance parameter is employed because it gives the ratio of heat transfer augmentation to friction augmentation in a form in which the pumping power is the same for both [44].

Performance parameter magnitudes for the present study are determined from the data in Figs. 2.16-2.18, and decrease from 1.66 to 1.42 as $Re_H$ increases from 10,000 to 90,000 in Fig. 2.19. Here, as in the previous figure, the best overall performance is provided by the Taslim 45° ribs at $Re_H<12,000$, and by Han 60° broken ribs at higher $Re_H$ greater than 12,000. Our performance parameters are then lower than these two data sets when compared at the same Reynolds number (in part, because of the different channel aspect ratios used). Performance parameter magnitudes
from the present study are then in approximate agreement with the $60^\circ$ continuous rib data from Han et al. [4] for $Re_H<40,000$. The present data then lie between the $60^\circ$ continuous rib data and the $60^\circ$ broken rib data at higher Reynolds numbers.

Overall, these rib turbulator channel performance parameters are a result of vortex induced secondary flows and shear layer re-attachments, which result in augmented three-dimensional turbulence transport, and increased secondary flow advection. Also important are the relatively high pressure losses and friction factors produced by the form drag which develops around the rib turbulators.

**SUMMARY AND CONCLUSIONS**

Spatially-resolved Nusselt numbers and flow structure are presented for a stationary channel with an aspect ratio of 4 and angled rib turbulators inclined at $45^\circ$ with perpendicular orientations on two opposite surfaces. The flow structure results include time-averaged distributions of local streamwise vorticity, local streamwise velocity, local total pressure, and local static pressure, surveyed over flow cross-sectional planes, as well as flow visualization images and friction factors. Results are given at different Reynolds numbers based on channel height from 270 to 90,000, and ratios of air inlet stagnation temperature to surface temperature ranging from 0.93 to 0.95. The ratio of rib height to hydraulic diameter is .078, the rib pitch-to-height ratio is 10, and the blockage provided by the ribs is 25 percent of the channel cross-sectional area.

Spatially-resolved local Nusselt numbers are highest on tops of the rib turbulators, with lower magnitudes on flat surfaces between the ribs, where regions of flow separation and shear layer
re-attachment have pronounced influences on local surface heat transfer behavior. Also important are intense, highly unsteady secondary flows and vortex pairs (evident in flow visualizations and time-averaged surveys of local flow structural characteristics), which act to increase secondary advection and turbulent transport over the entire channel cross-section.

Because of these phenomena, local and spatially-averaged rib turbulator Nusselt numbers, normalized by values measured in a smooth channel, generally increase on the rib tops as Reynolds number increases. These Nusselt number ratios then decrease on the flat regions away from the ribs, especially at locations just downstream of the ribs, as Reynolds number increases. Because of the normalization employed, this means that the observed Nusselt number increases with Reynolds number are slower than smooth-surface baseline Nusselt number increases with Reynolds number on flat regions, and more rapid on the ribs themselves. Such changes are partially due to increases in the size and strength of the flow re-circulation region, and the shear layer associated with it, as the Reynolds number increases. In both cases, local and spatially-averaged Nusselt number ratios are generally much higher than 1.0 on most of the test surface, including the flat regions between the ribs, irrespective of the value of Reynolds number employed.

Globally-averaged Nusselt number ratios vary from 3.36 to 2.82 as Reynolds number increases from 10,000 to 90,000. Thermal performance parameters also decrease somewhat as Reynolds number increases over this range, with values in approximate agreement with, or slightly higher than 60° continuous rib data measured by Han et al. [4] in a square channel. Such performance parameter magnitudes and Nusselt number variations are due augmented three-dimensional turbulence transport and increased secondary flow advection, as well as the pressure losses and friction factors produced by the form drag which develops around the rib turbulators.
ACKNOWLEDGEMENTS

The work presented in this paper was sponsored by the AGTSR Advanced Gas Turbine Research Program of the SCIES, the South Carolina Institute for Energy Studies of the U.S. Department of Energy. Mr. Eli Comeau, Mr. Arthur Fox, and Mr. Jacob Kingston helped in setting up the experimental apparatus employed in the investigation.

REFERENCES


PART III: NUMERICAL PREDICTION

Part 3 is conducted at Texas A&M University under the direction of Dr. H.C. Chen and Dr. J.C. Han.
ABSTRACT

Computations were performed to study three-dimensional turbulent flow and heat transfer in a rotating smooth and 45° ribbed rectangular channels for which heat transfer data were available. The channel aspect ratio (AR) is 4:1, the rib height-to-hydraulic diameter ratio \( (e/D_h) \) is 0.078 and the rib-pitch-to-height ratio \( (P/e) \) is 10. The rotation number and inlet coolant-to-wall density ratios, \( \Delta \rho/\rho \), were varied from 0.0 to 0.28 and from 0.122 to 0.40, respectively, while the Reynolds number was fixed at 10,000. Also, two channel orientations \( (\beta = 90° \text{ and } 135° \text{ from the rotation direction}) \) were investigated with focus on the high rotation and high density ratios effects on the heat transfer characteristics of the 135° orientation. These results show that, for high rotation and high density ratio, the rotation induced secondary flow overpowered the rib induced secondary flow and thus change significantly the heat transfer characteristic compared to the low rotation low density ratio case. A multi-block Reynolds-Averaged Navier-Stokes (RANS) method was employed in conjunction with a near-wall second-moment turbulence closure. In the present method, the convective transport equations for momentum, energy, and turbulence quantities are solved in curvilinear, body-fitted coordinates using the finite-analytic method.

NOMENCLATURE

\( AR \quad \text{channel aspect ratio} \)

\( D_h, D \quad \text{hydraulic diameter} \)

\( e \quad \text{rib height} \)

\( h \quad \text{heat transfer coefficient} \)

\( k \quad \text{thermal conductivity of coolant} \)

\( Nu \quad \text{local Nusselt number}, hD/k \)

\( Nu_o \quad \text{Nusselt number in fully-developed turbulent non-rotating tube flow}, hD/k \)

\( Pr \quad \text{Prandtl number} \)

\( Re \quad \text{Reynolds number}, \rho W_b D_h/\mu \)

\( Ro \quad \text{rotation number}, \Omega D_h/W_b \)

\( R_r \quad \text{radius from axis of rotation} \)

\( S \quad \text{streamwise distance} \)
\( T \)  
local coolant temperature

\( T_o \)  
coolant temperature at inlet

\( T_w \)  
local wall temperature

\( W_b \)  
bulk velocity in streamwise direction

\( \alpha \)  
rib angle

\( \beta \)  
angle of channel orientation measured from direction of rotation

\( \rho \)  
density of coolant

\( \Delta \rho/\rho \)  
inlet coolant-to-wall density ratio, \( (T_w - T_o)/T_w \)

\( \Omega \)  
rotational speed

\( \theta \)  
dimensionless temperature, \( (T - T_o)/(T_w - T_o) \)

\( \mu \)  
dynamic viscosity of coolant

1. INTRODUCTION

1.1 Motivation: To improve thermal efficiency, gas-turbine stages are being designed to operate at increasingly higher inlet temperatures. A widely used method for cooling turbine blades is to bleed lower-temperature gas from the compressor and circulate it within and around each blade. The coolant typically flows through a series of straight ducts connected by 180° bends and roughened with ribs or pin fins to enhance heat transfer. These cooling ducts may not only be square in cross section or normal to the rotational direction of the blade. In fact, the aerodynamic shape of the turbine blade dictates the use of cooling channels that are rectangular in cross section (with different aspect ratios) and are at an angle, \( \beta \), from the direction of rotation. Rotation of the turbine blade cooling passages adds another complexity to the problem. It gives rise to Coriolis and buoyancy forces that can significantly alter the local heat transfer in the internal coolant passages from the non-rotating channels. The presence of rib turbulators adds a further complexity since these ribs produce complex flow fields such as flow separation, reattachment and secondary flow between the ribs, which produce a high turbulence level that leads to high heat transfer coefficients.

1.2 Literature Review: Experimental Studies. The complex coupling of the Coriolis and buoyancy forces with flow separation/reattachment by ribs has prompted many investigators to study the flow and temperature fields generated in heated, rotating ribbed wall passages. Most
experimental studies on internal cooling passages have focused on non-rotating ducts. See, for example, Han and Park [1], Han et al. [2] Ekkard and Han [3] and Liou et al. [4] and the references cited there. Experimental studies on rotating ducts have been less numerous. Wagner et al. [5], Dutta and Han [6], Soong et al. [7] and Azad et al. [8] investigated rotating ducts with smooth walls. Wagner et al. [9], Johnson et al. [10 and 11], Parsons et al. [12] and Zhang et al.[13] reported studies on rotating square channels with normal and angled ribs. Azad et al. [8] also investigated the effect of channel orientation on rotating ribbed two pass rectangular channel. Griffith et al. [14] studied the effect of channel orientation on rotating smooth and ribbed rectangular channels with channel aspect ratio of 4:1. They investigated a broad range of flow parameters including Reynolds number (Re = 5000-40000), rotation number (Ro = 0.04-0.3) and coolant to wall density ratio ($\Delta \rho / \rho = 0.122$). Their experimental results provided a database for the present work.

1.3 Literature Review: Numerical Studies

1.3.1 Smooth Surfaces: In addition to the experimental studies mentioned above, several studies have been made to predict numerically the flow and heat transfer in radially rotating smooth and ribbed ducts. Iacovides and Launder [15], Prakash and Zerkle [16], Dutta et al. [17] and Bo et al. [18] studied one passage smooth ducts with normal channel orientation from the direction of rotation i.e., $\beta = 90^\circ$. Sathyamurthy et al. [19], Stephens et al. [20], Iacovides et al. [21] and Bonhoff et al. [22] reported numerical predictions for rotating smooth two passage ducts and $\beta = 90^\circ$. The differential Reynolds stress model (RSM) with wall function in FLUENT code was used in the calculation of Bonhoff et al. [22]. Chen et al. [23, 24] predicted the flow and heat transfer in a rotating smooth two-pass square channel which is the first two passages of the four-pass serpentine passage that was experimentally investigated by Wagner et al. [5]. They used two turbulence models: a two-layer $k$-$\varepsilon$ isotropic eddy viscosity model and a near-wall second-moment closure model. The near-wall second-moment closure model accurately predicted the complex three-dimensional flow and heat transfer characteristics resulting from the rotation and strong wall curvature. They provided the most reliable predictions in comparison with the data of Wagner et al. [5]. Al-Qahtani et al. [25] predicted the flow and heat transfer in a rotating smooth two-pass rectangular channel with a 180° sharp turn and an aspect ratio of 2:1 which was also experimentally investigated by Azad et al. [8]. Two channel orientations were
studied: $\beta = 90^\circ$ and $135^\circ$. They also investigated the effect of the rotation number, $Ro$, and inlet coolant-to-wall density ratio $\Delta \rho / \rho$.

1.3.2 Ribbed Surfaces: Stephens et al. [26, 27] studied inclined ribs in a straight non-rotating square duct. Stephens and Shih [28] investigated the effect of angled ribs on the heat transfer coefficients in a rotating two-passage duct using a low-Re number $k-\omega$ turbulence model. They studied the effects of Reynolds numbers, rotation numbers, and buoyancy parameters. Prakash and Zerkle [29], employing a high Reynolds number $k-\varepsilon$ turbulence model with wall function, performed a numerical prediction of flow and heat transfer in a ribbed rectangular duct ($90^\circ$ rib) with and without rotation. However, their calculations used periodicity and neglected buoyancy effects. They suggested that a low Reynolds number turbulence model is necessary to simulate real gas turbine engine conditions and a Reynolds stress model is required to capture anisotropic effects. Bonhoff et al. [22] calculated the heat transfer coefficients and flow fields for rotating U-shaped coolant channels with angled ribs ($45^\circ$). They used a Reynolds stress turbulence model with wall functions in the FLUENT CFD code. Using the periodicity of the flow, Iacovides [30] computed flow and temperature fields in a rotating straight duct with $90^\circ$ ribs. Two zonal models of turbulence were tested: a $k-\varepsilon$ with a one-equation model of $k$ transport across the near-wall region and a low-Re differential stress model. He concluded that the differential stress model thermal computations were clearly superior to those of the $k-\varepsilon$/one-equation model.

Using the same model and method of Chen et al. [23, 24], Jang et al. [31, 32] studied flow and heat transfer behavior in a non-rotating two-pass square channels with $60^\circ$ and $90^\circ$ ribs, respectively. Their results were in good agreement with Ekkad and Han’s [3] detailed heat transfer data which validated their code and demonstrated the second-moment closure model superiority in predicting flow and heat transfer characteristics in the ribbed duct. In a later study, Jang et al. [33] predicted flow and heat transfer in a rotating square channel with $45^\circ$ angled ribs by the same second-moment closure model. Heat transfer coefficient prediction was well matched with Johnson et al. [11] data for both stationary and rotating cases. Al-Qahtani et al. [34] predicted flow and heat transfer in a rotating two-pass rectangular channel with $45^\circ$ angled...
ribs by the same second-moment closure model of Chen et al. [23, 24]. Heat transfer coefficient prediction was compared with the data of Azad et al. [8] for both stationary and rotating cases. It predicted fairly well the complex three-dimensional flow and heat transfer characteristics resulting from the angled ribs, sharp 180° turn, rotation, centrifugal buoyancy forces and channel orientation.

This affirmed the superiority of the second-moment closure model compared to simpler isotropic eddy viscosity turbulence models. This model solves each individual Reynolds stress component directly from their respective transport equations. The primary advantage of this model is that it resolves the near-wall flow all the way to the solid wall rather than using log-law assumption in the viscous sublayer. With this near-wall closure, surface data like heat transfer coefficients and friction coefficients can be evaluated directly from velocity and temperature gradients on the solid wall.

In practice, the aerodynamic shape of the turbine blade dictates the use of cooling channels that are rectangular in cross section and are at an angle β from the direction of rotation. The effect of rotation, channel orientation and large channel aspect ratio on the secondary flow and heat transfer in rectangular channels may vary from the square channels. None of the previous studies predicted the characteristics of fluid flow and heat transfer in rotating rectangular channels that have an aspect ratio, AR, of 4:1 whether perpendicular or at an angle from the direction of rotation.

The objective of this study is to use the second moment RANS method of Chen et al. [23, 24] to (1) predict the three-dimensional flow and heat transfer for rotating smooth and ribbed one-pass rectangular ducts (AR = 4:1) and compare with the experimental data of Griffith et al. [14] and (2) to investigate the effect of high rotation and high density ratios on the secondary flow field and the heat transfer characteristics in a ribbed duct at 135° orientation.
2. DESCRIPTION OF PROBLEM

A schematic diagram of the geometry is shown in Figure 3.1. It has a rectangular cross section with channel aspect ratio, $AR$, of 4:1. Two geometries are investigated, one with smooth walls (Figure 3.1a) and the other one with ribs (Figure 3.1b). Two of the four side walls, in the rotational direction, are denoted as the leading and trailing surfaces, respectively, while the other two side walls are denoted as the top and bottom surfaces. The channel hydraulic diameter, $D_h$, is 0.8 in (2.03 cm). The distance from the inlet of the channel to the axis of rotation (Y-axis) is given by $R_r/D_h = 20.0$ and the length of the channel is given as $L/D_h = 22.5$. The channel consists of unheated starting smooth length ($L_1/D_h = 9.92$), heated smooth or ribbed section ($L_2/D_h = 7.58$) and unheated exit smooth section ($L_3/D_h = 5.00$). The arc length $S$ is measured from the beginning of the heated section to the end of it. In the ribbed section, the leading and trailing surfaces are roughened with nine equally spaced ribs of square cross section. The rib height-to-hydraulic diameter ratio ($e/D_h$) is 0.078 and the rib-pitch-to-height ratio ($P/e$) is 10. All ribs are inclined at an angle $\alpha = 45^\circ$ with respect to the flow. Two channel orientations are studied: $\beta = 90^\circ$ corresponding to the mid-portion of a turbine blade and $\beta = 135^\circ$ corresponding to the serpentine passages in the trailing edge region of a blade. A summary of the cases studied is given in Table 3.1.

<table>
<thead>
<tr>
<th>Case #</th>
<th>Surface</th>
<th>Ro</th>
<th>$\Delta \rho/\rho$</th>
<th>$\beta$</th>
<th>Compare with Exp.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Smooth</td>
<td>0.00</td>
<td>0.122</td>
<td>-</td>
<td>yes</td>
</tr>
<tr>
<td>2</td>
<td>Smooth</td>
<td>0.14</td>
<td>0.122</td>
<td>90°</td>
<td>yes</td>
</tr>
<tr>
<td>3</td>
<td>Smooth</td>
<td>0.14</td>
<td>0.122</td>
<td>135°</td>
<td>yes</td>
</tr>
<tr>
<td>4</td>
<td>Ribbed</td>
<td>0.00</td>
<td>0.122</td>
<td>-</td>
<td>yes</td>
</tr>
<tr>
<td>5</td>
<td>Ribbed</td>
<td>0.14</td>
<td>0.122</td>
<td>90°</td>
<td>yes</td>
</tr>
<tr>
<td>6</td>
<td>Ribbed</td>
<td>0.14</td>
<td>0.122</td>
<td>135°</td>
<td>NA</td>
</tr>
<tr>
<td>7</td>
<td>Ribbed</td>
<td>0.28</td>
<td>0.122</td>
<td>135°</td>
<td>NA</td>
</tr>
<tr>
<td>8</td>
<td>Ribbed</td>
<td>0.28</td>
<td>0.200</td>
<td>135°</td>
<td>NA</td>
</tr>
<tr>
<td>9</td>
<td>Ribbed</td>
<td>0.28</td>
<td>0.400</td>
<td>135°</td>
<td>NA</td>
</tr>
</tbody>
</table>

Table 3.1: Summary of cases studied, $Re = 10,000$. 
3. COMPUTATIONAL PROCEDURE

3.1 Overview

The Reynolds-Averaged Navier-Stokes equations in conjunction with a near wall Reynolds
stress turbulence model are solved using the chimera RANS method of Chen et al. [23, 24]. The
governing equations with the second-moment closure turbulence model were described in detail
by Chen et al. [23, 24] and will not be repeated here. The flow is considered to be
incompressible since the Mach number is quite low. However, the density in the centrifugal
force terms is approximated by \( \rho = \rho_o T_o / T \) to account for the density variations caused by the
temperature differences. \( \rho_o \) and \( T_o \) are the density and temperature at the inlet of the cooling
channel. In general, the density is also a function of the rotating speed because the centrifugal
force creates a pressure gradient along the duct. In the experiments of Griffith et al. [14], the
maximum pressure variation between the channel inlet and the exit is approximately 0.0113 atm
for the highest rotation number of 0.28 (i.e., \( \Omega = 550 \text{ rpm} \)) considered in the present study. This
gives a maximum density variation of only about 1.1% from the inlet to the exit of the duct at the
highest rotation number. It is therefore reasonable to omit the density variation caused by the
pressure gradients induced by the channel rotation. For completeness, the numerical method will
be briefly summarized in the following subsection.

3.2 Chimera RANS Method

The present method solves the mean flow and turbulence quantities in arbitrary combinations
of embedded, overlapped, or matched grids using a chimera domain decomposition approach. In
this approach, the solution domain was first decomposed into a number of smaller blocks to
facilitate efficient adaptation of different block geometries, flow solvers, and boundary
conditions for calculations involving complex configurations and flow conditions. Within each
computational block, the finite-analytic numerical method of Chen et al. [35] was employed to
solve the unsteady RANS equations on a general curvilinear, body-fitted coordinate system. The
coupling between the pressure and velocity was accomplished using the hybrid PISO/SIMPLER
algorithm of Chen and Patel [36]. The method satisfied continuity of mass by requiring the
contravariant velocities to have a vanishing divergence at each time step. Pressure was solved by
using the concept of pseudo-velocities and, when combined with the finite-analytic discretization
gives the Poisson equation for pressure. To ensure the proper conservation of mass and
momentum between the linking grid blocks, the grid-interface conservation techniques of Chen and Chen [37] were employed to eliminate the unphysical mass source resulting from the interpolation errors between the chimera grid blocks. In the present study, the numerical grids in the block overlap region are fully matched. Therefore, the grid interface conservation is automatically satisfied.

3.3 Boundary conditions

A uniform velocity profile was used at the inlet of the duct \((Z = 0)\). The unheated length \((L_1)\) was long enough for the velocity profile to be fully developed turbulent profile before the heating start-point \((Z = L_1)\). At the exit of the duct, zero-gradient boundary conditions were specified for the mean velocity and all turbulent quantities, while linear extrapolation was used for the pressure field. The coolant fluid at the inlet of the duct is air at uniform temperature \(T = T_o\) (i.e., \(\Theta = (T - T_o) / (T_w - T_o) = 0\)). The wall temperature of the unheated sections is kept constant at \(T = T_o (\Theta = 0)\) while the wall temperature of the heated section is kept constant at \(T = T_w (\Theta = 1)\).

3.4 Computational grid details

**Figure 3.2a and 3.2b** show the computational grid for the smooth duct and around the ribs for the ribbed duct. The grid was generated using an interactive grid generation code GRIDGEN [38]. It was then divided into five overlapped chimera grid blocks (three for the case of smooth duct) to facilitate the implementation of the near-wall turbulence model and the specification of the boundary conditions. To provide adequate resolutions of the viscous sublayer and buffer layer adjacent to a solid surface, the minimum grid spacing in the near-wall region is maintained at \(10^{-3}\) of the hydraulic diameter which corresponds to a wall coordinate \(y^+\) of the order of 0.5. The number of grid points in the streamwise direction from inlet to outlet is 50 for the smooth case and 394 for the ribbed duct. Whether smooth or ribbed, the number of grid points in the cross-stream plane is \(33 \times 75\). The number of grid points and their distributions in the present smooth duct were obtained based on extensive grid-refinement studies that were performed in Chen et al. [23, 24] and Al-Qahtani [25] for similar smooth channels of a square and rectangular cross sections. The interested reader is referred to references [23-25] for the details of the grid refinement studies performed on the smooth surface channels. The number of grid points and
their distributions in the present ribbed duct were obtained based on extensive grid-refinement studies that were performed in Jang et al. [31 and 32]. They performed grid refinement studies on 90° and 60° ribbed two-pass square channels with nine ribs and a total grid points of 1,060,000 and 1,020,000 (Re =30,000) respectively. Also, Jang et al. [33] performed a grid independence study on a 45° ribbed one-pass square channel with 13 ribs (Re =25,000). The numerical grids used in these studies were shown to yield nearly grid-independent results. Also, their results were in close agreement with the experimental data. Note that the number of grid points used in the present rectangular configuration is 33×75×394 grid points with a total number of approximately 1,000,000 points which is comparable to the above mentioned geometries. In addition, the Reynolds number used in the present study (Re = 10,000) is lower than the one used in the previous studies. Therefore, it is believed that the present grid will produce nearly grid-independent results with accurate resolution of the boundary layer profile and Nusselt number distribution. In all calculations, the root-mean-square (rms) and maximum absolute errors for both the mean flow and turbulence quantities were monitored for each computational block to ensure complete convergence of the numerical solutions and a convergence criterion of 10^{-5} was used for the maximum rms error.

4. RESULTS AND DISCUSSION

As summarized in Table 1, computations were performed for one Reynolds number (10,000), rotation numbers ranging from 0 to 0.28 and inlet coolant-to-wall density ratios Δρ/ρ ranging from 0.122 to 0.40 with two channel orientations of β = 90° and 135°. The Nusselt numbers presented here were normalized with a smooth tube correlation by Dittus-Boelter/McAdams (Rohsenow and Choi [39]) for fully developed turbulent non-rotating tube flow:

\[ N_{tu_0} = 0.023 \, Re^{0.8} \, Pr^{0.4} \]

4.1 Velocity and Temperature Fields

Before discussing the detailed computed velocity field, a general conceptual view about the secondary flow patterns induced by angled ribs and rotation is summarized and sketched in
Figure 3.3. The parallel angled ribs in the non-rotating duct (Figure 3.3a) produce symmetric counter rotating vortices that impinge on the top surface. The Coriolis force in the $\beta = 90^\circ$ rotating duct (Figure 3.3b) produces two additional counter-rotating vortices that push the cooler fluid from the core to the trailing surface. For the $\beta = 135^\circ$ rotating duct (Figure 3.3c), the Coriolis force produces two long vortices parallel to the ribbed surfaces and a third small vortex near the corner of the top-trailing surfaces. The effect of this rotation secondary flow is to combine destructively (opposite directions) with the rib induced secondary flow along the whole leading and trailing surfaces. This is an important concept that will help explain some of the coming flow and heat transfer characteristics.

4.1.1 Smooth Duct.

At two axial stations as defined in Figure 3.1a, Figures 3.4 through 3.6 show the calculated secondary flow vectors and constant temperature contours for the smooth cases as mentioned in Table 1. Note that these axial stations are viewed from upstream of the channel. It can be seen from Figure 3.4a that secondary corner vortices are generated as a result of the Reynolds stress anisotropy. It can be noticed from the corresponding temperature contour plots that the cooler fluid is located in the core region of the channel cross section. Further downstream (Figure 3.4b), the level of the secondary corner vortices is the same and the fluid in the duct core is heated more.

In Figure 3.5, the Coriolis forces produce a cross-stream two vortex flow structure (Figure 3.5a) that pushes the cold fluid from the core toward the trailing surface and then brings it back along the inner and outer surfaces to the leading surface. This means that the thermal boundary layer starts at the trailing surface, grows along the two side surfaces and ends at the leading surface. This results in small temperature gradient near the leading surface (hence lower heat transfer coefficients) and steeper one near the trailing surface (hence higher heat transfer coefficients) as seen from the corresponding contour plot of Figure 3.5a. Moreover, the cooler heavier fluid near the trailing surface will be accelerated by the centrifugal buoyancy force while the hotter lighter fluid near the leading surface will be decelerated to maintain the continuity in the streamwise direction. The Coriolis forces, in the $\beta = 135^\circ$ smooth duct (Figure 3.6a), produce a secondary flow that pushes the cold fluid away from the corner of the leading and top
surfaces. This produces two counter rotating vortices with the one near the leading surface stronger than the one near the trailing surface. It can also be noticed that a small vortex is generated at the corner of the top and trailing surfaces. As a result of this secondary flow, the fluid is pushed toward the bottom surface at which part of the secondary flow will move back along the trailing surface while the other part moves along the leading surface such that they meet again at the leading corner. This means that the thermal boundary layer starts at the bottom surface, grows along the trailing and leading surfaces and ends at the leading corner. This can be seen from the corresponding temperature contour plots where high temperature contours are located near the leading corner.

4.1.2 Ribbed Duct.

At several axial stations as defined in Figure 3.2a, Figures 3.7 through 3.10 show the calculated secondary flow vectors and constant temperature contours for the ribbed cases as mentioned in Table 1. Figure 3.7 shows the calculated secondary flow vectors and constant temperature contours for the non-rotating case (case 4). Since the ribs are oriented at a negative $45^\circ$ angle, the fluid adjacent to the top and ribbed surfaces will reach the ribs first and change direction along the ribbed surfaces toward the bottom surface (Figure 3.7a). It then returns back to the top surface along the centerline of the inclined cross-stream plane. In the same figure, one can also notice the early stages of two symmetric counter-rotating vortices, which become two full symmetric counter-rotating vortices in the midsection of any two ribs (Figure 3.7b). Along the streamwise direction, the size of these two vortices oscillates from the largest in the middle of each inter-rib distance to the smallest on the rib tops (Figure 3.7c). This pattern keeps repeating until the last rib (Figure 3.7d and 7e). The effect of the secondary flow on the temperature field is convecting the cooler fluid from the top surface and along the ribbed surfaces towards the bottom surface. It then moves back to the top surface which results in steep temperature gradients and high heat transfer coefficients on both the top and ribbed surfaces as seen in the corresponding temperature contours.

Figure 3.8 shows the cross-stream velocity vectors and temperature contours for case 5 ($\text{Ro} = 0.14$ and $\beta = 90^\circ$) at the same planes as in the non-rotating ribbed duct (case 4). As the flow approaches the first rib, this Coriolis force induced secondary flow starts to distort the secondary
flow started by the inclined ribs. This effect can be clearly seen by comparing Figures 3.8a through 8e with Figures 3.7a through 3.7e. From this comparison, the following conclusions can be drawn. (1) The magnitude of the Coriolis force induced secondary flow is weaker than the rib induced secondary flow. (2) In the midsections of each of two ribs, the rib induced vortex near the bottom surface is distorted slightly in the midsection of rib 1 and 2 (Figure 3.8b) but this distortion increases as the fluid proceeds downstream the duct (Figure 3.8d). (3) On the ribs (Figure 3.8c), both vortices shrink in size and get distorted only near the bottom. This pattern repeats itself until the last rib (Figure 3.8e). The general effect of the Coriolis force induced secondary flow is to distort the rib-induced vortices. Consequently, the temperature contours are shifted toward the trailing surface, which affects the heat transfer coefficients from both the leading and trailing surfaces as seen from the corresponding temperature contour plot.

Figure 3.9 shows the cross-stream velocity vectors and temperature contours for the low rotation low density ratio $\beta = 135^\circ$ (case 6) at the same planes as in cases 5 and 6. Comparing Figure 3.9 with Figure 3.8, the following can be noticed. Just before the ribbed section, the rotation induced secondary flow is still dominant as can be seen from comparing Figures 3.9a and 8a. However, from rib 1 on, this low rotation induced secondary flow is dominated by the rib induced secondary flow. A careful comparison between the secondary flow fields of case 6 and case 5 (e.g. Figure 3.9d with Figure 3.8d) shows that there is only minor change in the net effect of the secondary flow fields. This minor change appears more clearly in the temperature field. By comparing the temperature contours in Figure 3.9 with Figure 3.8, we notice that the cooler fluid is pushed back toward the leading surface, reducing the steep temperature gradients on the trailing surface.

As we increase the rotation number and density ratio, the strength of the rotation-induced secondary flow increases and gradually overcomes the rib induced secondary flow (recall Figure 3.3c). By reaching a rotation number of 0.28 and a density ratio of 0.40 as shown in Figure 3.10 (case 9), the rotation-induced secondary flow is found to be dominant over the rib induced secondary flow especially downstream of the channel. This is very clear by comparing the corresponding axial stations in Figures 3.10 and 3.9. This important result has its own consequence on the temperature field and thus the Nusselt number ratio distribution. The rib
induced secondary flow is not any more able to drive the secondary flow from the ribs leading side (near the top surface) to the ribs trailing side (near the bottom surface). On the contrary, the rotation induced secondary flow moves the cold fluid from the bottom surface along the ribbed surfaces with the secondary flow along the leading surface is much stronger than the one on the trailing surface. The temperature contours in Figure 3.10 indicate that the cold fluid is moved toward the bottom surface compared to Figure 3.9.

4.2 Detailed Local Heat Transfer Coefficient Distribution

4.2.1 Smooth Duct.

Figure 3.11a shows the $\frac{Nu}{Nu_0}$ contour plots on the leading and trailing surfaces for the non-rotating smooth case. The unheated sections were cut to focus on the heated section. The Nusselt number ratios near the beginning of the heated section are high due to the thinner boundary layers. Downstream, they decrease and asymptotically approach the fully developed value. Figure 3.11b and 3.11c show the $\frac{Nu}{Nu_0}$ ratio contours on the leading side for the $\beta = 90^\circ$ and $135^\circ$ rotating cases ($Ro = 0.14$ and $\Delta \rho / \rho = 0.122$). Compared to the non-rotating case, the heat transfer in the $\beta = 90^\circ$ case is lower because of the Coriolis force induced secondary flow which pushes the fluid away from the leading surface. For the $\beta = 135^\circ$ case, we notice that the $\frac{Nu}{Nu_0}$ ratios are high next to the bottom surface and then decrease toward the top surface. The reason for this is explained in the velocity section where it was mentioned that part of the cold fluid comes back from the bottom surface along the leading surface. This means that the thermal boundary layer grows on the leading surface as the secondary flow moves toward the top surface and thus heat transfer will be high at the bottom surface and then decreases towards the top surface. Figure 3.11d shows the $\frac{Nu}{Nu_0}$ ratio contours on the trailing side for the $\beta = 90^\circ$ rotating case ($Ro = 0.14$ and $\Delta \rho / \rho = 0.122$). The heat transfer is higher on this surface compared to the non-rotating case. This is again a result of the rotation induced secondary flow that pushes the cold fluid toward the trailing surface (see the velocity section). Figure 3.11e shows the $\frac{Nu}{Nu_0}$ ratios contours on the trailing side for the $\beta = 135^\circ$ rotating case ($Ro = 0.14$ and $\Delta \rho / \rho = 0.122$). Except for the entry region, the $\frac{Nu}{Nu_0}$ ratios are almost constant in the middle portion of the duct.

4.2.2 Ribbed Duct.
For various rotation numbers and density ratios, Figures 3.12 and 3.13 show the local Nusselt number ratio contours of the ribbed leading and trailing surfaces, respectively. The non-rotating case in Figure 3.12a (3.13a for the trailing surface) will be used as a baseline for comparison and discussion. Figures 3.12b through 3.12e (3.13b through 3.13e for the trailing surface) are for $\beta = 135^\circ$ while Figure 3.12f (13f for the trailing surface) is for $\beta = 90^\circ$. The entrance and exit regions were cut to focus on the ribbed heated section. First, the effect of the channel orientation on the Nusselt number ratios is discussed via comparing Figures 3.12b and 3.12f (3.13b and 3.13f for the trailing surface). Second, the effect of increasing the rotation number on the $\beta = 135^\circ$ Nusselt number ratios is discussed via Figures 3.12a through 3.12c (3.13a through 3.13c for the trailing surface). Third, the effect of increasing the density ratio on the $\beta = 135^\circ$ Nusselt number ratios is discussed via Figures 3.12c through 3.12e (3.13c through 3.13e for trailing surface).

In Figure 3.12a, the highest Nusselt number ratios were obtained on the top of the ribs, and the lower Nusselt number ratios were obtained right before and after the ribs. Between any two ribs, the Nusselt number ratios are highest near the top surface and decrease as we move towards the bottom surface. This is due to the rib induced secondary flow that moves from the top surface (and parallel to the ribbed walls) to the bottom surface.

*Effect of channel orientation on the leading and trailing surfaces:* For fixed rotation number and density ratio ($Ro = 0.14$ and $\Delta \rho/\rho = 0.122$), Figures 3.12b and 3.12f show the Nusselt number ratios contours on the leading side for $\beta = 135^\circ$ and $90^\circ$, respectively. Comparing these figures with the non-rotating leading side (Figure 3.12a), it is noticed that the Nusselt number ratios decreased in both cases with the decrease in the $\beta = 135^\circ$ case being the most (a 19% decrease compared to a 10% decrease in the $90^\circ$ case). Figures 3.13b and 3.13f show the Nusselt number ratios contours on the trailing side for $\beta = 135^\circ$ and $90^\circ$, respectively. Comparing these figures with the non-rotating trailing side (Figure 3.13a), it is noticed that the Nusselt number increased in both cases with the increase in $\beta = 135^\circ$ being the least (a 1% increase compared to a 5% increase in the $\beta = 90^\circ$ case). The reason why the Nusselt number ratios in the $\beta = 135^\circ$ case decreased more on the leading side and increased less on the trailing side compared to $\beta = 90^\circ$ case can be understood in light of the conceptual secondary flow diagram in Figure 3.3. The
rotation induced vortex in the $\beta = 135^\circ$ configuration move along the full face of the leading or trailing surfaces. However, the rotation induced vortex in the $\beta = 90^\circ$ configuration moves along only one half the face of the leading or trailing surfaces. With this in mind, we notice in Figure 3.3 that the two secondary flows produced by rotation and angled ribs for the rotating $\beta = 135^\circ$ duct combine destructively (opposite direction) and thus reduce heat transfer on both the leading surface and the trailing surface. On the other hand, the two secondary flows produced by rotation and angled ribs for the rotating $\beta = 90^\circ$ duct combine to 
(i) constructively (same direction) enhance heat transfer for only one half of each of the leading and trailing surfaces and 
(ii) destructively (opposite direction) reduce heat transfer for the other half of each of the leading and trailing surfaces.

**Effect of increasing the rotation number on the leading surface:** In Figure 3.12b, the rotation number is increased to 0.14 while the density ratio is kept fixed at 0.122. As discussed before, this causes the Nusselt number ratios to decrease by 19% compared to the non-rotating case (Figure 3.12a). But when the rotation number was increased to 0.28 (Figure 3.12c), the Nusselt number ratios decreased only by 10% compared to the non-rotating case. Moreover, it is noted that the high Nusselt number ratios regions are shifted to the middle of the ribbed surface. This is because of the rotation induced secondary flow getting stronger and gradually overcomes the rib induced secondary flow.

**Effect of increasing the density ratio on the leading surface:** In Figure 3.12d, the rotation number is kept fixed at 0.28 while the density ratio is increased to 0.20. It is seen from this figure that the high Nusselt number ratios regions are moved further toward the bottom surface. Increasing the density ratio further to 0.40 (Figure 3.12e), we notice that the high Nusselt number ratios regions are now existing next to the bottom surface with a total decrease of only 4% compared to the non-rotating case.

**Effect of increasing the rotation number on the trailing surface:** Figure 3.13 shows the same information as in Figure 3.12 but for the trailing surface. Figure 3.13a ($\text{Ro} = 0.00$) will be used as the baseline for comparison and discussion. As discussed before, increasing the rotation number to 0.14 (Figure 3.13b) causes the Nusselt number ratios to increase only by 1%
compared to the non-rotating case. In Figure 3.13c, the rotation number is increased further to 0.28 while the density ratio is kept fixed at 0.122. This causes the Nusselt number ratios to increase by 6% compared to the non-rotating case. Also, it is seen from this figure that the high Nusselt number ratios regions are spreading toward the bottom surface.

**Effect of increasing the density ratio on the trailing surface:** In Figure 3.13d, the rotation number is kept fixed at 0.28 while the density ratio is increased to 0.20. It is seen in this figure that the high Nusselt number ratios regions are pushed slightly more toward the bottom surface. Increasing the density ratio further to 0.40 (Figure 3.13e) causes the Nusselt number ratios to increase by 12% compared to the non-rotating case. It is also seen from this figure that, upstream of the channel, the high Nusselt number ratios are moved toward the bottom surface while downstream they dominate most of the inter-rib regions.

4.3 Spanwise-Averaged Heat Transfer Coefficients and Comparison with Experimental Data

4.3.1 Smooth Duct.

In Figure 3.14, comparisons of the spanwise-averaged Nusselt number ratios \((\text{Nu}/\text{Nu}_o)\) were made with the experimental data of Griffith et al. [14]. In order to compare the effects of the channel orientation on the heat transfer, Figure 3.14 shows the Nusselt number ratios for the three smooth cases: 1, 2 and 3. In this Figure, the inlet coolant-to-wall density ratio was held constant at value of 0.122. The effect of the model orientation can be seen by comparing the \(\beta = 135^\circ\) Nusselt number ratios with the \(\beta = 90^\circ\) ones. It can be seen that the \(\beta = 135^\circ\) Nusselt number ratios are higher on the leading and bottom surfaces, and lower on the trailing and top surfaces. This can be explained in terms of the secondary flow patterns and temperature contours shown in Figures 3.5 and 3.6. For the \(\beta = 90^\circ\) case, the cold fluid reaches the leading surface after it passes over the trailing surface and both of the two side surfaces. On the other hand, the cold fluid in the \(\beta = 135^\circ\) case moves directly to the bottom surface at which it splits and comes back along the leading and trailing surfaces. When the channel orientation was changed from \(\beta = 90^\circ\) to \(135^\circ\), more cold fluid was flowing to the leading surface while the trailing surface received less cold fluid. This has led to higher heat transfer on the leading surface and lower heat transfer on the trailing surface for the \(\beta = 135^\circ\) case. For the top surface, the lower Nusselt number ratios observed in the \(\beta = 135^\circ\) rotating case can be attributed to the fact that most of the top surface behaves as a leading surface in the sense that the fluid is moving away from this
surface. Similarly, the $\beta = 135^\circ$ bottom surface behaves as a trailing surface with high heat transfer since it receives the cold fluid directly from the duct core. Comparisons with the experimental values reveal the following: (1) for the non-rotating case, the matching between the experimental and prediction is good on all surfaces, (2) fair agreement on the leading, trailing, top and bottom sides is achieved for the rotating cases of $\beta = 90^\circ$ and $135^\circ$.

4.3.2 Ribbed Duct.

Figures 3.15 and 3.16 show the spanwise-averaged and regional-averaged Nusselt number ratios ($Nu/Nu_o$) for the ribbed cases 4 ($\beta = 90^\circ$) and 5 ($\beta = 135^\circ$). The rotation number and the inlet coolant-to-wall density ratio were held constant at values of 0.14, and 0.122, respectively. Note that the experimental regional-averaged Nusselt number in Griffith et al. [14] is based on the projected area of each copper plate rather than the true heat transfer surface area which includes the $45^\circ$ rib-increased area. However, the predicted regional-averaged Nusselt Number is based on the true heat transfer area for the test surfaces with $45^\circ$ ribs which is 1.25 times the projected area. Therefore, the experimental data were divided by 1.25 to reasonably compare with our regional-averaged Nusselt number, except for the inner and outer surfaces where there were no ribs. The predicted Nusselt number ratios on the leading and trailing surfaces are in good agreement with Griffith et al. [14] data for the non-rotating case (Figure 3.15) while relatively close to the experimental data in the rotating case (Figure 3.16). Downstream of the channel, the predicted Nusselt numbers on the top and bottom surfaces are mildly over-predicted and under-predicted, respectively. This may be partly attributed to the fact that the predicted Nusselt number ratios are based on a uniform wall temperature boundary condition while the experimental ones are based on a uniform wall heat flux boundary condition.

The spanwise-averaged Nusselt number distributions on the leading and trailing surfaces of Figures 3.15 and 3.16 show periodic spikes. The higher spikes which occur on the ribs tops are caused by the flow impingement on the ribs, and the lower spikes (which occur right before and after the ribs) are caused by the flow reattachment between the ribs. The Nusselt number ratios are high in the regions between the ribs. The Nusselt number ratios increase until the last rib, which is similar to the results obtained in Jang’s et al. [33] 45°-ribbed square channel and Al-Qahtani’s et al. [34] 45°-ribbed rectangular channel (AR = 2). This phenomenon is caused by
the rib-induced secondary flow becoming stronger along the duct as discussed in Figure 3.7. The Nusselt number distribution on the top surface of Figures 3.15 and 3.16 shows that it increases all the way to rib 9 as a result of the secondary flow that pushes the cold fluid towards the top surface. For the same reason, the Nusselt number distribution on the bottom surface is decreasing (although mildly) since it receives the heated fluid from the ribbed surfaces.

Figures 3.17 shows the spanwise-averaged Nusselt number ratios \( \left( \frac{Nu}{Nu_c} \right) \) for the \( \beta = 135^\circ \) ribbed cases 6 and 9 which presents a comparison between the low-rotation low-density ratio case and the high-rotation high-density ratio cases which is close to the real rotor cooling conditions. The following observations are obtained by comparing Figure 3.17a with 3.17b. (1) It is seen that the Nusselt number ratios on the top surface of case 6 were higher than the ones on the bottom surface due to the rib induced secondary flow which convects the cooler fluid along the ribbed surfaces and then back to the top surface resulting into higher Nusselt number ratios on the top surface. However, in case 9 which represents the high rotation high density ratio range, the situation is reversed where the Nusselt number ratios on the bottom surface are higher than the ones on the top surface. This is a direct result of the rotation induced secondary flow which pushes the cold fluid toward the bottom surface. For the leading and trailing surfaces, the Nusselt number ratios spikes are higher in case 6 compared to case 9, however, the Nusselt number ratios in the inter-rib regions are higher in case 9 compared to case 6. It is found that the overall Nusselt number ratios of case 9 on the leading and trailing surface are higher than those in case 6 by 18% and 11%, respectively.

5. CONCLUSIONS

A multi-block RANS method was employed to predict three-dimensional flow and heat transfer in a rotating smooth and ribbed rectangular channel with aspect ratio of 4:1 and for various rotation numbers and inlet coolant-to-wall density ratios. Two channel orientations are studied: \( \beta = 90^\circ \) and \( 135^\circ \). The present near-wall second-moment closure model results were compared with the experimental data of Griffith et al. [14]. It predicted fairly well the complex three-dimensional flow and heat transfer characteristics resulting from the large channel aspect
ratio, rotation, centrifugal buoyancy forces and channel orientation. The main findings of the study may be summarized as follows.

**A) Smooth duct:**

I. The Coriolis force induces secondary flow, in the $\beta = 90^\circ$ rotating case, which pushes the cold fluid from the leading to the trailing surface.

II. The Coriolis force induces secondary flow, in the $\beta = 135^\circ$ rotating case, which pushes the cold fluid from the leading corner to the bottom surface.

III. In the $\beta = 135^\circ$ rotating case, most of the top surface behaves as a leading side and thus the Nusselt number ratios on this surface are lower than the corresponding ones on the $\beta = 90^\circ$ rotating case. Similarly, most of the bottom surface behaves as a trailing side. Thus, the increase in the Nusselt number ratios is higher on the bottom surface when compared with their counterparts in the $\beta = 90^\circ$ rotating case.

**B) Ribbed duct:**

I. The inclined ribs start two counter-rotating vortices that oscillate in size along the streamwise direction. For case 4 (non-rotating), the secondary flow results in steep temperature gradients and high heat transfer coefficients on both the top and ribbed surfaces.

II. For case 5 ($\beta = 90^\circ$), the rotation-induced cross-stream secondary flow distorts the rib-induced vortices and consequently, rotation shifts the temperature contours and affects the heat transfer coefficients from both the leading and trailing surfaces.

For case 6, 7, 8 and 9 ($\beta = 135^\circ$):

III. The rib-induced vortices are slightly distorted by low rotation-induced secondary flow (case 6) but significantly changed by the high rotation high density ratio induced secondary flow. This results into reversing the flow on the leading surface and reduces significantly the magnitude of rib-induced secondary flow on the trailing surface.

IV. The effect of increasing the rotation number (with fixed density ratio) is to monotonically increase the Nusselt number ratio on the trailing surface. On the leading surface, the Nusselt number ratio decreases first (case 6) and then increases (case 7).

V. The effect of increasing the density ratio (with fixed rotation number) is to have higher and uniform Nusselt number ratio on the leading and trailing surfaces.
VI. From design point of view, it is clear that the rib angle and the direction of rotation should be chosen such that the secondary flows that are induced by the rib angle and rotation direction should combine constructively to give maximum heat transfer.

ACKNOWLEDGEMENT

The leading author, Mohammad Al-Qahtani, received a fellowship from King Fahd University; Saudi Arabia, for his Ph.D. study at Texas A&M University. The project was also supported by the DOE Advanced Gas Turbine Systems Research (AGTSR) Program through project number SR-082. The computations were performed on the Cray J90 at the Texas A&M Supercomputer Center under a supercomputer research grant, and Cray C90 of Cray Research Inc. under the sponsorship of Frank Kampe. The GRIDGEN software was supported by Pointwise Inc. The support of all of the above institutions is greatly appreciated.

REFERENCE


ONGOING WORK

Part I: Rotating Heat Transfer

• Dimple plates of three different dimple depths (10%, 20%, and 30% of dimple diameter) have been manufactured using and CNC mill.

• Heat transfer experiments are completed and reported here for AR=4:1 channel with 30% depth dimple plates.

• Acrylic pins have been obtained for the pin-fin experiments. These experiments are currently underway to obtain the effect of rotation on the cooling channel with pin fins.

• The entry effect will be explored after the pin-fin experiments are finished. All test section inserts necessary to modify the test section inlet have been manufactured.

• Stationary pressure drop experiments are completed to determine the pressure drop in three different dimple depths (10%, 20%, and 30% of dimple diameter). The test section is made of acrylic and will later be used for liquid crystal thermography experiments.

• Rotation effects on heat transfer in 4:1 aspect ratio channels with 45-degree angled ribs were completed and reported in June 2001.
Part II: Stationary Heat Transfer and Flow Field

- Construction of test sections: The pin fins are completed, the rib turbulators are completed, and the dimples are completed except for heater installation, which is presently underway by Electrofilm Corp. Valencia, Ca.

- Flow structure measurements: The pin fins, the dimples, and the rib turbulators are completed.

- Heat transfer measurements (data obtained at one temperature ratio and different Reynolds numbers): The rib turbulators are completed, the pin fins are almost completed. Also, the dimples are scheduled for the near future.

- Heat transfer measurements (data obtained at different temperature ratios and one Reynolds number): The rib turbulators are completed, and the pin fins are almost done. The dimples are scheduled for the near future.

Part III: Numerical Prediction

- Boundary-fitted numerical grids were generated for a rectangular channel (AR = 4:1) with pin fins using the GRIDGEN code.

- Calculations of flow field, heat transfer coefficients, and pressure drops is currently being performed for the pin fin configurations under various combinations of rotation number, coolant to wall temperature ratio, and channel orientation. Both the two-layer k-ε model and near-wall second-moment closure model will be used to quantify the effects of Reynolds stress anisotropy.
• The numerical results will be compared to the experimental data obtained in the above-mentioned flow and heat transfer measurements to assess the general performance of the RANS code and turbulence models.

• Boundary-fitted numerical grids will be generated for a rectangular channel (AR = 4:1) with dimples using the GRIDGEN code.

• Calculations of flow field, heat transfer coefficients, and pressure drops will be performed for the dimple configuration under various combinations of rotation number, coolant to wall temperature ratio, and channel orientation. Both the two-layer k-ε model and near-wall second-moment closure model will be used to quantify the effects of Reynolds stress anisotropy.

• The numerical results will be compared to the experimental data obtained in the above-mentioned flow and heat transfer measurements to assess the general performance of the RANS code and turbulence models.
APPENDIX I: FIGURES FOR PART 1
Figure 1. Sketch illustrating orientation of a 4:1 aspect ratio channel in a gas turbine blade

\[ \delta = \text{depth} = 0.3 \times \frac{1}{4} \text{ in} = 0.3 \times 0.64 \text{ cm} \]

Figure 2. Schematic of 4:1 dimpled test
Figure 1.3. Annotation and data legend for surfaces within the 4:1 Channel ($\beta=135^\circ$)

Figure 1.4. Dimple induced secondary flow (conceptualization)
Figure 1.5. Rotation-induced (Coriolis force) vortices in rectangular channel.
Figure 1.6. Nusselt number ratio for stationary smooth case.
Figure 1.7. Nusselt number ratio for rotation smooth case with $\beta = 90^\circ$.
Figure 1.8. Nusselt number ratio for smooth case with $\beta = 135^\circ$.
Figure 1.9. Nusselt number ratio for stationary dimpled case.
Figure 1.10. Nusselt number ratio for rotation dimpled case with $\beta = 90^\circ$. 
Figure 1.13. Streamwise averaged Nusselt number ratio for smooth and dimpled channel with $\beta = 135^\circ$. 
Figure 1.12. Streamwise Averaged Nusselt number ratio for smooth and dimpled channel with $\beta = 90^\circ$. 

![Graph showing streamwise averaged Nusselt number ratio](image)
Figure 1.13. Streamwise averaged Nusselt number ratio for smooth and dimpled channel with $\beta = 135^\circ$. 
APPENDIX II: FIGURES FOR PART 2
Figure 2.1. Schematic diagrams of: (a) the experimental apparatus used for heat transfer measurements, and (b) the experimental apparatus used for flow visualizations and measurements of flow structure.
Figure 2.2. Schematic diagram of the rib turbulator test surfaces, including coordinate system.

Figure 2.3. Baseline Nusselt numbers, measured with smooth channel surfaces and constant heat flux boundary condition, as dependent upon Reynolds number based on hydraulic diameter.

\[ \text{Nu}_0 = 0.023(\text{Re}_{Dh})^{.8} (\text{Pr})^{.4} \]
Figure 2.4. Local Nusselt number ratio $Nu/Nu_o$ distribution along the rib turbulator test surface for $Re_H=10,000$ and $T_{oi}/T_w=0.95$.

Figure 2.5. Local Nusselt number ratio $Nu/Nu_o$ distribution along the rib turbulator test surface for $Re_H=90,000$ and $T_{oi}/T_w=0.94$. $Nu/Nu_o$ scale is presented in Fig. 2.4.
Figure 2.6. Local Nusselt number ratios $\frac{Nu}{Nu_0}$ along the rib turbulator test surface at $Z/D_h=0.0$ for different Reynolds numbers $Re_H$ and $T_{in}/T_w$ of 0.93-0.95.

Figure 2.7. Local Nusselt number ratios $\frac{Nu}{Nu_0}$ along the rib turbulator test surface at $X/D_h=6.90$ for different Reynolds numbers $Re_H$ and $T_{in}/T_w$ of 0.93-0.95. Symbols are defined in Fig. 2.6.
Figure 2.8. Time-sequence of instantaneous flow visualization images illuminated in a spanwise-normal plane at $x=1462$ mm for $Re_H=480$. The spanwise center of each image is located at $z/D_h=0.0$. 
Figure 2.9. Instantaneous flow visualization images illuminated in a spanwise-normal plane at \( x = 1462 \) mm for different Reynolds numbers. The spanwise center of each image is located at \( Z / D_h = 0.0 \).
Figure 2.10. Time-averaged surveys of: (a) local streamwise velocity, (b) local total pressure, (c) local static pressure, and (d) local streamwise vorticity from surveys conducted in a spanwise normal plane at \( x = 1235 \) mm for \( Re_H = 10,500 \).
Figure 2.11. Schematic diagram of a portion of the bottom rib turbulator test surface showing the orientations and layout of several rib turbulators, and the coordinates $W/D_h$ and $L/D_h$, which are oriented perpendicular to and parallel to the rib turbulators, respectively.

Figure 2.12. Nusselt number ratios $\overline{Nu}/Nu_o$, for fully-developed conditions measured at the downstream end of the test section and averaged in the $W/D_h$ direction, as dependent upon the $L/D_h$ coordinate for different Reynolds numbers and $T_{oi}/T_w = 0.93-0.95$. Symbols are defined in Fig. 2.13.
Figure 2.13. Nusselt number ratios $\overline{Nu}/Nu_o$, for fully-developed conditions measured at the downstream end of the test section and averaged in the $L/D_h$ direction, as dependent upon the $W/D_h$ coordinate for different Reynolds numbers and $T_o/T_w = 0.93-0.95$.

Figure 2.14. Nusselt number ratios $\overline{Nu}/Nu_o$, for thermally developing flow measured at the upstream end of the test section and averaged in the $W/D_h$ direction, as dependent upon the $L/D_h$ coordinate for $Re_H=60,800$ and $T_o/T_w = 0.93-0.95$. Symbols are defined in Fig. 2.15.
Figure 2.15. Nusselt number ratios $\overline{\text{Nu}} / \text{Nu}_0$, for thermally developing flow measured at the upstream end of the test section and averaged in the $L / D_h$ direction, as dependent upon the $w / D_h$ coordinate for $Re_H = 60,800$ and $T_{oi} / T_w = 0.93-0.95$.

Figure 2.16. Rib turbulator channel globally-averaged Nusselt number ratios for fully developed flow, averaged over the surface area corresponding to one period of rib turbulator geometry, as dependent upon Reynolds number for $T_{oi} / T_w = 0.93-0.95$. Comparisons with results from other investigations [2,4,5] are included.
Figure 2.17. Rib turbulator channel friction factor ratios $f/f_o$ for fully developed flow conditions as dependent upon Reynolds number for $T_{oi}/T_w=0.93-0.95$. Symbols are defined in Fig. 2.16. Comparisons with results from other investigations [2,4,5] are included.

Figure 2.18. Rib turbulator channel globally-averaged Nusselt numbers for fully developed flow and $T_{oi}/T_w=0.93-0.95$ as dependent upon friction factor ratios, including comparisons with results from other investigations [2,4,5]. Symbols are defined in Fig. 2.16.
Figure 2.19. Rib turbulator channel globally-averaged performance parameters for fully developed flow and $T_w/T_\infty=0.93-0.95$ as dependent upon Reynolds number, including comparisons with results from other investigations [2,4,5]. Symbols defined in Fig. 2.16.
APPENDIX III: FIGURES FOR PART 3
Figure 3.1(a) Geometry of smooth rectangular duct with AR = 4:1.

- $R_r/D_h = 20.0$
- $L/D_h = 22.5$
- $L_1/D_h = 9.92$
- $L_2/D_h = 7.58$
- $L_3/D_h = 5.00$
- $a: Z/D_h = 15.01$
- $b: Z/D_h = 20.04$
Figure 3.1(b) Geometry of ribbed rectangular duct with AR = 4:1.
Figure 3.2 (a) Computational grid of smooth rectangular duct with AR = 4:1.
Figure 3.2 (b) Computational grid of ribbed rectangular duct with AR = 4:1.
Figure 3.3 Conceptual view of the secondary flow induced by angled ribs and rotation.

(a) Non-rotating
   Ro = 0.0
   Δρ/ρ = 0.122

(b) Rotating
   Ro = 0.14,
   Δρ/ρ = 0.122, β = 90°

(c) Rotating
   Ro = 0.14,
   Δρ/ρ = 0.122, β = 135°
Figure 3.4. Secondary-flows and dimensionless temperature $[\Theta = (T - T_o)/(T_w - T_o)]$ for smooth non-rotating duct, $Ro = 0.0$. 

(a) $Z/D_s = 12.33$

(b) $Z/D_s = 16.24$
Figure 3.5. Secondary flows and dimensionless temperature \( [(\Theta = (T - T_o)/(T_w - T_o))] \) for Ro = 0.14, \( \Delta \rho/\rho = 0.122 \) and \( \beta = 90^\circ \).
Figure 3.6. Secondary-flows and dimensionless temperature $[\Theta = (T - T_o)/(T_w - T_o)]$ for Ro = 0.14, $\Delta \rho/\rho = 0.122$ and $\beta = 135^\circ$. 
Figure 3.7 Secondary flow and temperature $[\Theta = (T \cdot T_w)/(T_w - T_o)]$ for non-rotating ribbed duct, $Ro = 0.00$. 
Figure 3.8 Secondary flow and temperature $[\Theta = (T - T_0)/(T_w - T_0)]$ for rotating ribbed duct, $Ro = 0.14$ and $\beta = 90^\circ$. 
Figure 3.9 Secondary flow and temperature \([\Theta = (T - T_o)/(T_w - T_o)]\) for rotating ribbed duct, \(Ro = 0.14, \Delta \rho/\rho = 0.122\) and \(\beta = 135^\circ\).
Figure 3.10 Secondary flow and temperature [$\Theta = (T - T_o)/(T_w - T_o)$] for rotating ribbed duct, $Ro = 0.28$, $\Delta\rho/\rho = 0.40$ and $\beta = 135^\circ$. 
Figure 3.11 Detailed Nusselt number distribution in smooth duct.
Figure 3.12 Leading surface detailed Nusselt number ratio distribution in ribbed duct.
Figure 3.13 Trailing surface detailed Nusselt number ratio distribution in ribbed duct.
Figure 3.14 Effect of rotation and channel angle on Nusselt number distribution for smooth duct, Re = 10,000.
Figure 3.15 Calculated and measured Nusselt number ratio distribution for non-rotating ribbed duct, Re = 10,000.
Figure 3.16 Calculated and measured Nusselt number ratio distribution for rotating ribbed duct ($Ro = 0.14$), $Re = 10,000$. 

- Leading Surface
- Trailing Surface
- Top Surface
- Bottom Surface

$Ro=0.14$
$\Delta \rho / \rho = 0.122$
$\beta = 90^\circ$

- Prediction: Spanwise average
- Prediction: Regional average
- Todd et al. [14]
Figure 3.17 Effect of rotation and density ratio on 135 ° Nusselt number ratio distribution for Re = 10,000.
Rotating Heat Transfer in High Aspect Ratio Rectangular Cooling Passages with Shaped Turbulators

J.C. Han
(Principal Investigator)
Department of Mechanical Engineering, Texas A&M University
Ph: (979) 845-3738 **Fax: (979) 862-2418 **E-mail: JCHan@mengr.tamu.edu

and

H.C. Chen
(Co-Principal Investigator)
Department of Civil Engineering, Texas A&M University

December 1, 2001
# TABLE OF CONTENTS

## PROJECT SUMMARY ................................................................. 2

Part I - Rotating Heat Transfer: ............................................... 2  
Part II - Computational Study: .............................................. 3

## PART I: ROTATING HEAT TRANSFER ........................................ 4

ABSTRACT .................................................................................. 5
NOMENCLATURE ........................................................................ 5
INTRODUCTION .......................................................................... 7
EXPERIMENTAL FACILITY ...................................................... 11
DATA REDUCTION ..................................................................... 13
RESULTS AND DISCUSSION ................................................... 15
  Smooth Channel Results....................................................... 19
  Parallel 45° V-Shaped Rib Case ........................................... 21
  Crossed 45° V-Shaped Rib Case .......................................... 24
  Channel Averaged Nusselt Number Ratio ........................... 27
CONCLUSIONS ........................................................................... 7
REFERENCES ............................................................................. 29

## PART II: COMPUTATIONAL STUDY .......................................... 32

GEOMETRY AND NUMERICAL GRIDS ................................... 33
NUMERICAL METHOD ............................................................. 35
RESULTS AND DISCUSSION ................................................... 36

ONGOING WORK ......................................................................... 43

Part I: Rotating Heat Transfer ................................................. 43
Part III: Numerical Prediction ................................................ 43
PROJECT SUMMARY

The objective of this two-part investigation is to provide the designers with new internal cooling data for improving the cooling performance and thermal efficiency of power generation and industrial gas turbine engines. More specifically, this investigation ventures into the heat transfer phenomenon of internal cooling channels with shaped turbulators. The research is divided into two parts: Part I - Rotating Heat Transfer and Part II - Numerical Prediction. This investigation is a collaboration between Dr. J.C. Han and Dr. H.C. Chen of Texas A&M University. This report details the first stage of this investigation, namely the rectangular channel with V-shaped rib turbulators. A more detailed breakdown of the investigation of each of the two parts follows.

Part I: Rotating Heat Transfer

The objectives of part I are to obtain experimental data from rectangular, two-pass internal cooling passages with higher aspect ratios of 2:1 and 4:1. The following parameters will be altered: (1) Surface geometry, (2) Reynolds numbers, (3) rotation number, (4) rotation angle, and (5) channel aspect ratio. Angled ribs, V-shaped ribs, and delta-shaped turbulators will be installed on the leading and trailing sides of a rectangular internal cooling passage with rotation. The ratio of inlet coolant temperature to surface temperature (TR) will be around 0.8 - 0.9. The experiments will be designed so that (a) regionally averaged heat transfer coefficients will be measured at different locations along the cooling passages with enhanced surfaces, and (b) pressure drops will be measured along the cooling passages under rotating conditions. The new heat transfer and pressure drop data will be correlated and compared with numerical predictions.
in part II. The existing rotating facility and instrumentation available in the Turbine Heat Transfer Laboratory of Texas A&M University will be employed for the proposed study.

**Part II: Computational Study**

The objectives of part II are to predict flow and heat transfer behaviors from rectangular, two-pass internal cooling passages with higher aspect ratios of 2:1 and 4:1. An ongoing Chimera Reynolds-Averaged Navier-Stokes (RANS) code together with an advanced state of the art second-order Reynolds stress (second moment) turbulence model will be used for the prediction of rotating rectangular cooling channels with angled ribs, V-shaped ribs, and delta-shaped turbulators. The present numerical model has been tested to provide much better flow and heat transfer predictions than the standard k-ε turbulence model for rotating multi-pass square channels with angled ribs. The numerical predictions will be calibrated/compared with the part I-rotation heat transfer data. The ultimate goal is to predict and optimize flow and heat transfer in rotating rectangular channels with various shaped turbulators at very high Reynolds number and buoyancy parameter conditions.
PART I: ROTATING HEAT TRANSFER

Part 1 is conducted at Texas A&M University under the direction of Dr. J.C. Han
ABSTRACT

An experimental study was made to obtain heat transfer data for a two-pass rectangular channel (aspect ratio=2:1) with smooth and ribbed surfaces for two channel orientations (90° and 135° to the direction of rotational plane). The rib turbulators are placed on the leading and trailing surfaces at 45° V-shaped orientation to the main stream flow. Two different arrangements of 45° V-shaped ribs are studied. The Reynolds number and rotation numbers were varied in 5000-40000, and 0.0-0.21, respectively. The rib height to hydraulic diameter ratio (e/D) is 0.094; the rib pitch-to-height ratio (P/e) is 10 and the inlet coolant-to-wall density ratio (Δρ/ρ) is maintained at 0.115 for all surfaces in the channel. The results show that the rotation induced secondary flow enhances the heat transfer of trailing surface first pass and leading surface second pass. However, the first pass leading and the second pass trailing surfaces show a decrease in heat transfer with rotation. The results show that 45° V-shaped parallel ribs arrangements produce better heat transfer augmentation than 45° V-shaped cross ribs, and a 90° channel orientation produces higher heat transfer effect over a 135° orientation.

NOMECLATURE

D        hydraulic diameter (m)
e        rib height (m)
h        heat transfer coefficient (W/m² k)
\( k \) \hspace{1cm} \text{thermal conductivity of coolant (W/mK)}

\( Nu \) \hspace{1cm} \text{local Nusselt number, } hD /k

\( Nu_o \) \hspace{1cm} \text{Nusselt number in fully-developed turbulent non-rotating tube flow without ribs}

\( P \) \hspace{1cm} \text{rib pitch (m)}

\( Pr \) \hspace{1cm} \text{Prandtl number}

\( q_{net} \) \hspace{1cm} \text{net heat at wall (W/m}^2 \text{)}

\( A \) \hspace{1cm} \text{surface area of the copper plate (m}^2 \text{)}

\( Re \) \hspace{1cm} \text{Reynolds number, } \rho VD /\mu

\( Ro \) \hspace{1cm} \text{rotation number, } \Omega D/V

\( T_{bx} \) \hspace{1cm} \text{local coolant temperature (°C)}

\( T_{bi} \) \hspace{1cm} \text{coolant temperature at inlet (°C)}

\( T_w \) \hspace{1cm} \text{wall temperature (°C)}

\( V \) \hspace{1cm} \text{bulk velocity in streamwise direction (m/s)}

\( X \) \hspace{1cm} \text{distance measured along the channel axis from the start of the heating (m).}

\( \beta \) \hspace{1cm} \text{angle of channel orientation with respect to the axis of rotation}

\( \Omega \) \hspace{1cm} \text{rotational speed (rad/s)}

\( \mu \) \hspace{1cm} \text{dynamic viscosity of coolant (Pa-s)}

\( \rho \) \hspace{1cm} \text{density of coolant (kg/m}^3 \text{)}

\( \Delta \rho \) \hspace{1cm} \text{inlet coolant-to-wall density ratio, } \frac{T_w-T_{bi}}{T_w}$
INTRODUCTION

To achieve high thermal efficiency, and low cost of a gas turbine engine, turbine inlet gas temperature should be increased. However, the penalty is high thermal load, which exceeds the durability of the turbine components. Therefore, turbine blade is equipped with improved cooling techniques such as film cooling and internal cooling. Internal cooling is achieved by low enthalpy air circulating in multi-pass flow channels inside the blade structure. To increase the effectiveness of the internal cooling, the internal surfaces usually are roughened by angled ribs to trip the boundary layer and increase the turbulence, which enhance the heat transfer. As the turbine blade rotates, Coriolis and buoyancy forces appear and cause different heat transfer behavior from the leading and trailing surfaces. Coriolis force produces secondary flow in planes perpendicular to the main flow direction, which encourages the migration of core region flow toward the trailing surface in the first pass and leading surface in the second pass.

Over the past few decades numerous studies have been made experimentally on the flow field and heat transfer in the internal coolant passage of gas turbine rotor blade. Metzger et al.\textsuperscript{1} studied forced convection in two-pass smooth rectangular channels by varying the divider location and the gap at the 180\textdegree
turn. Fan et al.\textsuperscript{2} extended the Metzger et al.\textsuperscript{1} work by varying the channel width and the conclusion was, increasing channel aspect ratio results in smaller azimuthal heat transfer variations and increased overall channel heat transfer. Han and Park\textsuperscript{3} performed experimental studies on heat transfer characteristics in a non-rotating rib roughened rectangular channel. Han et al.\textsuperscript{4} studied the effect of the rib angle orientation on heat transfer distributions and pressure drop in a non-rotating square channel with two opposite in-line ribbed walls. They
found that the 60° (or 45°) V-shaped rib performs better than the 60° (or 45°) parallel rib and, subsequently, better than the 60° (or 45°) crossed rib and the 90° rib. The V-shaped rib produced the highest heat transfer augmentation, while the crossed rib had the lowest heat transfer enhancement. Ekkad and Han performed a detailed study on heat transfer characteristics in a non-rotating square channels using liquid crystals techniques. The results show that the parallel, 60° V, and 60° V inverted ribbed channels produce similar levels of heat transfer enhancement in the first pass, while the 60° inverted V ribbed channel produced higher enhancement in the second pass. Wagner et al. conducted the detailed experimental study to determine the effects of rotation (buoyancy and Coriolis forces) on the local heat transfer of a multi-pass square channel with smooth walls. They concluded that in the first pass of the coolant passage rotation created a thinner boundary layer on the trailing surface and a thicker boundary layer on the leading surface but in the second pass the performance was different and opposite to the first pass. The leading surface Nusselt number ratios in the second pass were higher than the trailing surface Nusselt number ratios because of the reversal of the Coriolis force direction. Johnson et al. performed a systematic investigation of the effects of buoyancy and Coriolis forces on heat transfer coefficients distribution of four-pass square channels with trips angled to the flow (45° ribs). Han et al. investigated uneven wall temperature effect on local heat transfer in a rotating two-pass square channel with smooth walls. Zhang et al. analyzed the heating condition effects in a duct with angled rib turbulators with rotation. They suggested that an uneven wall temperature had a significant impact on the local heat transfer coefficients. Parsons et al. presented wall-heating effect on local heat transfer in a rotating two-pass square channel with orthogonal ribs. Johnson et al. and Parsons et al. studied the effects of channel orientation and wall heating condition on local heat transfer coefficient in a rotating two-pass square channel.
with ribbed walls. They found that the effects of the Coriolis force and cross-stream flow were reduced as the channel orientation changed from normal $\beta=90^\circ$ to an angled orientation $\beta=135^\circ$. Dutta and Han\textsuperscript{14} also investigated the local heat transfer coefficients in rotating smooth and ribbed two-pass square channels with three channel orientations. Dutta et al.\textsuperscript{15} presented experimental heat transfer results for turbulent flows through a rotating two-pass rib-roughened triangular channel, with two channel orientations with respect to the direction of rotation. Taslim et al.\textsuperscript{16, 17} studied the heat transfer characteristics in rib-roughened square and rectangular orthogonal rotating channels. They used a liquid crystal technique to study the effect of rotation on heat transfer distributions on the walls. First part, leading and the trailing walls of the test channel were roughened with staggered transverse ribs, while second part was, the opposite walls were rib-roughened at $45^\circ$ with respect to the main flow, in a criss-cross arrangement. They found that rotational effects were more pronounced in rib-roughened channels with a higher channel aspect ratio and a lower rib blockage ratio. They investigated heat transfer effects only in an orthogonally rotating single pass channel. However, they did not consider a two-pass channel and the effect of channel orientation. Prabhu and Vedula\textsuperscript{18} investigated the pressure drop distribution in a rotating rectangular channel with transverse ribs on one wall. They conducted experiments for a rotation number up to 0.21 and rib pitch-to-height ratios of 3, 5, 7.5, and 10, and a rib height-to-hydraulic diameter ratio of 0.15. They found that a rib array with a pitch-to-height ratio of 5 caused the largest pressure drop. Azad et al.\textsuperscript{19} experimentally investigated the heat transfer distribution in two-pass rectangular channels (AR=2:1) connected with a sharp $180^\circ$ turn. The results showed that the roughened surfaces exhibit better heat transfer distribution than the smooth surfaces. Also the results showed that parallel $45^\circ$ angled ribs produced higher heat transfer distribution than crossed $45^\circ$ angle ribs case. However, from the above-mentioned
research, few papers can be found in the open literature studied the rectangular or triangular cross section channel especially with rotation condition. Hence, the first motivation of this paper was to study two pass rectangular channels (AR=2:1) that is connected with a sharp 180° turn. The second motivation was to find different ribs configuration that trip the boundary layer and promote more cooling effect inside the two-pass rectangular channels. However, it was found from a previous study by Han et al. that the 45° V-shaped ribs show higher heat transfer performance in a one-pass non-rotating square duct compared to other ribs configurations (45° angled ribs or transverse 90° ribs). Thus, we have chosen 45° V-shaped ribs to be placed on the principle surfaces in the two pass rotating rectangular channels since they have shown a potential for higher heat transfer performance. Moreover, the effect of the channel orientation with respect to plane of rotation was investigated for two positions $\beta =$90°, 135°. Such experimental data is not available in the open literature, which shows the combined effect of the 45° V-shaped ribs induced secondary flows, and rotational induced secondary flows on the heat transfer enhancement in the two-pass rectangular cross sectional channels.
Figure 1.1 Schematic of the Rotating Test Rig

EXPERIMENTAL FACILITY

Figure 1.1 shows the schematic of the experimental test rig. Driven compressed air goes through a filter, an orifice meter, and passes through a rotary seal and a hollow-rotating shaft to feed the test section. The test section is mounted in a horizontal plane. Air travels outward first pass and inward second pass to exhaust into the atmosphere. Slip rings transfer thermocouple
outputs to data logger and power input from variac transformer to strip heaters, which is uniformly fixed under the copper plates. An electric motor with an adjustable frequency controller rotates the test section. A digital photo tachometer measures the rotational speed of the rotating shaft.

Figure 1.2 Cross sectional view of the two-pass rectangular test section.

Fig. 1.2 shows cross sectional view of the test section. The test section has two passes. Each pass is 12.7 mm by 25.4 mm in cross section. The first pass starts with an unheated teflon
entrance channel to establish a hydro-dynamically fully developed flow at the entrance to the heated channel. It has twelve (12) hydraulic diameters length to achieve the task. The heated channel length-to-hydraulic diameter ($L/D$) ratio is 18, while each pass length-to-hydraulic diameter ($L/D$) ratio is 9, connected by a sharp 180° turn. The flow in the first pass is radially outward and the flow in the second pass in radially inward. The heated section is divided into twelve longitudinal sections, six sections in first pass and six in second pass, to obtain local heat transfer coefficients. Each section has four copper plates on four walls of the channel. Each copper plate is surrounded circumferentially by a thin teflon strip that has (1.59 mm) thickness to insulated from their neighbors. This teflon strip works as an insulation to reduce heat conduction from one plate to another. Each copper plate has a single thermocouple that gives the regional surface temperature. The inlet and exit bulk temperatures are measured by Thermocouples. Each surface receives constant heat flux from an individual transformer to maintain the surface temperature around 65° C.

**DATA REDUCTION**

The local heat transfer coefficient is calculated from the local net heat transfer rate per unit surface area to the cooling air, the local wall temperature on each copper plate, and the local bulk mean air temperature as:

$$h = \frac{q_{net}}{A (T_w - T_{amb})}$$  \hspace{1cm} (1)

Local total net heat transfer rate is the electrical power generated from the heater ($q=VI$) minus the heat loss outside the test duct. The heat loss is determined experimentally by supplying
electrical power to the test section until a steady state condition is achieved for a no flow (without any airflow) condition. This is done for several different power inputs to obtain a relation between the total heat loss from each surface and the corresponding surface temperature. The highest heat loss is about 5% of the total power input for non-rotating cases and about 27% for a rotating low Reynolds number (Re=5000) case. To place the results on a common basis, the heat transfer area used in equation (1) was always that of a smooth wall. The local wall temperature is obtained from thermocouple that impeded in each copper plate. The bulk mean air temperatures entering and leaving the test section were measured by thermocouples. The local bulk mean temperature ($T_{bx}$) is used in equation (1) was calculated from the linear interpolation between the measured inlet and exit air bulk temperatures. Another way to find the local bulk mean air temperature is determined by marching along the test section and calculating the temperature rise from the local net heat input through each set of four heated surfaces. The differences between the calculated and measured outlet bulk mean temperature are between 1-2 °C in all of the cases.

Local Nusselt number was normalized by the Nusselt number for the fully developed turbulent flow in a smooth stationary circular pipe to reduce the influence of the flow Reynolds number on the heat transfer coefficient. The correlation is by Dittus-Boelter/McAdams or (Rohsenow and Choi 1961), as:

$$\frac{Nu}{Nu_0} = \frac{hD}{k \sqrt[4]{[0.023 \ast Re^{0.8} \ast Pr^{0.4}]}}$$  \hspace{1cm} (2)$$

The Prandtl number, Pr, for air is 0.71. Air properties are taken based on the mean bulk air temperature.
The uncertainty of the local heat transfer coefficient depends on the uncertainties in the local wall and bulk air temperature difference and the net heat input for each test run. The uncertainty increases with the decrease of the both local wall to bulk air temperature difference and the net heat input. Based on the method described by Kline and McClintock\textsuperscript{20}, the typical uncertainty in the Nusselt number is estimated to be less than 9\% for Reynolds number larger than 5000. The maximum uncertainty, however, could be up to 23\% for the lowest heat transfer coefficient at the lowest Reynolds number tested (Re=5000).

![Conceptual view of secondary flow vortices induced by the 45° V-shaped rib and 45° angled rib.](image)

**Figure 1.3** Conceptual view of secondary flow vortices induced by the 45° V-shaped rib and 45° angled rib.

**RESULTS AND DISCUSSION**

Figure 1.3 shows the 45° angled ribs that were divided at the centerline to make the 45° V-shaped ribs. There are two different orientations of the V-shaped rib. The first orientation is
called the 45° V-shaped rib and the second orientation is called the inverted 45° V-shaped rib. Figure 1.3 also shows the conceptual view of secondary flows that induced by the 45° angled rib, 45° V-shaped rib, and inverted V-shaped rib. The 45° angled rib induces secondary flow that moves parallel to the rib from the left side to the right side and return back to the left side making a counter rotating vortex (see Al-Qahtaini et al.\textsuperscript{21}).

It is conjectured that the 45° V-shaped rib creates two counter rotating vortices. As the fluid approaching the V-shaped rib, it splits into two streams. Each one moves parallel to the rib from the centerline to the either left side or right side and returns back to the centerline making a counter rotating vortex. Another observation can be drawn that as the 45° V-shaped rib is half the 45° angled rib then the boundary layer thickness for the fluid that moves parallel to one side of the 45° V-shaped rib is thinner than that produced by the 45° angled rib. Therefore, as the 45° V-shaped rib produces two counter rotating vortices that promoting more mixing in the bulk main stream and at the same time produces thinner boundary layer near the heated surface, higher heat transfer rate is expected compared to 45° angled rib. However, different situation can be observed in the inverted 45° V-shaped rib. As the fluid approaching the inverted 45° V-shaped rib the near surface fluid starts moving simultaneously from the left side and right side to the centerline creating a stagnation region and then returns back to the starting positions creating two counter rotating vortices. This stagnation region may weak the two counter rotating vortices. Thus, 45° V-shaped rib is expected to perform better than the inverted 45° V-shaped rib in non-rotating condition since there is no such stagnation region that reduces the counter rotating vortices effect.
Figure 1.4 Conceptual view of the secondary flow vortices induced by rotation, ribs, and channel orientation (dash line: rotation-induced vortices, solid line: rib-induced vortices).

Figure 1.4 shows conceptual views for the secondary flow patterns of a smooth and ribbed rotating two-pass rectangular channel. Fig. 1.4-a shows the smooth channel that rotates at $\beta=90^\circ$
with respect to the direction of rotation. Two symmetric cells of counter rotating secondary flow (dotted line) appear due to Coriolis force. In the first pass of the channel, the fluid moves in a radially outward direction and the effect of Coriolis force directs the coolant from the core toward the trailing surface. This causes an increase of the heat transfer from the trailing surface and a decrease in the heat transfer from leading surface. However, in the second pass, the opposite situation can be seen that the fluid moves in a radially inward direction and the Coriolis force directs the coolant toward the leading surface, and causes an increase of heat transfer from the leading surface and a decrease in the heat transfer from trailing surface. When the channel is twisted to $\beta=135^\circ$ orientation, the secondary flow vortices are asymmetric and migrate diagonally away from the corner region of the inner-leading surface toward the center in the first passage, and from the corner region of the inner-trailing surface toward the center in the second passage. Figure 1.4-b shows the parallel arrangement is attached to the leading and trailing surfaces in a parallel fashion so that they are directly opposite to each other. The inverted 45° V-shaped ribs are attached to the leading and trailing surfaces in the first pass and 45° V-shaped ribs in the second pass. Also, it shows the secondary flow (dotted line) that induced by rotational forces and the secondary flow (solid line) induced by the inverted 45° V-shaped ribs in first pass and 45° V-shaped ribs in the second pass. As the channel angle changed to $\beta=135^\circ$, the ribs secondary flow is not change, but the rotational secondary flows are shared between the principle surfaces (trailing, and leading) and side surfaces. Fig. 1.4-c shows the crossed rib case (the ribs on two opposite surfaces of the cooling channels are in crossed orientation). The cross orientation of the 45° V-shaped ribs coalesces the two pairs of the counter rotating secondary flow vortices to one pair of counter rotating secondary flow vortices. This reduction in number of secondary flow vortices limits the mixing between the near wall flow (hot fluid) and core flow.
(cold fluid), which causes less heat transfer rate. In case of rotation, a pair of counter rotating secondary flow vortices appears and moves opposite to the counter rotating secondary flow vortices generated by cross ribs. This negative interaction minimizes the rotation effect by suppressing flow impingement on the first pass trailing and second pass leading surfaces and restrict mixing with the core for both leading and trailing surfaces in both passes, which causes low heat transfer performance.

Figures 1.5-7 show the regionally average Nusselt number ratios (Nu/Nuo) from leading and trailing surfaces for four Reynolds number (5000, 10000, 25000, 40000), rotating and non-rotating, and two channel orientations (β=90°, 135°).

**Smooth Channel Results**

Figure 1.5 shows the results of Nusselt number ratios from leading and trailing surfaces. For the non-rotating (Re =5000) heat transfer ratio decreases monotonically for both leading and trailing surfaces from 2.2 near the thermal entrance of the first passage to about 1.4 near the downstream X/D = 5 and than recovers to 2 before the flow enters the sharp 180° turn. The non-rotating heat transfer ratio reaches the highest value 3.0 right after the sharp 180° turn around (X/D = 10) due to the induced secondary flows of the 180° turn. The Nusselt number ratio approaches the fully developed turbulent flow value 2 near the outlet region. The non-rotating results agree with the previous study of local heat transfer ratios in a typical multi-pass coolant passage with smooth surface (see Azad et al.19). However, in the rotation case, rotation induced Coriolis forces produce cross-stream flow patterns, which push the main stream flow from the core towards the trailing surface in first pass. The situation is opposite in second pass. Coriolis forces push the main stream flow towards the leading surface. Therefore, the trailing surface heat transfer is higher than the leading surface in the first pass and similarly, the leading surface heat transfer is
higher than trailing surface in the second pass. It is also noticed that the rotation effect on Nusselt number ratios for second channel leading and trailing surfaces is not as strong as first pass. This is because Coriolis and buoyancy forces move in the same direction in first pass while in second pass they move against each other (see Han et al.\textsuperscript{9}). At channel orientation $\beta=90^\circ$, rotation secondary flows vortices produced by Coriolis forces are impinging normally on trailing surface first pass and leading surface second pass. In contrast, at channel orientation $\beta=135^\circ$, the secondary flow vortices migrate diagonally away from the corner region toward the center of the passage.

![Figure 1.5 Nusselt number ratio distribution for case (a)](image-url)
In first pass, it migrates diagonally away from the corner of inner-leading surfaces toward the center of the passage, and in second pass from the corner of the inner-trailing surfaces toward the center of the passage (as explained in Fig. 1.4-a). Thus, the Nusselt number ratio for the trailing surface first pass and the leading surface second pass for channel orientation $\beta=90^\circ$ are higher than the ratio for channel orientation $\beta=135^\circ$. The opposite situation is observed in the leading surface first pass and trailing surface second pass: the $\beta=135^\circ$ case has higher Nusselt number ratio when compare to the $\beta=90^\circ$ channel orientation. The results also show that the effect of rotation decreases with increasing Reynolds number (or decreasing rotation number).

**Parallel 45° V-shaped Rib Case**

Figure 1.6 shows the Nusselt number ratio for case (b). The stationary case results show the highest Nusselt number ratio in the first pass occurs at the entrance region. This peak value is due to the fact that the flow is thermally developing and the two pairs of counter rotating secondary flow vortices that are generated by the inverted 45° V-shaped rib. Then the Nusselt number ratio decreases slightly as the flow becomes thermally fully developed. Immediately the Nusselt number ratio recovers to increase. This is due to the two pairs of counter rotating secondary flow vortices that are generated by the inverted 45° V-shaped ribs. Then, the Nusselt number ratio decreases as the secondary flows are influenced by the 180° turn. As the flow reaching the exit of the first pass, Nusselt number ratio continuously decreases due to 180° turn induced secondary flows. The inlet of the second pass starts from the sharp 180° turn and the Nusselt number profiles differs from the first pass. At the inlet of the second pass, Nusselt number ratio reaches the minimum value. This drop is due to the decay of the turbulence produced by the sharp 180° turn. Beside the 180° turn, the rib orientation may also be
responsible for this drop. Then, the Nusselt number ratio increases down stream of the second pass inlet as the secondary flow induced by the 45° V-shaped ribs start to develop after the influences from the 180° turn are damped.

![Figure 1.6 Nusselt number ratio distribution for case (b)](image)

With rotation the results show that the rotation significantly increases the Nusselt number ratio on the first pass trailing surface and the second pass-leading surface. The rotating Nusselt number ratios on the first pass trailing and second pass leading surface are higher than in the
non-rotating case, while the first pass leading and second pass trailing surfaces are lower. This is because of the combined effect of the rib induced secondary flow and the rotation induced secondary flow vortices, as shown in Fig. 1.4-b. The Nusselt number ratio has the same trend as stationary case except the leading surface first pass shows continuous decreasing and then increase at the 180° turn. In the 180° turn, the leading surface Nusselt number ratio shows higher value compared to trailing surface.

For 135° channel orientation, the result also shows that rotation enhances the heat transfer in the first pass-trailing surface and second pass leading surfaces, whereas the heat transfer decreases in the first pass leading and second pass trailing surfaces. In the 135° orientation, the heat transfer surfaces are not orthogonal to the plane of rotation, and the rotational effects are less significant when compared with 90° orientation, as shown in Fig. 1.4-b The effect of Coriolis forces on the first pass trailing and second pass leading surfaces for 90° rotation is more since the two counter vortices are normally incident on trailing surface first pass and leading surface second pass. Thus greater differences between leading and trailing Nusselt number ratio in each pass are observed in the 90° orientation. Hence, the differences in Nusselt number ratio between leading and trailing surfaces are not as significant for the 135° orientation as they are for the 90° orientation. In the 180° turn, the leading surface shows slightly better Nusselt number ratio than trailing surface. The results also show that the rotation effect decreases with increasing Reynolds number (or decreasing rotation number).
Figure 1.7 shows the Nusselt number distribution for crossed rib case (see figure 1.4-c). The result show that the non-rotating Nusselt number ratios unlike the parallel rib case. Both sides of the principle surfaces are not identical and they have lower Nusselt number ratios. The peak value in the first pass-trailing surface appears at the entrance, while the peak value on the leading surface appears at the middle of the channel. This variation is due to the different orientations of

**Crossed 45° V-shaped Rib Case**

Figure 1.7 shows the Nusselt number distribution for crossed rib case (see figure 1.4-c). The result show that the non-rotating Nusselt number ratios unlike the parallel rib case. Both sides of the principle surfaces are not identical and they have lower Nusselt number ratios. The peak value in the first pass-trailing surface appears at the entrance, while the peak value on the leading surface appears at the middle of the channel. This variation is due to the different orientations of
the 45° V-shaped ribs that are placed on the principle surfaces and cause the dissimilarity. It is also noticed that the 45° V-shaped ribs that are placed on the leading surface produce higher Nusselt number ratio than the inverted 45° V-shaped ribs on the trailing surface. This is because the stagnation region that was generated by inverted V-shaped ribs (see Figure 1.3). In the second pass, the Nusselt number distributions for both surfaces are almost similar in trend and different in values. The Crossed rib case as seen in Fig. 1.4-c generates two counter rotating vortices, which enhances the Nusselt number distribution. But this enhancement is low compared to the parallel rib case which, induces four counter rotating vortices to increase the mixing and resulting to better heat transfer enhancement. Figure 1.4-c shows that the rotation induced secondary flow vortex pushes the cold fluid toward the trailing surface in the first pass and the leading surface in the second pass. At the same time the ribs induced secondary flows work against the rotation secondary flow. This is limited the mixing between the near surface fluid and the mean stream flow, which reduces the Nusselt number ratio through the entire channel.
Figure 1.8 Averaged Nusselt number ratio for $\beta=90^\circ$
Channel Averaged Nusselt Number Ratio

Figure 1.8 presents the averaged Nusselt number ratio distribution for smooth, 45° angled rib, parallel and crossed 45° V-shaped ribs. The results show that the ribs surfaces provide higher Nusselt number ratios than smooth surfaces in both passes. This is because ribs trip the boundary layer and induce secondary flows that enhance the heat transfer rate. In stationary case, Parallel 45° V-shaped ribs show higher Nusselt number ratio distribution than crossed 45° V-shaped ribs because they generate two pair of counter rotating vortices, while crossed case produces one pair of counter rotating vortices. In case of rotation, trailing surfaces in the first pass are higher than the leading surface and leading surfaces in the second pass are higher than trailing surfaces due to Coriolis effects for both cases parallel and crossed. Also, crossed case shows less rotation effects. This is because the ribs induced secondary flows move against the rotational secondary flows.

CONCLUSIONS

The influences of 45° V-shaped ribs arrangements, and channel orientation on the leading and trailing Nusselt number ratios in a two-pass rectangular channel have been reported for rotation number from 0 to 0.21 and Reynolds number from 5000 to 40000. The findings are:

1. The general trend of the rotation effect is that increases the Nusselt number ratio in the first pass trailing surface and second pass leading surface, while opposite situation can be observed that low Nusselt number ratio distribution in the first pass leading surface and second pass trailing surface. This is due to the Coriolis and buoyancy forces were generated by rotation. However, the differences between the Nusselt number
distributions on the second pass leading and trailing surfaces are smaller than that in the first pass due to the opposite effects of the Coriolis and buoyancy forces.

2. The effects of the Coriolis force and cross-stream flow reduce as the channel orientation changes from $\beta=90^\circ$ to $\beta=135^\circ$. Thus, the Nusselt number ratios for the $\beta=135^\circ$ first pass trailing and second pass leading surface decreased when compared to their corresponding Nusselt number ratios for the $\beta=90^\circ$ orientation. The Nusselt number ratios for $\beta=135^\circ$ first pass leading and second pass trailing surfaces increased when compared to their corresponding Nusselt number for $\beta=90^\circ$.

3. The parallel 45° V-shaped ribs arrangement provides higher heat transfer enhancement compared to the 45° crossed ribs arrangement for both rotating and non-rotating conditions. The crossed rib arrangement shows less rotational effect compared to the parallel rib cases. This is because the 45° parallel rib develops a pair of counter rotating vortices of secondary flows, while the 45° crossed rib develops only a single vortex of secondary flows.

4. The 45° V-shaped rib shows better heat transfer enhancement compared to 45° angled rib (see Azad et al.\textsuperscript{19}). 45° angled rib generates two counter vortices, while 45° V-shaped ribs generates four counter rotating vortices, which promoting more mixing between the near surface fluid (hot) and core fluid (cold).

5. In general, the roughened surfaces in the rectangular channel perform similarly to smooth surfaces with increasing rotational speed. However, the average Nusselt number ratio in the roughened rectangular channel is much higher than the smooth surfaces.

6. For 45° V-shaped ribs cases, results show low Nusselt number ratio in the 180° turn. This is because there in no 45° V-shaped rib placed at the middle of the 180° turn.
ACKNOWLEDGEMENTS

The leading author, Luai AL-Hadhrami, received a fellowship from King Fahd University of Petroleum and Minerals, Saudi Arabia, for his Ph.D. study at Texas A&M University. This work was supported by DOE Advanced Gas Turbine Systems Research (AGTSR) program through project number SR-094. The support of the above institutions is greatly appreciated.

REFERENCES


PART II: COMPUTATIONAL STUDY

Part 2 is conducted at Texas A&M University under the direction of Dr. H.C. Chen and Dr. J.C. Han.
GEOMETRY AND NUMERICAL GRIDS

A schematic diagram of the geometry is shown in Figure 2.1. It has a rectangular cross section with channel aspect ratio $AR = 4:1$. Two of the four side walls, in the rotational direction, are denoted as the leading and trailing surfaces, respectively, while the other two side walls are denoted as the top and bottom surfaces. The channel hydraulic diameter, $D_h$, is 0.8 in (2.03 cm). The distance from the inlet of the channel to the axis of rotation (Y-axis) is given by $R_r/D_h = 20.0$ and the length of the channel is given as $L/D_h = 22.5$. The channel consists of unheated starting smooth length ($L_1/D_h = 9.92$), heated smooth or ribbed section ($L_2/D_h = 7.58$) and unheated exit smooth section ($L_3/D_h = 5.00$). The arc length $S$ is measured from the beginning of the heated section to the end of it. In the ribbed section, the leading and trailing surfaces are roughened with nine equally spaced V-shaped ribs of square cross section. The rib height-to-hydraulic diameter ratio ($e/D_h$) is 0.078 and the rib-pitch-to-height ratio ($P/e$) is 10. All ribs are inclined at an angle $\alpha = 45^\circ$ with respect to the flow.

![Figure 2.1. Geometry of AR = 4:1 rectangular duct with V-shaped ribs.](image-url)
Figure 2.2 shows the computational grid for the 4:1 rectangular duct around the V-shaped ribs. The grid was generated using an interactive grid generation code GRIDGEN. It was then divided into five overlapped chimera grid blocks to facilitate the implementation of the near-wall turbulence model and the specification of the boundary conditions. To provide adequate resolutions of the viscous sublayer and buffer layer adjacent to a solid surface, the minimum grid spacing in the near-wall region is maintained at $10^{-3}$ of the hydraulic diameter which corresponds to a wall coordinate $y^+$ of the order of 0.5. The number of grid points in the streamwise direction from inlet to outlet is 394, while $33 \times 75$ grid points were used in the cross-stream plane. In all calculations, the root-mean-square (rms) and maximum absolute errors for both the mean flow and turbulence quantities were monitored for each computational block to ensure complete convergence of the numerical solutions and a convergence criterion of $10^{-5}$ was used for the maximum rms error.
NUMERICAL METHOD

The Reynolds-Averaged Navier-Stokes equations in conjunction with a near wall Reynolds stress turbulence model are solved using the chimera RANS method together with near-wall second-moment turbulence closure. The flow is considered to be incompressible since the Mach number is quite low. However, the density in the centrifugal force terms is approximated by \( \rho = \rho_o T_o / T \) to account for the density variations caused by the temperature differences. \( \rho_o \) and \( T_o \) are the density and temperature at the inlet of the cooling channel. In general, the density is also a function of the rotating speed because the centrifugal force creates a pressure gradient along the duct. In the planned experiments to be conducted later, the maximum pressure variation between the channel inlet and the exit is approximately 0.0113 atm at the highest rotating speed of \( \Omega = 550 \text{ rpm} \). This gives a maximum density variation of only about 1.1% from the inlet to the exit of the duct at the highest rotation number. It is therefore reasonable to omit the density variation caused by the pressure gradients induced by the channel rotation.

The present method solves the mean flow and turbulence quantities in arbitrary combinations of embedded, overlapped, or matched grids using a chimera domain decomposition approach. In this approach, the solution domain was first decomposed into a number of smaller blocks to facilitate efficient adaptation of different block geometries, flow solvers, and boundary conditions for calculations involving complex configurations and flow conditions. Within each computational block, the finite-analytic numerical method was employed to solve the unsteady RANS equations on a general curvilinear, body-fitted coordinate system. The coupling between the pressure and velocity was accomplished using the hybrid PISO/SIMPLER algorithm. The method satisfied continuity of mass by requiring the contravariant velocities to have a vanishing
divergence at each time step. Pressure was solved using the concept of pseudo-velocities and, when combined with the finite-analytic discretization gives the Poisson equation for pressure.

A uniform velocity profile was used at the inlet of the duct \((Z = 0)\). The unheated length \((L_1)\) was long enough for the velocity profile to be fully developed turbulent profile before the heating start-point \((Z = L_1)\). Zero-gradient boundary conditions were used at the exit of the duct for mean velocity and all turbulent quantities, while linear extrapolation was used for the pressure field. The coolant fluid at the inlet of the duct is air at uniform temperature \(T = T_o\) (i.e., \(\Theta = (T - T_o) / (T_w - T_o) = 0\)). The wall temperature of the unheated sections is kept constant at \(T = T_o\) (\(\Theta = 0\)) while the wall temperature of the heated section is kept constant at \(T = T_w\) (\(\Theta = 1\)).

RESULTS AND DISCUSSION

Computations were performed for non-rotating \((Ro = 0)\) and rotating \((Ro = 0.14)\) rectangular channels with V-shaped ribs at Reynolds number \(Re = 10,000\), and inlet coolant-to-wall density ratios \(\Delta \rho / \rho = 0.122\). The Nusselt numbers presented here were normalized with a smooth tube correlation by Dittus-Boelter/McAdams for fully developed turbulent non-rotating tube flow:

\[
Nu_o = 0.023 \, Re^{0.8} \, Pr^{0.4}
\]

Figures 2.3(a) and 2.3(b) show the local Nusselt number ratio contours of the leading and trailing surfaces, respectively, for the non-rotating case. The entrance and exit regions were cut to focus on the ribbed heated section. It is seen that the highest Nusselt number ratios were obtained on the top of the ribs, and the lower Nusselt number ratios were obtained right before and after the ribs. Between any two ribs, the Nusselt number ratios are highest near the middle.
section of the channel and decrease gradually towards the sidewalls (i.e., inner or outer surfaces). This is due to the rib induced secondary flow that moves from the center of the V-shaped rib to the sidewalls. In Figure 2.4, the rotation number is increased to 0.14 while the density ratio is kept fixed at 0.122. For the rotating case, the Coriolis forces produce a cross-stream two vortex flow structure that pushes the cold fluid from the core toward the trailing surface and then brings it back along the inner and outer surfaces to the leading surface. This means that the thermal boundary layer starts at the trailing surface, grows along the two side surfaces and ends at the leading surface. This results in small temperature gradient near the leading surface (hence lower heat transfer coefficients) and steeper one near the trailing surface (hence higher heat transfer coefficients). Moreover, the cooler heavier fluid near the trailing surface will be accelerated by the centrifugal buoyancy force while the hotter lighter fluid near the leading surface will be decelerated to maintain the continuity in the streamwise direction. The Coriolis and centrifugal buoyancy forces cause the Nusselt number ratios on the trailing surface to increase compared to the non-rotating case. On the other hand, the Nusselt number ratios were reduced on the leading surface comparing to the non-rotating case.
Figure 2.3 Detailed Nusselt number distributions on trailing and leading surfaces;

Re = 10,000, Ro = 0, Δρ/ρ = 0.122
Figure 2.4. Detailed Nusselt number distributions on trailing and leading surfaces; Re = 10,000, Ro = 0.14, $\Delta \rho / \rho = 0.122$
Figures 2.5 and 2.6 show the spanwise-averaged Nusselt number ratios ($\frac{Nu}{Nu_0}$) for the non-rotating and rotating cases, respectively. The inlet coolant-to-wall density ratio was held constant at 0.122. The spanwise-averaged Nusselt number distributions on the leading and trailing surfaces show periodic spikes. The higher spikes which occur on the ribs tops are caused by the flow impingement on the ribs, and the lower spikes (which occur right before and after the ribs) are caused by the flow reattachment between the ribs. The Nusselt number ratios are high in the regions between the ribs. The Nusselt number ratios reached a peak value around the eighth rib. This phenomenon is caused by the rib-induced secondary flow becoming stronger along the duct. As noted earlier, the channel rotation leads to an increase of Nusselt number on the trailing surface and a decrease of Nusselt number on the leading surface. However, the rotation effects are quite small at $Ro = 0.14$ when comparing to the heat transfer enhancement caused by the rib-induced flows. Additional computations will be performed in the near future for higher rotation number and higher density ratio cases to provide more detailed assessment on the effectiveness of the V-shaped ribs.
Figure 2.5. Calculated Nusselt number ratio distribution for non-rotating duct with V-shaped ribs; \( \text{Re} = 10,000 \).
Figure 2.6. Calculated Nusselt number ratio distribution for rotating duct with V-shaped ribs; 

\[ Ro = 0.14, \ Re = 10,000. \]
ONGOING WORK

Part I: Rotating Heat Transfer

- Design, fabricate and instrument a two-pass smooth wall rectangular channel with an aspect ratios of 2:1 (AR = a/b = 2:1).
- Measured local heat transfer distributions and pressure drops in the smooth wall rectangular channels for the effects of Reynolds number (Re = 10,000-100,000), Rotation number (Ro = 0-0.4), Coolant to wall temperature ratio (TR = 0.8-0.9), and rotating angle $\gamma = 90^0$ and $120^0$).
- Have done the effect of rotation in a two-pass rectangular channel with aspect ratio of 2:1 with V-shaped rib turbulators for two rotating channel orientations as showed in this report.
- Install various high performance turbulence promoters (broken angled ribs, broken V-shaped ribs, and delta-shaped turbulators, respectively) on the leading and trailing surfaces of the smooth-wall and measure local heat transfer and pressure drops for the same range of parameters as for the smooth surface.
- Correlate the new data and compare with the existing data (AR = 1:1).
- Compare with predictions in part II for rotating channels.

Part II: Computational Study

- Calculate flow field, heat transfer coefficients and pressure drops in the rectangular channels with V-shaped ribs for higher rotation number, higher coolant to wall temperature ratio cases using two-layer k-$\epsilon$ model and second moment closure model.
• Calculate flow field, heat transfer coefficients and pressure drops in the rectangular channels with V-shaped ribs for different channel orientations by the two-layer k-\(\varepsilon\) model and second moment closure model.

• Validate the numerical predictions with above-mentioned flow, heat transfer, and pressure drop measurements (part I) for assessment of the general performance of RANS code and turbulence models.

• Extend above-mentioned calculations for a very high Reynolds number (Re = 500,000) and buoyancy parameter (Bo = 10) conditions.
Interim Report
Subcontract #01-01-SR089

Internal Cooling in Leading- and Trailing-Edge Passages with Rotation and Buoyancy

Louisiana State University
Baton Rouge
Louisiana 70803

PI: Sumanta Acharya
Mechanical Engineering Department

Submitted To

Department of Energy
Advanced Gas Turbine Systems Research Program
South Carolina Energy Research and Development Center
Clemson, SC 29634
# TABLE OF CONTENTS

SUMMARY ........................................................................................................................... 3
GOALS ............................................................................................................................... 4
PROGRESS IN COMPUTATIONAL RESEARCH.......................................................... 5
  RANS Simulations........................................................................................................... 6
  Large Eddy Simulations (LES)........................................................................................ 8
PROGRESS IN EXPERIMENTAL RESEARCH............................................................ 10
  Experimental Facility..................................................................................................... 10
  Models to be tested ........................................................................................................ 11
PROJECTED TIME SCHEDULE .................................................................................... 12
SUMMARY

This report summarizes the work accomplished in the first half of year 1 of the DOE-ATS project to LSU. The main goals of the research were to perform experiments and simulations for flow and heat transfer in smooth and ribbed channels at large aspect ratios, high Re, and high buoyancy parameters. The following have been the main accomplishments to date:

- Delivery and installation of the UTRC (united Technologies Research Corporation) rotating rig facility and instrumented models. Work is now ongoing to take data in a 1:4 model. We expect to report measurements at Re=500,000, Ro=0.4, and $\Delta \rho/\rho_\infty=0.2$ in the Feb. ATS meeting for an aspect ratio of 1:4.
- RANS simulations for the ribbed rotating channel. These have been completed for a number of cases. In the Feb. meeting, we expect to present results for 1:1, 1:4, and 4:1 aspect ratios for a range of Re, Ro and Buoyancy parameters.
- Large Eddy Simulations (LES) for periodically developed flow and heat transfer in a channel with buoyancy and rotational effects. These have been completed for a number of cases, and these results are now being used for model validation in the RANS studies.
GOALS
The proposed research deals with flow and heat transfer in rotating channels. This configuration is of interest in internal cooling of gas turbine blades. Efforts directed toward improving the performance of gas turbine engines through increased turbine inlet temperatures require a clear understanding of the flow and heat transfer mechanisms under rotational conditions, and for parameters relevant to actual gas turbine engines. These parameters include aspect ratio and shape of the coolant channel, the channel orientation relative to the rotational axis, the Reynolds number ($Re=UD_h/\nu$), the Rotation number ($Ro=\Omega D_h/U$) and the centrifugal buoyancy parameter ($BP=Ro^2 (R/D_h) (T_w-T)/T_w$). The majority of the published literature deals with square or 1:2 aspect ratio channels, and the majority of the data published is for buoyancy parameters less than 1. However, the coolant passages in turbine blades can have aspect ratios of the order of 1:4 (along the leading edge) and 4:1 along the trailing edge, and buoyancy parameters can be considerably higher than that reported in the literature. The present work will experimentally investigate heat transfer in high/low aspect ratio channels (1:4 or 4:1) with different orientations (relative to the rotational axis) for Reynolds number up to 500,000, Rotation number up to 1, and buoyancy parameter values up to 5. These measurements will be done in a rotating rig donated by UTRC, with instrumented test sections that will be fabricated or modified. Details of the velocity field, and further extensions of the above parameters (triangular leading edge cross-sections, Reynolds number up to a million, and buoyancy parameters up to 10) will be done using a validated CFD code.

The main goal of the proposed research is to provide the relevant data for parameter ranges relevant to engine operating conditions, and to provide an understanding under these conditions, of the flow and heat transfer mechanisms in a rotating coolant channel. A two-pass smooth and ribbed rotating channel (normal and angled trips) will be studied. Time-resolved heat transfer measurements in the rotating frame and computations are proposed to address the goals of the research. The heat transfer measurements will be made in a facility obtained from UTRC. Provisions are available for both thermocouple based heat transfer and liquid-crystal based heat transfer measurements. These measurements will be done under conditions of rotation, for Reynolds number, Rotation number and centrifugal buoyancy parameter relevant to gas turbine applications. Pressure drop measurements will also be performed to provide information on the thermal performance behavior. The calculations will involve the solution of the Reynolds-Averaged Navier-Stokes (RANS) with a validated turbulence model. The model validation will be performed primarily through comparisons with measurements.
PROGRESS IN COMPUTATIONAL RESEARCH

The numerical calculations will be performed in order to understand the basic flow mechanisms, and to extend the parametric range possible with the experiments. RANS simulations and LES (large Eddy Simulations) will be used to provide a description of the flow field for the various parameters where measurements of heat transfer are made in the experimental phase of work. Agreement between the computed and measured heat transfer field will provide confidence in the predictions, and the velocity predictions can be used in understanding the heat transfer behavior in the parameter ranges of interest. The RANS simulations will also be used to extend the parametric range beyond those considered in the experiments.

The specific goals of the computational research are as follows:

- Perform turbulence model validation for a ribbed coolant channel in the periodically-developed module at low buoyancy parameter and no-rotation. For model validation, use published data (heat transfer and velocity) and LES at lower Re values and the present heat transfer data at higher Re values.
- Perform RANS simulations, with the appropriate turbulence models, for the full range of parameters for a smooth channel. Evaluate the turbulence model by comparison with current data.
- Perform RANS simulations for the full range of parameters for the ribbed channel. Use the predictions to get the flowfield associated with the measured heat transfer information. The predicted flowfield and the measured heat transfer data provide a complete picture of the physical process.
- Use RANS simulations to extend the parametric range, and investigate triangular cavities, Reynolds number up to a million, rotation number up to 1 and buoyancy parameters up to 10.

The following achievements have been completed to date:

1. RANS simulations for a two pass coolant channel (square cross-section, 1:4, and 4:1) with smooth walls, normal trips and inclined trips. Results have been obtained for both stationary and rotating cases and compared with Johnson and Wagner’s data with good agreement.
2. LES of flow in smooth and ribbed (normal-trips) duct for stationary and rotating conditions. Results are compared with Johnson and Wagner’s data, and provide detailed understanding of flow and heat transfer physics, and how they are influenced by Coriolis and buoyancy forces.

Current activities are focusing on simulations in 1:4 and 4:1 aspect ratios at high Reynolds numbers (up to 500,000) and buoyancy parameters (up to 5).
RANS Simulations

The RANS simulations were performed initially for validation studies. In this regard, we selected the cases reported by Johnson and Wagner in a series of papers in the 1980’s for both smooth and ribbed ducts under stationary and rotating conditions.

We have performed the following simulations to date:

1. Stationary and rotating duct with aspect ratio of 1, smooth walls, normal trips and inclined trips, and a range of Re, Ro, and buoyancy parameters. Results obtained compare well with the measurements reported by Wagner and Johnson. Figure 1 below shows the comparison between the measured and predicted heat transfer in a smooth three-legged passage with no-rotation. As may be seen, the agreement is quite good. We are currently exploring improved turbulence models, and using LES to help in the model validation.

**Fig. 1:** Comparison of experimental and predicted Nusselt numbers in a three-legged passage for Re=25,000, and Ro=0, $\Delta \rho/\rho=0.12$

The following simulations are ongoing, and are at various degrees of completion.

1. Flow and heat transfer in channels of aspect ratio of 1:4 and 4:1, for a wide range of buoyancy parameters, and Re and Ro numbers. Representative results are presented below. Detailed results on these simulations will be presented in the upcoming contractors meeting in Feb.
The streamwise distributions of the average Nusselt number in the three radial passages for the 1:1, 1:4, and 4:1 cases are shown in Fig. 2 above. Lowest heat transfer rates are observed for the 1:4 case, while the square and 4:1 cases show comparable rates.

With rotation, both the 1:4 and 4:1 cases exhibit the behavior observed for the 1:1 case, that is, enhanced heat transfer (relative to the stationary channel) along the trailing surfaces of the radially-outward flow leg, and leading surfaces of the radially-inward flow leg (see Fig. 3 below). These differences are greater for the 1:4 case as seen in Fig. 3.

Figure 2: Nusselt number for 1:1, 1:4, and 4:1 aspect ratios, Re=25000, Ro=0, $\Delta \rho/\rho=0.12$

Figure 3: Nusselt number for the 1:4 aspect ratio case, Re=25000, Ro=0.23, $\Delta \rho/\rho=0.12$
Large Eddy Simulations (LES)

We have performed LES simulations periodically fully developed flow in stationary and rotating channels for a range of parameters. We present below just some illustrative results, and details of the work can be found in Saha and Acharya (2002).

The contours of streamwise vorticity ($\omega_x$) for the two different cases (stationary and rotating) have been presented in Figure 4. Comparison of the two figures clearly reveals that the stationary case has elongated streamwise vortical structures with almost equal activity near top and bottom walls. On the other hand, the rotational case shows smaller structures that are predominantly present near the bottom wall (leading or stable side), and more elongated structures present near the top wall (trailing or unstable side). The greater stretching of the structures near the trailing surface are due to the higher momentum of the fluid near the unstable side. Some of the structures are so elongated that they fill almost the entire streamwise periodic length.

Figure 4: Streamwise vorticity (a) Ro=0.0 and (b) Ro=0.12 and $\Delta\rho/\rho=0.13$

Figure 5: Temperature contours (a) Ro=0.0 and (b) Ro=0.12 and 0.13
Figures 5(a) and 5(b) are the non-dimensional temperature contours for the two cases. It is to be noted that the non-dimensionalisation of temperature is such that red zone indicates cold fluid and blue zone corresponds to hot fluid. For the stationary case, the incoming coolant fluid is distributed almost symmetrically between the two walls. For the rotational case, the temperature of the bulk fluid becomes asymmetric because high momentum cold fluid is pushed towards the trailing side by the Coriolis-induced secondary flows. Thus, for the stationary case, both walls and the experience fluid of comparable temperature, while for the rotational case, the rib as well as wall on the trailing side are exposed to colder fluid compared to its counterparts in the leading side. Thus the heat transfer on the trailing surface is expected to be higher. Due to the shedding of the vortices from the ribs (see Figure 4) there is continuous breakdown of thermal boundary layers on both the walls. The recirculatory regions (represented by the high temperature zones) behind the ribs are almost of equal size for the non-rotating case whereas they are of different sizes for the rotational case. It is already mentioned that the presence of ribs produces large scale unsteadiness. Due to large scale irregular flow separation and reattachment locally, there is higher rate of heat removal from the heated walls and this results in enhanced heat transfer rate compared to their smooth wall counterparts.

Currently two papers are in preparation, and the results will be presented in detail in the upcoming contractors meeting in February.
PROGRESS IN EXPERIMENTAL RESEARCH

It is planned to perform the proposed research in an experimental facility donated to us by United Technologies Research Corporation (UTRC). This facility was originally used by Bruce Johnson and Joel Wagner for making a series of heat transfer measurements under rotating conditions. Along with this facility, several fully instrumented coolant channel models were also obtained. Two of these models were in line with the models to be tested for the proposed work, and included a 4:1 model and a 6:1 model. These were both heat transfer models where temperature measurements were made using thermocouple and slip-ring assemblies.

The following tasks have been completed by December 2001.

1. Transport of the UTRC rotating rig facility to LSU
2. Installation of the facility
3. Preliminary testing is now ongoing

We anticipate to have test runs and data with the 1:4 test section by February 2002.

Experimental Facility

A schematic of the experimental facility during assembly is shown in the figure 6. Figure on the left shows the lower half of the containment vessel, while figure on the right shows the main shaft and the spinning arm. These pictures were taken during the assembly operations which are now complete.

Figure 6: Photograph of the test facility under assembly. Picture on the left shows the lower half of the containment vessel, while the picture on the right shows the instrumented shaft assembly. The big windows on the containment vessel are for liquid crystal imaging.
Models to be tested

Figure 7 below shows two heat transfer models to be initially tested. Figure on the left is a 1:4 aspect ratio model, with normal trips, which is fully instrumented with heater strips and thermocouple wires. The leads for the heater coils and thermocouple wires are clearly shown. We are currently setting up to perform test on this model. The model on the right is a two-pass model that is instrumented with pressure taps and a larger number of thermocouple wires. Both detailed pressure and heat transfer information can be obtained in this model.

Fig. 7: Picture of instrumented test sections to be used in the project. Left picture shows a two-legged 1:4 aspect ratio model instrumented with thermocouple wires. Picture on the right also shows a two-legged model instrumented with both pressure tap and thermocouple wires.
<table>
<thead>
<tr>
<th>Experimental task</th>
<th>2000</th>
<th>2001</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>QI</td>
<td>QII</td>
</tr>
<tr>
<td>Installation and assembly of the rotating rig facility.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Qualification tests for existing models (1:4 and 1:1) under stationary conditions</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Heat transfer measurements in 1:1; 1:2; and 1:4 aspect ratio channels with 90° and 60° angled ribs</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pressure Drop Measurements for the above channels</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Orient channels to obtain 2:1 and 4:1 aspect ratios for heat transfer and pressure drop measurements with 90° and 60° angled ribs</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Adjust Channel Orientation to Angles of 30°, 45°, and 60° for some cases for different channels to study the effect of rotation</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Computational Task</strong></td>
<td>QI</td>
<td>QII</td>
</tr>
<tr>
<td>Turbulence Model Validation (using LES and measured data) for smooth and ribbed channels without rotation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Perform RANS simulation for smooth channels with rotation. Evaluate turbulence model with comparisons from measurements</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Perform RANS simulations for ribbed channels with rotation and compare to experimental data</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Extend RANS simulations to high Reynolds number, High Ro cases and also investigate triangular cavities</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Analysis and Final report</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Legend:**
- **Completed**
- **Ongoing**
- **To Be Done**
PUBLICATIONS

Final Report
Contract Number: 98-02-SR069
July 2001

Turbine Blade Tip, Endwall, and Platform Heat Transfer Including Rotational Effects

Sanford Fleeter and Patrick B. Lawless
School of Mechanical Engineering
Purdue University
West Lafayette, IN 47907

Prepared For:

Advanced Gas Turbines Systems Research
South Carolina Energy Research and Development Center
386-2 College Avenue
Clemson, SC 29634-5180
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>NOMENCLATURE</td>
<td>iv</td>
</tr>
<tr>
<td>SUMMARY</td>
<td>v</td>
</tr>
<tr>
<td>1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>2. EXPERIMENTAL APPARATUS AND METHODS</td>
<td>17</td>
</tr>
<tr>
<td>3. DATA STRUCTURE AND ANALYSIS</td>
<td>55</td>
</tr>
<tr>
<td>4. GENERAL DESCRIPTION OF THE MEASURED FLOW FIELD</td>
<td>68</td>
</tr>
<tr>
<td>5. VANE WAKE AND ROTOR BLADE POTENTIAL FIELD EFFECT ON SEAL FLOW</td>
<td>72</td>
</tr>
<tr>
<td>6. 3D PIV MEASUREMENTS</td>
<td>100</td>
</tr>
<tr>
<td>7. SUMMARY AND CONCLUSIONS</td>
<td>113</td>
</tr>
<tr>
<td>LIST OF REFERENCES</td>
<td>117</td>
</tr>
</tbody>
</table>

## APPENDICES

- Appendix A  Four Plane, Axial Velocity Contours, High Seal Flow Rate 119
- Appendix B  Four Plane, Low Seal Flow Rate, Axial Velocity Contours 129
- Appendix C  Four Plane, High Seal Flow Rate Vorticity Contours 139
- Appendix D  Four Plane, Low Seal Flow Rate Vorticity Contour 149
Appendix E  Regions Of Negative Radial Velocity For The High Seal Flow Rate  159
Appendix F  Regions Of Negative Radial Velocity For The Low Seal Flow Case  169
Appendix G  2-Dimensional Representation Of Vector Field For Vane Position A
           At A High Seal Flow Rate ....................................................... 179
Appendix H  2-Dimensional Representation Of Vector Field For Vane Position A
           At A Low Seal Flow Rate .......................................................... 189
Appendix I  VBI3D Unsteady Flow Analysis of the Purdue Low Speed
           Turbine Research Rig 1st Stage ................................................. 199
## NOMENCLATURE

**Roman**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Flow Area</td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
</tr>
<tr>
<td>Q</td>
<td>Volumetric Flow Rate</td>
</tr>
<tr>
<td>T</td>
<td>Temperature</td>
</tr>
<tr>
<td>U</td>
<td>Axial Velocity</td>
</tr>
<tr>
<td>V</td>
<td>Tangential Velocity</td>
</tr>
<tr>
<td>Y</td>
<td>Adiabatic Expansion Coefficient</td>
</tr>
<tr>
<td>r</td>
<td>Pressure Ratio</td>
</tr>
</tbody>
</table>

**Greek**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Φ</td>
<td>Flow Coefficient</td>
</tr>
<tr>
<td>Ψ</td>
<td>Loading Coefficient</td>
</tr>
<tr>
<td>β</td>
<td>Relative Flow Angle, Venturi Diameter Ratio</td>
</tr>
<tr>
<td>γ</td>
<td>Specific Heat Ratio</td>
</tr>
</tbody>
</table>

**Subscripts**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>∞</td>
<td>Free Stream</td>
</tr>
<tr>
<td>0</td>
<td>Stagnation Condition</td>
</tr>
<tr>
<td>1</td>
<td>Inlet</td>
</tr>
<tr>
<td>2</td>
<td>Exit</td>
</tr>
<tr>
<td>M</td>
<td>Midspan</td>
</tr>
<tr>
<td>SCFM</td>
<td>Standard Flow Rate Designation</td>
</tr>
<tr>
<td>i</td>
<td>Measurement Station Index</td>
</tr>
<tr>
<td>r</td>
<td>Radius</td>
</tr>
</tbody>
</table>
SUMMARY

Low emission combustors produce turbine gas path radial temperature profiles which are relatively flat. This has resulted in endwall region over-temperature problems, specifically the blade and vane tip and platform regions. The desire to use less film cooling also puts an additional burden on the heat transfer design in these regions. Both data and comparison to CFD solutions are needed to provide a detailed and quantitative definition of both the complex 3-D flow and heat transfer phenomena.

The primary objective of this research is to investigate and quantify airfoil tip, platform and endwall region flow and heat transfer. Specific objectives include the acquisition and analysis of high-quality detailed data quantifying the complex flow in these regions in a research turbine with representative parameters, and comparison of these data to solutions obtained with state-of-the-art CFD tools.

Toward this end an investigation is performed on the first stage of the Purdue Research Turbine using Particle Image Velocimetry (PIV). The flow field is interrogated in the near-hub region of the intra-stage space, downstream of the first vane row. Purge air is introduced through a planar seal at three different flow rates which characterize the typical range of dimensionless seal flow rates encountered in practice. Two-dimensional (radial and axial) velocity data from four measurement planes spaced from vane pressure to vane suction side are acquired. These data are phase-locked to rotor position. The ensemble-averaged data from each rotor position provides a characterization of the effect of the rotor potential field on the emergent seal flow.
1. INTRODUCTION

Modern low emission combustors produce turbine gas path radial temperature profiles which are relatively flat. This has resulted in endwall region over-temperature problems, specifically in the blade tip and platform regions. Because of this, and the desire to improve system efficiency by cooling flow reductions, the issue of endwall heat transfer has taken on greater significance in turbine design. Predicting heat transfer on the blade platform is difficult because of the highly complex nature of the flow field, which features seal purge air introduced into regions where secondary flows are already significant (Sharma and Butler, 1987). In addition, high performance turbines feature closely spaced blade rows and thus are characterized by inherently unsteady intra-blade regions. Here, the seal air emerges into a low-momentum endwall flow dominated by convected wakes from upstream vanes and potential disturbances from highly loaded downstream rotors.

Thermal management of internal disc cavities and structures is a critical durability issue. The literature contains many interesting computational and experimental works addressing this issue. These studies have focused on a variety of phenomena including windage effects resulting from facing surfaces which are alternately stationary and rotating, conduction from wetted surfaces to internal engine structural components, and leakage of hot mainstream gas through rim seals. Due to the extremely high temperature of the turbine mainstream flow, the most significant issue in this thermal management problem is the prevention of hot gas ingress.
The primary function of the purge air flowing from the disk cavities through the vane-rotor seal is to prevent hot gas ingress into these cavities. Researchers have focused considerable effort on the seal purge flow from this perspective (Green and Turner, 1994; and Hills et al., 1997). The recent review of the literature by Johnson et al. (1994) gives a complete discussion of seal ingestion including the extent to which studies have focused on the influence of mainstream flows. Von Karman’s (1921) work on viscous pumping formed the basis for early approaches addressing the cavity seal flow problem. These studies expanded to include the effect of cavity geometry, surface irregularities (e.g. fasteners), gap sizes and forms, and rotational rates. These studies reveal some information about the expected nature of the flow leaving the seal, but are idealized to the extent that external flow effects were not considered. Thus, further studies of seal flow including the presence of external flows have been conducted with variations in the swirl of the external flow and pressure asymmetry of the external flow.

The intent of these studies has been to understand how to optimize the rim seal design. As this seal flow degrades the thermal efficiency of the gas turbine cycle, the goal is to minimize the seal flow rate required to eliminate ingress. However, less effort has been expended studying the effect of the seal flow on aerodynamics and heat transfer in the mainstream flow outside of the disk cavity.

Although the primary function of the seal flow is to prevent ingress, a beneficial side effect can be the enhancement of platform film cooling. However, the effectiveness of this cooling is dependent on the trajectory of the emergent seal flow, with this path not well understood. An understanding of the parasitic purge flow, which must be tolerated, may well allow optimization of ejection characteristics to maximize the platform cooling effect of this flow.

It is expected that the horseshoe vortex formed at the blade leading edge will be a significant mechanism in the transport of the seal cooling flow. The horseshoe vortex itself can play a crucial role in blade heat transfer, producing a scraping effect that enhances blade suction surface heat transfer and adversely affects surface temperature
Because of this, suppression of the horseshoe vortex has been a topic of interest in turbine design, and the purge flow may also have a role to play in this process.

This research effort is focused on developing a better understanding of the seal flow-mainstream flow interaction. With such knowledge, the motivation for rim seal design improvement may well grow to include not only the prevention of ingress, but also the optimization of blade heat transfer and aerodynamic performance.

1.1 Literature Survey

A single study by Granser and Schulenberg (1990) is concerned with the cooling effect of a purge flow introduced through a slot upstream of a blade row. This film cooling study stands alone in the literature as the only available work on the usefulness of purge flow in platform cooling. The issue remains of interest, however, as the work of Granser and Schulenberg was limited to a linear cascade of two stationary blades. Aside from this work, understanding of the complex flow issues and forces that contribute to the seal flow interaction with the mainstream flow must be garnered from other sources.

While most cavity flow work is dedicated to understanding heat transfer within the seal cavity or the cavity mechanisms that contribute to cavity ingress, there are some that consider the nature of external flows. Again these works concentrate on the cavity effects of the external flow and use highly simplified simulated mainstream flows. The review by Johnson et al. (1994) cites the literature as providing only one work including the effects of both vanes and blades. Nevertheless, for completeness, some of the work focused on cavity flows will be discussed here.

As the interaction of seal purge flow with secondary flow structures may influence the effectiveness of any hoped for external blade platform cooling benefits, information on secondary flow structures, film cooling, blade row flow mixing, and interaction of film cooling with secondary flows is of interest here.
The basic geometry to be considered is presented, Figure 1.1, in an engine cross section taken from Wilson et al. (1995). The typical inner engine cavities and cooling air flow paths are shown. Within the figure, the cavity space is called a ‘wheel-space’ and the cool air delivered to these cavities is called disc cooling air. Wilson et al. measured and predicted heat transfer within the cavity space, which is not of primary interest here, but contributed to an understanding of the motivation for these studies. They choose to study a simple cavity geometry which is common in the literature, and is sketched in Figure 1.2.

### 1.1.1 Seal Exit Profiles

Seal cavity flow work provides insight to possible velocity and temperature profiles of the flow exiting the seal gap. The study of Ko and Rhode (1992) is among those that present details of flow within the seal gap. They predict purge air exit velocity profiles into an external cross flow with near zero radial components at the upstream seal edge and very strong radial components at the downstream edge. All of their work generated time average results and did not predict ingress at any axial point along the seal exit plane. The computational studies by Hills et al. (1997) also provide insight to flow within the seal gap, adding the complication of external pressure asymmetry with upstream bluff bodies to simulate vanes and thus predicting a circumferential variation in the radial and tangential velocity components. Maximum ingress is predicted in the negative circumferential direction from the low pressure zone and maximum egress is predicted just to the positive side of the low pressure zone. The circumferential direction was taken as positive in the direction of rotor rotation. As with this study, all available information on the nature of the seal exit flow field has been predicted computationally.

Some experimental investigations that may add to understanding the nature of the seal exit flow have observed lobe patterns of recirculation within the wheel space. Work by Phadke and Owen (1988) and Nordquist et al. (1990) show non-uniformity existing within the wheel space flow field. These flow patterns certainly affect the way in which
the seal flow interacts with the main stream flow. Interaction with the external flow may
be one of the most important factors in understanding seal flows according to Chew et al.

Chew (1991) reviews selected works by Phadke (1982), Phadke and Owen
(1983a, 1983b, 1988), Bayley and Owen (1970) and others to develop a mathematical
model for prediction of cavity and seal flows. The work focused on the minimum seal
flow rate to prevent ingress at the seal. The developed theory was applied to the
experimental measurements of the investigators listed. Pressure ratio and seal gap
dimension both influence the ingress problem, but it was found that they are linked
together and not independent. Also Chew claims a non-linear relationship between
ingress and rotational Reynolds number, which had not been proposed in previous
theoretical studies. But the most significant conclusion, in light of the present research, is
that a need exists for understanding the impact that pressure asymmetries in the external
flow have on the level of ingress. He postulates that these pressure effects may be even
more important than the seal flow Reynolds number. Four different geometries were
considered in this discussion, but the main geometry of concern is of the same style as
that used in the Purdue Research Turbine, and sketched in Figure 1.2.

To address these questions Chew et al. (1994) developed a mathematical model to
describe cavity flow which included flow within the seal gap and the influence of an
external flow described by an experimentally determined pressure variation due to
upstream nozzle guide vanes. The result is a distinct circumferential variation in the seal
gap through flow, with a location of inflow which is unexpected, and attributed to a
strong circumferential component of velocity within the seal gap. Their primary
conclusion from the study of an asymmetric external flow is that low seal flows are
dominated by the vane induced pressure field, with the higher seal flow rate interaction
with the mainstream flow is postulated to be important in influencing seal effectiveness.

Hartland et al. (1998) are concerned with the effect of circumferential pressure
distributions on the seal flow. When the mainstream is of relatively high pressure then
the seal flow pressure must be high to prevent ingress. Conversely, when the mainstream flow is of relatively low pressure than the high pressure seal flow will leak heavily into the mainstream. Thus some work has been done to profile the upstream endwall surface to cause a local reduction in pressure in those regions where the mainstream is normally high pressure. Hartland et al. make the point within theory development that circumferential pressure variations result from pressure waves that travel nearly normal to the flow direction or the exit blade angle, with the high pressure wave initiation point at the blade trailing edge. The low pressure waves are clocked one half of a vane pitch from the high pressure wave, and emanating from the vane suction surface.

Also influencing seal effectiveness, and hence the strength with which the purge will emerge into the main flow, is the swirl of the mainstream flow. Izenson et al. (1994) conclude that the magnitude of the external tangential velocity relative to the wheel speed can be increased to the point where significant increase in seal flow rates is required to maintain seal effectiveness.

Unfortunately, the seal flow literature is focused on predicting ingress, and the limited information available that presents egress details is far from enough to give a picture of the flow profile that should be expected from a seal. Additionally, a study of the circumferential temperature profile of the exiting seal flow may be useful but has not been found in the literature.

With a similar cavity geometry to that shown in Figure 1.2, a numerical study on heat transfer within the cavity was conducted by Ko and Rhode (1992). They predict a pattern of strong flow recirculation just within the cavity under the seal gap. This recirculation is credited for an area of increased temperatures, since the recirculation introduces thermal mixing, taking warm fluid near the exit and mixing it with cool fluid further inside the cavity. This indicates that care must be taken in using the rim seal flow for cooling as this mixing effect will minimize surface film cooling efficiency, with the fluid exiting the seal gap having been warmed within the cavity. The heat transfer studies within literature concentrate on effects applied to surfaces within the cavity.
1.1.2 Vane Exit Flows

The vane exit flow is convected across the seal gap. Vane exit flow structures and the vane developed pressure field are of particular interest. It is important to know the pressure field and the velocity field into which the seal flow is introduced, since the interaction of the two may influence downstream flow development.

Measurements of pressure, velocity and flow angle from a low speed single stage turbine nozzle guide vane by Zaccaria et al. (1996) were made at stations near the exit of the nozzle passage. The measurements give a clear picture of the flow structures near the endwall. The suction side of the passage shows strong vortical structures at both the tip and the endwall. These structures result from the passage vortex, as explained by Sharma and Butler (1987), Langston (1980) and Sieverding (1985). The structure results from a pressure gradient from the pressure side of the vane to the suction side of the vane which initiates a cross-passage flow within the boundary layers of the endwalls. By continuity, this wall flow sets-up two regions of circulation of opposite rotational direction within the passage. Once established, these circulations are then pushed towards the suction side of the passage, so that a survey of the passage exit shows that these structures have a strong vortex core on the suction side of the passage.

Figure 1.3 taken from Zaccaria et al. (1996) shows a secondary flow velocity vector plot at the suction surface and endwall corner. Point P is at the center of a counter clockwise rotation described as the passage vortex flanked by a weaker rotation labeled H_{ss}, which is conjectured to be the remnant of the suction side leg of the horseshoe vortex.

An earlier study by Zaccaria and Lakshminarayana (1995) surveyed the development of these nozzle exit structures at downstream locations where the pressure side flow and the suction side flow interact. The interesting result was to see that the strong pressure differential within the wake served to annihilate the passage vortex core found in the suction side – endwall corner. Their results, Figure 1.4, show a series of
structures of opposite rotation of the passage vortex across the entire blade span. The represented measurement location is 0.025 chords downstream of the trailing edge. These structures shown in the secondary (perturbation) velocity plot of Figure 1.4 are very tight vortices with the strongest appearing at the hub wall. It is this flow structure that is convected downstream toward the seal gap and into the rotor blade passage.

These experimental investigations of the vane flow field can be used to understand the type of structures expected to exist within the measurement region of the intra-stage space. It is important to note that the passage vortex which might be expected to impact the seal flow is not present at the downstream seal location, but rather the wake flow field which has structures counter to the rotation of the passage vortex exist within the measurement region.

1.1.3 Rotor Blade Passage Flow

These vane studies also serve as a model for the type of flow structures that exist within a rotor blade passage. The flow structure tracking within a vane row show that boundary layer fluid is swept into the passage vortex. The complementary rotor blade passage structure is then likely to capture all of the cooling flow emanating from the seal gap and entering the rotor passage.

An ethylene injection and detection technique is used by Denton and Usui (1981) to track flow through a low speed turbine. They claim that tracking through the nozzle row shows that “all fluid that enters the nozzle row close to the casing wall ends up in the core of the passage vortex.” In fact the presented concentration distributions show a distinct migration from pressure to suction side for both the casing side injection and the hub side injection. The most interesting finding here is that only the lower half of the boundary layer is completely swept to the suction side, and the upper portion of that same boundary layer remains on the pressure side, reattaches and forms a new boundary layer beyond the passage vortex separation line.
When the tracking fluid passed through the downstream rotor row, it was expected that the rotor surface boundary layers with their low momentum fluid would result in sweeping the tracking fluid through a great circumferential distance. The fact that this was not witnessed in large concentrations resulted in the conclusion that very thin boundary layers exist on these surfaces, thus maximizing the effect that mainstream flow can have on surface heat transfer.

In fact the heat transfer studies of Hamabe et al. (1993) show high values of Nusselt number on the blade suction side near the trailing edge and the hub wall. This is the same location where the passage vortex is observed. The alignment of this flow structure with the region of high Nusselt numbers suggests that the scraping effect of the passage vortex is enhancing heat transfer at this point. In addition to high heat transfer at the trailing edge, their studies show relatively large Nusselt numbers at the leading edge, just to the suction side of the stagnation line, on the hub wall. These two areas are important to consider when projecting the influence that seal flow could have on heat transfer.

1.1.4 Film Cooling and Secondary Flows

Angled injection holes positioned at various vane endwall axial locations were used by Goldman and McLallin (1977) for a study of the secondary flow effects of platform film cooling. Vane row exit pressure profiles were used to determine the size of the vortex cores convecting off of the vane trailing edge along the end wall. These regions were measured for both endwall cooled and endwall un-cooled cases. The results were shown as pressure contours which indicate the size of the vortex core generated at the vane end wall interface along the suction side of the vane. These clearly show that the size of the vortex core is affected by the cooling flow. It appears that high pressure ratio coolant injection results in weaker vortex cores, given by a smaller area of reduced pressure. The investigators attribute the improvements to higher momentum fluid being introduced in the case of high injection pressure ratios. The injection pressure ratio is
measured as the ratio of the inlet pressure of the coolant to the inlet pressure of the primary flow. It should also be noted that this coolant flow had an impact on the secondary flows. The lower momentum injection was carried along by the primary flow induced secondary structures, but higher momentum injection tended to disrupt the secondary structures generated by the primary flow stream.

Goldman and McLallin (1977) also note that a strong vortical structure is generated at the vane suction surface-end wall interface. The coolant delivered upstream is carried into these structures by the end wall flow migration from pressure surface to suction surface. Also note that the coolant injection studied was concerned with the angle of coolant injection, not the inclination of the hole which was 15° to the endwall, but with the angle of injection relative to the streamline direction. Directing the coolant at an angle relative to the flow direction resulted in redirecting the through-flow streamlines.

The direct influence of endwall cooling on secondary flow structures was summarized by saying, “[that] increased coolant momentum decreased under-turning in the passage vortex regions.” Two studies on endwall flow injection trajectory in a turbine, Moore and Smith (1984) and Gaugler and Russell (1984), revealed little more than additional qualitative understanding of where path-lines typically exist within a single vane passages.

Granser and Schulenberg (1990) present results from a two vane cascade experiment. Their results identify the rim seal flow as useful for film cooling of the vane platform. The inlet flow in this case is uniform. Thus the measured film cooling effectiveness is uniform across the passage, and at the passage inlet. However, due to the passage vortex, the downstream cooling effectiveness is skewed advantageously towards the suction side of the passage. Flow in the near seal area is not presented in detail, but the effect of blowing rate and momentum addition is discussed with a focus on the limit at which the blowing ratio has a beneficial cooling effect. The resultant of cooling flow
penetration through the mainstream boundary layer is an identifiable limit on the cooling effectiveness.

1.2 Research Objective

The overall objective of this research program is to gain an understanding of how purge flow introduced through a first stage turbine rim seal interacts with the external flow field so that downstream flow development can be understood for the purpose of generating meaningful heat transfer predictions. If improved platform cooling and/or secondary flow suppression is to be garnered from what is a necessary evil from a performance perspective, then a capability must be demonstrated to model these complex seal flows.

Unsteady viscous Computational Fluid Dynamics (CFD) simulations thus become the obvious tool of choice, and investigations of similar flows have already shown promise in compressor geometries (Heidegger et al., 1994). However, without data with which to confirm the predictions and calibrate the codes, the usefulness of such simulations is limited. To address this, the current research program is focused on providing unique experimental data to characterize the vane-blade seal purge flow behavior and its effect on secondary flow structures. To recreate the nature of this complex unsteady flow field, experiments are conducted in the Purdue Research Turbine, with the flow field in the seal region downstream of the first vane and upstream of the first rotor being characterized with Particle Image Velocimetry (PIV).

The specific objectives of the experimental work are to quantify the velocity field in the intra-stage space, including the velocity field that develops downstream of this seal injection point, since this initial interaction of the main flow and the injection flow reveals mixing information from the seal flow interaction with the horseshoe vortex. Measurements within this region are made such that variations of the flow field with circumferential location are captured. The measurements are intended to reveal the
influence of upstream flow structure generated from the vane row, on flow development downstream of the seal flow.
Figure 1.1  A typical turbine cross-section with cooling flow paths (from Wilson, 1995).
Figure 1.2 A typical seal cavity geometry.
Figure 1.3 Flow field within a turbine nozzle (from Zaccaria et al., 1996).
Figure 1.4 Flow field at the exit of a turbine nozzle (Zaccaria and Lakshminarayana, 1995).
2. EXPERIMENTAL APPARATUS AND METHODS

A cold turbine with blading typical of high-pressure stages was modified specifically for an investigation of seal purge flow using Particle Image Velocimetry. An existing turbine facility was retrofitted with a new blade set, resulting in a required modification of the load control system for the purpose of operating at the design point of the new blading. An expanded optical access window was added to the case of the machine for imaging the flow path of interest.

2.1 Turbine Facility

The Purdue Low-Speed Research Turbine, Figure 2.1, was designed for the experimental investigation of unsteady flow phenomena in turbines (Weaver, 1993), and was here modified for the purpose of secondary flow and cooling flow investigations. The primary facility components are the turbine, the main air flow system, the seal flow system, and the turbine loading system.

2.1.1 Turbine

The Purdue Low – Speed Research Turbine is a two stage, axial flow turbine with constant hub and tip diameters. The blading is of forced vortex design, and was designed by the Allison Engine Company (currently Rolls-Royce plc) to recreate the essential aerodynamic features of modern low-solidity high pressure turbine stages. The significant parameters of the blade and vane rows are presented in Table 2.1.
Table 2.1 Turbine Design Geometry.

<table>
<thead>
<tr>
<th>Number of Stages</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tip Diameter</td>
<td>19.18 in.</td>
</tr>
<tr>
<td>Hub/Tip Diameter Ratio</td>
<td>0.739</td>
</tr>
</tbody>
</table>

Geometry at 50% span

<table>
<thead>
<tr>
<th></th>
<th>Stator 1</th>
<th>Rotor 1</th>
<th>Stator 2</th>
<th>Rotor 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Blades</td>
<td>24</td>
<td>32</td>
<td>28</td>
<td>32</td>
</tr>
<tr>
<td>Meridional Chord (in.)</td>
<td>2.93</td>
<td>2.23</td>
<td>2.54</td>
<td>2.27</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>0.85</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>Solidity</td>
<td>1.288</td>
<td>1.12</td>
<td>0.982</td>
<td>1.100</td>
</tr>
<tr>
<td>Camber Angle (deg)</td>
<td>86</td>
<td>98.0</td>
<td>98.0</td>
<td>98.0</td>
</tr>
<tr>
<td>Thickness/Chord (%)</td>
<td>16</td>
<td>21</td>
<td>21</td>
<td>18</td>
</tr>
</tbody>
</table>

The facility features a constant tip diameter of 19.18 in. (0.49 m), a hub/tip ratio of 0.739, and a nominal rotational speed of 2500 rpm. Figure 2.2 depicts the turbine blade row configuration, with a model of the actual turbine blades and cross sections shown in Figure 2.3. The design point predicted midspan static pressure distributions are presented in Figure 2.4.

2.1.2 Main Air Flow System

Air at ambient temperature and pressure is drawn into the turbine inlet by a Novenco 800 centrifugal blower powered by a Marathon 100-hp (75-kW) 3565-rpm electric induction motor. The turbine inlet is an annular contraction with an area ratio of nine and a length-to-inlet diameter ratio of 0.75. Downstream of the turbine is an annular diffuser with an area ratio of 1.79. A 12-ft (3.7 m) length of 20-in. (0.5 m) diameter PVC pipe connects the turbine cascade to the blower. A Badger Meter, Inc. type BVT-IF 20-in. (0.5 m) diameter venturi flow meter is located just upstream of the blower inlet. At the blower inlet, air flow rates of 3,000–5,400 cfm (1.4 – 2.6 m³/s) are regulated by a variable inlet vane damper allowing blade row axial velocities ranging from 40–110 ft/sec (12.2–33.5 m/s) to be achieved. A Remote Controls, Inc. model RC40SR-60 90°
rotation spring return pneumatic actuator is used for damper vane actuation, controlled by a Kytronics, Inc. model DLU80 digital positioner. The Turbine rotational speed is monitored with a Honeywell model UDC 3000 process controller which closes the damper should the turbine rotational speed exceed 2,700 rpm. The blower provides a maximum suction of –2.4 psig (16.5 kPa) at blower inlet. Figure 2.5 schematically depicts the air flow system.

2.1.3 Turbine Loading System

Turbine loading is controlled by a Go-Power Systems model EDP-316 water-brake dynamometer. At the highest turbine loading 40 hp (30 kW) is absorbed by the loading system at a steady torque of approximately 70 ft-lb (95 m-N) when operating at 2500-rpm. A Morse type HV chain drive transmits power from the turbine shaft to the water brake.

Turbine loading is regulated by control of the water flow rate through coarse-adjustment and fine-adjustment needle valves. Both the flow rate and pressure supplied to the water brake vary with control valve setting. As the current studies mandated tight speed regulation, a closed-cycle water system was installed to insulate the system from variations in water main pressure. A Gould model JS09 1-hp jet pump is used to supply pressures of 0–70 psig (0-482.6 kPa) at flow rates as high as 35 gpm (2.1 L/s) to the water brake. The pump draws from an 80 gallon (300 liter) supply reservoir. A bypass line allows a continual stream of water to be pulled through the pump even when the water brake supply is highly restricted, thus preventing a pump motor thermal overload and unexpected pump shutdown. Such a water pump shutdown would result in uncontrollable turbine loading and possible turbine over-speed.

Because the water-brake performance is sensitive to exit pressure restriction, the discharge water must exit below the water brake to atmospheric pressure. To achieve this, a discharge pan is employed along with a scavenge pump to transfer discharge water
to the supply reservoir. A schematic of the water system for the water brake is shown in Figure 2.6. The thermal energy added to the water system from the extracted turbine work must be removed to keep the system within the individual component temperature limits. As ambient convection is not sufficient, warm water is continually drained from the system and made-up with cool water from the laboratory mains.

2.2 Turbine Operation

Steady turbine operation is described by dimensionless work flow parameters that serve as set points and are quantified during turbine operation.

2.2.1 Performance Parameters

The turbine operating point is specified by the midspan values of the blade loading coefficient $\Psi$ and flow coefficient $\Phi$. These dimensionless parameters set the aerodynamic loading of the turbine stage:

$$\Psi \equiv \frac{(V_{a2} - V_{a1})\Omega r_M}{(\Omega r_M)^2} \quad (2.1)$$

$$\Phi \equiv \frac{U_{\infty}}{\Omega r_M} \quad (2.2)$$

The blade loading coefficient $\Psi$ represents the specific work extracted from a turbine nondimensionalized by the square of the wheel speed. The work term is calculated from the difference in rotor inlet (1) and exit (2) tangential velocities at midspan ($M$). The turbine stage through-flow is represented by the flow coefficient, which is the ratio of the area-average axial velocity $U_{\infty}$ at the inlet to the wheel speed. These dimensionless parameters form the turbine operating map and for ideal flow are related as the stage characteristic:
\[ \Psi = \Phi (\tan \beta_1 + \tan \beta_2) \quad (2.3) \]

where \( \beta \) is the relative flow angle at the rotor blade inlet (1) and exit(2).

2.2.2 Performance Instrumentation

Figure 2.7 schematically diagrams the turbine performance instrumentation. Stagnation temperature measurements are made with three thermistor probes sensing midspan temperature at Stations 0, 2, and 4. The thermistor probes, Figure 2.8, are constructed from Omega model 44032 precision thermistors matched by the manufacturer to a standardized resistance-temperature curve to within ±0.2 °F (±0.2 °C) accuracy. Temperature is determined by first measuring the thermistor resistance via a voltage divider circuit, then determining the temperature from the standardized resistance-temperature curve.

Stagnation pressure measurements are made with a United Sensor model PAC-8-KI Pitot probe and two stagnation pressure probes that are aligned with the mean flow to sense midspan pressures at Stations 0, 2, and 4 respectively. The pressure probes are constructed of stainless steel tubing and connected by pneumatic tubing to a 5 psi (34.5 kPa) Scanivalve Corporation differential pressure transducer. The Scanivalve sensor is calibrated with a water manometer. Pitot and stagnation pressure probe geometries are depicted in Figure 2.9.

The volumetric flow rate measurements for determining the flow coefficients are made with a venturi flow meter. The following equation correlates the manufacturer’s venturi flow calibration.

\[ Q_{SCFM} = 43655 \left[ \frac{P_0}{T_0} \left( P_0 - P \right) \right]^{1/2} Y \quad (2.4) \]
\begin{align*}
Y &= \left[ \frac{r^{2/\gamma} \gamma \left(1 - r^{(\gamma-1)/\gamma}\right) \left(1 - \beta^4\right)}{(\gamma - 1)(1 - r) \left(1 - \beta^4 r^{-2/\gamma}\right)} \right]^{1/2} \tag{2.5}
\end{align*}

where \( Q_{SCFM} \) is the volumetric flow rate in scfm, \( P_0, P, \) and \( T_0 \) are the stagnation pressure, static pressure, and stagnation temperature at the venturi in psia and °R. The parameter \( Y \) is the adiabatic expansion coefficient, with \( r = P/P_0 \), venturi diameter ratio \( \beta = 0.41 \), and specific heat ratio \( \gamma = 1.40 \).

Stagnation and static pressure taps from the venturi are connected via pneumatic tubing to a Scanivalve pressure transducer. Stagnation temperature at the venturi is assumed to be equal to the measurement made at Station 4 downstream of the last turbine stage since convection time is only approximately 0.3 sec from that point to the flowmeter. With the volumetric flow rate known, the area average axial velocity \( U_{\infty, i} \) at station \( i \) is found from

\begin{equation}
U_{\infty, i} = \frac{14.696}{P_i} \frac{T_i}{519.67} \frac{Q_{SCFM}}{A} \frac{1}{60} \tag{2.6}
\end{equation}

where the annular area of the turbine \( A = 0.90972 \text{ ft}^2 \), static temperature and static pressure are in °R and psia, yielding \( U_{\infty, i} \) in ft/sec.

Rotational speed measurements of the turbine rotor are made with a one-per-revolution optical trigger mounted on the turbine. The photodiode is connected to a Schmidt Trigger Integrated Circuit. The resulting TTL output waveform has a fundamental harmonic at the rotor rotation rate. A National Instruments NB-MIO-16XL PC-based data acquisition board samples this frequency signal and measures the time between pulses, which is the period of one rotor revolution. This waveform from the trigger is also routed through a frequency to voltage converter to the Honeywell UDC controller which protects the system from rotor over-speed.
2.3 Seal Flow System

The first stage vane blade gap features a planar seal as shown in Figure 2.10. Since the experimental turbine facility is a cold flow facility and was not initially designed to model seal flow behavior, there was no inherent need for the cooling flow passages that would normally be integral to a high temperature turbine. In this original configuration, the inner cavity that would exist on a turbine engine was instead replaced with an open-ended rotor drum facing the upstream center-body and the hollow inlet nosecone. Thus addition of internal walls and flow passages was required to allow for the seal flow studies.

2.3.1 Seal Flow Path and Cavity Geometry

A dimensioned drawing of the seal geometry is presented in Figure 2.11. The nominal width of the seal at the measurement location is 18 mil (0.45 mm), and the axial gap to disc radius ratio is $2.4 \times 10^{-3}$. The vane-rotor gap varies during rotor rotation due to run-out of the rotor shaft. This run-out is measured statically as 3 mil (0.0762 mm) on the 6 in. (0.152 m) diameter drum/shaft mounting point. Further variation in seal gap size exists during operation due to dynamic effects and minor variations in drum-shaft mounting. To account for these variations, the actual seal gap has been measured both with optical techniques and with feeler gages.

The optical measurements are dependent on the optical set-up described within Section 2.4.1. Using the PIV camera, a view of the seal and measurement plane is digitized, and each image pixel has a given length for the specified optical magnification. The number of image pixels that fit within the seal gap is then the measure of the seal width. A sampling of this value was taken for both static conditions and under operational conditions. The range of seal widths as measured optically is 18-26 mil.
For comparison, the range of seal widths measured with feeler gages ranged from 17 - 19 mil (0.43-0.48 mm). It is postulated that the optical measurements did not measure the actual minimum seal width, but rather a wider portion that exists within the imaged area, due to rounded edges of the seal.

The model in Figure 2.12 shows the cavity form, which is below the seal gap. The instrumentation ring is shown with three spokes supporting a center disc. The open space in this ring is sealed with a rigid, heat shrinkable plastic film that is applied over the entire surface of the ring. At the center of the ring an annular pattern of electrical connectors are raised from the surface by 0.11 in. (2.8 mm) and are located at a 5.56 in. (141 mm) diameter at the outer most point. Thus, the plastic sealing film forms a conical shape leading up to the planar annulus at the outer ring of the wheel. On the opposite side of the seal, an acrylic sheet is mounted to the seal ring and forms the fixed facing wall of the seal cavity. The maximum cavity width is 0.35 in. (8.9 mm) at the outer radius.

At the center of the fixed seal wall is an annular opening of 2.5 in. (63.5 mm) outer diameter and 1.375 in. (34.9 mm) inner diameter through which the pressurized seal flow is delivered. The seal flow is delivered to this opening through a section of telescoping PVC pipe mounted within the inlet nose cone. Any misalignment between the center of the nose cone and the center of the pipe coupling is absorbed in the o-ring that seals to two pipe sections. At the upstream edge of the pipe section, duct putty is used to seal the pipe with the rough interior wall of the nose cone.

This seal flow feed pipe is attached to a centrifugal blower powered by a 5-hp (3.75-kW) electric motor. This blower is only required for the highest seal flow rate used in these experiments, as in the other two cases the natural pressure differential and rotor pumping action was sufficient to supply the required flow rates. A throttle plate on the blower inlet, a throttle valve on the delivery pipe inlet, and pipe friction were all used to control the seal flow rate. For the three seal flow rates investigated, two different size seal flow supply lines were used. For the larger flow rate a 3 in. (76.2 mm) supply pipe
was used, and for the smaller flow rates a 1 in. (25.4 mm) supply pipe was used. A photograph of the seal flow delivery system is given in Figure 2.13, and a schematic drawing of the details of the system is provided in Figure 2.14.

2.3.2 Seal Flow Rate Measurement

Seal air flow rates of 90-120 cfm (0.042-0.056 m\(^3\)/s) are inferred by velocity measurements made with a United Sensor model PAA-“a”-KL 1/16 inch (1.6mm) Pitot-static probe. A small probe diameter was selected for the purpose of minimizing end wall boundary layer interference effects. At a distance of over 10 probe diameters from the flow passage wall, there is no measurable effect of the endwall on accuracy. Interference of an accurate measurement would normally arise from a venturi affect developed within the probe – wall space. Additionally, the flow Reynolds number could be of concern. For low flow rates delivered to a small impact hole, viscous effects may be an issue. United Sensor Corporation publishes 12 ft/sec (3.65 m/s) as the critical velocity for air at standard atmospheric conditions impacting a hole of 0.010 in. (0.25 mm) in diameter. The smallest velocities are measured around 40-50 ft/sec (12 – 15 m/s) in the 3 in. (76.2 mm) supply pipe, which is far above the speeds that result in error.

The Pitot-static probe was mounted at 80 in. (2.03 m) from the pipe leading edge for the 3 in. (76.2 mm) supply pipe, and a development length of 80 in. (2.03 m) was used for turbulent boundary layer calculations. For the smaller pipe, the Pitot static probe was mounted at 50 in. (1.27 m) from the pipe leading edge. The probe location was selected so that the flow was fully developed far upstream of the core velocity sampling point. In all three seal flow cases, the pipe Reynolds number was sufficient to use turbulent pipe flow correlations for predicting the boundary layer thickness and displacement thickness. Displacement thickness at the probe location was determined to represent less than 0.4% error in flow rate calculations.
The two pressure measurements for the Pitot-static probe are taken with a PDCR23D Scanivalve pressure transducer, which is made for a maximum 5 psi (34.5 kPa) differential measurement and outputs a $\pm 12$ V signal. The static accuracy of the device is published as $\pm 0.06\%$ or $\pm 0.003$ psi (20 Pa). The differential pressure measurements were taken relative to atmospheric conditions, and were over 200 times greater than the smallest measurable pressure, so the certainty of the individual measurements is not in question. The differential between the two measurements of total pressure and static pressure within the seal flow delivery pipe must be greater than the accuracy of the device. The predicted pressure differential for a flow at 45 ft/sec (13.72 m/s) is 0.0173 psi (199 Pa), resulting in an uncertainty in the velocity measurement of 8.7% or $\pm 3.9$ ft/sec (1.19 m/s). An empirical study of this measurement revealed that the measured pressures were consistent with micro-manometer measurements, and thus the true accuracy was higher than predicted.

Pipe flow temperature is measured with an unshielded type-T thermocouple bead. The measurement is electronically cold junction compensated using an Omega Electronic Ice Point model MCJ. The compensated signal was acquired through a National Instruments data acquisition board NB-MIO-16XL on a Macintosh Quadra 950 computer.

2.4 Particle Image Velocimetry

The Dantec Inc. PIV system utilized for these experiments consists of a 30 mJ NewWave Research Minilase III Nd:YAG laser, a high-resolution Kodak Megaplus ES 1.0 digital camera, and a dedicated PC controlled PIV 2100 Processor. The laser has twin oscillators operated in single Q-switch mode and are capable of delivering a 5-7 ns duration pulse with a wavelength of 532 nm (visible green light) at a repetition rate of 10 Hz. The digital camera has a 1008 x 1018 CCD array operated in cross-correlation mode, with the images corresponding to the first and second pulse of the laser recorded separately, with the minimum allowable time between these frames of 1 µsec. Both
images are then transferred to the PIV 2100 Processor which provides near real-time vector processing of the images using Fast Fourier Transform (FFT) correlation techniques. This unit also synchronizes the camera and laser, and is capable of resolving the particle displacement to within one-tenth of a pixel through the use of sub-pixel interpolation schemes. PC controlled software is used to perform off-line validation and post-processing of the vector maps, with directional velocity information unambiguously determined since the initial and final particle positions are recorded as separate images.

2.4.1 Optical Path

To image the seal flow region, a 39.4 mil (1 mm) thick laser light sheet is introduced through the bellmouth inlet section using a combination of cylindrical lenses. Two angled surface mirrors deflect the laser beam to a height such that the beam is at a 15% span location within a radial-axial plane oriented 45 degrees from horizontal. Alignment of the beam with the turbine axis was achieved by mounting a pair of targets at parallel upstream and downstream locations. The bellmouth mounting flange and the diffuser exit flange were mount points for these targets, which carried angular and spanwise demarcations. Thus a gage was established for aligning the laser sheet at a known angular and radial location, on a plane that passed exactly through the axis of the turbine shaft.

The cylindrical lenses chosen are of focal length: f=+80mm, f=-250mm, and f=+250mm, listed in order from upstream to downstream. All three are positioned such that the upstream surface is the planar lens surface. The three lenses are spaced at approximately 8 in. (200mm), but precise adjustment must be made to establish the desired sheet thickness and height. The positive lenses are used to control the sheet thickness, and the negative lens is used to spread the beam into a sheet that is of a designed height. The height of the light sheet must be sufficient to cover the entire measurement region without over-expanding the beam. Over-expanding the beam will
result in intensity reduction so that particle reflections can not be distinguished from background noise.

This light sheet is brought from the turbine inlet through the first vane row to the seal region. Due to the typical large turning in this vane row, the removal of a single vane was necessary, thus introducing a local reduction in solidity. The measurement plane, typical of that shown in Figure 2.15, spans the distance between vane endwall trailing edge and rotor leading edge. The flow is illuminated in the axial-radial plane, with the measurement thus resolving those components of the flow. The measurement plane dimensions are 0.8386 in. (21.3 mm) in the axial direction by 0.8464 in. (21.5 mm) in the radial direction.

To image this region, the digital camera views the flow from an oblique angle through a window located over the first rotor row. The optical access window is centered over vertical and the measurement location is 45 degrees off of the vertical. Because of the imaging angle and the window thickness, the camera cannot focus on the entire object plane, with the resulting astigmatic distortion shown in Figure 2.16. For the purpose of correcting the astigmatism, a calibration target was designed and built, as shown in Figures 2.16 and 2.17. The target was a square grid print, mounted on an aluminum block which was clamped to a single rotor blade and fitted to the hub radius such that the target is set in a radial-axial plane. The target was positioned within the plane of the laser sheet by rotating the drum until the rotor-clamped target swept through the laser sheet and the target print itself was aligned with the laser light.

Adjustment of camera focus was accomplished through an adjustment ring on the lens extension body which would lengthen or shorten the separation between the lens and the image focal plane. Focusing the camera on the grid resulted in either the axial grid lines or the radial grid lines coming into focus at two different focal distance settings, but never would they both be in focus simultaneously. As previously stated, this behavior is due to an astigmatism as shown in Figure 2.16, and is a result of the optical access window and the angle with which it is viewed.
To correct the astigmatism, three different window designs were attempted, since a proper design of the window outer surface could completely correct the distortion. The original window design had a planar outer surface, with the inner surface matching the outer radius of the flow annulus. The second iteration on window design was based on the basic lens maker formulas and used a window surface with the same radius as the inner surface. This second design resulted in an astigmatism of opposite sign and half strength of the astigmatism generated by the first window, i.e. the focus points changed order for the axial and radial focus distances. Previous experience had shown that concentric window surfaces minimized distortion, so a third window was attempted which used concentric curves. This final window design achieved a reduction of the astigmatism sufficient that a simple optic element could be used to provide the final correction.

The corrective optic element required to eliminate the remaining astigmatic distortion is a planar acrylic sheet slanted at the opposite angle from that made between the window plane and the image plane. The case window which gives access to the flow path is 1.25 in. (31.75 mm) thick with concentric curvature on the inner and outer window surfaces. The corrective window is 1.4 in. (35.56 mm) thick and skewed at 33 degrees from the camera axis. Because of some difficulty in finding a supplier for thick acrylic and the prohibitive cost of custom glass, the corrective optic was not a single piece but rather, three separate pieces, each approximately 0.46 inch thick, welded together. This three part optic was a compromise as it introduced error in certain measurement situations. The number of material transitions that exist in the light path affects the magnitude of the distortion, and thus the resulting quality of the image. The interior interfaces of the three part optic acted as light tunnels, introducing shadow images in the measurement. Figure 2.19 is a sample of the reflection issues which were introduced. The rotor leading edge image which is located in the lower right hand corner of all the images is duplicated by a shadow image at the mid-height of the image. This reflection was however only present over a small range of rotor positions.
2.4.2 Flow Seeding

Reliable PIV measurements require that the flow be seeded with tracer particles small enough to accurately track the flow and large enough to scatter sufficient light to be detected with the imaging system. These particles must be introduced in a manner that does not significantly disturb the flow field and at the same time provide a sufficient and uniform seeding density in the region of interest. Due to the high flow rate through the turbine, a Rosco 1600 Fog Machine is used to generate the seed particles. This is a thermal aerosol generator that produces a high volume of seed particles by discharging a heated and pressurized glycol based mixture into the atmosphere where it immediately vaporizes and then condenses into a fine mist of monodisperse particles. A uniform seeding density in the test section is achieved by introducing these seed particles upstream of the bellmouth inlet and allowing them to disperse into the ambient air prior to being drawn through the facility.

One stream of high seed density flow from the fogger and another of dry air is pumped into a mixing plenum using a small blower, with the ratio between the two streams setting the final seed density. One side of the mixing plenum is open and covered with a honeycomb flow straightener, allowing seeded flow to discharge at low velocity. The exit of the mixing plenum is located approximately 9ft (3 m) from the turbine inlet, and the position adjusted until the appropriate stream tube feeding the turbine is intercepted.

In the PIV experiments, seeding of the seal flow stream was not attempted. Although the seeding of all streams is common for PIV so that full field measurements can be made, the degree of mixing that is present in the turbulent unsteady flow field of an intra-stage turbine space was sufficient that seeding of only the main stream was needed to provide the particle density required for full field PIV measurements. This was advantageous, as flow visualization studies performed with a seeded seal flow showed
that run time was quite limited due to seed fluid deposition. The glycol seed fluid quickly collects on any surface that has become wetted by the fluid particles. Such accumulation is especially problematic in small spaces where the flow path takes sharp turns, and thus significant accumulation occurred in the seal flow delivery path during these studies.

2.5 Vane Relative Measurement Plane Positioning

The measurement plane location was fixed by the optics set-up. Thus to resolve different measurement locations relative to the airfoil positions, the first vane row was indexed. The vane position is measured by the fixed position of the laser generated light sheet on a gauge block which is attached to the indexed vane support ring in place of a single vane. The gauge block mounting surface is contoured to fit the vane ring and to align the center of the gage block with the mounting hole. The center position of the gauge is then precisely known, relative to the vane circumferential location. The gauge surface features a series of angular reference marks spaced at half degree increments, over which a removable piece of photo paper is mounted. When the gauge was in position the laser would expose the photo paper, and thus allow the exact light sheet location relative to the vane to be resolved. Within the half degree marks finer resolution was achieved with standard rule gauges. The gage blocks and associated laser image marks are shown in Figure 2.20.

2.6 Data Acquisition and Processing

The seal flow rates employed in these experiments were selected by dimensionless flow rate (or seal flow Reynolds number) defined as:

\[ C_w = \frac{\dot{m}}{\mu r_o} \]  

(2.9)
where the actual mass flow rate has been scaled with the fluid viscosity and outer radius of the seal.

Typical values for turbine seals range from $C_w = 5 \times 10^3$ to $2 \times 10^4$ (El-Oun et. al., 1988). For these experiments, three different seal flow rates, $C_w = 1.2 \times 10^4$, $9.2 \times 10^3$, and $4.6 \times 10^3$ were chosen. The turbine operating point during the experiments was near the design point at a flow coefficient of $\phi = 0.515$ and a loading coefficient of $\psi = 1.59$. The rotational speed was 2500 rpm. This results in a mean inlet velocity to the turbine of 94.2 ft/sec (28.7 m/s).

The PIV images corresponding to the first and second laser pulses are divided into rectangular interrogation areas, with cross-correlation software used to determine an average particle displacement for each region. To obtain a high signal-to-noise ratio, the interrogation area must be small enough so that the flow velocity is homogeneous within each region and at the same time large enough to encompass a sufficiently large population of particle pairs. The FFT processing algorithm that computes the cross-correlations generates artificial cyclic background noise at the edges of each interrogation area since this approach assumes that the sampled regions are periodic in space. This can result in the loss of particle pairs due to a low signal-to-noise ratio at the boundaries, with particles near the edges not used in the velocity calculation. However, this information is recovered by over-sampling the images using overlapping interrogation regions. This process does not increase the fundamental spatial resolution, but generates additional vectors as suitable interpolations. For the present investigation 32 x 32 pixel interrogation areas with 50% overlap are used to process the image maps resulting in 3844 raw vectors per image.

Due to the combination of a physically small measurement area and moderately large velocities, the minimum time between laser pulses was required. With the pulse delay set to 1 µs, the finest measurement resolution was limited by the displacement of particles within each interrogation region, resulting in particle displacements of approximately 30% of the length of an interrogation region. A resolution of 32 x 32 pixel
measurement regions was chosen. Smaller interrogation regions resulted in too many particles traveling across regions during the time delay between pulses, resulting in an inability to resolve the mean velocity from the background noise.

To characterize the unsteady flow field generated by the rotor potential field, the instantaneous flow field is imaged at a fixed time delay from a known position on the rotor drum. This is achieved by triggering the PIV acquisition sequence with a signal generated from a photo-optic sensor on the turbine shaft. A LaserStrobe 165 Phase Delay Generator introduces the appropriate phase delay calculated from shaft trigger readings. Vernier marks on the rotor drum are also imaged directly with this system to verify the accuracy of the triggering system. The timing system in conjunction with the Dantec PIV system was used to activate a Kodak E.S 1.0 type 10 camera and a Spectra Physics PIV 200 as the flash for capturing the instantaneous rotor position at the specified delay time. The laser beam passed through two cylindrical lenses at a $90^\circ$ offset thus producing a conical light pattern for illuminating a large area over the top of the first rotor row, including marks on the rotor drum and the fixed centerbody. The instantaneous images for specified delay times were used to determine the rotor blade position nearest the measurement plane, and the variation in rotor position with delay time.

The locations where the data are taken are indicated by the delay time, as shown in Figure 2.21. The accuracy of the rotor relative measurement position is quantified as $\pm 0.25\mu$s ($\pm 0.00375$ degrees) for a rotor speed variation of $\pm 15$rpm about the design speed, and at the longest delay period.
Figure 2.1 Purdue 2-Stage Research Turbine.
Figure 2.2 Turbine blade row configuration.
Figure 2.3  Turbine blade sections.
Figure 2.4  Predicted blade surface pressure profiles.
Figure 2.5  Turbine air flow system schematic.
Figure 2.6 Turbine loading system schematic.
Figure 2.7 Turbine performance instrumentation.
Figure 2.8  Thermistor stagnation temperature probe geometry.
Figure 2.9  Pitot and stagnation pressure probe geometry.
Figure 2.10  Planar seal between the first vane and the first rotor.
Figure 2.11  Seal section hardware drawing.
Figure 2.12  Cross section of seal, seal cavity, and delivery flow path.
Figure 2.13 Experimental facility with laser light sheet and seal supply pipe passing through inlet.
Figure 2.14  Schematic of delivery path for air supplying the seal flow.
Figure 2.15 Vane relative PIV measurement planes.
Figure 2.16  Calibration target as viewed through optical access window.
Figure 2.17 Optical calibration target and mounting block.
Figure 2.18  Astigmatism sectional view
(from Melles Griot Inc., 1999).
Figure 2.19  Rotor blade leading edge reflection due to reflective optics.
Figure 2.20  Vane plane positioning tool.
Figure 2.21  Timing of the rotor leading edge position relative to the measurement plane, with Plane A shown from over-head.
3. DATA STRUCTURE AND ANALYSIS

To investigate the effects of rim seal purge flow on the development of secondary flows and heat transfer along the endwall of a turbine, an experimental research program was performed in the Purdue Low-Speed Research Turbine for a range of seal flow conditions and vane index positions. Velocity fields developed within the intra-stage space were measured, ensemble averaged, assessed for significance, and then resolved into field representations of velocity magnitudes, velocity vectors, and vorticity magnitudes.

The primary parametric variables for this study are seal flow rate, turbine stator vane position, and turbine rotor blade position. The seal flow rate determines the radial momentum which the seal flow carries into the mainstream flow, and thus is indicative of the depth of the penetration of the seal jet and subsequent boundary layer disruption. The stator vane position determines how close the vane wake structures pass relative to the measurement plane above the seal. Strong wake vorticity convected with the mainstream flow can be expected to interact with the emergent seal flow. The rotor blade position relative to the measurement plane governs the strength of the upstream propagating potential field generated by the rotor. This potential field can be envisioned as a sinusoidal spatial variation of pressure anchored with a maximum at the rotor blade leading edge stagnation point and rotating with the blade. The streamwise pressure gradient associated with this potential field can be expected to selectively retard low momentum regions of the flow near the endwall and associated with seal flow structures.
Particle Image Velocimetry measurements were taken at four different vane positions, ten different rotor positions, and two different flow rates. Additional data were acquired at a third, intermediate flow rate for the vane relative position designated as Plane A. The vane relative plane designations are shown in Figure 2.15. The total data matrix of Table 3.1 presents 90 data sets each of a radial-axial measurement plane with 3844 vectors, of which a limited amount are rejected based on the analysis acceptance criteria.

<table>
<thead>
<tr>
<th>Blade Position – Time (µs)</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
</tr>
</thead>
<tbody>
<tr>
<td>000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>075</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>150</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>225</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>300</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>375</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>450</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>525</td>
<td>P</td>
<td></td>
<td></td>
<td></td>
<td>P</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>600</td>
<td>P</td>
<td></td>
<td></td>
<td></td>
<td>P</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>675</td>
<td>P</td>
<td></td>
<td></td>
<td></td>
<td>P</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
3.1 Data Validation

Each data set is identified first by a seal flow rate setting, second at a vane position and third at a rotor position. Before close analyses of the data are performed, the integrity of the data set is investigated by comparing various measures of mean velocities. Consider three different types of velocity measurements, represented in four forms. Turbine flows were measured via PIV interrogation of a specific area, Pitot probe measurement of the inlet midspan velocity, and venturi metering of the mass flow through the machine. A comparison is then made between individual PIV data planes, between groups of PIV data planes, and between the three flow measurements.

First consider a comparison between the three flow measurement types. The calculated average of all axial components of all the valid vector information from PIV results in a mean velocity of 88.6 ft/sec (27 m/s). This matches well with the midspan Pitot probe measured axial inlet velocity of 93.5 ft/sec (28.5 m/s), and the annulus area averaged inlet velocity from mass flow measurement which was 94.2 ft/sec (28.7 m/s). From this conclusion it can only be verified that the PIV measurements are of a very reasonable magnitude. A more direct comparison cannot be easily made between these inlet axial velocities and the PIV measurement cases, since the vector information is not measured at the vane inlet and includes only the slowest 33% of the span.

In addition to acceptable comparison of the axial inlet velocity to the PIV measured data, plane to plane comparisons, rotor position comparisons and seal flow rate comparisons can be made of the PIV data. Figure 3.1 shows data plane average axial velocity values for each rotor position and vane position at the high and low seal flow rates, where each plot point is the average of all valid vector axial components for that data plane. The graph abscissa denotes rotor position according to the data acquisition delay time for the measurement plane. The four vane positions are marked by line styles and labeled as Plane A, B, C and D. Two seal flow rates are represented in Figure 3.1, with a dark line indicating the high seal flow rate and a light gray line marking the low seal flow rate.
Figure 3.2 shows the average radial velocity for each data plane in the same way that Figure 3.1 shows average axial velocity. In analyzing these data, it was noted that for the case of the low flow rate, at measurement Plane D, plot peaks occur at times 375 µs and 525 µs. It would be expected that these values should align with the Plane D high flow rate peaks at time 450 µs and 600 µs. The discrepancy is more noticeable in the axial velocity plot, where Plane D data for the low flow rate lags behind Planes A, B and C. The conclusion that this data set is out of phase with the other data at the indicated blade pass times is also supported from analysis of corresponding vorticity plots and vector plots [Appendix A,B,C and D: compare Figure B.5 and Figure B.6 vs. Figure A.6 and A.7 or Figure D.5 and Figure D.6 vs. Figure C.6 and Figure C.7]. Thus, the data in question is marked with a “P” (partial set) in Table 3.1 and care must be taken in drawing conclusions from this data set (low seal flow rate, Plane D). For the purpose of studying circumferential flow variations, the high flow rate case must be used if the full range over which data was acquired is to be included.

Other than the peak lag in Plane D, vane plane to plane variations in radial averages are insignificant considering the effect of secondary flow on the radial component of velocity within the measurement region. However, the variation in average axial velocity with vane position is significant, with a total range of over 65 ft/sec (20 m/s) from vane Plane A to vane Plane D, and at a fixed rotor position the variation is still as much as 40 ft/sec (12 m/s). These differences are likely a result of turning variation within a vane passage, a hypothesis that is consistent with increased axial velocities at positions further from the vane surface.

The oscillation in average velocity for rotor position is a result of the rotor blade flow blockage. As the blade passes the measurement plane, the flow decelerates axially, and turns around the approaching flow obstruction. This event leads the rotor potential field maximum event which occurs at time t=675µs. The fact that the rotor leading edge pass event lags the time of minimum average axial velocity can be rationalized by understanding that the measurement plane is skewed 50° from the rotor relative inlet flow.
angle thus the streamline direction turns toward the tangential direction prior to rotor pass and then back towards axial after rotor pass as it accelerates around the blade edge. Continuity arguments then explain the variation in axial velocity around the rotor pass event.

Trends in average axial velocity between two different seal flow rate cases are not as easily explained. The observed changes are likely due to the turbine set point, since duplicating the flow coefficient and loading coefficient setting when changing seal flow rate required a slightly different set of blower damper and load control settings. There is thus a small variation in turbine operating point from case to case. Overall the data sets presented are determined to be acceptable for meaningful comparisons of flow development in the seal region.

3.1.1 Ensemble Averaged Data

Each data set presented in the Table 3.1 is a result of an average performed on a collection of approximately 100 instantaneous measurements at each vane position, phase locked to a single rotor position. This ensemble averaging process serves to remove aperiodic unsteadiness that exists within a turbulent flow, while preserving all harmonics of the rotor pass event. Thus, the effects of the first stage rotor blade potential field are extracted from background unsteadiness. The degree of unsteadiness in the flow field is quantified in Figure 3.3, where a weighting scheme is presented to show the standard deviation of the axial velocity components within each interrogation area. The standard deviation of the 100 instantaneous measurements is a measure of the uncertainty of the resulting average value which is due to the flow unsteadiness.

Table 3.2 presents the key to this weighting scheme, which also includes a consideration of the quantity of vectors from each of the 100 instantaneous measurement points that were validated through the PIV software. A vector is considered valid according to the online PIV processing if it was within a magnitude range limit of $-65.6$
to +196.85 ft/sec to (−20 to +60 m/s) and the cross-correlation primary peak to secondary peak ratio is greater than $R_p=1.1$, which is a measure of the signal-to-noise ratio. Previous PIV investigations (Keane and Adrian, 1992) have shown that $R_p=1.2$ is a good limit for the peak ratio, however low seed density, uneven lighting and strong secondary flows require a less restrictive limit for minimizing loss of data.

In the presence of a vane wake, and with a strong rotor potential field acting on the measurement region, Figure 3.3 shows that rather wide velocity distributions exist within the unsteady flow. Although the highest standard deviations are greater than 63% of the overall average axial velocity, discarding data on this basis could remove vortical structures of interest from the processed vector field. Thus ensemble averaged velocity vectors were included in the results analysis as long as greater than 40 valid instantaneous vectors existed at the particular interrogation region (weighting scale level 8).

<table>
<thead>
<tr>
<th>Weighting Scale</th>
<th>Scale Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>Geometry Interferes with Image</td>
</tr>
<tr>
<td>8</td>
<td>Less than 40 valid vectors available</td>
</tr>
<tr>
<td>7</td>
<td>Less than 50 valid vectors available</td>
</tr>
<tr>
<td>6</td>
<td>Standard deviation &gt; 55.7 ft/sec (17 m/s)</td>
</tr>
<tr>
<td>5</td>
<td>Standard deviation &gt; 52.5 ft/sec (16 m/s)</td>
</tr>
<tr>
<td>4</td>
<td>Standard deviation &gt; 49.2 ft/sec (15 m/s)</td>
</tr>
<tr>
<td>3</td>
<td>Standard deviation &gt; 45.9 ft/sec (14 m/s)</td>
</tr>
<tr>
<td>2</td>
<td>Standard deviation &gt; 42.6 ft/sec (13 m/s)</td>
</tr>
<tr>
<td>1</td>
<td>Standard deviation &gt; 39.4 ft/sec (12 m/s)</td>
</tr>
</tbody>
</table>
3.2 Vorticity Calculations

A simple first-order center differencing scheme was used for vorticity calculations. At locations where data is bounded by geometric imaging interference as well as locations where data is bounded by invalidated (i.e. removed) data, vorticity data is not reported.
3.3 Data Presentation Format

Both two dimensional and three dimensional representations of the acquired data are presented. Each vane relative measurement plane was assigned an angle associated with its circumferential location. These values were all referenced to the solid-geometry model that was generated for the test section. This geometry is shown in Figure 3.4

Table 3.3 Vane Relative Rotor Blade Delay Times.

<table>
<thead>
<tr>
<th>Plane A (µs)</th>
<th>Plane B (µs)</th>
<th>Plane C (µs)</th>
<th>Plane D (µs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>000</td>
<td>600</td>
<td>450</td>
<td>300</td>
</tr>
<tr>
<td>075</td>
<td>675</td>
<td>525</td>
<td>375</td>
</tr>
<tr>
<td>150</td>
<td>000</td>
<td>600</td>
<td>450</td>
</tr>
<tr>
<td>225</td>
<td>075</td>
<td>675</td>
<td>525</td>
</tr>
<tr>
<td>300</td>
<td>150</td>
<td>000</td>
<td>600</td>
</tr>
<tr>
<td>375</td>
<td>225</td>
<td>075</td>
<td>675</td>
</tr>
<tr>
<td>450</td>
<td>300</td>
<td>150</td>
<td>000</td>
</tr>
<tr>
<td>525</td>
<td>375</td>
<td>225</td>
<td>075</td>
</tr>
<tr>
<td>600</td>
<td>450</td>
<td>300</td>
<td>150</td>
</tr>
<tr>
<td>675</td>
<td>525</td>
<td>375</td>
<td>225</td>
</tr>
</tbody>
</table>

Simultaneous analysis of all four vane relative measurement planes allows circumferential flow variations to be determined. Accurate spacing measurements between vane-relative positions allow all four planes to be referenced to a point in time specified by rotor position. Plane to plane spacing is 2 $\frac{7}{32} \pm \frac{1}{64}$ degrees. At the design speed of 2500 rpm, it thus takes a rotor blade 148 $\pm 1$ µs to sweep from one measurement plane to the next. Thus, for the purposes of simultaneously analyzing all four measurement planes, a traverse time was taken at $\Delta t = 150$ µs. Table 3.3 shows the timing sequence for comparison of all vane positions. Each row represents a single instant in
physical time or, equivalently, a given rotor blade position. The specified acquisition delay time provided in each cell of that row ensures that the rotor blade is in the proper position relative to that specific measurement plane. For example, if Plane A was imaged at a delay time $t=450 \, \mu s$ then Planes B, C and D would be at the proper distance from the rotor at delay times $t=300$, 150, and 0 $\mu s$, respectively. This allows information to be displayed at all four vane positions at a single instance in time, for example as shown in Figure 3.4.
Figure 3.1 Mean axial velocity components.
Figure 3.2 Mean radial velocity components.
Figure 3.3 Axial velocity weighting map of typical flow field unsteadiness.
Figure 3.4 Perspective for presentation of measurements at all spatial locations.
4. GENERAL DESCRIPTION OF THE MEASUREMENT FLOW FIELD

The primary parametric variables for this study are seal flow rate, turbine stator vane position, and turbine rotor blade position. The distinct changes in the steady and unsteady flow field attributed to the variation of these parameters are the subject of this research. To better portray these changes, it is first necessary to understand the similarity in flow patterns present in all test cases.

First note the orientation of a measurement plane relative to the test hardware, typical of what is presented in Figure 4.1. The single measurement plane is presented with the mainstream flow from left to right. Upstream of the measurement plane is the vane trailing edge, with an approximate distance of 0.264 in. (6.7 mm) between the vane trailing edge and the plane at the hub wall. At the downstream edge of the measurement plane is the rotor leading edge. The measurement plane is positioned such that the seal is located at an axial distance of 0.287 – 0.307 in. (7.3 – 7.8 mm) from the upstream edge of the measurement plane.

The measurement plane is subject to flow structures convected downstream from the vane row, to a pressure field developed by the vane row, to a fluctuating pressure field introduced by the passing rotor blade, and to the emergent seal flow. All this is in addition to the shear forces at the hub wall.

There are then specific flow patterns along the upstream edge, the hub wall and the downstream edge of the measurement plane. The upstream edge is affected by the vane exit flow field, and the downstream edge by the rotor potential field. Additionally,
these upstream and downstream effects can extend well into the center of the measurement region and can be significant in the low momentum flow along the hub wall.

A typical velocity field is presented in Figure 4.1. Along the upstream edge is radial upward flow which is evident to a certain extent in all of the data sets. Along the downstream edge is a low momentum region which may or may not contain a vortex core, depending on the rotor blade position. Along the hub surface is reduced axial velocity, and a slightly large radial component of velocity at the axial location of the seal gap.

The information presented in Figure 4.1 includes both velocity vectors and the resolved vorticity values at those points. The scale on the vorticity plots is in m/s/mm, this scale being used because of the small physical size of the interrogation regions. The spacing from one velocity vector to the next is 13.8 mil (0.35mm). Thus the scale for each structure is relatively small.

The vortex core which exists at the leading edge of the rotor blade is evidence of the start of the rotor horseshoe vortex. The pressure field around the blade provides the potential for vorticity generated at the wall to contribute to a vortical structure at the blade leading edge. Flow acceleration around the blade blockage then sweeps the vortex around the blade. Additionally, the negative vorticity at the upstream edge of the measurement plane matches with the wake vortex structure presented in Figure 4.2 and described by Zaccaria and Lakshminarayana (1995). High values of vorticity are also common at the seal gap downstream edge since this is where the seal flow penetrates into the wall boundary layer and turns to align with the mainstream flow.

The upper half of the measurement plane is, in general, far removed from the rotor blade effects at the downstream edge, the vane exit flow effects at the upstream edge, and the seal flow at the hub wall. It is the lower half of the measurement region where fluid momentum is low and subject to influence by outside forces.
Figure 4.1  Sample of the typical intra-stage space flow field.
Figure 4.2 The vane exit flow field for the Purdue Turbine first stage vane.
5. VANE WAKE AND ROTOR BLADE POTENTIAL FIELD EFFECTS ON SEAL FLOW

The seal flow was characterized with data taken at three seal mass flow rates, referred to here as high \( (C_w = 1.2 \times 10^4) \), midrange \( (C_w = 9.2 \times 10^3) \), and low \( (C_w = 4.6 \times 10^3) \). In all cases the momentum of the emergent flow, the presence of a vane wake, and the passing rotor potential-field impact the flow development along the hub wall downstream of the seal. To characterize these effects, data in measurement planes A and C are considered, where Plane A is completely removed from the vane wake convection path and Plane C intersects the vane wake convection path.

The radial velocity magnitude in the near-seal region of the flow field for the three mass flow cases is presented in Figure 5.1 for rotor position \( t = 375 \mu s \) at vane Plane A. For the high flow case, shown in the top frame of Figure 5.1, a positive purge flow is observed exiting the seal. No gas ingress into the cavity was observed at any rotor position for this case. As the seal flow was reduced to the midrange point, a significant reduction in the mass flow leaving the seal is observed, with the strongest purge appearing at the seal leading edge. When the seal flow is reduced to the minimum value, air ingestion into the seal becomes the dominate phenomena during portions of the blade pass event, as shown in the bottom frame of Figure 5.1.

5.1 Rotor Potential Field Effects in the Absence of a Vane Wake

The effect of the rotor pass event on the axial velocity profiles near the rotor for each of the three seal mass flow rate events are presented in Figure 5.2. At axial
locations upstream of the rotor, the velocity profiles for all three cases are relatively insensitive to rotor position. However, near the blade the deceleration of the flow as it approaches the potential field is evident. At times $t=450$ µs and $t=600$ µs, near the blade, the potential field has strongly affected the axial velocity at all radial positions. In the high flow case, the blockage caused by the penetrating seal flow has reduced the momentum of the flow from near the hub when compared to the other two cases. This low momentum fluid is most strongly affected by the approaching blade, as seen at time $t=450$ µs.

Figures 5.4 and 5.5 show the axial velocity profiles for the low and high seal flow cases at five rotor positions. In each frame, three different axial planes have been chosen, with $x=0.185$ in. (4.7 mm) furthest from the rotor and $x=0.665$ in. (16.9 mm) nearest the rotor. For the low flow case, Figure 5.3, there is acceleration of the flow near the hub when the measurement plane is located near the middle of the rotor passage. As the potential field increases, $t \geq 300$ µs, there is uniform velocity reduction and the ‘bulge’ in the profiles shrinks, until by $t=600$ µs all profiles are similar. From an examination of Figure 5.1 this behavior is not unexpected, as the strength of the adverse pressure gradient has become sufficient to produce a negative purge flow (seal ingress) which is removing low momentum fluid near the endwall and flattening the profile.

As shown in Figure 5.4, the high seal flow rate case exhibits dramatically different behavior than that observed in the previous case. Here, as previously noted, the seal flow emerges with high radial momentum and penetrates deeper into the mainstream flow. This results in hub blockage and low momentum flow in an extended region to approximately $r=5$ mm from the hub. Other than this effect, the profiles are quite nominal until $t=450$ µs, where there is a large axial velocity deficit forming at radial locations higher than $r=5$ mm. This deficit then disappears again by $t=600$ µs. This behavior is best explained by observing the streamlines presented in Figure 5.5 for time $t=450$ µs. At the high flow rate (left frame), we see the start of horseshoe vortex roll-up approaching the seal region. This would introduce both local acceleration on the top
surface due to clockwise rotation and significant blockage near the hub for the approach flow. This results in a large increase in radial velocity near the seal, and continuity arguments can thus defend the deficit in velocity observed in the profile of Figure 5.4.

Supporting these arguments for the effect of rotor blade potential field on the purge flow, velocity magnitude and vorticity contours for two blade times are presented: a time when the blade potential field is near minimum, $t=300\mu s$, and a time when the blade leading edge is aligned with the measurement plane, $t=675\mu s$.

The velocity field for the measurement plane outside of the vane wake (Plane A) and with the minimal effect of the rotor potential field ($t=300\mu s$) is shown in Figures 5.6 and 5.7 for both the high and low seal flow rate respectively. The corresponding results for the normal vorticity are presented in Figures 5.8 and 5.9. The evidence of the seal flow can be realized by a low momentum region near the hub wall just downstream of the seal gap as well as by a region of high normal vorticity at the downstream edge of the seal. In the high seal flow case, the influence of the purge air is much more significant, whereas with the low seal flow rate little disruption downstream of the seal is seen.

The velocity magnitude contours of Figure 5.6 for the high seal flow show that in the absence of a vane wake, the strong seal flow causes an area of low momentum fluid immediately downstream of the seal. Additionally, analysis of the vector field demonstrated that the upstream flow turns radially upward due to the seal flow generated blockage. This effect is amplified by the presence of the rotor, as shown in Figure 5.10, where the rotor blade potential field effects are increased ($t=675\mu s$). As the blade passes, the low momentum area along the hub wall is slowed yet further by the blade potential field (i.e. the blade “bow” wave). Vector information reveals that the flow at the wall reverses, creating a clockwise recirculation region extending from $x=0.590$ to 0.787 in. ($x=15$ to 20 mm) on the axial scale. This is most easily seen in the corresponding vorticity contour given in Figure 5.11. This recirculation can be attributed to a rotor
blade leading edge horseshoe vortex which exists within an area 0.197 in. (5 mm) in diameter at its greatest.

The growth of this structure with rotor pass and the interaction with the roll-up on the lee side of the purge flow is clearly portrayed in the streamline history of Figure 5.12. The instantaneous axial-radial streamlines for the high seal flow case at ten rotor positions are shown, demonstrating the evolution of the secondary flow structures during the rotor pass event. The image plane movement relative to the rotor is indicated by the arrow, beginning with the image plane centered between rotor blades at the top center frame and moving counter-clockwise as the image plane approaches the blade which occurs at the bottom center frame. This structure increases the endwall blockage and subsequent radial deviation of the seal approach flow. As the blade nears, the seal flow interacts with the rotor blade horseshoe vortex which grows until near the blade the vortex swallows the emergent seal flow. This behavior was not observed at lower seal flow rates.

When the two streamline patterns of Figure 5.5 are compared, the enhancing role of the seal flow in the secondary flow structure becomes obvious. At low seal flow rates, only a small vortex structure is seen near the blade. This may well be due to an invigoration of the boundary layer that occurs when the seal flow is introduced with positive flow but low radial momentum, or negative flow, over a portion of the blade pass cycle. When the seal flow rate is increased such that it possesses sufficient radial momentum to disrupt the hub boundary layer, it encourages roll-up and results in a large horseshoe vortex structure dominating the near-hub flow. However, even without the blade potential effect, Figure 5.8 shows that the purge flow still has a distinct impact on the flow profile downstream of the seal.
5.2 Wake Interaction with Seal Flow

A significant alteration of this velocity field occurs when a vane wake crosses the hub seal. For the measurements made in the presence of a vane wake (Plane C) and with high seal flow rate, an area of increased axial velocity over the seal is observed. This high flow condition is likely due to the interaction of the emergent seal flow and the convected wake structures which carry opposite signs of rotation. Higher momentum in the flow near the wall downstream of the seal is observed for both blade positions, as shown by a comparison of the velocity contours of Figures 5.13 and 5.14 to those of the corresponding no-wake case (Figures 5.10 and 5.6). The wake structure is clearly visible from the negative vorticity observed over the seal in Figures 5.15 and 5.16.

Due to this effect, the seal flow penetration into the mainstream is reduced, resulting in increased axial momentum downstream of the seal and less blockage to the approach flow. The seal flow appears to remain close to the wall and the blade potential effect is reduced to the degree that the size of the horseshoe structure which develops with the blade pass is significantly reduced, as is the magnitude of normal vorticity upstream of the rotor leading edge. However, as shown by the seeded seal flow images of Figures 5.17 and 5.18 for this case, the purge fluid actually does not remain as close to the wall as implied by the velocity and vorticity data. Instead, within 3mm axial distance from the seal, rapid mixing entrains seeded seal fluid into the mainstream. The mixing process is enhanced by the rotor pass event as shown in Figure 5.17.

The low seal flow case evaluation of velocity magnitude contours shows that in the absence of a vane wake, there is very little boundary layer disruption that can be attributed to the seal flow. Figures 5.19 and 5.7 present this no-wake case (Plane A), which shows that in the immediate area of the seal there is a slight reduction in axial velocity. This reduction is more noticeable during the blade pass event. In Plane C, with the vane wake, (Figures 5.20 and 5.21) a larger magnitude of axial velocity over the
entire measurement field is seen. This is likely due to flow under turning that occurs in the intra-stage space far from the vane exit, as it is an effect that is evident in both the high flow and low flow cases for all rotor positions.

It would be reasonable to expect the same influence of the wake for the low seal flow rate case as that for the high seal flow rate case, but surprisingly Figures 5.22 and 5.23 shows otherwise. In this low flow case, the field of high negative normal vorticity associated with the wake appears to be considerably diminished in size. This suggests that the magnitude of the seal flow itself contributes to the size of the wake-generated structure and the resulting degree of influence that the seal – wake interaction has on the downstream flow. In this low flow case, the structure appears to do more than keep the seal flow close to the hub wall, it may act to reinvigorate the platform boundary layer. In fact, referring back to the velocity magnitude plots shown in Figures 5.20 and 5.21 for the wake case, an area of increased axial velocity downstream of the seal gap coincides with the region of high negative normal vorticity. Velocity profiles plotted for discrete axial locations around $x=12\text{mm}$ showed an increasingly smaller boundary layer thickness with increasing axial position.
Figure 5.1 Radial velocities around the seal region. high (top), midrange (middle) and low (bottom) seal flow rates.
Figure 5.2  Effect of seal flow rate near the rotor leading edge as the rotor moves toward the measurement plane.
Figure 5.3 Low seal flow velocity profiles at 5 rotor positions and 4 different axial positions.
Figure 5.4 High seal flow velocity profiles at 5 rotor positions and 4 different axial positions.
Figure 5.5  Streamlines at high flow rate (left) and low flow rate (right) at rotor position \( t=450 \).
Figure 5.6 2-D velocity field, minimum effect of blade potential field (t=300µs), wake absent (Plane A). High seal flow case.
Figure 5.7 2-D velocity field, minimum effect of blade potential field (t=300µs), wake absent (Plane A). Low seal flow case.
Figure 5.8 2-D vorticity field, minimum effect of blade potential field (t=300µs), wake absent (Plane A). High seal flow case.
Figure 5.9  2-D vorticity field, minimum effect of blade potential field (t=300µs), wake absent (Plane A). Low seal flow case.
Figure 5.10 2-D velocity field, maximum effect of blade potential field (t=675µs), wake absent (Plane A). High seal flow case.
Figure 5.11 2-D vorticity field, maximum effect of blade potential field (t=675µs), wake absent (Plane A). High seal flow case.
Figure 5.12 Streamlines at each rotor position for the high seal flow case at vane position ‘A’.
Figure 5.13  2-D velocity field, maximum effect of blade potential field ($t=675\mu s$), wake present (Plane C). High seal flow case.
Figure 5.14 2-D velocity field, minimum effect of blade potential field (t=300µs), wake present (Plane C). High seal flow case.
Figure 5.15  2-D vorticity field, maximum effect of blade potential field (t=675µs), wake present (Plane C). High seal flow case.
Figure 5.16  2-D vorticity field, minimum effect of blade potential field (t=300µs), wake present (Plane C). High seal flow case.
Figure 5.17 Smoke seeded flow, maximum effect of blade potential field ($t=675\mu s$), wake present (Plane C). High seal flow case.
Figure 5.18 Smoke seeded flow, minimum effect of blade potential field (t=300μs), wake present (Plane C). High seal flow case.
Figure 5.19  2-D velocity field, maximum effect of blade potential field (t=675µs), wake absent (Plane A). Low seal flow case.
Figure 5.20  2-D velocity field, maximum effect of blade potential field (t=675µs), wake present (Plane C). Low seal flow case.
Figure 5.21 2-D velocity field, minimum effect of blade potential field (t=300µs), wake present (Plane C). Low seal flow case.
Figure 5.22 2-D vorticity field, maximum effect of blade potential field (t=675µs), wake present (Plane C). Low seal flow case.
Figure 5.23  2-D vorticity field, minimum effect of blade potential field (t=300µs), wake present (Plane C). Low seal flow case.
6. 3D PIV MEASUREMENTS

6.1 Overview

PIV is a whole field measurement technique that provides quantitative two or three dimensional velocity data over a planar region. The flow is seeded with tracer particles, with the two-dimensional plane of interest illuminated by a laser sheet generated by a timed double pulse of a high-power laser. In the digital PIV technique, CCD cameras are synchronized to the laser for recording images of the particles within the light sheet for both laser pulses. The camera images are divided into rectangular interrogation regions and correlation algorithms, which operate on the seed particle image, are used to determine an average two-dimensional displacement vector for each region. Vectors are then determined by dividing each displacement vector by the specified time between pulses. An additional vector-summing algorithm is used to combine pairs of two-dimensional vector maps into three-dimensional vector maps.

Camera alignment, measurement of the optical magnification factor, and generation of a suitable light sheet are the three main difficulties in instrumenting for 2d-PIV. Care must be taken in ensuring that the image acquisition camera is perfectly aligned normal to the plane of light produced by the illumination laser. Failure to attain this alignment will result in many data acquisition difficulties, and reduced data confidence. Without alignment, the experimentalist is sure to encounter increased population of ‘bad’ vectors across the measurement plane and an increase in measurement uncertainty. Reduced success results from uneven focus across the
measurement plane, thus affecting the effective particle image size from point to point within the image, and reducing the primary correlation peak to the noise modulus. Before a camera can be properly aligned a viable light sheet must be generated, guidelines for such work has been presented by Adrian et. al. (1991) among others, and includes the necessity to ensure perfect co-planarity of the two pulsed sheets.

Likely, the most difficult component of the PIV technique is ascertaining a measurement uncertainty. The calibration process used for measuring the magnification factor of the imaging system drives uncertainty. A magnification value for the system can be resolved from knowledge of the distance between imaging components. Distances from the object plane (measurement plane) to the image plane (CCD chip plane) and to the camera lens principle points must be known precisely. However, in practice, geometric calculation of the magnification factor is a much less reliable than image calibration.

To calibrate magnification factor for a PIV system, an object of known dimensions must be placed in the plane of the illumination light sheet. The certainty of aligning the calibration object with the light sheet is in itself a task that is critical for minimizing measurement uncertainty. Imaging of the calibration object allows measurement of the magnification value if the CCD camera pixel pitch is known. The calibration image then provides a ratio of image pixel to pixel width to object space width, which is the practical magnification factor used in resolving object space particle displacements.

6.2 Practical Implementation for Extension of 2D-PIV to 3D-PIV

Quite simply 3d-PIV differs from 2d-PIV in the need for a second camera. These two cameras view the measurement region from equal angle offsets with respect to the normal view. The introduction of a second camera is not so simple in practice. Greater complexity of component alignment exists, as well as a greater complexity in calibration.
Camera to camera alignment is the first challenge for 3D-PIV. The goal in camera alignment is to maximize the camera-to-camera image area overlap. By imaging a crosshair calibration object, perfect alignment of the cameras is possible. Test images of the calibration crosshair should show that the crosshairs align. This alignment ensures a maximum overlap and proper angular orientation of the camera about its focal axis. This alignment requires precision duplicates of all camera mounting hardware. Proper camera mounting allows the line of sight of both cameras to fall in the same plane. This plane must also be normal to the light sheet, and should be defined by one of the coordinate directions of the measurement plane.

Precise mounting of the calibration object is also required. The calibration object is used for defining the measurement volume, the camera-to-camera alignment, individual camera focusing, the setting of the Scheimpflug condition, and may also be used to verify the orientation of the light sheet with respect to the flow field depending on the mounting method. The calibration object should be mounted on a traversing mechanism, which allows a full traverse of the light sheet thickness.

Parallel alignment of the calibration object to the light sheet is the first step to this calibration. This is accomplished either by fixing the light sheet position and then matching the calibration object position, or the reverse. Care must be taken in ensuring that the traversing axis lies in the camera plane, which is perfectly normal to the light sheet. These tasks are complicated by the fact that light sheets diverge, so that alignment with one surface of the light sheet will certainly result in a lack of alignment at the opposite surface. It is advisable to first align the calibration object mounting hardware to a solid reference plane, preferably the same reference that was used in fixing the light probe. Then a check of the alignment directly with the light sheet should be made. Also, minimizing the degree of light sheet divergence over the measurement area is desirable.

Camera focus should be optimized for best results in both the near surface and the far surface of the finite thickness light sheet. Thus, the calibration object should be traversed through the light sheet thickness, and imaged at multiple traverse points, so as
to verify a nearly uniform focus through out. This is further motivation for precision in aligning the calibration object parallel to the plane of the light sheet.

For best focusing of each camera, the image plane, object plane and lens plane must intersect at a line. This optimal focus can only be approximated through geometric calculations, since the precise lens plane location may not easily be known. However, a geometric calculation for setting the camera image plane, which is the plane of the CCD optical chip, serves as a useful guide for manually arriving at optimal focus.

The ultimate purpose of the calibration object is to precisely define the measurement volume for 3D-PIV. Upon completing the alignment process for all mounting hardware associated with the cameras, light sheet, and calibration object, the measurement volume calibration images can be acquired. The center, the top surface, and the bottom surface of the light sheet should have previously been located with the calibration block traversing mechanism so that these positions can be found with favorable repeatability. And image pairs of the calibration object should be taken in these three known positions, at least. These images can be used for generating the coefficients in the final post-processing vector summation algorithm.

A proper setup will maximize the number of valid vectors possible for a given magnification factor, by ensuring the maximum number of stereo pairs for the total number of interrogation regions. Also, a minimization of systematic error for vector summation post processing can also be expected, due to a most accurate definition of the measurement volume.

6.3 Experimental Setup

To image the seal flow region, a 0.41 mm thick laser light sheet is introduced through the turbine case using a combination of cylindrical lenses the last of which is imbedded into a probe that is placed in middle of the second vane row. The laser source,
which produces a 5mm diameter beam, is placed 30 inches from the probe exit. This minimal distance reduces the degree of beam expansion, and thus allows the thinnest possible light sheet. Between the laser and light probe is first a lens which reduces the beam spreading so that it can pass into the probe opening, and then a lens that sets the thickness of the light sheet.

This sheet is brought from the vane row upstream through the inter-stage space. Due to the typical large turning in these blade rows, a single vane was removal to introduce the light probe, thus introducing a local reduction in solidity. This vane removal was not ultimately necessary, but was considered easiest for this test of the measurement technology. The measurement plane, typical of that shown in Figure 6.1, sets at 99.65% span (0.3mm from the end wall) and covers the area between rotor-1 trailing edge and vane-2 leading edge. The flow is illuminated in an axial-tangential plane, with the measurement thus resolving those components of the flow. The useful measurement plane area dimensions are 17 mm in the axial direction by 17 mm in the tangential direction.

To image this region, the digital cameras view the flow from an oblique angle through a window located over the first rotor row and the downstream inter-stage space. The optical access window was cut precisely to the inner diameter of the turbine case, and the outer diameter of the window was cut from the same centerline, since this was shown to minimize distortions. Attempts to adjust the outer window radius to smaller values, closer to that predicted by the simple lens maker formula, proved to worsen the distortion cause by viewing at oblique angles. The window thickness is approximately 1.25 in. thick. And the attempts at varying the outer radius showed that a value larger than that produced for a co-axial cut would improve the distortion at stronger oblique viewing angles. Because of the imaging angle, astigmatic corrective optics were expected to be needed but were shown to be unnecessary due to the roughly parallel alignment of the measurement plane with the window. Corrective optics was used in an earlier application of 2D-PIV for which the object plane was tilted 45 degrees from the present case.
The two cameras used Rodenstock 120 series flat field lenses, designed at two-times magnification for minimum distortion. A Nikon PB-6 bellows system was used for mounting the lens to the Kodak cameras. And a specially designed flat profile bellows/camera adapter was fabricated to maximize light transmission for the case of a tilted camera with respect to the lens.

The two cameras were positioned using a three axis positioning system. The precision camera positioners were mounted to a Velmex ball screw positioner for translation along the axis of the turbine. An optical rail fastened to this base and oriented normal to the flow direction held 6 Melles Griot nano-positioners. Precise overlap of the measurement images was guaranteed by actuation of the linear nano-positioners. Angular orientation of the lens rail/camera mount assembly was controlled by rotary nano-positioners, and object to image focal distance was controlled by a second set of linear nano-positioners. The angular adjustment of the CCD image plane was made by Velmex rotary tables A5990TS mounted atop the nano-positioners. Using a three-axis mill with a spindle mounted dial indicator of accuracy 0.00005 in, the positioner components were precisely mounted on the base optical rail.

For the near 2x magnification of the 18mm square object, the cameras were placed on a rail located approximately 24 in. radially outward from the measurement plane. Camera-to-camera included angle was set to 60 degrees. An interactive control system was designed for camera positioning using the nano-positioner Labview drivers. Based on three measurements: the desired flow passage span measurement location, the desired included camera angle, and the turbine axis to optical rail axis distance, the camera positioners could be activated for shifting the cameras into precisely the correct position. Focus adjustment, aperture adjustment, and Scheimpflug angle adjustments were made manually.
6.4 Calibration

Calibration required that the turbine drum shaft be loosened from the restraining bolt and slid forward. The rotor row just upstream of the measurement plane is so close to the downstream stator row, that the calibration block traversing mechanism could not be set into place, while the rotors where in the operational axial position. Temporary displacement of the rotor forward by 10mm was required.

The calibration target measured 21.5mm x 21.5mm and was constructed from a commercial engraving blank with white surface and black substrate. Endmills of Diameter 0.015, 0.020, and 0.025 inch diameter were used to remove the white layer by a depth of 0.2 mm. The dot spacing was set at 1.625mm. This calibration object was mounted atop a Melles-Griot vertical translation stage with a range of 4mm and drum dial indications of 0.01 mm. Interchangeable calibration objects with vertical, horizontal and cross striping were used for checking camera focus and calibrating the magnification factor. All calibration blocks were fabricated on a precision numerical controlled mill.

6.5 Results

After calibration images were successfully acquired, PIV could be attempted. This was first accomplished without spinning up the turbine. A circulation fan placed over the rig exhaust duct was used to produce a small mass flow, sufficient to pull through seed particles. This simple test was necessary to detect problems with light reflections and camera focus. Often with 2D-PIV the experimentalist would like to adjust the focus so as to improve the pixel images, thus placing the histogram of pixel widths within the desired range. However, this process is not directly applicable to 3D-PIV, as any adjustment to focus, would require that the calibration process be repeated using the new camera focal settings. During this investigation, it was found that the calibration procedure itself was very successful in determining the best possible seed image focus.
Preliminary tests of the system with the turbine rotor locked, allowed a smooth test of the system at part speed and full speed. At the nominal design operating conditions, data was acquired in the amount of 115 instantaneous acquisitions. Note that the 115 pairs of data resulted from an actual acquisition of over 250 pairs, which was then sorted according to good/bad particle seeding. Each instantaneous data set was checked for vector magnitudes within 150% of the mean, with the remaining information discarded. Additionally, each vector correlation peak was checked to be at least 1.2 times higher than any secondary peak. The final data set pair was then matched and summed to arrive at 115 instantaneous 3D vector fields. The data was acquired for 32x32pixel areas with 50% overlap and without interrogation area displacement.

The results of this analysis are shown in Figure 6.2 and 6.3, and give an relatively high range of values for the out of plane velocity component. As work is still progressing on the error analysis based on the calibration coefficients produced by the Dantec system, this data is still considered preliminary. The direction of the out of plane component is given by the color legend. Red is radially outward and out of the page, and blue is radially inward and into the page. The directional trends show an upward flow beginning on the pressure side of the blade and a downward flow on the suction side of the blade. This is consistent with the known rotational direction of a tip vortex. However the point of downward turning is much farther away from the suction side than expected. The single ensemble average out of plane vector set appears to be reasonable from a qualitative approach. Note that the small segment of blue vectors in the top center of the area is a result of a strong reflection on the rotor blade. Also, notice the flow angles in the axial-tangent plane. These vectors are within the expected velocity range, and have angles that show a turning of the flow to align with the camber angle of the vane-2 leading edge.

The final three-dimension perspective of the flow in Figure 6.4, shows the rotor trailing edge looking nearly straight on along the metal angle from downstream toward upstream. The displayed vectors are plotted three-dimensionally, and with every second vector dropped from the view for improved clarity.
After these data were acquired, the turbine axis relative location of the set-up was adjusted so that data acquisition could be attempted in the tip gap. This last attempt was the ultimate goal, but it failed due to the difficulty in placing the light sheet far enough from the rotor tip surface so as to prevent reflection noise from the rough finish on the rotor tip surface.
Figure 6.1. Measurement Plane
Figure 6.2 Raw 3D-PIV Image
Figure 6.3  3-D vector map of rotor tip trailing edge flow.
Figure 6.4  3-D vector field perspective of rotor tip trailing edge flow.
7. SUMMARY AND CONCLUSIONS

An experimental characterization of the seal flow region between the vane and rotor of the first stage of the Purdue Research Turbine has been accomplished. The data acquired characterize the axial-radial velocity field both inside and outside the vane wake convection path. Data were acquired at three seal mass flow rates and ten rotor positions relative to the measurement plane.

The seal flow interaction with vane wakes and the passing rotor blades develop a circumferential periodicity in the hub flow patterns. The vane wake serves to deflect the emergent seal purge away from the mainstream flow, whereas the rotor pass event serves to increase the purge flows radial migration. The strong negative normal vorticity of the wake generated structures reduces the radial growth of the seal flow structures in the near gap region and promotes boundary layer reattachment while inhibiting the strength of the purge at the seal gap.

Flow development downstream from the seal is also strongly influenced by the seal mass flow rate. Lower flow rates are consistent with increased instances of seal ingestion over portions of the blade pass cycle. This ingestion scenario actually removes low energy boundary layer fluid and thus has a stabilizing effect. Additionally, in those portions of the blade pass cycle with seal flow egress, the purge is fed into the hub boundary flow but appears not to disturb it to the extent seen in higher flow rate cases, and thus has very little influence on the secondary flow structure. As the seal flow rate is
increased to prevent ingress, the disturbance of the boundary layer by the emergent fluid results in low momentum endwall fluid, with a large horseshoe vortex observed to dominate the flow as the rotor blade moves through the measurement plane. This domination is significantly altered by the vane wake convection through the seal flow path. These data thus demonstrate a large participatory role of the seal flow in the secondary flow structure. With this information, it can be envisioned that in many high temperature turbines strong secondary flows may be enhanced by purge flows which are maintained at higher than necessary flow rates in order to provide a safety margin for ingress prevention.

Many of the physical insights provided by these data are somewhat limited to the specific geometry of the research turbine and necessary limitations of the testing environment. However, these experiments are of significant value in that they capture unique data that describe the unsteady interaction of an emergent seal flow. These data set thus allow computational studies to be calibrated and extended to other geometries and more realistic testing environments where data of this nature would be impossible to obtain. With both the evidence provided by these data and the value in its use as a test case for comparison to computational modeling, it is envisioned that future significant improvements in gas turbine performance and durability can be achieved through sophisticated management of the seal flow-mainstream flow interaction.

Additionally, a significant contribution has been made to the state-of-the-art understanding within the field of turbine secondary flows. This is the first study to contribute experimental measurements of seal flow interaction with major secondary flow structures. These data are also among a small body of literature available concerning a seal purge exiting into a cross-stream which is passing through a full turbine stage. While studies exist of film cooling interaction with secondary flows, an important insight has been offered on the influence of rim seal purge flows on the endwall flow development. Clearly, rim seal purge flow serves to augment the endwall flow in a way that cannot be ignored during engine design. It is expected that a body of additional work will be developed in light of this revelation.
LIST OF REFERENCES


Appendix A  Four Plane, Axial Velocity Contours, High Seal Flow Rate

Figure A.1  Vane plane A is at time t=600.
Figure A.2  Vane plane A is at time t=675.
Figure A.3  Vane plane A is at time t=000
Figure A.4 Vane plane A is at time $t=075$. 
Figure A.5  Vane plane A is at time t=150.
Figure A.6 Vane plane A is at time $t=225$. 
Figure A.7  Vane plane A is at time $t=300$. 
Figure A.8 Vane plane A is at time $t=375$. 
Figure A.9 Vane plane A is at time t=450.
Figure A.10  Vane plane A is at time t=525.
Appendix B  Four Plane, Low Seal Flow Rate, Axial Velocity Contours

Figure B.1  Vane plane A is at time $t=600$. 
Figure B.2 Vane plane A is at time $t=675$. 
Figure B.3  Vane plane A is at time t=000.
Figure B.4  Vane plane A is at time t=075.
Figure B.5  Vane plane A is at time t=150.
Figure B.6  Vane plane A is at time $t=225$. 
Figure B.7  Vane plane A is at time t=300.
Figure B.8 Vane plane A is at time $t=375$. 
Figure B.9 Vane plane A is at time t=450.
Figure B.10  Vane plane A is at time $t=525$. 
Appendix C  Four Plane, High Seal Flow Rate Vorticity Contours

Figure C.1  Vane plane A is at time \( t=600 \).
Figure C.2 Vane plane A is at time $t=675$. 
Figure C.3  Vane plane A is at time t=000.
Figure C.4  Vane plane A is at time t=075.
Figure C.5 Vane plane A is at time $t=150$. 
Figure C.6 Vane plane A is at time $t=225$. 
Figure C.7 Vane plane A is at time $t=300$. 
Figure C.8  Vane plane A is at time $t=375$. 
Figure C.9 Vane plane A is at time $t=450$. 
Figure C.10  Vane plane A is at time $t=525$. 
Appendix D  Four Plane, Low Seal Flow Rate Vorticity Contour

Figure D.1  Vane plane A is at time t=600.
Figure D.2 Vane plane A is at time t=675.
Figure D.3  Vane plane A is at time t=000.
Figure D.4 Vane plane A is at time t=075.
Figure D.5 Vane plane A is at time t=150.
Figure D.6  Vane plane A is at time t=225.
Figure D.7 Vane plane A is at time $t=300$. 
Figure D.8  Vane plane A is at time $t=375$. 
Figure D.9  Vane plane A is at time t=450.
Figure D.10 Vane plane A is at time $t=525$. 
Appendix E  Regions Of Negative Radial Velocity For The High Seal Flow Rate

Figure E.1  Vane plane A is at time $t=600$. 
Figure E.2  Vane plane A is at time t=675.
Figure E.3 Vane plane A is at time t=000.
Figure E.4 Vane plane A is at time $t=075$. 
Figure E.5  Vane plane A is at time t=150.
Figure E.6  Vane plane A is at time t=225.
Figure E.7 Vane plane A is at time $t=300$. 
Figure E.8  Vane plane A is at time t=375.
Figure E.9 Vane plane A is at time t=450.
Figure E.10 Vane plane A is at time $t=525$. 
Appendix F Regions Of Negative Radial Velocity For The Low Seal Flow Case

Figure F.1 Vane plane A is at time t=600.
Figure F.2 Vane plane A is at time $t=675$. 
Figure F.3 Vane plane A is at time t=000.
Figure F.4 Vane plane A is at time t=075.
Figure F.5 Vane plane A is at time $t=150$. 
Figure F.6  Vane plane A is at time t=225.
Figure F.7 Vane plane A is at time $t=300$. 
Figure F.8 Vane plane A is at time $t=375$. 
Figure F.9 Vane plane A is at time $t=450$. 
Figure F.10 Vane plane A is at time $t=525$. 
Appendix G  2-Dimensional Representation Of Vector Field For Vane Position A At A High Seal Flow Rate

Figure G.1  Vane plane A is at time t=600.
Figure G.2  Vane plane A is at time $t=675$. 
Figure G.3 Vane plane A is at time t=000.
Figure G.4  Vane plane A is at time $t=0.75$. 
Figure G.5  Vane plane A is at time $t=150$. 
Figure G.6 Vane plane A is at time $t=225$. 
Figure G.7 Vane plane A is at time $t=300$. 
Figure G.8  Vane plane A is at time t=375.
Figure G.9 Vane plane A is at time t=450.
Figure G.10  Vane plane A is at time $t=525$. 
Figure H.1 Vane plane A is at time $t=600$. 

Appendix H  2-Dimensional Representation Of Vector Field For Vane Position A At A Low Seal Flow Rate
Figure H.2  Vane plane A is at time $t=675$. 
Figure H.3  Vane plane A is at time t=000.
Figure H.4 Vane plane A is at time t=075.
Figure H.5  Vane plane A is at time $t=150$. 
Figure H.6 Vane plane A is at time $t=225$. 
Figure H.7  Vane plane A is at time $t=300$. 
Figure H.8 Vane plane A is at time $t=375$. 
Figure H.9  Vane plane A is at time $t=450$. 
Figure H.10 Vane plane A is at time t=525.
TECHNICAL DATA REPORT

TITLE
VBI3D Unsteady Flow Analysis of the Purdue Low Speed Turbine Research Rig 1st Stage

Introduction
This report summarizes Computational Fluid Dynamics (CFD) predictions for the Purdue University Low Speed turbine research rig. The U.S. Department of Energy (DOE) provided funding for this work, with the intent being to gain a more complete understanding of endwall flow and heat transfer in turbines. This motivation is driven, in part, by the recent development of low emission combustors needed to meet strict environmental standards. Low emission combustors produce turbine gas path radial temperature profiles which are relatively flat. This design feature results in problems of endwall region over-temperature. The need for film cooling and the desire to minimize the amount of film cooling flow puts an additional burden on the heat transfer designs in the blade tip and endwall regions. The primary objective of the experimental and computational investigations is to provide a detailed and quantitative definition of the complex 3-D flow and heat transfer phenomena.

The computational fluid dynamics (CFD) investigation of the Purdue low speed turbine research rig, which began in 2000, is part of a combined experimental and computational study to quantify airfoil tip, platform, and endwall region flow and heat transfer. Experimental data were taken at the Purdue University Zucrow Labs turbine facility. Steady and unsteady CFD predictions were completed at Rolls-Royce in Indianapolis using the Vane-Blade Interaction (VBI) CFD code.

The Purdue low speed turbine research rig consists of two stages with constant radius endwalls (hub radius 7.09 inches, shroud radius 9.59 inches). The turbine rotors had a rotational speed of 2500 rpm. The blade counts for the two stages were 1st vane row -24 blades, 1st rotor -32 blades, 2nd vane - 28 blades, 2nd rotor 32 blades. At the design operating point, the rig mass flow was designed to be 6.0 lb/s with a total to total expansion ratio of 1.147 and efficiency of 90.3%. The 1st stage expansion ratio was designed to be 1.073 with a stage efficiency of 90.5%.

The Vane-Blade Interaction (VBI) code is a 3-D Navier-Stokes based flow analysis code that can be used to predict both the steady (time-averaged) and the unsteady (time-dependent) characteristics of flow through a turbine stage. The code has been validated using test data from the VBI Rig at Ohio State University and the Joint Strike Fighter (JSF) counter-rotating HP/LP turbine rig. The VBI code uses a single blade passage representation for each blade row in the stage and unsteady phase-lagged boundary conditions to predict the unsteady flow through each of the blade rows. Phase lagging refers to the storage and use of boundary values that are periodic in time. For each of the two blade rows, sufficient data must be stored along the interface and at the periodic boundaries to provide information for a temporal period, which is equal to the blade passing frequency of the adjacent blade row. With this approach, a single blade passage for each row can be used to predict the unsteady flow through a stage. Exact blade counts are used, and there is no airfoil scaling or reduced blade counts.

The 1st stage of the Purdue Turbine Rig was chosen for analysis because much of the initial test data was taken at the interstage location between the 1st vane and the 1st rotor. In addition, the leakage or hub cavity flow was introduced into the rig flow at the hub in between the first two blade rows. The objectives for the computational study were:
1. complete an unsteady calculation of the 1st stage at the design operating condition
2. complete code upgrades to have the capability to include leakage mass flow addition
3. complete an unsteady calculation of the 1st stage with hub mass flow addition at conditions corresponding to one of the test cases run at Purdue.

All of these objectives were met during this study. This report focuses on the two unsteady calculations. Comparisons between the two calculations will be made to point out relevant flow physics, and to point out the effect that radial hub mass flow addition can have on the flow in the downstream blade row, which in this case was the 1st rotor. Ultimately, comparisons with test data will be made to validate the present computational results.
Grid system for VBI analysis

The composite mesh for the VBI calculations used an O-grid embedded in a background H-grid to model each blade passage. For rotors, a degenerate O-grid was used to model the flow in the tip clearance region. A 2-D section of the grid showing the midspan geometry is shown in Figure 1. Every other grid point in each of the grids is shown. The H-grid dimensions for the vane were 112x54x50. The H-grid dimensions for the rotor were 92x46x50. The dimensions for the O-grid in both blade rows were 146x24x50. The rotor tip clearance grid was 146x12x6. The total number of points used for the vane was 477,600. The total number of points used for the rotor was 397,512.

Figure 1. Illustration of the Composite mesh for the 1st stage of the Purdue low speed turbine rig showing the embedded O-grid in a background H-grid.

Boundary information for the outer boundary of the O-grids and the hole boundary of the H-grids was passed at each time step using trilinear interpolation. This grid scheme was chosen initially to provide an orthogonal mesh with adequate resolution for wake tracking. It was particularly attractive because there is no shear in the H-grid regardless of the blade stagger angle. This scheme is unattractive because the combined flow solver/grid system is not particularly conservative. An additional problem arises with this approach when applied to compressor blading. The problem with compressors is that the thin, high stagger blades require a very dense background grid, and there is a high probability that the code which determines the interpolation stencils for boundary points (PEGSUS) will not be able to provide a unique stencil for all points. Even for turbines, the preprocessing step to find all of the interpolations can be time-consuming. The most vulnerable aspect of the subsequent calculation is the lack of strict conservation.
Solution Procedure for an Unsteady Calculation

Once a composite grid for the two blade rows was generated, and the interpolations between the H-grids and the O-grids was established, a steady isolated flow solution for each blade row was calculated. The flow conditions at the inlet and the exit of each blade row were taken from an OS609 turbine design system throughflow run. At the inlet to each blade row, total pressure, total temperature, and flow angles were specified. At the exit, static pressure at the hub and the radial equilibrium equation were used to specify a static pressure radial profile. The steady isolated blade row calculations were run to convergence, and the steady predictions were used as an initial guess for the unsteady calculation.

For the unsteady calculation, the interface between the vane and the rotor allowed direct transfer of unsteady boundary values between the blade rows. For the pitchwise periodic boundaries, boundary data for the number of iterations equal to one rotor passing period was stored and used for the vane. Boundary data for the number of iterations equal to one vane passing period was stored and used for the rotor. The time periodic boundary values for each enable the blade passage solution for each blade row to become periodic in time. Figure 2 shows a plot of the static pressure for reference points on the surface of the vane and the blade at midspan. The vane reference point was near the trailing edge. The blade reference point was near the leading edge.

Figure 2. Unsteady pressure history for reference points on the 1st vane and the 1st blade of the Purdue low speed turbine rig.

Figure 2 shows that the unsteady period was 120 time steps for the vane and 160 time steps for the rotor. A close inspection of the values of unsteady pressure in Figure 2 show that the level of unsteadiness on the vane was only 0.1% and the level of...
unsteadiness on the rotor was 0.22%. This is exceptionally small relative to other cases run with the VBI code. For highly loaded transonic cases involving shocks, a difference between the maximum pressure and minimum pressure on the rotor of 30% is not uncommon. The very low Mach numbers in the Purdue rig presented difficulties for the VBI code because the flow solver in the code is optimized for compressible flow. The general rule is that the code is best suited for Mach numbers above 0.2. The inlet Mach number for the Purdue University low speed turbine rig was about 0.08, well below the normal limit. Because of this, several of the standard input parameters used for transonic cases could not be used. For example, if the standard number of 4 sub-iterations per real time step was used, instabilities near the inlet developed and were visible as active spots of alternating high and low flow variable values. Increasing the number of sub-iterations per time step to 10 and reducing the CFL number per time step stabilized the calculation. These changes also increased the required run time by three times compared to transonic cases, making each case require at least three weeks run time on a Silicon Graphics Power Challenge machine.

Discussion of the VBI Calculation for the Baseline Case

The initial calculation, which will be referred to as the baseline case, predicted the unsteady flow for the 1st stage of the Purdue rig at design conditions without hub leakage flow. Because the levels of unsteadiness were low, snapshots of the 3-D flow can be used to highlight relevant flow phenomena in the vane and the rotor. A comparison of the flow with and without leakage will follow. The grid system for the VBI code was developed to provide a high-resolution orthogonal grid near the blade surface along with an orthogonal H-grid to capture wakes and shocks. There were no shocks in the Purdue rig, but Figure 3 shows a snapshot in time of Mach number and entropy contours at midspan for the baseline case. Both images in the figure show that the analysis produced a seamless solution at the interface and pitchwise periodic boundaries. The plot of entropy contours show the combined grid and flow solver did a good job of capturing and convecting the wake from the 1st blade row through the 2nd blade row. The calculated expansion ratio for the 1st stage was 1.075 with a stage efficiency of 85%. The efficiency was lower than the design value of 90.5% because of some flow separation on the rotor suction surface. Previous steady 3-D calculations run with the ADPAC code did not indicate 1st rotor suction surface separation. For the steady ADPAC calculation, both stages of the rig were run in a multistage environment with mixing plane boundary procedures used at the interface between blade rows. Static pressure from the Rolls-Royce design code (OSO69) was specified at the exit of the grid system downstream of the 2nd rotor. The steady ADPAC run predicted a mass flow of 5.97 lb./s. For the unsteady VBI run, static pressure was specified downstream of the 1st rotor. The mass flow for the VBI run was 6.05 lb./s at the inlet of the vane and 6.25 lb./s at the exit of the 1st rotor. Mass flow for the VBI run is discussed later in this report. To explain the reason for the VBI suction surface separation, Figure 4 compares the Mach number and flow angle distribution at the exit of the 1st vane.
Figure 3. Purdue low speed rig midspan entropy contours (left) and Mach number contours (right with legend) for the baseline case demonstrating the capability of the analysis to clearly capture blade-to-blade flow features.

VFI Flow Analysis of the Purdue Low Speed Turbine Rig

Figure 4. Comparisons of radial variation of predicted and design value Mach number (left figure) and flow angle (right figure) for the Purdue low speed turbine rig (1st vane exit).
VB3D Unsteady Flow Analysis of the Purdue Low Speed Turbine Research Rig 1st Stage

Figure 4 shows that the predicted Mach numbers at the exit of the 1st vane from the VBI run were higher than those from the ADPAC steady run and the design system. The data also show that predicted VBI flow angles from 10% to 60% span were slightly higher. Together, the plots indicate that the flow coming into the 1st rotor for the VBI run had a higher Mach number with higher incidence angle. This difference was enough to cause flow separation on the suction surface of the blade. Additional runs with the design system tools confirmed that this behavior was expected. If the flow angles into the 1st rotor are specified, the design system blade row flow solver does not indicate separation, and the blades were designed using that option. If the V6 velocity component into the 1st rotor is specified, the design system indicated flow separation over the lower and midspan sections of the blade. The 3-D predictions and the design system runs indicated that the flow through the 1st rotor probably would not separate at design conditions, but that there were clear indications that off design conditions could cause flow separation. This flow separation results in a loss in rotor efficiency and stage efficiency.

Figure 5 shows the envelope of unsteady pressures over the surface of the 1st vane at 10%, 50%, and 90% span. At each spanwise location, these plots have the maximum, minimum, and time mean values for instantaneous pressure vs. surface distance at every grid point on the blade surface. For the vane, the maximum, minimum, and time mean were practically equal, except on the suction surface near the trailing edge. The relative Mach number entering the 1st rotor was about 0.14, and the rotor potential field was very weak. Only the suction surface of the vane was affected by the passing of the rotor and the effect was minimal. Figure 6 shows the envelope of unsteady pressure over the surface of the 1st blade at 10%, 50%, and 90% span. There was considerably more unsteadiness on the blade than on the vane, but the levels were still quite low. The principal source of the unsteadiness in the blade was the wake from the vane. For this low Mach number case the flow is nearly incompressible, and the velocity deficit through the wake was the only mechanism for changes in pressure.

Figure 5. Illustrations of the envelope of the maximum, minimum, and time mean static pressure on the surface of the 1st vane at 10%, 50%, and 90% span.
Figure 6. Illustrations of the unsteady envelope of the maximum, minimum, and time mean static pressure on the surface of the 1st blade at 10%, 50%, and 90% span.

Figure 7 is an image of the near surface particle traces for the vane suction surface and the blade pressure surface showing the extent of secondary flow in the blade passage, particularly near the endwalls. Near the vane suction surface, flow from the endwall region was driven radially toward midspan. On the pressure surface of the blade, flow was pushed radially toward the endwalls. The flow at the tip was forced into and through the tip clearance region. Figure 8 shows particle traces near the pressure surface of the vane and the suction surface of the blade. The fluid on the pressure surface of the vane was pressed towards the endwalls, but there was not a great deal of flow migration. On the other hand, the suction surface of the blade indicates that as the flow moves downstream, there was flow migration from the hub towards midspan with probable flow separation near the hub. The secondary flow near the tip, which also migrates toward midspan, was heavily influenced by the tip clearance flow.
Figure 7. Particle traces near the suction surface of the vane and the pressure surface of the blade.

Figure 8. Particle traces near the pressure surface of the vane and the suction surface of the blade.
In Figure 9, particle traces are used to display some of the flow through the tip clearance region. In Figure 9a, the view is from the passage slightly upstream of the rotor looking circumferentially at the flow coming through the clearance region to the suction side of the blade. In Figure 9b, the view is from downstream of the rotor looking upstream at the flow coming through the clearance region to the suction side of the blade. Some of the tip clearance flow closest to the tip of the blade became wrapped up tightly in a vortex close to the tip of the blade and the suction surface. Flow that was closer to the shroud was swept radially downward and away from the suction surface. The downstream location of the flow depends on where the flow came through the clearance region.

Figure 9. Particle traces showing the flow through the 1st stage blade tip clearance region.
Effect of Hub Leakage Flow

The second case examined included mass flow addition on the hub at an axial location just upstream of the midpoint between the vane trailing edge and the rotor leading edge. The leakage flow was introduced into the primary stage flow using a boundary procedure that treated specified points on the hub as if they were inlet boundary points. Total pressure, total temperature, and flow angles that allowed the leakage flow to be vectored were specified. Variations in the leakage flow total pressure were used to control the leakage mass flow rate. The values for total pressure and total temperature for the cases presented here were provided by Kirk Gallier from Purdue. Experimental data were taken for three cases run at the Purdue aerodynamics lab. The "high flow" leakage case was chosen for the VBI unsteady run. The leakage flow model was used to model an open cavity with a gap of 0.025 inches between the 1st vane and the 1st blade. The flow was directed normal to the hub making it purely radial with no axial or tangential component. For this case, the total pressure of the hub leakage flow was 1.04 times the total pressure at the inlet to the 1st vane. The total temperature of the hub leakage flow was 1.046 times the total temperature at the inlet to the 1st vane.

Figure 10 shows a circumferential view of velocity vectors and total temperature contours in the vicinity of the cavity. The axial location of the cavity is marked by the dotted line. The solid line shows the location of the interstage axial station, chosen as the location to compare mass-averaged flow quantities with the baseline case. These are presented in the next section. The cavity flow introduced into the passage flow created a complex three-dimensional separation region downstream of the cavity. The passage flow at the exit of the vane had a tangential velocity of 300 ft/s and an axial velocity of 100 ft/s. The high total pressure specified in the thin cavity produced a radial component of 400 ft/s at the hub. Immediate flow separation caused the high temperature cavity flow to become entrained in a complicated vortical flow near the hub. The vortical flow entered the rotor and was convected downstream more or less intact, making it possible to track the high temperature flow from the cavity through the rotor.

Figure 10. Circumferential view of velocity vectors (a) and total temperature contours (b) in the vicinity of the cavity. The dotted line shows the axial location of the cavity.
Figure 11. Total temperature contours at axial stations from just downstream of the hub leakage flow location to mid-chord in the blade passage, illustrating the migration of the leakage flow through the 1st blade.

Figure 11 is a series of plots, viewed from upstream, showing total temperature contours at axial stations through the rotor. Figure 12 includes two plots viewed from downstream which also show total temperature contours at axial stations near the trailing edge of the blade and at the exit of the blade H-grid. The six contour plots in Figures 11 and 12 should be viewed as part of a single series. Together, figures 11 and 12 show the downstream convection and radial migration of the cavity flow. The flow features in figures 7 and 8 are helpful in explaining the migration of the core of leakage flow from the hub at the interstage axial location to 40% span by the exit of the rotor. This migration was determined for the most part by the physics of the flow through the rotor. As the passage flow entered the rotor passage, the flow near the pressure surface of the rotor was forced radially toward the endwalls. As a consequence of the secondary flow, fluid near the hub was driven across the passage to the suction side of the adjacent blade. Near the suction surface, the flow was forced radially toward midspan. The leakage flow was split at the leading
edge of the blade and organized into a clearly defined patch of fluid that stayed close to the rotor suction surface. As the patch convected downstream, it migrated radially toward midspan.

Figure 12. Total temperature contours at axial stations near the 1st rotor trailing edge and at the exit of the computational domain showing that the hub cavity flow had migrated to nearly 40% span.

Downstream of the blade trailing edge, the migration continued, and the core of the patch reached 40% span by the exit of the blade grid.

Although the secondary flow patterns were determined in large part by the design of the rotor and physics of the rotor flow, the hub cavity flow provided the mechanism to make these features more pronounced, particularly near the leading edge of the blade. Figure 13 compares the secondary flow patterns for the baseline case without leakage flow and the case with hub leakage flow. The secondary patterns were created using restricted particle traces, which neglected the axial velocity component. Figure 13a shows that just downstream of the blade leading edge on the pressure side of the blade; there was a tight leading edge vortical flow region rotating clockwise when viewed from upstream. In the leakage flow case, 13b, the region covered 25% of the span, and a counter-rotating region is evident off the suction surface. The leakage flow, in essence, amplified the secondary flow that existed in the baseline case.
Figure 13. Restricted particle traces compare the secondary flow patterns just downstream of the leading edge of the 1st blade for the case without hub leakage flow (a) and the case with hub leakage flow (b).

Comparison of radial distributions of performance parameters

Figure 14 compares radial profiles of mass-averaged total pressure and total temperature for the baseline case and the case with hub leakage flow. The interstage location, shown in Figure 10 was 0.06 inches downstream of the location of the hub leakage flow. The stage exit was 0.2 inches downstream of the rotor trailing edge. The baseline case is called 'no leak' in the legends.
Figure 14. Comparison of predicted total pressure and total temperature profiles at the interface between the 1st vane and 1st blade for the baseline 1st stage case and the case with hub leakage flow.

In the plot of total pressure vs. span, the baseline case shows a small spike in total pressure very near the hub. This feature occurs because the axial station chosen for these comparisons was in the blade grid and the blade grid was rotating. With a no-slip condition on the hub, the total pressure near the hub was greater than the inlet total pressure. The case with leakage flow shows a significant drop in total pressure at the interstage location from 10% span down to the hub. The interstage location for these plots was downstream of the cavity where the radial leakage flow from the cavity caused the flow to separate. The region near the hub at this axial location was in a recirculating flow region, which produced total pressure loss. At the stage exit, the differences between the two are not as evident, but the baseline case provided a more uniform total pressure distribution. The comparison of total temperature vs. span shows that at the interstage location high temperature flow dominated the hub endwall region. The cavity flow caused separation, and the recirculating flow downstream of the cavity consisted of high temperature fluid. This was also shown in Figure 10(b). The nondimensional total temperature returned to 1.0 (the reference condition) at 17% span. At the stage exit, the influence of the high temperature leakage flow had reached 50% span and the core of the leakage flow had migrated from 4% span to 40% span.

Figure 15 compares mass-averaged values of axial velocity, tangential velocity, and radial velocity for the baseline case and the leakage flow case at the interstage station and the stage exit. In the baseline calculation, the axial velocity was fairly uniform at near 100 ft/s. For the leakage flow case, the interstage station was in a separated flow region, and Figure 15a shows negative axial velocity below 2% span and a peak value of 165 ft/s at 8% span. In addition, the axial velocity at all spanwise locations was affected. This implies that the leakage flow was strong enough to effectively redistribute the mass flow through the stage. At the stage exit, the baseline axial velocity values were again fairly constant and near 100 ft/s, but there was a local minimum at 18% span. For the leakage flow case, the profile from the interstage station seems to have been convectively downstream with outward radial migration just as was seen previously with the high total temperature fluid that originated near the hub. This resulted in an accentuation of the local minimum at 20% span and the development of a local maximum at 54% span.

Figure 15b shows that the cavity flow had a profound effect on the tangential velocity at the interstage station up to 15% span, and had some effect on the rest of the span. The cavity flow, which was purely radial, caused considerable loss of tangential velocity near the hub and some loss through the remainder of the span. This had a detrimental effect on the overall performance of the blade. At the stage exit, the characteristic shape of the tangential velocity profiles is similar with small differences in the value and spanwise locations of the local minimum and maximum. The net result was that the blade in the leakage case did considerably less work than the baseline blade.

Figure 15c compares the radial velocity profiles, and shows why the cavity flow had such an influence on the flow into and through the blade. At the interstage station, the baseline case plot shows that the 1st vane provided an almost uniform radial profile with a peak value of only 5 ft/s. The radial velocity profile for the leakage flow case had a peak value of 168 ft/s at 6% span. The radial velocity coming out of the cavity just upstream of the interstage station was 400 ft/s. The magnitude of the total velocity coming out of the vane was 320 ft/s. Although the cavity gap was only 0.025 inches and modeled using a single line of grid points at the hub, the magnitude of the cavity flow velocity and the fact that it was strictly radial caused considerable disruption in the flow downstream of the cavity.
Figure 15. Comparison of predicted axial velocity (a), tangential velocity (b), and radial velocity (c) at the interstage station and at the stage exit for the baseline case and the case with hub leakage flow.

The VBI analysis has proven to be a valuable tool, and it has considerable potential to become much more useful. However, there is a shortcoming with the use of the grid system shown in Figure 1 that needs to be addressed. That shortcoming or problem is the lack of conservation, which results from the transfer of interpolated boundary data between the O-grids and the H-grids. Figure 16 shows a comparison of the predicted mass flow through the stage from the inlet of the vane to the exit of the rotor. The values were calculated from three vane passages and four blade passages at a single instant in time. Averaging the mass flow
through each passage over the unsteady time period is actually a rigorous and more accurate determination. That capability is not currently in the code, and will be added with the next series of upgrades. The post-processing code was not sophisticated enough to calculate the mass flow in the region of the grid where the O-grid is embedded in the H-grid. A linear curve from the calculated mass flow value upstream of the O-grid to the value downstream of the O-grid is shown in the figure. If mass flow were exactly conserved, there would be no difference between the two. The fact that there was a difference raises a legitimate concern.

![Purdue Low Speed Rig 1st Stage VB3D Prediction](image_url)

Figure 16. Variation of predicted mass flow with axial station through the low speed rig 1st stage for the baseline case and the case with hub leakage flow.

In both cases, the flow from the inlet of the H-grid to the beginning of the H-grid and O-grid overlap region was constant, showing good mass flow conservation for the pure H-grid region. In the overlap region of the vane O-grid and H-grid, both predictions show a gain of 0.12 lb/s or about 2% of the primary flow. For the baseline case, the flow was constant from the end of the vane O-grid through the interface to the beginning of the blade O-grid. This behavior is expected, and showed that mass flow was conserved through the interface. For the case with hub mass-flow addition, there was a rise in mass flow at the location of the cavity, but this rise was not constant in the rotor H-grid. This behavior is probably due to the limited number of blade flow passages included in the post-processing or due to lack of numerical convergence. For the baseline case, there was an increase in mass flow through the rotor O-grid and H-grid overlap region. For the case with hub mass flow addition, there was a decrease through this region of the blade. Both cases show a blade exit flow of about 6.25 lb/s. From this plot, two conclusions can be drawn. (1) The mass flow through the stage was determined by the exit conditions. For both cases the specified exit static
pressure determined the flow exiting the stage. (2) Flow was not conserved through the overlap or Chimera grid region. This was likely not solely because of inadequate convergence. This characteristic has been present to some degree in all VBI calculations. In addition, the combined H-grid and O-grid for the vane used 477,600 points and 397,312 for the rotor. With the current tools for O-grid generation and with the leeway to use that number of points for each blade row, a pure O-grid VBI scheme which is conservative, provides better shock and wake resolution and is easier to use could be developed. The effort would be small compared to the benefits of a much-improved analysis.

Animation of the unsteady Calculations

Six animations of the unsteady flow in the two cases were created. The animations were created by saving individual color images made with Plot3D or FAST, and combined sequentially using the Silicon Graphics software MEDIA CONVERT. Five of the animations showed the change in blade-to-blade flow quantities with time. At midspan, a Mach number contour animation showed how the flow in the interstage region was affected by the passing of the rotor. Animation of entropy contours was very useful in showing convection of the vane wake downstream. The blade sliced through and cut the vane wake into patches of flow that were transported downstream. The wake was stretched through the blade passage because the flow near the suction surface of a given blade was convected faster than the flow near the pressure side of the adjacent rotor. The passing of the vane wake also had a noticeable effect on the blade suction surface boundary layer.

Blade-to-blade animations of entropy, radial velocity, and tangential velocity contours at 10% span for the case with cavity flow showed the introduction of cavity flow in the hub endwall region. These animations demonstrated the passing of the rotor and illustrated how the rotor potential field caused the cavity flow to be unsteady even though the conditions specified at the cavity were constant. The sixth animation was made for visual comparison with test data. From test data, Purdue researchers made an animation of the velocity vectors in a circumferential plane between the trailing edge of the vane and the leading edge of the rotor. From the simulation, we made a similar animation of velocity vectors from the inlet of the rotor grid to the leading edge of the rotor. Both animations compared well with each other and demonstrated how the passing of the rotor affects the cavity flow and the velocity field in the interstage region.

Summary

A detailed set of time-dependent, vane/blade aerodynamic interaction simulations was compiled using the VBI Navier-Stokes analysis code for the Purdue low speed turbine research rig. The simulations depict the time-dependent features of this complicated aerodynamic problem both with and without hub leakage flow. Decomposition of the data depicts several key features of the interactions between the leakage flow, and primary and secondary flow features through the blade passage. Animations of the time-dependent simulations improve our understanding of this problem and lead insight for measured data. Future efforts will concentrate on correlating test and prediction data more thoroughly and examining other features of this complex flow.
Final Report For The Project Entitled

Edge Cooling Heat Transfer On Turbine Blades

for
Advanced Gas Turbine Scientific Research Consortium
South Carolina Institute for Energy Studies (SCIES)
Clemson University
Clemson, SC
No. I/98-01-SR071/DOE

by
R. J. Goldstein
Heat Transfer Laboratory
Department of Mechanical Engineering
University of Minnesota
111 Church St. SE
Minneapolis, Minnesota

and

C. Camci
Department of Mechanical Engineering
Pennsylvania State University
University Park, Pennsylvania

February 19, 2002
Contents

1 Introduction ........................................................................... 8

2 Experimental Facilities, Instrumentation and Procedures .......... 10
   2.1 Experimental Facilities .................................................. 10
       2.1.1 Wind tunnel ......................................................... 10
       2.1.2 Test section ......................................................... 11
       2.1.3 Linear turbine cascade with tip clearance ................. 11
       2.1.4 Test blade .......................................................... 12
       2.1.5 Qualification of the test section .............................. 13
   2.2 Experimental Instrumentation ......................................... 13
       2.2.1 Pressure measurement ........................................... 13
       2.2.2 Temperature measurement ...................................... 14
       2.2.3 Data acquisition system ....................................... 15
       2.2.4 Setup for the naphthalene blade ............................ 15
       2.2.5 Hot wire anemometry ........................................... 16
   2.3 Experimental Procedures ............................................... 16
       2.3.1 Procedures for data reduction ............................... 16

3 Flow Field Characterization and Visualization ....................... 26
   3.1 Flow Field Characterization .......................................... 26
       3.1.1 Laminar boundary layer flow on the tip-endwall .......... 26
       3.1.2 Turbulent boundary layer flow on the tip-endwall ...... 26
       3.1.3 Grid generated high turbulence ............................ 27
       3.1.4 Static pressure distribution and flow ..................... 27
       3.1.5 Near-tip static pressure distribution ..................... 28
   3.2 Flow Visualization .................................................... 28
       3.2.1 Surface flow visualization .................................... 28
       3.2.2 Surface flow near the endwall with no tip clearance .... 29
       3.2.3 Surface flow on the tip-endwall with tip clearance ... 30
       3.2.4 Surface flow on the tip surface ............................ 31
       3.2.5 Surface flow on the blade near-tip surfaces ............ 32
       3.2.6 Smoke wire visualization ..................................... 32
   3.3 Summary ............................................................... 34
4 Mass/Heat Transfer Measurement
   4.1 Mass/Heat Transfer from Blade Surfaces (No Clearance) .............. 52
   4.2 Mass/Heat Transfer from the Near-Tip Blade Surfaces ............... 55
   4.3 Mass/Heat Transfer from the Tip Surface .......................... 59
      4.3.1 Effect of tip clearance level .................................. 59
      4.3.2 Effect of mainstream stream Reynolds number and turbulence ... 60
   4.4 Summary .......................................................... 61

5 Tip Mass Transfer Measurements With Endwall Motion .................. 116
   5.1 Description of the moving wall apparatus .......................... 116
   5.2 Mass transfer field on the tip ................................... 118
   5.3 Summary .......................................................... 121

6 Flow and Heat Transfer in the Tip Region and Endwall Region of an
   Annular Cascade .................................................... 135
   6.1 Turbine research facility modifications performed for the specific research
        program .......................................................... 136
   6.2 Project Results ................................................... 137
      6.2.1 Endwall coolant injection from the wheelspace cavity and its influence
              on turbine mainstream aerodynamics .......................... 138
      6.2.2 Turbine tip aerodynamics under the influence of coolant injection
              from a tip trench ............................................. 140
      6.2.3 Heat transfer measurements on the tip surface using liquid crystal
              thermography ............................................... 141
# List of Figures

2.1 Plan view of the test section ........................................... 19  
2.2 Cascade coordinates and parameters .................................... 20  
2.3 Sketch of the naphthalene blade ....................................... 20  
2.4 Cascade blade balance test data ....................................... 21  
2.5 Incoming flow velocity and turbulence intensity .................. 22  
2.6 Flow chart of data acquisition system (Wang, 1997) .............. 23  
2.7 Four-axis table (Wang, 1997) ......................................... 24  
2.8 Calibration curves for LVDT PCA-220-010 gauge .................... 25  
3.1 Laminar boundary layer flow on the tip-endwall .................. 36  
3.2 Turbulent boundary layer on the tip-endwall ....................... 37  
3.3 Turbulent boundary layer on the tip-endwall (grid turbulence) .. 38  
3.4 $C_p$ and Cascade flow at midspan ................................... 39  
3.5 $C_p$ near the blade tip for different tip clearances ................ 40  
3.6 Surface flow visualization without tip clearance ................. 41  
3.7 Tip-endwall surface flow visualization ............................... 42  
3.8 Tip surface flow visualization ......................................... 43  
3.9 Suction side surface flow visualization .............................. 44  
3.10 Pressure side surface flow visualization ............................ 45  
3.11 Secondary flows with no tip clearance .............................. 46  
3.12 Tip leakage vortex and secondary flows at $\tau/C = 0.86\%$ ........ 47  
3.13 Tip leakage vortex and secondary flows at $\tau/C = 1.72\%$ ........ 48  
3.14 Tip leakage vortex and secondary flows at $\tau/C = 3.45\%$ ........ 49  
3.15 Leakage flow inside the tip clearance at $\tau/C = 3.45\%$ ........ 50  
3.16 Secondary flow model proposed by Wang (1997) ................... 51  
4.1 Mid-span average $Sh$ .................................................. 65  
4.2 Mid-span average $Sh/Re_{ex}^{1/2}$ ...................................... 66  
4.3 Mid-span RMS $Sh$ ...................................................... 67  
4.4 Local $Sh$ on the mid-span pressure surface ......................... 68  
4.5 Local $Sh$ on the mid-span suction surface .......................... 69  
4.6 $Sh$ contour on the pressure surface with no tip clearance .......... 70  
4.7 $Sh$ surface plot on the pressure surface with no tip clearance .... 71  
4.8 Local $Sh$ on the pressure surface with no tip clearance ........... 72
4.9 Local $Sh/Re_{ex}^{1/2}$ on the pressure surface with no tip clearance ........... 73
4.10 Sh contour on the suction surface with no tip clearance ................. 74
4.11 Sh surface plot on the suction surface with no tip clearance .......... 75
4.12 Local Sh on the suction surface with no tip clearance ................. 76
4.13 Local $Sh/Re_{ex}^{1/2}$ on the suction surface with no tip clearance ......... 77
4.14 Spanwise average Sh in 3D region .................................... 78
4.15 Spanwise average $Sh/Re_{ex}^{1/2}$ in 3D region ........................... 79
4.16 Sh contour on the pressure surface at different tip clearance levels ...... 80
4.17 Sh surface plot on the pressure surface at different tip clearance levels .. 81
4.18 Local Sh on the pressure surface at different tip clearance levels ........ 82
4.19 Local $Sh/Re_{ex}^{1/2}$ on the pressure surface at different tip clearance levels .... 83
4.20 Average Sh on the pressure surface at different tip clearance levels ...... 84
4.21 Sh contour on the pressure surface at $\tau/C = 0.86\%$ ...................... 85
4.22 Sh surface plot on the pressure surface at $\tau/C = 0.86\%$ ................. 86
4.23 Local Sh on the pressure surface at $\tau/C = 0.86\%$ ...................... 87
4.24 Local $Sh/Re_{ex}^{1/2}$ on the pressure surface at $\tau/C = 0.86\%$ .......... 88
4.25 Local $Sh/Re_{ex}^{1/2}$ on the pressure surface: effect of turbulence ......... 89
4.26 Average Sh on the pressure surface at $\tau/C = 0.86\%$ ................. 90
4.27 Sh contour on the suction surface at different tip clearance levels ...... 91
4.28 Sh surface plot on the suction surface at different tip clearance levels .... 92
4.29 Local Sh on the suction surface at different tip clearance levels .......... 93
4.30 Local $Sh/Re_{ex}^{1/2}$ on the suction surface at different tip clearance levels .... 94
4.31 Average Sh on the suction surface at different tip clearance levels ...... 95
4.32 Sh contour on the suction surface at $\tau/C = 0.86\%$ ...................... 96
4.33 Sh surface plot on the suction surface at $\tau/C = 0.86\%$ ................. 97
4.34 Local Sh on the suction surface at $\tau/C = 0.86\%$ ...................... 98
4.35 Local $Sh/Re_{ex}^{1/2}$ on the suction surface at $\tau/C = 0.86\%$ ............. 99
4.36 Local $Sh/Re_{ex}^{1/2}$ on the suction surface: effect of turbulence .......... 100
4.37 Average Sh on the suction surface at $\tau/C = 0.86\%$ .................... 101
4.38 Tip surface coordinates ................................................. 102
4.39 Sh contour on the tip surface ............................................ 103
4.40 Sh contour on the tip surface ............................................ 104
4.41 Local Sh on the tip surface at different clearance levels .................. 105
4.42 Local Sh on the tip surface at different clearance levels (continued) .... 106
4.43 Local Sh on the tip surface at different clearance levels .................. 107
4.44 Local Sh on the tip surface at different clearance levels (continued) .... 108
4.45 Sh contour on the tip surface at $\tau/C = 0.86\%$ .......................... 110
4.46 Local Sh on the tip surface at $\tau/C = 0.86\%$ .......................... 111
4.47 Local Sh on the tip surface at $\tau/C = 0.86\%$ (continued) ............... 112
4.48 Local $Sh/Re_{\tau}^{0.8}$ on the tip surface at $\tau/C = 0.86\%$ ............... 113
4.49 Local $Sh/Re_{\tau}^{0.8}$ on the tip surface at $\tau/C = 0.86\%$ (continued) .... 114
4.50 Averaged Sh on the tip surface ........................................... 115
5.1 Moving wall apparatus used to simulate rotation
5.2 Comparison of Sherwood number data for $\tau/C = 6.9\%$, EES=0 with flat endwall
5.3 Sherwood number contours for $Re = 2.7 \times 10^5$
5.4 Sherwood number contours for $Re = 2.7 \times 10^5$
5.5 Sherwood number contours for $Re = 2.7 \times 10^5$
5.6 Effect of endwall motion on Sherwood number for $\tau/C = 0.6\%$
5.7 Effect of endwall motion on Sherwood number for $\tau/C = 0.6\%$
5.8 Effect of endwall motion on Sherwood number for $\tau/C = 0.86\%$
5.9 Effect of endwall motion on Sherwood number for $\tau/C = 0.86\%$
5.10 Effect of endwall motion on Sherwood number for $\tau/C = 1.72\%$
5.11 Effect of endwall motion on Sherwood number for $\tau/C = 1.72\%$
5.12 Effect of endwall motion on Sherwood number for $\tau/C = 3.45\%$
5.13 Effect of endwall motion on Sherwood number for $\tau/C = 3.45\%$
5.14 Effect of endwall motion on Sherwood number for $\tau/C = 6.9\%$
5.15 Effect of endwall motion on Sherwood number for $\tau/C = 6.9\%$

6.1 Perspective view of blade in AFTRF
6.2 Earlier work using Inconel heater
6.3 Thermogram of tip surface
6.4 Sketch of proposed design showing connector strips
6.5 Coordinates of blade, mm
6.6 Enlarged view of fig. 6.5
6.7 An alternative heater foil boundary that has extensions near the trailing edge.
List of Tables

2.1  Tip clearance levels for the linear cascade .......................... 12
2.2  Turbine linear cascade parameters ................................. 18
3.1  Characteristic of inlet boundary layer flow on the tip-endwall ....... 27
4.1  Surface averaged Sh measured at different tip clearance levels ($Re_{ex} = 5.8 \times 10^5, Tu=0.2\%$) ........................................ 63
4.2  Tip surface averaged Nu, measurement from Azad et al. (2000) ($Re_{ex} = 1.1 \times 10^6, Tu=6.1\%$) ........................................ 63
4.3  Tip surface averaged Nu, measurement from Bunker et al. (2000) ($Re_{ex} = 2.57 \times 10^6, Tu=9\%$) ........................................ 63
4.4  Surface averaged Sh at $\tau/C = 0.86\%$ ................................ 63
4.5  Summary of naphthalene sublimation tests .......................... 64
5.1  Area-averaged Sherwood numbers for the different cases studied. .... 121
6.1  Power density (airfoil area=1600 mm$^2$) ............................ 144
Chapter 1

Introduction

This document summarizes a collaborative investigation between the Pennsylvania State University and University of Minnesota on aerodynamics and heat transfer performed near the tip region and endwall of a turbine rotor blade with and without coolant ejection. For the design of gas turbines, designers need reliable flow and heat transfer information for the hot section components. Such information is still missing for several locations on the gas flow path which are exposed to deterioration and destruction by the hot gas. A recent research proposal from the University of Minnesota highlights three needed areas:

- Flow and Heat Transfer in the tip region of turbine blades.
- Heat Transfer caused by misalignment or leakage between turbine blades or with adjoining flow channels at the base region.
- Reduction of the strength of the horseshoe-passage-vortex system.

The current University of Minnesota study uses a linear cascade facility and special instrumentation for naphthalene sublimation based mass/heat transfer measurements as well as tracers in the fluid stream and on surfaces. The proposed investigation includes validation of CFD models by using the measured data. The computational models are then suggested to extend the range of conditions to high Mach numbers, large temperatures and high rotation rates. The final outcome of the Minnesota study is a set of design recommendations for industry. The details of the specific approach is explained in a proposal "Edge Cooling Heat Transfer on Turbine Blades," by R.J.Goldstein, S.V.Patankar and E.R.G.Eckert submitted to AGTSR, SCERDC (RFP No. AGTSR 97-01). The Pennsylvania State University currently operates a large scale Axial Flow Turbine Research Facility in its Turbomachinery Laboratory. The turbine rotor is instrumented for internal aerodynamics and heat transfer studies including cooling in the rotor frame of reference. The collaborative investigation was in the areas of assessing the additional effects caused by rotational effects in a turbine rotor frame of reference.

The organization of this report is as follows: chapter 2 describes the measurement facilities used in the University of Minnesota for flow and mass transfer measurements, and the flow visualization procedure. Chapter 3 presents the effects of tip clearance on the pressure distribution around the blade, and also gives flow visualization results. Chapter 4
describes mass transfer from the tip and near-tip surfaces with the presence of tip clearance. Chapter 5 presents the effects of the relative motion between the tip surface and the endwall. Chapter 6 describes the results obtained at the Axial Flow Turbine Research Facility at PSU.
Chapter 2

Experimental Facilities, Instrumentation and Procedures

In this chapter, experimental facilities are introduced first, followed by the instrumentation for various measurements. The focus is on the qualification of the linear cascade as well as the setup for naphthalene sublimation measurement. Finally, the procedures for running the experiment are presented.

2.1 Experimental Facilities

The experiment facilities employed in the study include a blowing type wind tunnel, and a test section, which consists of a straight and triangular section, a linear cascade with variable tip clearances, and a tailboard section.

2.1.1 Wind tunnel

The wind tunnel used in the present study is a multi-purpose blowing type wind tunnel built by Engineering Laboratory in 1991 in the Heat Transfer Laboratory at the University of Minnesota.

At normal operating conditions, air at room temperature is drawn through a filter by a 22.4kW blower, flowing through a long diffuser and a heat exchanger before entering a couple of settling chambers with screens inside. After passing through a second screen layer, the air flow is guided through a squared contraction nozzle with an area ratio of 6.25 and an exit area of $45.7 \times 45.7cm^2$ into the test section, which will be described in the next section. Exiting the test section, the air is discharged into the room and then outside the building through windows. As the exit of the contraction, the air flow can attain a velocity of about $40m/s$ with 0.2% turbulence intensity as designed by the manufacturer.

The mainstream velocity in the wind tunnel is controlled by a variable torque system and can be tuned by adjusting the rotating frequency of the blower.
2.1.2 Test section

The test section as shown in fig. 2.3.1 is connected to the exit of the contraction of wind tunnel and is composed of the straight and triangular sections, the linear cascade with different tip clearances, and the tailboard section. All the walls of the test section except the top and bottom wall of the cascade section are made of 1.9cm thick Plexiglas. The straight section with a cross-section of $45.7 \times 45.7\,\text{cm}^2$ and a length of 61.0cm, has slots reserved for generation of mainstream turbulence by inserting various grid turbulence generators, and is followed by the triangular section connecting to the linear cascade. In present study, a bar grid turbulence generator is inserted into one of the slots to obtain a high free stream turbulence level.

To obtain a fully developed turbulent boundary layer on the endwall, a 1mm diameter trip wire is placed at the exit of the contraction section, which is 82.5cm ahead of the stagnation point of the central blade. At the incoming flow measurement plane, which is at $X = 67.0\,\text{cm}$, ahead of of the leading edge of the center blade in the direction of the incoming flow, the incoming flow velocity and turbulence intensity are measured using a hot wire anemometry and a traverse system. The incoming flow dynamic pressure (velocity), static pressure and temperature are also observed at this position, using a pitot tube and a T type thermocouple respectively. Measurement plane is at $X = -20.8\,\text{cm}$, ahead and parallel to the leading edges of the blades, where the boundary layer on the tip-endwall is measured at three different locations at half pitch distance. Velocity and turbulence intensity are also measured at measurement plane2 ($X = 28.2$), which is downstream of the linear cascade and parallel to the trailing edge of the blades.

The tailboard section has two tailboards attached to the trailing edges of two outside blades. It has two endwalls connected to endwalls of the cascade section.

2.1.3 Linear turbine cascade with tip clearance

The linear turbine cascade consists of five 45.7cm long high pressure turbine rotor blades made of aluminum with central blade configuration (blade 3 in fig. 2.3.1). The blade profile data and other cascade geometries are listed in table 2.2 and shown in fig. 2.2. Basically, the blade has a profile of the high performance turbine blade in high pressure stage and the cascade has a blade chord of 18.4cm with a solidity of 0.75 and a high turning angle of 107.5°.

The two outside blades (blade 1 and 5) fixed by screws from the bottom and the top walls, together with two bleeds and two downstream tailboards, form the sidewalls for the mainstream. The space between the bleeds and the outside blades as well as the orientation of the tailboards are adjustable to obtain periodic flows in blade passages. Three middle blades (blade 2, 3 and 4) are fixed by screws at the bottom wall. The central blade is composed of three parts, one of which is the test blade and can be either pressure blade or naphthalene blade placed near the tip-endwall. The other two are solid aluminum blades with equal height of 13.0cm (5.125 inch) and can be placed between the bottom endwall and the test blade during the measurement. However, the test blade can also be placed between the other two solid part for two-dimensional mid-span measurement.

The bottom and the top wall of the linear cascade are 1.27cm thick aluminum plates,
Table 2.1: Tip clearance levels for the linear cascade

<table>
<thead>
<tr>
<th>spacers</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>no spacer</th>
</tr>
</thead>
<tbody>
<tr>
<td>height(cm)</td>
<td>1.27</td>
<td>1.11</td>
<td>0.95</td>
<td>0.64</td>
<td>0.0</td>
</tr>
<tr>
<td>$\tau/C(%)$</td>
<td>0.0</td>
<td>0.86</td>
<td>1.72</td>
<td>3.45</td>
<td>6.90</td>
</tr>
</tbody>
</table>

and are attached to the endwalls of the triangular section and the endwalls of the tailboard section.

The top wall (tip-endwall) simulates the casing of the turbine and the tip clearance region near the central blade is the main focus of this research. The space between the three middle blades and the tip-endwall forms the blade tip clearance, which can be varied by adding aluminum spacers with different thicknesses between the blades and the bottom wall. The thicknesses of the spacers and the corresponding tip clearances employed in this study are listed in table 2.1. A rectangular Plexiglas window covering most of the tip clearance region is opened on the tip-endwall for observation use.

2.1.4 Test blade

As mentioned in the previous section, two test blades are used: one is the pressure blade for static pressure distribution measurement on the blade surface, and the other is the naphthalene blade for mass transfer measurement. Both blades have the same high pressure blade profile.

The pressure blade is hollow and assembled from two aluminum sheets with the high pressure blade surface profile and two caps through pins and screws. It has a height of 19.69 cm (7.75 inch). There are eight rows of pressure tap holes on the blade surfaces along the span, with the first row 2.2 cm (0.875 inch) away from its top tip and each row consisting of 30 taps, of which 28 are used in this study. From the interior of the pressure blade, pressure taps are connected to an outside pressure manometer through small-diameter plastic tubes. The pressure blade is first used for balancing the flow through blade passages by comparing the static pressure on the central blade surface with that of neighboring blades and adjusting the bleeds and the tailboards accordingly. It is also used for measuring the effect of tip clearance on the static pressure distribution on the blade surface.

The naphthalene blade made essentially of solid aluminum, consists of a top hollow cap, a thin bottom cap, and a main part with a spanwise through hole (fig. 2.3), with an overall height of 18.7 cm (7.375 inch). The through hole and the hollowed cap are designed for cooling air injection from the tip for future study. The side and top surfaces of the naphthalene blade are 2.5 mm deeper than the rims at top and bottom of the blade. These small gaps leave room for liquid naphthalene to cast around the blade side and top surfaces by a mold. Some grooves are cut on those deeper surfaces for holding the naphthalene cast.

A specially designed mold and modified casting procedures are employed for the naphthalene blade with tip surface. To obtain a smooth naphthalene-cast profile, inner surfaces of the mold with the high pressure blade profile are well polished. The naphthalene tip
surface can be obtained between the depressed tip of the blade and a flat spacer with a smooth inner surface and a thickness of 0.95cm (0.375 inch), which also makes the total height of the blade and the spacer compatible with that of the mold. Thus, during the casting, the liquid naphthalene will first go through a small hole in the sink to form the tip naphthalene surface, and then fill the gap between the metal surfaces of the naphthalene blade and mold to obtain smooth naphthalene surfaces, and finally exit from the top air vent holes. It is found that the temperature of the mold may have some effects on the smoothness of the casting since the mold temperature is not uniform during the solidification process.

2.1.5 Qualification of the test section

Periodic flow in the wind tunnel is obtained by adjusting the bleeds and the tailboards, obtaining static pressure coefficients from the pressure blade replacing the three middle blades respectively, and ensuring the coefficients are close to the data from the inviscid flow solution.

It can be seen from the figure that the static pressure distributions from the three middle blades are close to each other and agree with the inviscid solution fig. 2.4, especially for the pressure side. This implies that passage flows around the center blade in the two neighboring passages have almost the same volumetric flows. Therefore, the periodic flow in the blade passages is established and the linear cascade test section is qualified for this study.

The incoming flow from the wind tunnel is also measured at the incoming flow measurement plane in fig. 2.3.1 using hot wire anemometry to ensure the mainstream is uniform and steady. It is shown in fig. 2.5 that the incoming flow has the typical mean flow and turbulence characteristics of a duct flow at different inlet speeds.

2.2 Experimental Instrumentation

The present experiment measures four fundamental parameters: time, pressure, temperature and sublimation depth. During the measurement, the time is recorded using a hand timer of 1 sec precision for each step. The main flow velocity and turbulence characteristics are measured by hot wire anemometry while the incoming flow dynamic pressure and static pressure are obtained with a pitot tube. A special data acquisition system is used for sublimation depth measurement.

2.2.1 Pressure measurement

Pressure is a fundamental parameter in determining the density, diffusivity, and velocity, hence should be measured accurately during the experiment.

The atmospheric pressure ($p_{atm}$) is measured by a PRINCO NOVA™ barometer in the wind tunnel room with a reading precision of 0.1 mmHg and can be obtained by:

$$p_{atm} = \rho H g \Delta H_{atm}$$

(2.1)

The dynamic ($\Delta p$) and static pressure ($p_s$) of the incoming flow in the wind tunnel is measured by a pitot tube inserted at the incoming flow measurement plane. The dynamic
pressure, which is the pressured difference between the total pressure \( p_0 \) and static pressure in wind tunnel, is obtained by a Dwyer Microtector manometer with a precision of 0.01\textit{mm}H\textsubscript{2}O.

\begin{align*}
p_s &= p_{\text{atm}} - \rho H_2O g \Delta H_{sl} \\
\Delta p &= p_o - p_s = 2\rho H_2O g \Delta H_{dyn}
\end{align*}

The density of air in the wind tunnel can be calculated from the ideal gas law:

\[ \rho = \frac{M_{\text{air}} P}{RT} \]

in which the \( P \) and \( T \) are the static pressure and temperature of the incoming flow in the wind tunnel.

The incoming main flow velocity and Reynolds number at the cascade exit based on chord and exit velocity can then be obtained from:

\begin{align*}
U_\infty &= \sqrt{2\Delta p / \rho} \\
Re_{ex} &= \frac{\rho U_{\infty} C}{\mu} = \frac{\rho AR U_{\infty} C}{\mu}
\end{align*}

\( AR \) is the area ratio of inlet to exit of the blade passage (=2.72).

The static pressures measured on the pressure blade is read from a manometer system with a precision of 1.3\textit{mm} water height.

\subsection*{2.2.2 Temperature measurement}

The properties of air are functions of temperature. The pressure and density of naphthalene vapor are also sensitive to the naphthalene surface temperature, which in turn affects the sublimation rate and the accuracy of local mass transfer coefficient. Therefore, it is important to attain high accuracy and precision in the temperature measurement, especially near the surface of naphthalene layer.

The temperature in the wind tunnel room is obtained from a thermometer with a 0.2\textdegree C reading precision, which is located near the barometer. The temperature in the data acquisition system room is also monitored by a thermometer with the same reading precision. These temperatures are used for naphthalene natural convection loss estimation.

The incoming main flow temperature in the wind tunnel \( (T) \), the naphthalene surface temperature at two different locations \( (T_w) \) around the mid-blade on the pressure and the suction side, respectively, are measured by three T-type thermocouples. The signals are then transferred to a multiple scanner (Keithly 7001) and a Keithly 196 digital multimeter, from which the voltage readings are collected by a Linux workstation through an IEEE-488 (GPIB) bus during the testing. All of the thermocouples were calibrated in a previous study (Goldstein et al., 1999).
2.2.3 Data acquisition system

To measure the naphthalene sublimation depth locally on the naphthalene blade surface, a microcomputer-controlled four-axis automated data acquisition system is used, which is also described by Wang (1997) in detail.

The four-axis data acquisition system consists of a four-axis table, a motor-controller (Superior Electric MODULYNX) with four terminals, a signal conditional (Schaevitz ATA-101), a Linear Variable Differential Transformer (LVDT) depth gauge (Schaevitz PCA-200-010 LVDT), a Digital multimeter (Fluke model 8840A), and an IBM-XT microcomputer with UNIX operating system and GPIB card. A flow chart of the system is shown in fig. 2.6.

The four-axis \((X-Y-Z-\Theta)\) table described in fig. 2.7 is used to position the naphthalene blade and LVDT gauge. The XY-table (Design Components Inc. DC-1212) is similar to that described in Jin (1998). The rotary table is a model 6R90 of Design Components. The Z-table (Velmax B4015) holds the LVDT gauge while the rotary table controls the orientation of the naphthalene blade. All the tables are driven by step motors connected to four terminals of the motor-controller from Superior Electric.

An LVDT is an electromechanical device that produces an electrical output proportional to the displacement of a separate movable core. The LVDT depth gauge made by Schaevitz Engineering (PCA-220-010) has a standard deviation of 225\(\mu m\) in a linear range of \(\pm 0.25\mu m\) as shown by calibration and mock measurement. The LVDT gauge is calibrated by measuring a series of precision gauge blocks at 25.4\(\mu m\) intervals (fig. 2.8(a)). The random errors of the LVDT gauge and the four-axis table positioning system are estimated by mock sublimation depth measurement on the metal rings of the naphthalene blade. The blade is aligned and scanned twice as for a real naphthalene sublimation testing. Due to the random errors in electronic signals as well as in repositioning, the sublimation depth is not ideally zero, instead it has a Gauss distribution as shown in fig. 2.8(b).

The signal from the LVDT is sent to the Schaevitz signal conditioner (model ATA-101) and amplified to a DC voltage within range of \(\pm 13 V\), and is then input to the Fluke (model 8840A) digital multimeter.

The IBM XT computer is connected to the four-axis system and the digital multimeter through an IEEE-488 (GPIB) interface. The command issued by a data acquisition program from the computer is transferred through the IEEE bus to position the gauge and the naphthalene blade. The measured signal from the LVDT gauge is then converted by the conditioner and displayed by the multimeter, and finally collected by the computer through the IEEE-488 (GPIB) interface.

2.2.4 Setup for the naphthalene blade

The naphthalene blade is fixed on a circular plate by two screws from bottom. Pins are used to make sure that the same position can be maintained for before and after runs. The circular plate is supported by three cylinders of the same height on the rotary table and connected to the rotary table by four long screws so that the plate and the fixed blade can rotate together with the rotary table. The height of the cylinders and the tightness of the screws must be adjusted to make sure the blade stands straight vertically. A pre-scanning of the blade rims of the side surface before the formal scanning can put the gauge in an
accurate position when approaching the blade surface. For the tip surface measurement, a
counter program is also developed so that the gauge can move accurately along the tip
surface between the concave and convex rims.

2.2.5 Hot wire anemometry

A single constant temperature hot wire probe (TSI model-1210) is used for turbulent flow
field measurement in the wind tunnel. Boundary layer measurement at measurement plane
is conducted using a boundary layer probe (TSI model-1218).

2.3 Experimental Procedures

The naphthalene sublimation technique is used for obtaining mass transfer coefficients on
the naphthalene blade surfaces, which are exposed to the flow in the linear turbine cascade.
Following steps are required for the measurements.

1. naphthalene casting for the blade surface and tip.
2. naphthalene blade and tip surface profile measurement before-run.
3. wind tunnel run with naphthalene blade installed in the linear cascade.
4. naphthalene blade and tip surface profile measurement after-run.
5. data reduction after the experiment,

From the beginning of the second step, a hand timer is started to record the time for each
procedure conducted until the start of the forth step. This is used during the data reduction,
in order to account for the sublimation due to natural convection during the setup.

2.3.1 Procedures for data reduction

The data reduction process is fully computerized. The basic procedures are listed below.

1. input data file preparation Input the data from the before-run and after-run profile
   measurement, the temperature and pressure measured in the wind tunnel run, the
time recorded in the whole measurement process.

2. data conversion Convert the voltage readings in the profile measurements into
   depth using the LVDT calibration curve for each measurement position and compare
   them with the straight line formed by the two reference points measured on the
   rims to obtain the sublimation depth.

3. natural convection loss estimation Estimate the naphthalene natural convection loss
during the profile measurements, the storage, setup and dismount process in wind tun-
nel, and subtract it from the total sublimation depth to get the net sublimation depth
due to the forced convection. For side surfaces measurement, the natural convection
correlations from Wang (1997) is used. For tip surface, the natural convection loss is
estimated by sitting the blade on the 4-axis table and scanning the tip surface every hour. Usually, the average natural convection loss is about 3 – 4μm, about 5% of the total sublimation depth.

4. *output data calculation* Calculate the local Sherwood number and Stanton number from the net sublimation depth and the time spent in the run.

5. *data visualization* Use data visualization software such as Matlab and Techplot to analyze and visualize the obtained data.
Table 2.2: Turbine linear cascade parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades</td>
<td>5</td>
</tr>
<tr>
<td>Chord length of blade - $C$</td>
<td>18.4 cm</td>
</tr>
<tr>
<td>Axial chord of blade - $C_a$</td>
<td>12.96 cm</td>
</tr>
<tr>
<td>Pitch of cascade - $P$</td>
<td>13.8 cm</td>
</tr>
<tr>
<td>Height of cascade - $H$</td>
<td>45.7 cm</td>
</tr>
<tr>
<td>Aspect ratio (Span/Chord) - $H/C$</td>
<td>2.48</td>
</tr>
<tr>
<td>Solidity (Pitch/Chord) - $P/C$</td>
<td>0.75</td>
</tr>
<tr>
<td>Blade inlet angle - $\beta_1$</td>
<td>35°</td>
</tr>
<tr>
<td>Blade outlet angle - $\beta_2$</td>
<td>-72.49°</td>
</tr>
<tr>
<td>Inlet/Exit area ratio of the cascade (AR)</td>
<td>2.72</td>
</tr>
<tr>
<td>Area ratio of the contraction</td>
<td>6.25</td>
</tr>
<tr>
<td>Exit Reynolds number - $Re_{e,x} \times 10^{-5}$ range</td>
<td>4.5-6.9</td>
</tr>
</tbody>
</table>
Figure 2.2: Cascade coordinates and parameters

Figure 2.3: Sketch of the naphthalene blade
Figure 2.4: Cascade blade balance test data
incoming flow velocity distribution

incoming flow turbulence intensity distribution

Figure 2.5: Incoming flow velocity and turbulence intensity
Figure 2.6: Flow chart of data acquisition system (Wang, 1997)
Figure 2.7: Four-axis table (Wang, 1997)
Figure 2.8: Calibration curves for LVDT PCA-220-010 gauge
Chapter 3

Flow Field Characterization and Visualization

Boundary layer flow characteristics of the incoming flow on the tip-endwall and the static pressure distribution on the blade surface near the tip are presented in this chapter, followed by flow visualization results.

3.1 Flow Field Characterization

Inlet boundary layer flow on the tip-endwall and high turbulence mainstream are characterized using a single-sensor hot wire probe upstream of the cascade. The static pressure on the blade surface is measured with the pressure blade and a manometer. The various boundary layer characteristics as well as mainstream $Re_{ex}$ and $Tu$ are summarized in table 3.1. In all cases, the boundary layer thickness is defined at 99.5% of mainstream velocity.

3.1.1 Laminar boundary layer flow on the tip-endwall

The boundary layer is measured at flow measurement plane1 ($X/C_x = -1.6$). The mainstream velocity is 2.7m/s with a low turbulence intensity of 0.2%. As mentioned by Wang (1997) and shown by following results, the flow is laminar even with the trip wire placed upstream.

As plotted in fig. 3.1, the boundary layer flow on the tip-endwall is clearly laminar: the velocity profile shown in the upper figure is less steeper close to the wall and the velocity distribution in terms of the wall coordinates obviously follows the velocity profile of the sublayer (lower figure). The boundary layer displacement thickness at the point of measurement is 0.55cm with a shape factor of 2.32. The mainstream Reynolds number based on the blade chord and exit velocity is $7.0 \times 10^4$.

3.1.2 Turbulent boundary layer flow on the tip-endwall

At normal wind tunnel operating conditions, the mainstream velocity is about 20m/s with a turbulence intensity of 0.2% in the absence of tip clearance. The exit Reynolds number
at this condition is $6.2 \times 10^5$. The boundary layer characteristics, including velocity distribution, velocity profile in the wall coordinates, and the wake characteristics, are measured at three different locations on the plane1 ($X/C_w = -1.6$) at a distance of half pitch from each other on the tip-endwall, as displayed in fig. 3.2.

In fig. 3.2 it shows that the velocity profile near the wall is very steep and assumes the power law with an index of 5.7, while the profile in the wall coordinates follows the log law of the wall. The boundary layer shape factor is 1.38 with a displacement thickness of 0.17cm. The solid line of turbulence intensity is used as an inlet condition in the numerical simulation.

In summary, the data shows that at normal condition, the boundary layer flow on the tip-endwall is uniform and turbulent, following the log law of the wall and the law of the wake, and the displacement thickness is typical of that in a gas turbine engine (1% of chord, Hodson and Dominy (1986)). Although the boundary layer thickness may have influence on the heat/mass transfer on the blade surface as suggested by Graziani et al. (1980), the experiment from Chan et al. (1994) showed that the inlet boundary layer thickness may not have significant effects on tip leakage flow and vortex. Therefore, the effects of inlet boundary layer thickness are not considered further in this study.

3.1.3 Grid generated high turbulence

To simulate the high turbulence flow from the gas turbine combustor, a bar grid turbulence generator with an open area ratio of 0.6 is employed. The free stream turbulence intensity is about 14.0% at an exit Reynolds number of $6.1 \times 10^5$. The turbulent boundary layer on the tip-endwall downstream of the bar grid turbulence generator is also measured at $X/C_w = -1.6$ ahead of the leading edge of the blade 3. The results are plotted in fig. 3.3, which show that the velocity profile fits the log law of the wall well.

3.1.4 Static pressure distribution and flow

Static pressure measurement on the blade surface is performed using the pressure blade. For the two dimensional flow, the middle row of pressure taps is used for obtaining the mid-span static pressure on the blade surface. During the measurement, the mainstream velocity is about 20m/s with a exit Reynolds number of $6.0 \times 10^5$. The measured distribution of static pressure coefficients at mid-span of the central blade, as well as the static pressure coefficients predicted from inviscid flow solution, are plotted in fig. 3.4(a). The measured coefficients on pressure side agree well with the predicted curve from the inviscid prediction,
while on the suction side the measured data are also close to the 2-D prediction with little deviation.

3.1.5 Near-tip static pressure distribution

To measure the effect of tip clearance on the static pressure distribution, a row of pressure taps closest to the blade tip is used, which is 2.2cm (0.12\%C) away from the blade tip. The repeatable static pressure coefficients near the tip are plotted in fig. 3.5 together with the midspan data and the inviscid solution at different clearance levels.

The effect of secondary flows are the apparent cause for the difference between the data for zero tip clearance and those for mid-span, especially close to the trailing edge, as shown in fig. 3.5(b). For measurements with tip clearances, the unloading effect (decrease in pressure difference) from the tip clearance can be clearly found from the plot, even though the pressure taps are not very close to the tip. From the leading edge up to $S/C \approx 0.5$, the blade loading (i.e. the pressure difference) decreases with increase of tip clearance, as the pressure coefficients increase on the suction side but decreases on the pressure side. In effect, very close to the leading edge ($S/C < 0.05$), the loading for the two largest tip clearances become negative, indicating that there may be leakage flowing into (instead of out of) the tip clearance from the suction side (fig. 3.5(e) and (f)). The loading decreases substantially for the all tip clearances in the second half of the blade, where the pressure coefficients increase on both the pressure and the suction surfaces compared with the mid-span data (except for the largest tip clearance), but with larger changes on the suction side, resulting in larger unloading effect at larger tip clearances due to the presence of leakage vortex on the suction side. However, for the largest tip clearance ($\tau/C = 6.90\%$), the pressure coefficients reduces drastically downstream of $S_p/C = 0.6$ on the suction surface, which is possibly caused by the blocking of the strong tip leakage vortex, leading to the reverse of unloading. Similar effect also occurs at smaller clearances ($\tau/C = 0 - 3.45\%$) close to $S_p/C \approx 0.9$.

3.2 Flow Visualization

Two methods of flow visualization are employed in the present study, i.e., oil and lampblack surface flow visualization and laser light sheet smoke wire technique.

3.2.1 Surface flow visualization

For surface flow visualization, a mixture of paint powder and oil is applied to blade surfaces near the tip, the tip surface, and the tip-endwall covered by a piece of tightly fitted white sheet, respectively, at different tip clearances to visualize the surface flow pattern in the linear turbine cascade.

Viscosity of the mixture is adjusted by trial and error to allow the powder particles to follow the direction of shear stresses and to display the shear force pattern on the wall. Patterns on the sheet are photographed using a Richon camera immediately after wind tunnel test at inlet mainstream speed of about 20m/s ($Re_{ex} = 6.0 \times 10^6$) to avoid fading of the pattern.
3.2.2 Surface flow near the endwall with no tip clearance

The surface flow visualizations without tip clearance on the pressure, suction, and endwall surfaces are shown in fig. 3.6. The incoming flow is from left to right for the side surfaces with the mid-span at the bottom.

The endwall surface flow pattern is shown in fig. 3.6(a). The incoming turbulent boundary layer (shown by the arrow), blocked by the blade leading edge, splits around the saddle point A1 and lifts up along the separation curves S2p and S2s to form the pressure and the suction leg of horseshoe vortex \( V_{ph} \) and \( V_{sh} \) in fig. 3.16. The less obvious separation lines S1p and S1s are due to the boundary layer separation ahead of the horseshoe vortex (Sierding, 1985) and usually are difficult to detect, but is clearly shown in Wang et al. (1997) that both the pressure leg of the horseshoe vortex and this boundary layer separation vortex become part of the large passage vortex \( V_p \) in fig. 3.16. The suction leg horseshoe vortex \( V_{sh} \) is swept onto the suction surface not far from the leading edge \( \approx 15\%C_e \) and then lifts off, moving around the passage vortex, \( V_p \), and toward the trailing edge along S2s. The pressure side horseshoe vortex \( V_{ph} \), however, is accelerated by the traverse pressure gradient, displayed by the shear stress pattern formed between S2p and R5, toward the suction surface of the neighboring blade, and becomes part of the passage vortex separating along the suction surface (S4). Suction side corner vortices \( V_{sc} \) in fig. 3.16 are identified by the separation line S3, formed by the incoming endwall boundary layer meeting with the suction surface downstream of 40\%\( C_e \). Large shear stress is shown in this region. Pressure side corner vortices \( V_{pc} \) in fig. 3.16 can be detected by the reattachment line R5, where a new endwall boundary layer starts to develop and is driven to the neighboring blade.

On the suction side, the surface flow pattern is also complicated (fig. 3.6(b)). The suction leg of the horseshoe vortex \( V_{sh} \) separates along the separation line S2s as it lifts off from the endwall and moves toward the trailing edge. The trace of the passage vortex \( V_p \) including the pressure side horseshoe vortex \( V_{ph} \) is shown by S4 as it separates above the suction leg of horseshoe vortex from the surface, inducing large shear stress on the blade surface. Further downstream on the suction surface, the pattern under S4 is difficult to track, which could be caused by the wall vortex \( V_{wip} \) in fig. 3.16 induced by the passage vortex (Wang et al., 1997) or by the suction leg of the horseshoe vortex moving around the passage vortex. The sense of rotation for the passage vortex can be interpreted from the inclination of the streamline on the surface. A reattachment line R3 shows the existence of suction side corner vortex \( V_{sc} \) with an opposite sense of rotation with respect to the passage vortex. Beyond the S2, the surface flow is essentially two dimensional from the leading edge until it separates at S7, and later reattaching at R7 after transition.

The surface flow pattern on the pressure side (fig. 3.6(c)) displays a high shear stress region (light colored) close to the leading edge (away from endwall), followed by a low shear stress region (deep dark) induced by flow separation (S6). In this separation zone (0.05-0.15 \( S_p/C \)), the oil-particle trace is not clear, showing little movement of fluid particles. The reattachment line (R6) is also not very obvious in the surface flow pattern, and is possibly located somewhere downstream the dark separation region that displays relative higher shear stresses. Downstream of the separation and reattachment region, the flow is essentially two dimensional and follows the streamline till the trail edge. The existence of
leading edge corner vortex \((V_{pLe} \text{ in fig. 3.16})\) is obvious in the flow pattern. The incoming boundary layer obstructed by the blade leading edge becomes stagnant and induces a low shear region at the junction of the leading edge and the endwall. Further downstream, the interaction of the endwall boundary layer and the pressure surface boundary layer results in weak separation line R5 near the endwall, showing existence of a corner vortex \((V_{pc} \text{ in fig. 3.16})\).

The secondary flow system near the endwall interpreted from the present surface flow visualization is similar to those from previous studies (Sieverding (1985), Hodson and Dominy (1986), Goldstein and Spores (1988), Chung and Simon (1990), Wang et al. (1997)).

### 3.2.3 Surface flow on the tip-endwall with tip clearance

With the presence of tip clearance, the surface flow patterns are changed appreciably at different tip clearance levels.

The surface flow patterns on the tip-end wall at the tip clearance level of \(\tau/C = 0.86\%\) in fig. 3.7(a) is similar to that of without tip clearance. The incoming flow approaching the blade leading edge, splits into two parts at the saddle point \((A1)\), forming the horseshoe vortex system composed of pressure side leg \((V_{ph} \text{ separating along S2p})\) and suction side leg \((V_{sh} \text{ separating along S2s})\) as in the case of zero tip clearance. Similarly, the pressure side leg of the horseshoe vortex \((V_{ph})\) and the endwall boundary layer flow starting along AR are accelerated by the traverse pressure gradient toward the suction side of neighboring blade, as shown by the stress pattern on the surface. The suction leg of the horseshoe vortex \((V_{sh})\) lifts up around the 15\%\(C_r\) from the leading edge and moves around the passage vortex \((V_p)\) along the suction surface \((S2s)\). The other part of boundary layer flow starting along AR is accelerated into the tip clearance due to the pressure difference between the pressure and the suction sides. Large shear stresses exist on the endwall along the pressure edge of the tip apparently caused by the leakage flow passing through the narrow passage between the endwall and the separation bubble formed on the pressure edge of the tip surface. The leakage flow then separates on the endwall, forming the dark low stress streak along the pressure edge of the tip on the endwall, then it reattaches and passes through the clearance, and separates along LS, which coincides with the suction side edge from 15\% to 70\%\(C_r\), to become the tip leakage vortex \((V_d)\) before meeting the passage vortex \((V_p)\) separating along PS. The flow between LS and PS has relatively large shear stresses, and the leakage vortex separation line LS becomes weak and finally merges with the passage vortex separation line PS near the exit of the cascade, probably showing that the tip leakage vortex starting from about 15\%\(C_r\) is rather weak at this small clearance level.

At the tip clearance of 1.72\%\(C\) (fig. 3.7(b)), the surface flow pattern is close to that of 0.86\%\(C\), though the horseshoe vortex system (separation lines S2p and S2s) is more difficult to identify. It can be found that more boundary layer fluids on the endwall are sucked into the tip clearance along AR as it moves from near the pressure side in previous case to about one third of pitch into the passage from the pressure side. A wider high-shear stress zone exists near the pressure edge of the tip on the endwall, perhaps indicating a larger separation bubble on the tip along the pressure edge. Along the separation line on the endwall immediately after the high shear stress zone near the pressure edge of the tip,
there is a small accumulating spot around $0.75 C_{x, t}$, probably showing the existence of a low pressure zone there. The tip leakage vortex ($V_{\theta}$ separating along LS) apparently starts separating along the suction edge of the tip from around $0.2 C_{x, t}$, and the area between the leakage flow separation (LS) and the passage vortex separation (PS) widens as they move further downstream into the blade passage. It is interesting to see the leakage flow on the endwall opposite to the first half of the tip surface splits into several small branches after reattachment. The surface flow patterns at two small tip clearance levels are similar to the results from Sjölander and Amrud (1987) and Moore and Tilton (1988), which suggested that the separation on the endwall is caused by the adverse pressure gradient created by the convergent-divergent passage formed by the separation bubble on the tip pressure edge, and that the separation on the endwall is laminar due to the acceleration of the leakage flow into the clearance.

At $\tau/C = 3.45\%$ (fig. 3.7(c)), the horse vortex system is no longer evident. Part of the incoming boundary layer flow is driven into the tip clearance from the suction side near the leading edge, mixing with the boundary layer flow from the pressure side along AS. The dividing line AR moves further into the blade passage, indicating more endwall boundary layer flow is sucked into the pressure side of tip clearance. The high shear stress zone along the pressure edge of the tip becomes even larger and the leakage flow separates on the endwall as it gets through the clearance in several large groups around the mid-chord tip surface, to form a complicated flow pattern, before separating along LS into the tip leakage vortex ($V_{\theta}$). The leakage vortex separates near the suction edge around $0.3 C_{x, t}$ and soon deviates away from the suction edge into the blade passage, at the same time the separation of passage vortex ($V_p$ separating along PS) also moves further downstream in the blade passage, possibly indicating a weaker and underdeveloped passage vortex compared to the tip leakage vortex, since more fluids are sucked into the tip leakage flow to become the leakage vortex instead of the passage vortex.

### 3.2.4 Surface flow on the tip surface

For the surface flow pattern on the tip surface in fig. 3.8, we can find that a separation bubble exists immediately along the pressure edge of the tip at $\tau/C = 0.86\%$. Relatively larger shear stresses show up in the rest part of the tip surface, caused by the high speed shear flow inside the tip clearance. For the larger tip clearance of $1.72\% C$, this separation bubble increases in size as the separation line moves a little away from the pressure edge of the tip, and a small accumulation appears on the pressure edge near the mid-chord of tip surface, indicating a low pressure zone near the pressure side edge similar to that found by Bindon (1987). A second separation bubble is also evident on the suction side of tip starting from the leading edge up to $20\% C_{x, t}$. The leakage flow inside the tip clearance travels generally in the direction close to that of the incoming flow at two smallest tip clearances, and the leakage flow exits from the clearance after $X/C_{x, t} = 0.2$.

At the larger tip clearance of $3.45\% C$, low shear stresses and accumulation areas lie along the pressure side edge and near mid-chord respectively on the second half of the tip surface, showing perhaps the existence of separation exists near the pressure edge and reverse flows on the second half of the tip surface.
3.2.5 Surface flow on the blade near-tip surfaces

In fig. 3.9 on the suction side at small tip clearances \((\tau/C = 0.86\%, 1.72\%)\), the surface flow patterns are close to that of without tip clearance. Separation curves S2s and S4 due to the suction leg of the horseshoe vortex \(V_{hc}\) and the passage vortex \(V_p\) can be clearly identified. However, S4 starts later on the suction surface, perhaps indicating the late development of the passage vortex due to the presence of tip clearance. The separation and reattachment/transition zone (S7, R7) decreases with increase in tip clearance level. When compared with the case of zero tip clearance, a major difference is the disappearance of the corner vortex \(V_{cu}\) reattachment line (R3) at suction surface. It is replaced by the reattachment line of the leakage vortex \(V_{ul}\) along LR). For larger tip clearances \((3.45\%C \text{ and } 6.90\%C)\), however, the separation curves of the suction leg of the horseshoe vortex and the passage vortex are difficult to track and totally disappeared at the highest tip clearance level 6.90\%, at the same time it can be observed that the region affected of the tip leakage vortex \(V_{ul}\) grows with the increase in tip clearance level, from very close to the tip edge at small clearance level to almost one third of suction surface height at the largest clearance level, which suggests that the leakage vortex is strong and probably pushed the passage vortex \(V_p\) away from the suction surface. The tip leakage vortex \(V_{ul}\) flow patterns on the suction surface at the two largest tip clearances are similar to that described by Sjolander and Amrud (1987): the surface flow first deviates from the tip at 0.4\(C\) and then is driven toward the tip on the second half of the suction surface. This pattern is perhaps caused by the roll-up of the leakage vortex and could incorporate boundary layer flow on the suction surface into the leakage vortex at large tip clearance levels.

On the pressure surface in fig. 3.10, the separation (S6) and reattachment (R6) zone still exists at all four tip clearance levels. The flow inside the separation zone is difficult to interpret, and larger shear stresses can be observed in reattachment zone at larger tip clearance levels. Downstream of this zone and away from the tip, the surface flow is still essentially two dimensional. No corner vortex \(V_{ulCe} \text{ and } V_{pe}\) exists near the junction of the leading edge close to blade tip, as in the case of without tip clearance, even for the smallest tip clearance level of 0.86\%. The sink flow effect of tip clearance can be clearly seen by the traces of leakage flow accelerated toward the tip edge. The larger the tip clearance, the larger angle of the leakage trace to the tip edge, at almost right angle for the largest tip clearance \((\tau/C = 6.90\%)\), indicating that the boundary layer flow on the pressure surface can become part of the leakage flow. It can also be observed that at the two largest tip clearance levels, the flow is strongly accelerated from the separation zone near the tip into the tip clearance, which may cause high rates of local heat/mass transfer.

3.2.6 Smoke wire visualization

A multiple smoke wire composed of eight 0.1\(mm\) diameter 304 stainless steel is placed horizontally near the tip-endwall and just ahead of trip wire on the tip-endwall. Each smoke wire is attached to springs at two ends to ensure it is parallel to the endwall and allow extra expansion during heating. The distance of the smoke wires from the endwall are 3.2, 6.4, 12.7, 19.1, 25.4, 35.8 and 50.8 mm respectively.

A thin layer of light oil from Life-Line Products, INC is used to put on the wires and
generate traces of smoke when they are electrically heated by power from a variable voltage transformer. A laser sheet expanded from a He-Ne laser beam by a glass rod is used to illuminate the cross-sectional plane inside the turbine cascade. Different planes can be easily illuminated by manipulating the orientations and positions of the glass rod. In the study, the light sheet is often positioned at planes parallel to the leading edge of the cascade ($X/C_x = 0.15, 0.45$ and $0.90$) or a plane parallel to the incoming flow ($x/C \approx 0.45$).

For the smoke wire flow visualization, the mainstream flow speed must be low enough for easy-identifying the vortices on the illuminated plane but high enough to avoid the smoke traces being affected by buoyancy as well as the back flow from the exit of the wind tunnel and to obtain straight smoke traces. In this study, the inlet main flow speed is about $2.7m/s$, resulting a laminar flow on the tip-endwall with an exit Reynolds number of $7.0 \times 10^4$.

The vortex patterns produced the smoke wire are video-taped by a Panasonic camera at different viewing angles.

**Secondary flows and tip leakage vortex**

At zero tip clearance in fig. 3.11, the passage vortex $V_p$ develops soon after the flow enters the blade passage at $X/C_x = 0.15$ with a clockwise sense of rotation when looking in the direction of flow in fig. 3.11(b)-(e). The suction leg of the horseshoe vortex $V_{sh}$ can also be detected from the same figures. Though weak, it lies close to the suction surface but is lifted off from the endwall, with an opposite sense of rotation to the passage vortex. On the plane at $x/C = 0.45$ in fig. 3.11(g)-(j), the vortex system is complex - a series of vortices integrates to form one strong passage vortex ($V_p$). The distinction between the passage vortex and the suction leg of the horseshoe vortex is difficult to tell, probably due to the fact that the suction leg of the horseshoe vortex revolves around the passage vortex and could be considered as part of the larger passage vortex, as suggested by Wang et al. (1997). At position $X/C_x = 0.90$ in fig. 3.11(l)-(o), we can see the large passage vortex $V_p$ emerging from the blade passage near the trailing edge is strong and rotates in the counter-clock direction when looking upstream from the trailing edge.

At tip clearance level of $0.86\% C$ in fig. 3.12(b)-(e), the passage vortex $V_p$ is still very strong near the leading edge ($X/C_x = 0.15$) and the suction leg of the horseshoe vortex $V_{sh}$ can also be detected, while the leakage flow is difficult to see in the pictures. At the middle blade positions of $X/C_x = 0.45$ and $x/C = 0.45$ in fig. 3.12(f)-(o), the large passage vortex ($V_p$) rotating in the clockwise direction can be clearly identified, although the tip leakage vortex $V_{ld}$ is rather weak, some fluids emerging from the tip clearance at the suction side is appears to be incorporated into the passage vortex in the pictures.

For tip clearance level of $\tau/C = 1.72\%$, we can find the passage vortex $V_p$ coexists with the small amount of tip leakage vortex ($V_{ld}$) emerging from the suction side of the tip clearance at $X/C_x = 0.15$ in fig. 3.13(b)-(e). The leakage vortex has the same sense of rotation as the suction leg of the horseshoe vortex $V_{sh}$, and perhaps combines with the horseshoe vortex during the development, while the passage vortex remains strong and rotates in the opposite direction. In fig. 3.13(g)-(j) at $X/C_x = 0.45$, a relatively strong passage vortex still can be observed. Similar to that observed by Yamamoto (1988), the
weak leakage vortex $V_d$ from the suction side of the tip clearance is drawn into the large passage vortex, which eventually develops into one large passage vortex $V_p$ in pictures at position $x/C = 0.45$ in fig. 3.13(l)-(o).

At a higher tip clearance level of $\tau/C = 3.45\%$ in fig. 3.14(b)-(d), the passage vortex $V_p$ is not well developed at $X/C_x = 0.15$. In fig. 3.14(f)-(h), the passage vortex is much simpler and in a pattern quite different from those with small tip clearance at position $X/C_x = 0.45$. We can also observe that the tip leakage vortex $V_d$ becomes stronger and bends down in an opposite sense of rotation to that of the passage vortex, pushing the passage vortex away from the suction surface. At $X/C_x = 0.90$ in fig. 3.14(j)-(l), the much smaller passage vortex rotates under the tip leakage vortex coming out of the tip clearance with a different sense of rotation is displayed.

**Tip leakage flow inside large tip clearances**

Tip leakage flow inside the tip clearance for $\tau/C = 3.45\%$ is shown in fig. 3.15. Near the leading edge at $X/C_x = 0.15$, a separation bubble can be found near the pressure edge of the tip. Flow from the blade passage comes into the tip clearance from the suction side of blade and separates on the tip, as shown by the curved flow pattern near the suction side in the tip clearance (fig. 3.15(d)). At $X/C_x = 0.45$, a second much smaller separation bubble is formed near the suction edge of the tip, in addition to the separation bubble near the pressure edge on the tip surface as illustrated in fig. 3.15(h).

It is clear that the leakage flow inside the large tip clearance could be fairly complicated at large tip clearance levels. Near the leading edge, the leakage flow can come from either the pressure side or the suction side. Around the middle tip surface, a double counter-rotating vortex is developed. From previous sections on the surface flow visualization, we can also observe the reverse flow and accumulation on the second half of tip surface.

### 3.3 Summary

In this chapter, the flow field in the linear cascade test section is characterized and visualization. Some important results are summarized:

1. At normal operating conditions, the turbulent boundary layer on the tip-endwall has a displacement thickness of 0.24 cm (1.3\%C), which is typical in a gas turbine engine.

2. From the analysis of mid-span flow near the blade, it is clear that a separation region exists on the pressure surface from 0.05 to 0.15 $S_p/C$. Transition may occur on the suction side downstream of $S_x/C = 1.2$.

3. The unloading effect of tip clearance can be observed from the static pressure distribution on the near-tip surfaces. This effect becomes larger with the increase in tip clearance level, especially near the leading edge, where loading could be negative at the largest tip clearance.

4. From the surface flow visualization, it is found that the horseshoe vortex system still appears at the smallest tip clearance level. A separation bubble forms along the
pressure side edge on the tip and then separation occurs on the tip-endwall due to the divergent flow in the clearance. The leakage vortex separates from the endwall initially at about 20% $C_e$ along the suction edge and deviates from the suction edge further downstream at small tip clearances.

5. The suction flow effect can be clearly found on the pressure surface at all tip clearance levels. On the suction surface, the effect of passage vortex gradually disappears as the leakage vortex becomes stronger and pushes the underdeveloped passage vortex into the blade passage.

6. Separation bubble also exists along the suction edge on the tip close to the leading edge at large tip clearance levels. Incoming flow enters the tip clearance from suction side directly while the development of the passage vortex is delayed. The surface flow pattern on the second half of the tip surface at large tip clearance levels is complicated, including accumulation, separation and reverse flows.

7. From the smoke wire visualization, we can find that the leakage vortex is weak and probably incorporated into the passage vortex at small tip clearance levels.
Figure 3.1: Laminar boundary layer flow on the tip-endwall
Figure 3.2: Turbulent boundary layer on the tip-endwall
Figure 3.3: Turbulent boundary layer on the tip-endwall (grid turbulence)
(a) $C_{ps}$ at midspan
Figure 3.5: $C_{ps}$ near the blade tip for different tip clearances
Figure 3.6: Surface flow visualization without tip clearance

(e) Pressure side

(f) suction side

(g) Tip-endwall
Figure 3.7: Tip-endwall surface flow visualization
Figure 3.8: Tip surface flow visualization
Figure 3.9: Suction side surface flow visualization

\( \frac{U}{U_c} = 6.90\% \) \( (p) \)

\( \frac{U}{U_c} = 3.45\% \) \( (c) \)

\( \frac{U}{U_c} = 1.72\% \) \( (q) \)

\( \frac{U}{U_c} = 0.36\% \) \( (q) \)
Figure 3.10: Pressure side surface flow visualization

(a) $R/C = 0.86$

(b) $R/C = 1.72$

(c) $R/C = 3.45$

(d) $R/C = 6.30$
Figure 3.11: Secondary flows with no tip clearance
Figure 3.12: Tip leakage vortex and secondary flows at $\tau/C = 0.86\%$
Figure 3.13: Tip leakage vortex and secondary flows at $\tau/C = 1.72\%$
Figure 3.14: Tip leakage vortex and secondary flows at $\tau/C = 3.45\%$
Figure 3.15: Leakage flow inside the tip clearance at \( r/C = 3.45\% \).
Figure 3.16: Secondary flow model proposed by Wang (1997)
Chapter 4

Mass/Heat Transfer Measurement

In this chapter, the mass/heat transfer data from the turbine blade in the linear cascade with tip clearance are presented and analyzed in detail. The experiments are conducted using the heat/mass analogy and the naphthalene sublimation technique on the pressure, suction and tip surfaces of the turbine blade, at different clearance levels, mainstream Reynolds numbers, and turbulence intensities, respectively. For each case, at least two runs are performed to obtain repeatable data. Detailed information on the data selected to present in the chapter is listed in table 4.5.

4.1 Mass/Heat Transfer from Blade Surfaces (No Clearance)

Mass/heat transfer on the pressure and the suction sides of a turbine blade was studied by Wang (1997) in a linear cascade with a central passage configuration. In the present study, mass transfer is measured from the pressure and the suction sides of a similar high pressure turbine blade in the linear cascade with a central blade configuration. The results can serve as base cases for comparison with measurements at different tip clearance levels.

Two-dimensional mass transfer from blade surfaces

The spanwise averaged (i.e. in the z direction around the middle blade) mass transfer Sherwood number from the mid-span blade surfaces are plotted in 4.1 for different mainstream Reynolds numbers and turbulence intensities.

On the pressure surface (fig. 4.1(a)), the mass transfer rates decrease drastically from the leading edge up to the separation at about $S_p/C = 0.05$ for all cases. After the reattachment and perhaps transition for the high mainstream turbulence intensity case ($Tu=12.0\%$) at about $S_p/C = 0.15$, the average Sh number distribution becomes less smooth, especially at high mainstream Reynolds number, due to the existence of unsteady Taylor-Görtler vortices. Downstream of the reattachment, higher mass transfer rates are obtained at higher mainstream Reynolds number. The effect of high mainstream turbulence intensity is also obvious: earlier separation and much higher mass transfer rates after separation. The effects of mainstream Reynolds number and turbulence intensity on the turbulent transition near the trailing edge are, however, not clearly shown on the pressure side.
In fig. 4.1(b) on the suction surface, the mass transfer rates also descend fast from the leading edge. Further away from the leading edge, the higher mainstream Reynolds number causes higher mass transfer rates and earlier turbulent transition. It can be found that the high mainstream turbulence intensity induces much earlier transition to turbulence on the suction surface.

In fig. 4.2, the same Sh numbers are normalized by the square root of the mainstream exit Reynolds numbers, respectively. On both pressure and suction surfaces, the data for low mainstream turbulence intensity (Tu = 0.2%) but at different mainstream Reynolds numbers collapse to almost one curve before transition near the trailing edge. It clearly shows that the flows on the suction surface remains laminar from the leading edge until transition at about $S_p/C = 1.2$ for these cases, which agrees with the analysis of 2D flow near the blade done using Texstan. The high mainstream turbulence intensity flow of 12.0% causes earlier transition to turbulent flow on the suction surface. On the pressure surface, the flows are also laminar from the leading edge until the separation region around $S_p/C = 0.15 - 0.20$, downstream of which the reattached flows are affected by the unsteady Taylor-Gortler vortices for the low turbulence level cases.

In the same figures, data from Wang (1997) are also plotted for comparison. Same general trend is followed for all the data, which validates the present measurements.

**Unsteady/transitional mass transfer from blade surfaces**

To study the effect of unsteady flow near the mid-span on the pressure side caused by the Taylor-Gortler vortices, the local standard deviation of Sherwood numbers along the pressure surface are plotted for different cases in fig. 4.3(a). We can find high variations in Sh after reattachment at $S_p/C = 0.2$ for the cases at different mainstream Reynolds numbers, showing that the flows are rather unstable due to the Taylor-Gortler vortices. For the high mainstream turbulence intensity case, the Sh’ variation dies down from the leading edge and remains low afterwards, which indicates the flow becomes perhaps laminar near the leading edge and after separation and reattachment the Taylor-Gortler vortices do not appear in this case.

The effects of mainstream Reynolds number and turbulence intensity on the unsteady flow on the pressure surface can also be observed in fig. 4.4, where local Sh near the mid-span are plotted. The local distributions of Sh for the high mainstream turbulence case are always smooth until close to the trailing edge ($S_p/C > 1.0$). For other cases with low mainstream turbulence intensity but different Reynolds numbers, the fluctuations of local Sh numbers are apparent, and the higher the mainstream Reynolds number, the earlier occurrence of the fluctuation, and the more irregular the fluctuations become further downstream.

On the suction side, Sh’ are shown in fig. 4.3(b) along the blade surface in the curvilinear coordinate. For different mainstream Reynolds numbers, the variation becomes much higher near and after the transition ($S_s/C \approx 1.2$). However, it seems the high mainstream turbulence intensity does not have an obvious effect on the variation Sh number until very near the trailing edge. Local fluctuations near the mid-span on the suction surface are displayed in fig. 4.5. For the higher mainstream Reynolds number cases, the fluctuation in Sh starts earlier and becomes more irregular downstream. For all cases, similar results can
also be found in Wang (1997).

Three-dimensional mass transfer from near-endwall surfaces

The contour and surface plots of mass transfer Sherwood number on the pressure surface are shown in fig. 4.6 and fig. 4.7, respectively, for different mainstream Reynolds numbers and turbulence intensities. From these plots we can observe that the unsteady Taylor-Görtler vortices induce uneven mass transfer rates on the pressure surface away from the endwall for the low mainstream turbulence level (Tu=0.2%) cases with various Reynolds numbers. The higher mass transfer rates near the pressure surface-endwall junction can be attributed to the pressure side corner vortices \( V_{pc} \) and \( V_{pLe} \), of which the leading edge corner vortex generates a small semi-circular high mass transfer zone \( (S_p/C < 0.25) \) near the junction of the leading edge and the endwall. This high mass transfer zone increases in size at higher Reynolds number. For the high mainstream turbulence intensity (Tu=12.0%) case, the Taylor-Görtler vortices seem to disappear on the surface and the effects of corner vortices near the endwall are limited to the region very close to the endwall.

In fig. 4.8 and fig. 4.9, local \( Sh \) and \( Sh/Re_{ex}^{1/2} \) are plotted along the span starting from the endwall at different curvilinear locations along the pressure surface, for different Reynolds numbers and turbulence intensities respectively. In fig. 4.8, higher mass transfer rates are obtained at higher mainstream Reynolds numbers and the mass transfer rates descend quickly close to the leading edge \( (S_p/C < 0.04) \). Further downstream the effect of Reynolds number is not obvious. We can also observe that the effect of corner vortices starts from \( S_p/C = 0.04 \), where a maximum mass transfer rate occurs at the endwall \( (Z/C = 0.0) \) and a minimum mass transfer rate appears at about \( Z/C = 0.05 \). Upstream of \( S_p/C = 0.04 \), the trend is reversed, with minimum mass transfer at the endwall, perhaps due to the stagnant flow at the leading edge and the endwall junction. The Taylor-Görtler vortices induced unsmoothness away from the endwall can also be clearly seen. At the high turbulence level of 12%, the effects of both corner vortices and Taylor-Görtler vortices are not apparent. For the normalized \( Sh \) numbers in fig. 4.9, we can find that the curves for the low turbulence intensity cases collapse to almost one near the endwall. Higher mainstream turbulence intensity increases the mass transfer rates up to 100% from the lower turbulence cases downstream of \( S_p/C > 0.09 \).

For the suction surface, fig. 4.10 and fig. 4.11 shows the contour and surface plot of Sherwood numbers for different mainstream Reynolds numbers and turbulence intensities, respectively. The effect of secondary flows on the mass transfer near the endwall can be identified from the similar large triangular region of high mass transfer in all cases. This triangular region remains almost the same for all low turbulence cases with \( Tu = 0.2% \) but becomes smaller for the high mainstream turbulence case. A mass transfer peak around \( S_s/C = 0.4 - 0.5 \) at the junction of the suction surface and the endwall is perhaps induced by the interaction between the suction leg of the horseshoe vortex \( V_{sh} \) and the suction surface when the horseshoe vortex turns around the blade and lifts off from the endwall. It is also clear that the transition to turbulence becomes earlier as the mainstream turbulence intensity and Reynolds number increases.

The local \( Sh \) and \( Sh/Re_{ex}^{1/2} \) are plotted against the \( Z/C \) at different locations of \( S_s/C \) in
fig. 4.12 and fig. 4.13 for different cases, respectively. The mass transfer rates also decrease steadily from the leading edge, and for \( S_s/C < 0.09 \), the effect of mainstream Reynolds number is more evident in fig. 4.12. As mentioned earlier, the sharp peak at \( S_s/C = 0.41 \) and \( S_s/C = 0.50 \) at the endwall is caused by the suction leg of horseshoe vortex \( (V_{sh}) \), whose effect near the endwall starts from \( S_s/C = 0.09 \). Upstream of \( S_s/C = 0.09 \), the mass transfer has a minimum at the endwall, perhaps due to the stagnant flow near the leading edge. Downstream of \( S_s/C = 0.61 \), we can observe a second peak in the triangular zone of high mass transfer, this peak is probably caused by the passage vortex \( (V_p) \) separating from the suction surface. After \( S_s/C = 0.94 \), the second mass transfer peak becomes flat and a third one appear at \( S_s/C = 1.15 \), which probably shows the effects of the complex vortices including the suction leg of the horseshoe vortex \( (V_{sc}) \) and the wall vortex \( (V_{wip}) \). Also downstream of \( S_s/C = 0.94 \), mass transfer for the high turbulence case is much higher than those of low turbulence cases outside the triangular region due the earlier transition. In fig. 4.13, the normalized Sh for different Reynolds numbers collapse to one, except for the case with highest Reynolds number, which may have errors in measurement at \( S_s/C = 0.94 \) and \( S_s/C = 1.07 \). The effect of high mainstream turbulence level is only obvious close to the leading edge and downstream of \( S_s/C = 0.94 \) after transition.

The overall three-dimensional effects on mass transfer on the pressure surface and the suction surface are plotted in fig. 4.14 and fig. 4.15. We can see that from the figures the three-dimensional effect is not very obvious on the pressure surface for both low and high mainstream turbulence intensities, except around \( S_p/C = 0.2 \). On the suction surface, the three-dimensional effect are very strong, resulting in higher mass transfer rate from \( S_s/C = 0.5 \) up to transition for both low and high mainstream turbulence intensity cases. However, after the turbulent transition, the mass transfer rate goes down in the three-dimensional region for both cases.

4.2 Mass/Heat Transfer from the Near-Tip Blade Surfaces

Mass transfer measurements on the pressure and the suction surfaces near the tip clearance are conducted at four different tip clearance levels, and at different mainstream Reynolds numbers and turbulence intensities for the smallest tip clearance \( (\tau/C = 0.86\%) \) in the present study. The measurements are performed using the naphthalene sublimation techniques with LVDT probe in horizontal position and exposing the blade in the linear cascade with central blade configuration.

Pressure surface near the tip clearance

Effects of tip clearance level on the mass transfer on the pressure surface are shown in the contour and surface plots of fig. 4.16 and fig. 4.17, respectively. Similar to the results for with no tip clearance, the unsteady Taylor-Gortler vortices appear evidently on the pressure surface for all tip clearance levels except the largest one. However, the effect of corner vortices disappear near the tip edge. Instead, the relatively higher mass transfer region near the tip edge after \( S_p/C > 0.15 \) is caused by the leakage flow sucked into the tip clearance. This effect of acceleration is so strong that much higher and probably unstable mass transfer
rates are induced at the two largest tip clearance level at about $S_p/C = 0.15$ near the
tip edge and extended further away from the tip in the span direction at $\tau/C = 6.90\%$, which can also be observed from the surface flow visualization in fig. 3.10(d). This strong
acceleration perhaps also causes the disappearance of the Taylor-Gortler vortices on the
pressure surface at the largest tip clearance level.

Local $Sh$ and $Sh/Re_{ex}^{1/2}$ at different $S_p/C$ locations starting from the tip in the span
direction are plotted in fig. 4.18 and fig. 4.19, respectively, for different tip clearance levels.
The trend of mass transfer near the tip on the pressure surface is quite different for the case with no tip clearance: the mass transfer rates have maximum at the tip and decrease gradually in the span direction for all non-zero tip clearances in fig. 4.18. However, the effect of different tip clearance on the mass transfer near the tip is not very strong upstream of $S_p/C = 0.09$. Between $S_p/C = 0.09$ and $S_p/C = 0.48$, the two largest tip clearance levels generate much higher and highly fluctuating mass transfer rates near the tip than the two
smaller tip clearance levels. Further downstream of $S_p/C = 0.48$, the mass transfer rates near the tip increase as tip clearance level becomes larger and is much higher than the case
with no tip clearance, though the effect of tip clearance is limited to the region close to the
tip edge and generally within $Z/C < 0.15$. For the normalized Re plots, this trend can also be seen clearly in fig. 4.19. For $S_p/C > 0.48$, the mass transfer curves tend to collapse to one after $Z/C = 0.1$ disregarding the effect of the Taylor-Gortler vortices. Generally, the
effect of tip clearance is limited to $Z/C < 0.1$ for the small tip clearance levels.

Spanwise and streamwise averaged $Sh$ on the pressure surface are plotted in fig. 4.20 for
different tip clearance levels. From fig. 4.20(a), we can find that the effect of tip clearance is
evident only for the two largest tip clearance around $S_p/C = 0.2$. As suggested earlier, it
is caused by the highly fluctuated suction flow at the larger tip clearances. The variation of streamwise averaged $Sh$ vs. $Z/C$ in fig. 4.20(b) shows the effect of tip clearance. For the
case without tip clearance, the minimum mass transfer rate around $Z/C = 0.05$ is induced by the corner vertices ($V_{te}$), while the mass transfer rate increases monotonically toward the
tip for all tip clearance cases. For the largest the tip clearance of $6.90\%$, the fluctuating $Sh$, caused by the Taylor-Gortler vortices for smaller tip clearance cases, disappears due to
the existence of highly accelerated suction flow on the pressure surface.

At the same tip clearance level of $\tau/C = 0.86\%$, the effects of mainstream Reynolds
number and turbulence intensity on the mass transfer on the pressure surface are displayed in
contour and surface plots of fig. 4.21 and fig. 4.22, respectively. Similar to the results
without tip clearance, the existence of Taylor-Gortler vortices can be found for the low
mainstream turbulence cases while for the high mainstream turbulence case, the effect of unsteady vortices disappears. High mass transfer region near the tip edge can be observed and the mass transfer rates in this region increase with the increase of mainstream Reynolds
number. The high mainstream turbulence seems not to greatly affect this relatively high
mass transfer region near the tip.

The local mass transfer $Sh$ against $Z/C$ at different curvilinear locations on the blade
are shown in fig. 4.23. Near the leading edge ($S_s/C < 0.04$), the higher the mainstream
Reynolds number, the higher mass transfer rate can be obtained. Further downstream, the
effect of Reynolds number is not very obvious. However, the high mainstream turbulence
level increases the mass transfer rate up 100\% at $S_s/C = 0.15$. The same trend is revealed
by normalized Sh plots in fig. 4.24. Mass transfer curves for different Reynolds numbers at low turbulence level collapse further downstream of the leading edge, with the unsmoothness caused by the unsteady Taylor-Görtler vortices. The effect of leakage flow sucked into the tip clearance is limited well within $Z/C < 0.1$ for all case and at most two times as large in Sh as that near mid-span. The effect of mainstream turbulence level is also plotted in fig. 4.25. We can find that at high turbulence level, higher mass transfer rates are obtained with tip clearance than that of without.

Spanwise and streamwise averaged Sh on the pressure surface are shown in fig. 4.26 at the same tip clearance of 0.86% $C$ for different Reynolds numbers and mainstream turbulence intensities. In fig. 4.26(a), we can clearly see that higher mainstream Reynolds number and turbulence intensity cause high mass transfer on the pressure surface in streamwise direction. The same effects can also be found in in fig. 4.26(b) for streamwise averaged Sh vs. $Z/C$. The high mainstream turbulence also totally eliminates the Taylor-Görtler vortices on the pressure surface.

The present measurements on the effects of tip leakage flow on the mass/heat transfer on the pressure surface is different from the results from Metzger and Russ (1989)'s sink flow measurements, in which higher heat transfer rates (up to 100% near the edge) were found to extend to 30% of the span. In present study, the effect of leakage sink flow on mass transfer is found to limited with 10% of the chord at small clearances.

Suction surface near the tip clearance

Effects of tip clearance level on the mass transfer on the suction surface are plotted in fig. 4.27 and fig. 4.28 in the form of contour and surface, respectively. From the plots, we can see that the triangular high mass transfer region caused by the secondary flows decreases in size as the tip clearance level increases. At tip clearance of 0.86% $C$, the tip leakage vortex exiting from the suction side of the tip clearance produces a small high mass transfer zone centered around $S_t/C = 0.7$ close to the tip edge. The effects of secondary flows still exist near the edge of the triangular region and seem to be pushed away from the tip edge. For the tip clearance of $r/C = 1.72\%$, the region affected by the tip leakage vortex extends both downstream and in the span direction with its peak at $S_t/C = 0.5$ very close to the tip edge, while the region affected by the secondary flows becomes smaller, though still detectable. At higher tip clearance of 3.45% $C$, the effect of secondary flows disappears and the smaller triangular region is mainly affected by the tip leakage vortex with two peaks of high mass transfer, probably indicating multiple tip leakage vortices.

Local Sh and $Sh/R^1/2$ at different $S_t/C$ locations along the span are shown in fig. 4.29 and fig. 4.30, respectively, at different tip clearance levels. As can be found in fig. 4.29, the effect of tip clearance on the mass transfer on the suction surface is not evident till $S_t/C = 0.18$ and the mass transfer distribution near the tip edge are rather flat, though higher than that with no tip clearance, indicating almost no existence of leakage vortex near the leading edge of the blade. Further downstream, the effect of tip clearance can be seen that the smaller tip clearance induces larger mass transfer rate near the tip. This peak near the tip edge is caused by the tip leakage flow reattaches on the surface and the second peak occurs at $S_t/C = 0.83 - 1.15$ for the two smaller tip clearance levels probably indicates the

57
passage vortex separating from the surface, pushed away by the tip leakage vortex. For the
two larger tip clearances this second peak is not evident and the triangular region become
smaller as the effect of secondary flows disappears. The same trend can be observed for the
normalized Re in fig. 4.30, where the mass transfer data collapse outside of the triangular
region at all streamwise locations.

Spanwise and streamwise averaged Sh on the suction surface are plotted in fig. 4.31 for
different tip clearance levels. In fig. 4.31(a), the effect of tip clearance is evident: the smaller
the tip clearance, the higher Sh for $S_s/C > 0.5$. The variation of streamwise averaged Sh vs.
Z/C in fig. 4.31(b) clearly shows the effect of tip leakage vortex ($V_l$) especially at smallest
tip clearance of 0.86%C.

At the same tip clearance level of $\tau/C = 0.86\%$, the effects of mainstream Reynolds
number and turbulence intensity on the mass transfer on the suction surface can be seen
as contour and surface plots in fig. 4.32 and fig. 4.33, respectively. We can find that the
high mass transfer region caused by the tip leakage vortex increase in size at the highest
mainstream Reynolds number ($Re_{ex} = 7.18 \times 10^5$), while the effect of secondary flow as
well as the size of the triangular high mass transfer region decrease at the high mainstream
turbulence level ($Tu = 12\%$). The high mainstream turbulence level also causes the earlier
transition on the suction surface away from the tip.

The local mass transfer Sh vs. Z/C at different curvilinear locations is shown in fig. 4.34.
We can find that near the leading edge ($S_s/C < 0.09$), the higher the mainstream Reynolds
number, the higher mass transfer rate can be obtained, and the leakage flow has almost no
effect on the mass transfer near the tip. Downstream of $S_s/C = 0.09$, the effect of Reynolds
number is not quite evident outside the triangular region. The mass transfer curves in the
triangular region follow the same trend at different Reynolds numbers: the first peak near
the tip is caused by the leakage vortex reattachment while the second peak is probably
induced by the passage vortex separation from the surface. However, the effect of high
mainstream turbulence level on the mass transfer rate is not important until $S_s/C = 0.94$,
where transition to turbulence begins. A very similar trend is revealed by the normalized
Sh plot in fig. 4.35. Mass transfer curves for different Reynolds numbers at low turbulence
level collapse downstream of the leading edge and outside the triangular region. The effect
of mainstream turbulence level is also plotted in fig. 4.36. Again the high turbulence level
can effectively reduce the size and mass transfer peak of the triangular region of high mass
transfer.

Spanwise and streamwise averaged Sh on the suction surface are displayed in fig. 4.37 at
the same tip clearance of 0.86%C for different Reynolds numbers and mainstream turbulence
intensities. In fig. 4.37(a), we can find that mass transfer rates increase with mainstream
Reynolds number while high turbulence at the same exit Reynolds number doesn’t cause
high mass transfer on the suction surface in streamwise direction. From fig. 4.37(b) for
streamwise averaged Sh vs. Z/C, the high mainstream turbulence induces much higher Sh
for $Z/C > 0.4$ due to the early turbulent transition.
4.3 Mass/Heat Transfer from the Tip Surface

In the present study, mass transfer measurements on tip surfaces are also obtained at four different tip clearance levels, and at different mainstream Reynolds numbers and turbulence intensities for $\tau/C = 0.86\%$. The measurements are conducted in the blade chord coordinates showing in fig. 4.38(a) with the LVDT gauge in vertical position. The dots in the figure are positions the measurements take place and they are aligned in lines parallel to the $y/C$ direction. In the following sections, the local distributions of Sherwood number are presented in the blade chord coordinates starting from the beginning of the naphthalene surface ($y_o$). For contour plots, the blade chord coordinate is converted to the axial coordinate, as shown in 4.38(b). As we can see, in the axial coordinates the measurement lines are close to the direction of incoming flow that is at $35^\circ$ to the axis.

4.3.1 Effect of tip clearance level

Effects of tip clearance level on the mass transfer on the tip surface are shown in the contour plots of fig. 4.39 and fig. 4.40. At tip clearance of $\tau/C = 0.86\%$, high mass transfer rates occur immediately at the pressure edge of the tip surface, especially near the mid-chord of the surface, where the Sh number descends from a value of 10000 at the pressure side to about 3500 near mid-chord of the tip. For the tip clearance of 1.72$\%$, high mass transfer rates still exist on the second half of the tip surface with peak values at a small distance downstream of the pressure edge, and near the leading edge. These peaks are resulted from the reattachment of the leakage flow after separation bubble. A large low mass transfer zone also appears on the first half of the tip surface near the leading edge and close to the suction side, which is probably caused by the low rate of leakage flow passing through that area as shown from the surface flow visualization the leakage vortex separates from the tip-endwall around $X/C_x = 0.2$. There are small peaks of high mass transfer on the first half of the tip and near the pressure edge, inducing irregular mass transfer patterns.

At higher tip clearance of $\tau/C = 3.45\%$, the high mass transfer region remains near the pressure edge close to the leading edge, while the low mass transfer region increases in size on the first half of the tip surface, probably induced by the leakage flow coming from the suction side at this relatively large tip clearance level. Relatively low mass transfer rates exist on the second half of the tip surface due to the accumulative and perhaps reverse flows occurring there. For the largest tip clearance of 6.90$\%$, the high mass transfer region appears along the pressure and the suction edge of the tip surface close to the leading edge. At the mid-chord surface and in the second half of the tip surface, low mass transfer is caused by the reverse flow. The low mass transfer region affected by the leakage flow from the suction side become larger in size on the first half of the tip surface.

Local Sh at different $x/C$ locations along $(y - y_o)/C$ are plotted in fig. 4.41 and fig. 4.42, at different tip clearance levels. For $\tau/C = 0.86\%$, maximum mass transfer rates always occur at the start of the naphthalene surface $(y-y_o = 0)$, very close to the pressure side edge for all $x/C$ locations, showing the leakage flow separates and reattaches on the tip surface immediately at the pressure side edge of the tip. At tip clearance level of 1.72$, mass transfer peaks exist at a small distance ($\approx 0.02C$) from the start of naphthalene surface for all $x/C$ locations, probably indicating the leakage flow reattaches over a larger separation
bubble close to the pressure side edge of the tip. For the two largest tip clearances, the peaks of mass transfer Sh move further away from the pressure side edge and toward the suction side upstream of $x/C = 0.41$, showing a even larger separation bubble on the first half of the tip surface. The reason for a valley of Sh occurring around $0.05(y - y_o)/C$ from $x/C = 0.13$ to 0.33 for the largest tip clearance is not very clear. It may have something to do with the interaction between the leakage flow from suction side and the pressure side at larger tip clearance levels. After $x/C = 0.41$, the separation bubble is perhaps large enough to cover the tip thickness, leading to the accumulation or even reverse flow on second half of the tip surface. In fact, the mass transfer patterns for the two largest tip clearances are changed after $x/C = 0.41$, with peaks of mass transfer near the suction side edge of the tip, showing the effect of reverse flow.

Local Sh at different $x/C$ locations are plotted in fig. 4.43 and fig. 4.44 against $(y - y_o)/\tau$, at different tip clearance levels. For $x/C < 0.43$, we can clearly observe that the leakage flow reattaches at about $2 - 3\tau$ from the beginning of the naphthalene surface for tip clearance levels 0.86-3.45%C, the smaller the clearance level, the higher the peak value. For the largest tip clearance of 6.90%C, the reattachment occurs closer to the pressure edge. After $x/C = 0.43$, we can see that the reattachment of the leakage lies at about $2\tau$ for almost all $x/C$ locations and at all clearance levels. For the smallest tip clearance of 0.86%C, the entrance flow effect on the mass transfer is quite clear, with the maximum at the leakage flow entrance 3-4 times as high as that of further downstream. For other larger clearances, this entrance flow effect is also evident but less effective. It also shows for the two largest tip clearances, the separation bubble is so big in the second half of the tip surface that it covers the whole tip. The second peaks at large $(y - y_o)/\tau$ at larger tip clearance level for $x/C < 0.21$ are probably caused by the second separation bubble on the suction side near the leading edge, as shown in the flow visualization.

The averaged Sherwood number along $x/C$ for different tip clearances is plotted in fig. 4.50(a). We can see that the averaged mass transfer rate for $\tau/C = 1.72$% is higher than that of $\tau/C = 0.86$%. Similar result was also obtained by Teng et al. (2000) and Azad et al. (2000) and can be explained by the combined effects of shear flow and total leakage flow rate inside the tip clearance. For the two largest tip clearance, the first peak near the leading edge is due to the high mass transfer on the pressure and the suction edge of the tip, while the valley and peak around $x/C = 0.5 - 0.65$ is caused by the separation and reverse flow. The peaks around $x/C = 0.4$ for all cases are caused by the in accurate measurement near the injection hole of the naphthalene casting.

4.3.2 Effect of mainstream stream Reynolds number and turbulence

At the same tip clearance level of $\tau/C = 0.86$%, the effects of mainstream Reynolds number and turbulence intensity on the mass transfer on the tip surface are displayed in contour plots fig. 4.4 and 4.45. The mass transfer pattern is quite similar for all Reynolds number cases, with higher mainstream Reynolds number resulting in higher mass transfer rates. The high mainstream turbulence intensity seems not affecting the mass transfer pattern very much but reduces the local mass transfer rate.

The local mass transfer Sh along the span at different $x/C$ locations is shown in fig. 4.46
and fig. 4.47 in terms of $(y - y_0)/\tau$. The entrance flow characteristics is obvious at this smallest tip clearance levels. The figures show that the mass transfer Sh generally increases with increasing mainstream Reynolds number, while the high mainstream turbulence intensity leads to lower mass transfer rate compared to that of lower turbulence level at a similar Reynolds number. At low mainstream Reynolds number ($Re_{ex} = 4.65 \times 10^5$ or high mainstream turbulence level ($Tu=12\%$), the separation bubble occurring close to the pressure edge of the tip is apparently larger and the peaks of mass transfer locate at a small distance downstream of the start of naphthalene surface on the tip. In the normalized local Sh plots in fig. 4.48 and fig. 4.49, mass transfer Sherwood numbers over $Re_{ex}^{0.8}$ collapse to one curve very well for all cases with different Reynolds numbers at almost all $x/C$ locations, which suggest that though the leakage flow may separate laminarly due to the large acceleration rate at inlet of the clearance, the leakage flow after reattachment is turbulent inside the small tip clearance of 0.86%C.

Averaged Sherwood number along $x/C$ for different Reynolds numbers and turbulence levels is shown in fig. 4.50(b). As stated earlier, the higher mainstream Reynolds number, the higher averaged mass transfer rates on the tip surface. The high mainstream turbulence intensity suppress the mass transfer rate on the tip surface compared with that of lower turbulence levels at similar mainstream exit Reynolds number.

### 4.4 Summary

The local mass transfer measurement from the blade near-tip and the tip surfaces are presented in previous sections of this chapter for different tip clearance levels, mainstream Reynolds numbers and turbulence intensities. To obtain an overall sense of those effects, the surface averaged mass transfer Sh are presented in table 4.1 and table 4.4. From the first table, we can see that surface averaged mass transfer Sh is about four and six times as high on the tip surface as on the suction and the pressure surface respectively. A similar trend can also be found in table 4.4 for different mainstream Reynolds numbers and turbulence intensities.

Some most recent data on the tip surface heat transfer from Azad et al. (2000) and Bunker et al. (2000) are also listed in table 4.2 and table 4.3 respectively. The heat transfer coefficient data are averaged from the curves and contour plots from the above papers, and then converted into Nu number based on their blade chord. For both cases, much higher Reynolds numbers and turbulence intensities are employed in their experiments. Considering the effect of Reynolds number and a conversion factor for $(Sc/Pr)^{0.33}$ for turbulent flow, those results are comparable to the mass transfer Sh numbers on the tip surface for $\tau/C = 0.86\%$ and 1.72%. This shows the validity of the present experimental study.

The important results from the mass/heat transfer experiments are summarized as following:

1. The spanwise averaged mass transfer Sh on the mid-span blade surfaces are validated by comparing with previous results. The mainstream Reynolds number and turbulence intensity effects are also characterized by the mid-span measurements with no
tip clearance.

2. The effects of the corner vortices on the mass transfer near the endwall of the pressure surface with no tip clearance, as well as the effects of the secondary flows on the mass transfer near the endwall on the suction surface with no tip clearance, are identified and analyzed.

3. The effect of tip clearance on the mass transfer on the pressure surface is limited to 10% at smaller tip clearances, while at the largest tip clearance high mass transfer rates are induced at 15% $S_p$ by the strong acceleration of the fluids into the clearance.

4. The effect of tip clearance on the mass transfer is not very evident for $S_p/C < 0.20$ on the suction surface. However, much higher mass transfer rates are caused at $S_p/C = 0.60 - 1.0$ by the tip leakage vortex at the smallest tip clearance level. At the largest tip clearance level, the high mass transfer region becomes smaller, probably because the strong leakage vortex pushes the passage vortex away from the suction surface.

5. The entrance flow effect induces mass transfer rates 3-4 times as large along the pressure edge of the tip surface at the smallest tip clearance level of 0.86% $C$. It is also found that the leakage flow after the reattachment is turbulent. As the tip clearance level increase the peaks of the mass transfer move away from the pressure edge due to the increase of separation bubble size near the pressure edge. At the two largest tip clearances, the separation bubble could cover the whole width of the tip on the second half of the tip surface. A low mass transfer region on the first half of the tip is caused by low leakage flow rate at small tip clearance levels, but by leakage flow from suction side at large tip clearances. The average mass transfer rate is highest at the tip clearance level of 1.72% $C$.

6. The high mainstream turbulence level of 12.0% reduces the local as well as averaged mass transfer rates on the near-tip and the tip surfaces, while the higher mainstream Reynolds number generates higher local and average mass transfer rates on the near-tip and the tip surfaces. The Reynolds number and mainstream turbulence intensity also affect the entrance flow at the smallest tip clearance level by changing the separation bubble size and position of reattachment on the pressure side edge of the tip.
Table 4.1: Surface averaged Sh measured at different tip clearance levels ($Re_{ex} = 5.8 \times 10^5, Tu=0.2\%$)

<table>
<thead>
<tr>
<th>$\tau/C(%)$</th>
<th>0.00</th>
<th>0.86</th>
<th>1.72</th>
<th>3.45</th>
<th>6.90</th>
</tr>
</thead>
<tbody>
<tr>
<td>pressure</td>
<td>645.4</td>
<td>662.1</td>
<td>727.4</td>
<td>746.7</td>
<td>828.3</td>
</tr>
<tr>
<td>suction</td>
<td>1067.1</td>
<td>1169.1</td>
<td>1120.1</td>
<td>1036.4</td>
<td>764.2</td>
</tr>
<tr>
<td>tip surface</td>
<td>-</td>
<td>4341.3</td>
<td>4835.5</td>
<td>3673.7</td>
<td>3526.5</td>
</tr>
</tbody>
</table>

Table 4.2: Tip surface averaged Nu, measurement from Azad et al. (2000) ($Re_{ex} = 1.1 \times 10^6, Tu=6.1\%$)

<table>
<thead>
<tr>
<th>$\tau/C(%)$</th>
<th>1.0</th>
<th>1.5</th>
<th>2.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\tau/H(%)$</td>
<td>0.98</td>
<td>1.46</td>
<td>2.44</td>
</tr>
<tr>
<td>heat transfer coeff.</td>
<td>850</td>
<td>970</td>
<td>1080</td>
</tr>
<tr>
<td>Nu (C=12.5cm)</td>
<td>4071</td>
<td>4646</td>
<td>5172</td>
</tr>
</tbody>
</table>

Table 4.3: Tip surface averaged Nu, measurement from Bunker et al. (2000) ($Re_{ex} = 2.57 \times 10^6, Tu=9\%$)

| $\tau/C(\%)$ | 0.85 (1.27mm) | 1.3 (2.03mm sharp-edge) | 1.8 (2.79mm)
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>heat transfer coeff.</td>
<td>875</td>
<td>975</td>
<td>1050</td>
</tr>
<tr>
<td>Nu (C=14.9cm)</td>
<td>4995</td>
<td>5566</td>
<td>5994</td>
</tr>
</tbody>
</table>

Table 4.4: Surface averaged Sh at $\tau/C = 0.86\%$

<table>
<thead>
<tr>
<th>$Re_{ex} \times 10^{-3}$</th>
<th>4.5</th>
<th>5.8</th>
<th>7.1</th>
<th>5.6 (Tu=12.0%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>pressure (no tip)</td>
<td>557.9</td>
<td>645.4</td>
<td>782.2</td>
<td>961.8</td>
</tr>
<tr>
<td>pressure (with tip)</td>
<td>555.5</td>
<td>662.1</td>
<td>807.6</td>
<td>918.6</td>
</tr>
<tr>
<td>suction (no tip)</td>
<td>906.7</td>
<td>1067.1</td>
<td>1271.2</td>
<td>1317.5</td>
</tr>
<tr>
<td>suction (with tip)</td>
<td>915.7</td>
<td>1169.1</td>
<td>1421.7</td>
<td>1378.1</td>
</tr>
<tr>
<td>tip surface</td>
<td>3628.1</td>
<td>4341.3</td>
<td>5081.0</td>
<td>3890.3</td>
</tr>
<tr>
<td>$\tau/C$ (%)</td>
<td>Test No.</td>
<td>Surface</td>
<td>$Re_{ex} \times 10^{-3}$</td>
<td>Tu (%)</td>
</tr>
<tr>
<td>-------------</td>
<td>----------</td>
<td>---------</td>
<td>-----------------</td>
<td>-------</td>
</tr>
<tr>
<td>0.00</td>
<td>Prun11</td>
<td>pressure</td>
<td>4.36</td>
<td>0.2</td>
</tr>
<tr>
<td>0.00</td>
<td>Prun09</td>
<td>pressure</td>
<td>5.75</td>
<td>0.2</td>
</tr>
<tr>
<td>0.00</td>
<td>Prun15</td>
<td>pressure</td>
<td>7.01</td>
<td>0.2</td>
</tr>
<tr>
<td>0.00</td>
<td>Prun39</td>
<td>pressure</td>
<td>5.79</td>
<td>12.0</td>
</tr>
<tr>
<td>0.86</td>
<td>Prun32</td>
<td>pressure</td>
<td>4.79</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Prun31</td>
<td>pressure</td>
<td>5.95</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Prun34</td>
<td>pressure</td>
<td>7.17</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Prun42</td>
<td>pressure</td>
<td>5.33</td>
<td>12.0</td>
</tr>
<tr>
<td>1.72</td>
<td>Prun27</td>
<td>pressure</td>
<td>5.89</td>
<td>0.2</td>
</tr>
<tr>
<td>3.45</td>
<td>Prun18</td>
<td>pressure</td>
<td>5.56</td>
<td>0.2</td>
</tr>
<tr>
<td>6.90</td>
<td>Prun16</td>
<td>pressure</td>
<td>5.50</td>
<td>0.2</td>
</tr>
<tr>
<td>0.00</td>
<td>Srun11</td>
<td>suction</td>
<td>4.36</td>
<td>0.2</td>
</tr>
<tr>
<td>0.00</td>
<td>Srun09</td>
<td>suction</td>
<td>5.75</td>
<td>0.2</td>
</tr>
<tr>
<td>0.00</td>
<td>Srun14</td>
<td>suction</td>
<td>7.05</td>
<td>0.2</td>
</tr>
<tr>
<td>0.00</td>
<td>Srun40</td>
<td>suction</td>
<td>5.72</td>
<td>12.0</td>
</tr>
<tr>
<td>0.86</td>
<td>Srun33</td>
<td>suction</td>
<td>4.44</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Srun31</td>
<td>suction</td>
<td>5.95</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Srun35</td>
<td>suction</td>
<td>7.18</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Srun41</td>
<td>suction</td>
<td>5.55</td>
<td>12.0</td>
</tr>
<tr>
<td>1.72</td>
<td>Srun23</td>
<td>suction</td>
<td>5.77</td>
<td>0.2</td>
</tr>
<tr>
<td>3.45</td>
<td>Srun29</td>
<td>suction</td>
<td>5.79</td>
<td>0.2</td>
</tr>
<tr>
<td>6.90</td>
<td>Srun16</td>
<td>suction</td>
<td>5.50</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Trun47</td>
<td>tip</td>
<td>4.65</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Trun51</td>
<td>tip</td>
<td>5.88</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Trun50</td>
<td>tip</td>
<td>7.32</td>
<td>0.2</td>
</tr>
<tr>
<td>0.86</td>
<td>Trun44</td>
<td>tip</td>
<td>5.37</td>
<td>12.0</td>
</tr>
<tr>
<td>1.72</td>
<td>Trun53</td>
<td>tip</td>
<td>6.04</td>
<td>0.2</td>
</tr>
<tr>
<td>3.45</td>
<td>Trun55</td>
<td>tip</td>
<td>5.83</td>
<td>0.2</td>
</tr>
<tr>
<td>6.90</td>
<td>Trun56</td>
<td>tip</td>
<td>5.54</td>
<td>0.2</td>
</tr>
</tbody>
</table>
Figure 4.1: Mid-span average Sh

65
Figure 4.2: Mid-span average $Sh/Re^{1/2}$
Figure 4.3: Midspan RMS Sh'
Figure 4.4: Local Sh on the mid-span pressure surface
Figure 4.5: Local Sh on the mid-span suction surface
(a) $Re_{xx} = 4.36 \times 10^5$, Tu=0.2%

(b) $Re_{xx} = 5.75 \times 10^5$, Tu=0.2%

(c) $Re_{xx} = 7.01 \times 10^5$, Tu=0.2%

(d) $Re_{xx} = 5.79 \times 10^5$, Tu=12.0%

Figure 4.6: Sh contour on the pressure surface with no tip clearance
Figure 4.7: SH surface plot on the pressure surface with no tip clearance

(a) $Re_x = 4.36 \times 10^5$, $Tu=0.2\%$

(b) $Re_x = 5.75 \times 10^5$, $Tu=0.2\%$

(c) $Re_x = 7.01 \times 10^5$, $Tu=0.2\%$
Figure 4.8: Local Sh on the pressure surface with no tip clearance
Figure 4.9: Local $Sh/Re_{ex}^{1/2}$ on the pressure surface with no tip clearance
Figure 4.10: Sh contour on the suction surface with no tip clearance
(a) $Re_{e_x} = 4.36 \times 10^5$, Tu=0.2%

(b) $Re_{e_x} = 5.75 \times 10^5$, Tu=0.2%

(c) $Re_{e_x} = 7.05 \times 10^5$, Tu=0.2%

(d) $Re_{e_x} = 5.72 \times 10^5$, Tu=12.0%

Figure 4.11: Sh surface plot on the suction surface with no tip clearance
Figure 4.12: Local Sh on the suction surface with no tip clearance
Figure 4.13: Local $Sh/Re_{ex}^{1/2}$ on the suction surface with no tip clearance
Figure 4.14: Spanwise average Sh in 3D region
Figure 4.15: Spanwise average $Sh/Re_{ex}^{1/2}$ in 3D region
Figure 4.16: Sh contour on the pressure surface at different tip clearance levels

(a) $\tau/C = 3.45$

(b) $\tau/C = 1.72$

(c) $\tau/C = 0.86$

(d) $\tau/C = 6.99$
Figure 4.17: Sh surface plot on the pressure surface at different tip clearance levels.
Figure 4.18: Local Sh on the pressure surface at different tip clearance levels
Figure 4.19: Local $Sh/Re_{ex}^{1/2}$ on the pressure surface at different tip clearance levels.
Figure 4.20: Average Sh on the pressure surface at different tip clearance levels
Figure 4.21: Sh contour on the pressure surface at $\tau/C = 0.86\%$

(a) $Re_x = 4.79 \times 10^5$, $Tu=0.2\%$
(b) $Re_x = 5.95 \times 10^5$, $Tu=0.2\%$
(c) $Re_x = 7.17 \times 10^5$, $Tu=0.2\%$
(d) $Re_x = 5.33 \times 10^5$, $Tu=12.0\%$
Figure 4.22: Sh surface plot on the pressure surface at $\tau/C = 0.86\%$
Figure 4.23: Local Sh on the pressure surface at \( \tau/C = 0.86\% \)
Figure 4.24: Local $Sh/Re_{ex}^{1/2}$ on the pressure surface at $\tau/C = 0.86\%$
Figure 4.25: Local $Sh/Re_{ex}^{1/2}$ on the pressure surface: effect of turbulence
Figure 4.26: Average Sh on the pressure surface at $\tau/C = 0.86\%$
Figure 4.27: Suction contour on the suction surface at different tip clearance levels.

(a) \( \tau/C = 0.86\% \)
(b) \( \tau/C = 1.72\% \)
(c) \( \tau/C = 3.45\% \)
(d) \( \tau/C = 6.90\% \)
Figure 4.28: Sh surface plot on the suction surface at different tip clearance levels
Figure 4.29: Local Sh on the suction surface at different tip clearance levels
Figure 4.30: Local $Sh/Re_{ex}^{1/2}$ on the suction surface at different tip clearance levels
Figure 4.31: Average Sh on the suction surface at different tip clearance levels
(a) $Re_x = 4.44 \times 10^5$, $Tu=0.2\%$

(b) $Re_x = 5.95 \times 10^5$, $Tu=0.2\%$

(c) $Re_x = 7.18 \times 10^5$, $Tu=0.2\%$

(d) $Re_x = 5.55 \times 10^5$, $Tu=12.0\%$

Figure 4.32: Sh contour on the suction surface at $\tau/C = 0.86\%$
(a) \( Re_{ex} = 4.44 \times 10^5 \), \( Tu=0.2\% \)

(b) \( Re_{ex} = 5.95 \times 10^5 \), \( Tu=0.2\% \)

(c) \( Re_{ex} = 7.18 \times 10^5 \), \( Tu=0.2\% \)

(d) \( Re_{ex} = 5.55 \times 10^5 \), \( Tu=12.0\% \)

Figure 4.33: Sh surface plot on the suction surface at \( \tau/C = 0.86\% \)
Figure 4.34: Local Sh on the suction surface at $\tau/C = 0.86\%$
Figure 4.35: Local $Sh/Re_{ex}^{1/2}$ on the suction surface at $\tau/C = 0.86%$
Figure 4.36: Local $Sh/Re_{cz}^{1/2}$ on the suction surface: effect of turbulence
Figure 4.37: Average Sh on the suction surface at $\tau/C = 0.86\%$
(a) chord coordinate

(b) axial coordinate

Figure 4.38: Tip surface coordinates
Figure 4.39: Sh contour on the tip surface
Figure 4.40: Sh contour on the tip surface
Figure 4.41: Local Sh on the tip surface at different clearance levels
Figure 4.42: Local Sh on the tip surface at different clearance levels (continued)
Figure 4.43: Local Sh on the tip surface at different clearance levels
Figure 4.44: Local Sh on the tip surface at different clearance levels (continued)
(a) $Re_x = 4.65 \times 10^5$, $T_u = 0.2\%$

(b) $Re_x = 5.88 \times 10^5$, $T_u = 0.2\%$
Figure 4.45: Sh contour on the tip surface at $\tau/C = 0.86\%$

(c) $Re_{ex} = 7.22 \times 10^5$, $Tu=0.2\%$

(d) $Re_{ex} = 5.37 \times 10^5$, $Tu=12.0\%$
Figure 4.46: Local Sh on the tip surface at $\tau/C = 0.86\%$
Figure 4.47: Local Sh on the tip surface at $\tau/C = 0.86\%$ (continued)
Figure 4.48: Local $Sh/Re^{0.8}$ on the tip surface at $\tau/C = 0.86\%$
Figure 4.49: Local $Sh/Re_x^{0.8}$ on the tip surface at $\tau/C = 0.86\%$ (continued)
Figure 4.50: Averaged Sh on the tip surface

115
Chapter 5

Tip Mass Transfer Measurements With Endwall Motion

This chapter presents data on mass transfer from the tip surface of the turbine blade for the cases with and without relative motion between the tip and the endwall.

5.1 Description of the moving wall apparatus

The moving wall apparatus used for simulating rotation of the blades against the stationary casing in a turbine consists of a 2kW 3-phase induction motor, a rubber belt, three rollers on which the moving wall rotates, and two pitch pulleys which transmit power from the motor to the belt. The whole apparatus rests on an aluminium base that fits on to the top wall of the linear cascade section.

The moving wall uses a belt made of N-8 Sampla synthetic rubber that is placed flush with the inner top wall of the test section. The belt moves from the direction of the suction side of the stationary blades to the pressure side. This simulates the relative motion in a turbine where it is the rotation of the blades that causes the relative motion. The belt rotates on 3 aluminium rollers of diameter 75mm supported on bearings housed in an aluminium frame. There are two primary rollers and a tertiary roller; one of the two primary rollers is driven by a 2kW 3-phase induction motor (also bolted to the frame) through a system of pitch pulleys. A tertiary roller with an adjustable axis of rotation is used to keep the belt tight and flat when moving in the wind tunnel. The vertical position of the bearings supporting the tertiary roller can be changed by using screws to move each bearing housing independent of the other in a slot provided on the frame. This mechanism is used to prevent ‘tracking’ of the belt from one end of the roller to another. It was experimentally verified that the ‘flapping’ effect of the belt due to vibrations during operation resulted in a variation of tip clearance of not more than 0.1% of chord. A schematic diagram of the moving wall apparatus is shown in fig. 5.1.

The motor speed (and hence the belt ‘rotation’ speed) is controlled by means of an Altivar ATV 18 speed drive controller. The apparatus is designed so that air leakage can occur only from the gap between the roller and the frame, and the very low tolerances used
Figure 5.1: Moving wall apparatus used to simulate rotation
during fabrication ensure that there is virtually no such leakage. With the motor operating at its rated speed of 3250 rpm, the reduction in speed during transmission by the pulleys ensures that the maximum belt speed that can be achieved is 25 m/s. In order to simulate engine conditions, where the blade tangential velocity is twice the axial throughflow velocity, the wind tunnel is operated at an inlet speed of about 12.5 m/s.

5.2 Mass transfer field on the tip

A baseline test was conducted at the conditions $\tau/C = 6.9\%$, $Re_{ex} = 5.5 \times 10^5$, $EES = 0$, with the moving wall apparatus mounted on the tunnel top. The data obtained are compared with those obtained under the same conditions but without the moving wall. Figure 5.2 shows the local variation of the two sets of data over the forward half of the blade. As can be seen, the data obtained in the current study are quite close to those obtained earlier. This validates the measurement procedure used, and also implies that the introduction of the moving wall does not alter the flowfield in any way.

Figure 5.3 shows the Sherwood number contour plots for $\tau/C = 0.6\%$ and $\tau/C = 0.86\%$. There is no observable effect of endwall motion on the mass transfer distribution for either clearance. The distribution of Sherwood number in general conforms to the flow pattern obtained through oil-lampblack visualization. The thin region of very high mass transfer on the pressure edge of the blade corresponds to the flow reattaching after initial separation.

As the clearance is increased to $\tau/C = 6.9\%$ through intermediate values of clearance level, an area of low mass transfer is apparent on the thickest part of the blade, near the leading edge. Figure 5.4 shows that the area of low mass transfer increases as the clearance increases. This corresponds to the 'sweet spot' observed by Bunker et al. (2000) for a clearance of 2.3mm with a blade axial chord of 12.45cm compared to 13mm in this study. This feature is essentially the result of low shear stress caused by reduced leakage at this location, as borne out by the flow visualization. Also observed at the two largest clearances is a region of high mass transfer on the suction side near the leading edge, caused by the flow entering directly from the suction side of the blade, then undergoing separation and reattachment.

Figures 5.6 and 5.7 show the local Sherwood number plotted against the clearance flow path expressed in terms of tip clearance lengths, $(y-y_0)/\tau$, for different chordwise locations. Here $y_0$ is the y-coordinate of the pressure edge at a particular chordwise location.

For the case of $\tau/C = 0.6\%$, it is seen that very close to the leading edge, the Sherwood number decreases monotonically as $y$ increases, indicating reattachment at the edge, outside the naphthalene surface. Figures 5.6 and 5.7 show that for downstream chordwise locations, up to $x/C = 0.2$, reattachment occurs at roughly 4 clearance heights. The effects of endwall relative motion start becoming apparent at $x/C = 0.23$. There is a slight reduction in the Sherwood number, from values of about 5000 to nearly 4000. However, towards the trailing edge, there is very little effect of wall motion on the Sherwood number.

It must be noted that the experimental error in Sherwood number (see appendix C) is of the order of 8\%, while the difference in Sherwood number between the cases of endwall motion and no motion is never more than 10\%, except at $x/C = 0.23 - 0.31$. Thus, the effect of endwall motion is, at best, very small at such tight clearances.
Figure 5.2: Comparison of Sherwood number data for $\tau/C = 6.9\%$, EES=0 with flat endwall.
However, the fact that Sherwood numbers for the case of a moving endwall were consistently lower than those for the case of no motion indicates that there does exist a slight effect of endwall motion that acts to reduce $Sh$. A possible explanation for this phenomenon is that at small clearances, viscous effects become sufficiently strong as to slightly drag the leakage vortex towards the suction side and alter the driving pressure gradients. The leakage flow may be slightly reduced in magnitude, resulting in smaller velocities and smaller mass transfer coefficients. The movement of the reattachment point towards the pressure edge corroborates this.

At $\tau / C = 0.86\%$, the picture is the same as that for $\tau / C = 0.6\%$ on the first half of the blade, as seen in figure 5.8. Reattachment occurs at roughly the same $y$-position as for $\tau / C = 0.6\%$. However, there is virtually no effect of endwall motion on the Sherwood numbers on the first half of the blade. Sherwood numbers are lower for the case of endwall motion, as for the smallest clearance, with the difference lying within the experimental error limits of this study. However, figure 5.9 shows that there is a slight effect on the second half of the blade, with the Sherwood numbers slightly increasing with the introduction of wall motion. This is somewhat difficult to explain, since both the flow visualization and the pressure measurements point to a reduced flow velocity near the trailing edge at this clearance level as compared to $\tau / C = 0.86\%$. However, it was confirmed that the data were repeatable. The reattachment point seems to shift towards the suction side, indicating a slight increase in the size of the separation bubble, which is not observed from the flow visualization.

As the clearance is increased to $\tau / C = 1.72\%$ (figs. 5.10 and 5.11), the trend of reducing Sherwood numbers continues. As the flow moves downstream towards the trailing edge, the point of maximum Sherwood number shifts towards the suction side, indicating an increase in the size of the separation bubble near the trailing edge. At the very last $x/C$ location shown, there is no maximum of $Sh$, indicating that the bubble covers the entire width of the tip. There is no effect of endwall motion at this clearance level.

At $\tau / C = 3.45\%$ (figs. 5.12 and 5.13), the overall Sherwood numbers decrease further. For this clearance level, the separation bubble covers the entire width of the tip starting from $x/C = 0.42$, as seen by the continuous increase of $Sh$ from pressure side to the suction side. As expected, based on the previously discussed clearance levels, there is no effect of endwall motion on the mass transfer.

At $\tau / C = 6.9\%$ (figs. 5.14 and 5.15), the size of the separation bubble become larger. Near the leading edge, two peaks of Sherwood number are observed, corresponding to the two reattachment lines caused by flow entering from pressure and suction sides. The size of the midchord separation bubble increases, and there is no observable effect of endwall motion on the mass transfer.

A summary of the area-averaged Sherwood number on the tip surface for each of the cases studied is given in table 5.1. It can be seen that except for the case of smallest clearance, the difference in Sherwood number caused by wall motion is within the error limits of this experiment (see appendix C).
<table>
<thead>
<tr>
<th>$\tau/C$</th>
<th>Run</th>
<th>$Sh_m, EES = 0%$</th>
<th>Run</th>
<th>$Sh_m, (EES = 100%)$</th>
<th>Percent change</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\tau/C = 0.6%$</td>
<td>61</td>
<td>2557</td>
<td>67</td>
<td>2293</td>
<td>3.04</td>
</tr>
<tr>
<td>$\tau/C = 0.86%$</td>
<td>43</td>
<td>2814</td>
<td>48</td>
<td>2660</td>
<td>5.5</td>
</tr>
<tr>
<td>$\tau/C = 1.72%$</td>
<td>51</td>
<td>2417</td>
<td>52</td>
<td>2495</td>
<td>-3.22</td>
</tr>
<tr>
<td>$\tau/C = 3.45%$</td>
<td>38</td>
<td>1876</td>
<td>58</td>
<td>2009</td>
<td>-7.07</td>
</tr>
<tr>
<td>$\tau/C = 6.9%$</td>
<td>14</td>
<td>2234</td>
<td>17</td>
<td>2117</td>
<td>5.22</td>
</tr>
</tbody>
</table>

Table 5.1: Area-averaged Sherwood numbers for the different cases studied.

### 5.3 Summary

The effects of relative endwall motion at 100% EES can be summarized as follows:

1. At a clearance of $\tau/C = 0.6\%$, there appears to be some effect of wall motion on the Sherwood number at certain midchord locations, with Sherwood number decreasing with the introduction of rotation. This may be caused by the leakage vortex moving closer to the suction surface. This would cause reduced pressure gradients across the tip compared to the case of no wall motion, and lead to lower leakage velocities.

2. At a clearance of $\tau/C = 0.86\%$, there appears to be little effect of wall motion on the Sherwood number, except very near the trailing edge where $Sh$ increases.

3. At all higher clearances, there is no effect of relative motion on $Sh$, indicating that at such clearances the flow is essentially pressure-driven and viscous effects are negligible.
\[ \tau/C = 0.6\%, \ EES = 0 \]

\[ \tau/C = 0.6\%, \ EES = 100\% \]

\[ \tau/C = 0.86\%, \ EES = 0 \]

\[ \tau/C = 0.86\%, \ EES = 100\% \]

Figure 5.3: Sherwood number contours for \( Re = 2.7 \times 10^5 \)
$\tau/C = 1.72\%, \ EES = 0$

$\tau/C = 1.72\%, \ EES = 100\%$

$\tau/C = 3.45\%, \ EES = 0$

$\tau/C = 3.45\%, \ EES = 100\%$

Figure 5.4: Sherwood number contours for $Re = 2.7 \times 10^5$
\( \tau/C = 6.9\%, \ EES = 0 \)   \[ \tau/C = 6.9\%, \ EES = 100\% \]

Figure 5.5: Sherwood number contours for \( Re = 2.7 \times 10^5 \)
Figure 5.6: Effect of endwall motion on Sherwood number for $\tau/C = 0.6\%$
Figure 5.7: Effect of endwall motion on Sherwood number for $\tau/C = 0.6\%$
Figure 5.8: Effect of endwall motion on Sherwood number for $\tau/C = 0.86\%$
Figure 5.9: Effect of endwall motion on Sherwood number for $\tau/C = 0.86\%$
Figure 5.10: Effect of endwall motion on Sherwood number for $\tau/C = 1.72\%$
Figure 5.11: Effect of endwall motion on Sherwood number for $\tau/C = 1.72\%$
Figure 5.12: Effect of endwall motion on Sherwood number for $\tau/C = 3.45\%$
Figure 5.13: Effect of endwall motion on Sherwood number for $\tau/C = 3.45\%$
Figure 5.14: Effect of endwall motion on Sherwood number for $\tau/C = 6.9\%$
Figure 5.15: Effect of endwall motion on Sherwood number for $\frac{\tau}{C} = 6.9\%$
Chapter 6

Flow and Heat Transfer in the Tip Region and Endwall Region of an Annular Cascade

The proposed PSU research as part of this project was as follows:

Proposed PSU research in year 1:

- Plan and coordinate the parallel effort between UM and PSU experiments.
- Provide tip section profiles to UM.
- Modify the coolant delivery system (from stationary to rotating frame of reference) in AFTRF.
- Remove the rotor, replace four solid blades with four tip cooled blades.
- Balance the rotor and re-assemble the turbine rig.
- Construct the heat transfer surface at the tip section of cooled turbine rotor blade.
- Construct the heat transfer surface at the endwall of cooled turbine rotor blade.
- Connect the heat transfer surfaces in the rotor to stationary frame instrumentation via an existing slip-ring device.
- Exchange information with UM.

Proposed PSU research in year 2:

- Complete liquid crystal thermography at the tip of the turbine rotor blade
- Complete liquid crystal thermography at the endwall region of the turbine rotor blade (vary the nozzle/rotor coolant leakage flow).
- Reduce heat transfer data using an existing image processor/software.
• Exchange information with UM.

Proposed PSU research in year 3:

• Start Aerodynamic loss measurements for the turbine stage due to tip cooling and
  nozzle/rotor coolant leakage flow near the endwall.

• Determine change in the total to total efficiency of the turbine stage.

• Exchange information with UM.

• Compile a final report for the investigation.

6.1 Turbine research facility modifications performed for the specific research program

The Pennsylvania State University currently operates a highly loaded axial flow turbine
stage for aerodynamic and heat transfer research. The fully operational facility is of large-
scale low-speed nature with modern turbine design attributes. The Axial Flow Turbine
Research Facility (AFTRF) is an open circuit facility 91.4 cm in diameter and a hub to
tip radius of 0.73, with advanced axial turbine blading configurations as designed by GE
(Evendale). The facility consists of a large bellmouth inlet, a turbulence generating grid
followed by a test section with a nozzle vane row and a rotor. There are 23 nozzle guide
vanes and 29 rotor blades followed by outlet guide vanes. A variable through-flow is provided
by two auxiliary, adjustable pitch axial flow fans and an aerodynamically designed exhaust
throttle. This system allows accurate control of the mass flow through the turbine stage
up to a maximum of 22,000 cfm. The two fans connected in series produce a pressure rise
of 74.7 mm Hg (40 inches of water height) with a mass flow rate of 10.4 cubic meter per
second under nominal operating conditions. The power generated is absorbed by an eddy
current brake. The rotor and nozzle vane passages are instrumented with high frequency
instrumentation to measure steady and unsteady pressures and shear stresses. A three
component LDV system is currently operational in the facility. The facility is instrumented
with two modern probe traversing systems in the rotating frame. Measurements in the
rotor passages and downstream of the rotor can be taken in the radial direction and in
the circumferential direction using a stepping motor controlled custom designed traverser.
The facility also includes a 150 channel coin and brush type slip ring device to transfer
data from the rotating frame of reference to the stationary frame for further processing.
The current research program required extensive modification to turbine blades. Design
manufacturing and assembly of a new cooling air transfer system from the stationary frame
to rotating frame was necessary. An air-transfer device including two custom manufactured
seals working against plasma coated rotating surfaces were contracted out to a specialist
manufacturing facility. A special calibrated orifice and robust pressure vtranslucers were
installed in the rotor frame of reference for accurate coolant mass flow rate measurements
of the coolant air injected from the tip trench. Cooling plenum chambers, radial cooling
holes, tip trenches and discrete cooling holes were precision machined in computer controlled
five-axis machine tools. The tip trench and cooling system sizing was completed under the supervision of engineering staff at Solar Turbines Inc. Since extensive rotor modifications were completed, a dynamic balancing of the turbine rotor was also performed after the re-assembly process. An instrumented disk cavity flow simulator allowing the researcher to create realistic cavity flow scenarios was used. Different cavity flow patterns were generated by a root cooling, disk impingement and radial injection system. The disk cavity flow system were integrated in the tight axial space between the rotor disk and the nozzle guide vane assembly. The coolant/leakage flows originating from the disk cavity space were monitored by a set of total pressure and temperature sensors located at the inlet and exit of the turbine stage. The change in the total to total efficiency of the turbine stage due to endwall coolant/leakage was properly documented. The research program also included the measurement of the three dimensional aerodynamic field in the rotating frame by using sub-miniature five hole probes. This action required operating a computer controlled radial-circumferential probe traverser that can move with respect to the turbine rotor. The facility was also modified for the proposed convective heat transfer coefficient measurements using liquid crystal thermography. This effort required obtaining color liquid crystal images of the rotor tip that is in rotational motion at 1330 RPM. A high resolution digital camera and a stroboscope system resulted in the acquisition of the color images that are currently under investigation. The previous P S U studies dealing with the acquisition of liquid crystal images in rotational environment and the use of Inconel surface heater foils were extremely supportive of the current study.

### 6.2 Project Results

In the framework of the collaborative investigation described in the original proposal, the Pennsylvania State University provided turbine airfoil details and the relative flow conditions that were carefully replicated at the linear cascade facility of the University of Minnesota.

During the initial stages of the program, the turbine facility (AFTRF) went through an extensive modification for the installation of the coolant delivery system that allowed the researchers to inject from the tip sections of rotating turbine blades (four blades). Since the rotor assembly was taken out of the rig for extensive cooling system modifications, a comprehensive evaluation of the rotor after the assembly was required. The rotor needed to be dynamically balanced after the re-assembly.

During that initial period the rotor instrumentation for heat transfer measurements was also completed. The slip-ring channels that transmit high current (up to 5 Amps) were connected to the electrodes imbedded in the hub endwall surface of the rotor. The Inconel heater foil that was cut in the form of the tip surface was connected to electrodes imbedded in the hub endwall surface via thick metallic foils that were non-intrusively attached to the blade surfaces.

The Penn State program focused on measuring the detailed aerodynamics field in the rotor frame of reference and in the stationary frame of reference when coolant injection near the inner hub surface and blade tip was performed.
6.2.1 Endwall coolant injection from the wheelspace cavity and its influence on turbine mainstream aerodynamics

Aerodynamic Measurements In The Stationary Frame

The relative aerodynamic and performance effects associated with rotor - NGV gap coolant injections were investigated in the Axial Flow Turbine Research Facility (AFTRF). This study quantifies the effects of the coolant injection on the aerodynamic performance of the turbine for radial cooling, impingement cooling in the wheelspace cavity and root injection. Overall, it was found that even a small quantity (1%) of cooling air can have significant effects on the performance character and exit conditions of the high pressure stage. Parameters such as the total-to-total efficiency, total pressure loss coefficient, and three-dimensional velocity field show local changes in excess of 5%, 2%, and 15% respectively. It is clear that the cooling air disturbs the inlet end-wall boundary layer to the rotor and modifies secondary flow development thereby resulting in large changes in turbine exit conditions. It was concluded that:

- The wheelspace coolant mixes with the mainstream flow and produces measurable changes in loss coefficient, velocity field, exit angle, and total-to-total efficiency. Local changes can be significant. The wheelspace coolant should not be neglected on the aerodynamic analysis of turbine blade stages. Overall, root injections showed the strongest effects although radial and impingement cooling showed measurable changes in loss coefficient, velocity, and efficiency.

- The cooling flow caused significant local perturbations in the pressure coefficients. Root injection showed the largest changes followed by radial cooling, and impingement cooling. In all cases the strongest effects were below midspan but dwindling effects exist out to the tip regions. All three cooling methods caused significant local changes and a general redistribution of the data over the entire passage in both radial and tangential directions. Maximum effects ranged from 1.70%, 0.86%, and 2.57% for radial cooling, impingement cooling, and root injection respectively. Although the local perturbations were quite high, when passage average data was evaluated the changes were found to be small for radial cooling and impingement cooling. Root injection was able to affect the overall pressure coefficient as well as cause a redistribution of pressure coefficient data. The amount of change was almost five times that of impingement cooling and 30 times that of radial cooling.

- The cooling flow was responsible for modifying the velocity profiles. Radial cooling and impingement cooling shifted the velocity profiles radially outward while root injection was able to decrease the overturning and underturning. The point of maximum overturning was shifted radially by 10% for radial cooling and 5% for impingement cooling. The radial cooling is injected normal to the mainstream flow and would energize and thicken the rotor inlet boundary layer more than impingement cooling. The thicker boundary layer displaces the core flow and the passage vortex towards the tip region. The root injection shows the most significant effects due to the coolant
injection and is fundamentally different from radial cooling and impingement cooling. Root injection has the ability to affect the magnitude of the overturning and underturning. The amount of overturning and underturning is reduced.

- Overall root injection showed the strongest changes in total-to-total efficiency. The passage averaged efficiency increase was over 1.5% for root injection. The root injection efficiency increase was localized to the midspan region of the blade. With an injection hole size on the order of the trailing edge thickness and with the injection inclined at 45° to the hub wall, the root injection can significantly energize the nozzle wake region. Radial cooling showed irregular changes in total-to-total efficiency. For 1% coolant flow the change was positive, at 1.25% the change was negative and at 1.5% the change was positive again. Impingement cooling caused the passage averaged total-to-total efficiency to drop.

Detailed information and reduced data for this part of the study is presented in Appendix A:


Aerodynamic Measurements In The Rotating Frame

This part of the research program deals with the aerodynamic measurements in the rotational frame of reference of the Axial Flow Turbine Research Facility (AFTRF) at the Pennsylvania State University. Stationary frame measurements of "Mainstream Aerodynamic Effects Due to Wheelspace Coolant Injection in a High Pressure Turbine Stage" were presented in the previous page. The relative aerodynamic effects associated with rotor - nozzle guide vane (NGV) gap coolant injections were investigated in the rotating frame. Three-dimensional velocity vectors including exit flow angles were measured at the rotor exit. This study quantifies the secondary effects of the coolant injection on the aerodynamic and performance character of the stage main stream flow for root injection, radial cooling and impingement cooling. Current measurements show that even a small quantity (1%) of cooling air can have significant effects on the performance and exit conditions of the high pressure turbine stage. Parameters such as the total pressure coefficient, wake width, and three-dimensional velocity field show significant local changes. It is clear that the cooling air disturbs the inlet end-wall boundary layer to the rotor and modifies secondary flow development thereby resulting in large changes in turbine exit conditions. Effects are the strongest from the hub to midspan. Negligible effect of the cooling flow can be seen in the tip region.

- The effects of small (1%) coolant injection into the free-stream of a high pressure turbine stage can be very significant and should not be neglected on the aerodynamic analysis of turbine stages. The cooling air is affecting the structure of the three-dimensional secondary flow and inlet rotor boundary layer, which in turn has a large effect on the exit three-dimensional flow and stage performance.
• The changes observed were large and three-dimensional in nature. Their neglect in
stage blading design could lead to serious miscalculations. Parameters such as pressure
coefficient, wake width, three-dimensional velocity field, and exit angles were observed
to change significantly.

• The cooling was able to produce significant changes in the total pressure coefficient.
The large changes in pressure coefficient are due to a shift in the wake position and
wake width rather than a change in the wake peak or trough magnitudes. In the
rotational frame, root injection reduced the width of the wake while radial and im-
pingingment cooling increased the width. The effects of the rotational frame width
changes are smoothed out in the stationary frame measurements. To understand the
physical mechanism responsible for the loss coefficient changes (stationary frame) it
is necessary to look in the rotational frame.

• Root injection tended to have an opposite effect on the three-dimensional velocity
field then radial and impingement cooling. This is consistent with the stationary
frame data provided in Part I of this publication. Again, it is necessary to examine
both the stationary and rotational frames to have a complete understanding of the
physics controlling the velocity field.

• The changes in rotor exit flow are highest at the midspan. The effects of the cooling
are convected to the midspan by the action of the passage vortices. It is believed that
the cooling air is energizing the boundary layer ahead of the rotor thereby effecting
the development and effects of the secondary flows.

Additional research should be performed to detail the complex cooling-mainstream mixing
process and its effects on the inlet rotor boundary layer and the resulting secondary flows.

Detailed information and reduced data for this part of the study is presented in Appendix-
B:

"Mainstream Aerodynamic Effects Due to Wheelspac Coolant Injection in a High Press-
ure Turbine Stage: Part II- Aerodynamic Measurements in the Rotating Frame," (Christopher McLean, Cengiz Camci, Boris Glezer), The ASME Transactions, Journal of Turboma-

6.2.2 Turbine tip aerodynamics under the influence of coolant injection
from a tip trench

Rotational frame velocity and pressure measurements were made downstream of a tip cooled
rotor blade row in AFTRF. The specific emphasis was given to study discrete hole cooling
injection from a a square cross section trench machined on the tip surface of turbine blades.
Measurements were taken at two axial locations to track the development of the secondary
flow in the blade tip region using a five hole probe. Coolant mass flow was injected at
several locations on the rotor tip to investigate the effect of coolant air on the secondary
flow. The ultimate objective is to reduce losses by the introduction of high momentum
air in the tip gap. Results indicate that the passage and the leakage vortices retain their
structures up to 46 % chord-length downstream of the rotor trailing edge. The cooling air,
which is 0.3 % of the total mass-flow, appears to be well mixed with the leakage flow and makes little difference to the flow field downstream of the rotor.

Cooling is indispensable in rotors. It is also expensive, since cooling air is high pressure air that does no work. One would like to use cooling air for reasons other than heat transfer damage protection; for instance it could be used for the reduction of total pressure losses. The present study attempted to find whether coolant could be used for reducing the losses associated with tip leakage flow. The idea was to block the way of leakage flow entering into the tip gap, hence reducing the leakage mass flow. Results indicate that the concept might prove beneficial, but a relatively large amount of coolant is necessary to make significant improvements. Results from rotating frame measurements indicate that the leakage vortex is much stronger than the passage vortex, and is expected to be the most dominant vortical structure entering the downstream stage. Efforts are being made to increase the coolant mass flow rate as well as to measure heat transfer characteristics of the flow.

Detailed information and reduced data for this part of the study is presented in Appendix-C:


6.2.3 Heat transfer measurements on the tip surface using liquid crystal thermography

Convective heat transfer rates on the tip surface and on the endwall surface of the turbine rotor blades are presented in the form of convective heat transfer coefficients from steady state liquid crystal measurements. The Turbomachinery Heat Transfer Laboratory of the PSU has expertise in measuring heat transfer rates on surfaces with arbitrarily specified external boundaries as explained in Wiedner and Camci [1997] "Determination of Convective Heat Flux on State Heat Transfer Surfaces with Arbitrarily Specified Boundaries," the Transactions of the ASME, Journal of Heat Transfer, Vol.118, No.4, pp:1-8, November 1996. The approach described in this publication is extremely relevant to the success of the current heat transfer program because of the complicated external boundaries existing on the tip section of the rotor blade and on the endwall surface. The surface heater used in this study is formed from a 0.010 inch thick INCONEL 600 and attached to the tip section. The heat transfer surface is non-intrusive. The electric heater foil provides a prescribed wall heat flux boundary condition at the fluid solid interface. The liquid crystal coating on the heater surface provides the local wall temperature information. Local heat flux, wall temperature and free stream temperature information defines the heat transfer coefficient. The color information on the liquid crystal surface in the rotating frame is recorded from the stationary frame with the help of a stroboscope illumination system that was recently proven to be extremely useful in rotating frame applications. The Turbomachinery Heat Transfer Laboratory at PSU currently has all the components to complete liquid crystal thermography in the rotating frame. Although the project is currently terminated by the AGTSR, an additional heat transfer measurement effort is continuing with funds available
from the Dept. of Aerospace Engineering at Penn State. The turbine blade conduction heat loss determination was completed in January 2002. We are expecting to continue our heat transfer efforts in Spring 2002 using limited funds available from the department. Some of our more current efforts in this area are summarized as follows:

**A constant heat flux surface for turbine heat transfer research**

Current Task and the Facility: A turbine blade mounted in the axial flow turbine facility of the Pennsylvania State University is shown in fig. 6.1. Twenty-nine rotor blades rotate at about 1330 rpm in air at ambient temperature (typically 20 – 30°C). One of the 29 blades needs to be instrumented for convective heat transfer coefficient measurements. The specific research interest is the tip platform surface. The electrical heater is powered by DC current transmitted through a slip-ring device. The current is limited to 5 Amps because of the slip-ring limitations.

![Perspective view of blade in AFTRF](image)

**Figure 6.1:** Perspective view of blade in AFTRF

Inconel heater surface: Figure 6.2 shows our previous effort using a 0.001 inch thick Inconel sheet acting as a heater. Due to the specific tip geometry local heat flux is not constant. Currently we numerically calculate the local voltage and heat generation from the shape of the heater. The specific heat transfer measurement technique we use requires surface temperature measurements at the fluid-solid interface. We spray a thin black paint layer on top of the heater and then another thin layer of thermochromic liquid crystals. The red/yellow color band can be moved over the top surface by increasing the current passing through the heater.

Figure 6.3 shows a thermogram of the tip surface. The colors playing between black and blue/violet are calibrated for temperature. In figure 6.3 red/yellow color shows up at
about 42°C. All of the colors between the red/yellow and blue/violet represent a temperature bandwidth of about 1°C. The liquid crystal color against temperature is precisely calibrated. The local temperature from liquid crystal coating and the local heat flux from the Inconel heater leads to the calculation of h (convective heat transfer coefficient) on the tip surface. Although this is an excellent method of measuring tip heat transfer, the surface wall heat flux distribution is not constant. This method requires matching a calculated wall heat flux with the measured wall temperature from the liquid crystal coating.

A constant heat flux heater could be used to perform convective heat transfer coefficient measurements on the tip surface shown in Figures 6.1-6.3. The airfoil area in fig. 6.4 marked with dashed lines need to generate CONSTANT HEAT FLUX. The thickness of the heater system should be between 0.003 and 0.004 inches. The high aspect ratio rectangular strips near the leading edge and trailing edge are just electrical connectors with negligible resistance. The airfoil area is about 2.4 $in^2 = 1600 \ mm^2$. We are expecting to have an overall resistance of 10 OHMS in the airfoil area. This will require 10W, 40W and 160W at voltage settings of 10V, 20V and 40 V. The corresponding heat flux will be 4.2 $W/in^2$, 16.7 $W/in^2$ and 66.7 $W/in^2$. The wedge shaped trailing edge zone of the airfoil is the most important region in this research. Therefore we need to make sure that we can generate "constant heat flux" in this almost triangular area. We are hoping that we can obtain a constant heat flux surface in the zone marked with dashed line.

Figure 6.6 shows an enlarged view of Figure 6.5. Note that the trailing edge region between $y=50 \ mm$ and $100 \ mm$ is the most important measurement region. We need a heat flux distribution as uniform as possible in this region. We also need a good heat flux distribution near the leading edge ($160 < y < 180 \ mm$).
Figure 6.3: Thermogram of tip surface

**An alternative strategy**

Due to the extremely narrow geometry near the trailing edge (25 < y < 100), it may be worthwhile to investigate a heater shape as shown in fig. 6.7. The plan is to bend the heater surface around the trailing edge wedge (from the dotted lines). The heater surface is extended into the almost square regions added (starting from the dashed lines). High aspect ratio rectangular strips are electrical connectors (with negligible resistance).

Power Density (AIRFOIL AREA =1600 MM²=2.4 SQ.INCH)

<table>
<thead>
<tr>
<th>V=10 V</th>
<th>R=2 Ohms</th>
<th>R=5</th>
<th>R=10</th>
<th>R=15</th>
<th>R=20</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>I=5 A</td>
<td>I=2 A</td>
<td>I=1 A</td>
<td>I=0.66 A</td>
<td>I=0.5 A</td>
</tr>
<tr>
<td>I.V WATTS</td>
<td>50</td>
<td>20</td>
<td>10</td>
<td>6.6</td>
<td>5</td>
</tr>
<tr>
<td>POWER DENSITY W/in²</td>
<td>4.2W/in²</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>V=20 V</th>
<th>R=2 Ohms</th>
<th>R=5</th>
<th>R=10</th>
<th>R=15</th>
<th>R=20</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>I=10 A</td>
<td>I=4 A</td>
<td>I=2 A</td>
<td>I=1.33 A</td>
<td>I=1 A</td>
</tr>
<tr>
<td>I.V WATTS</td>
<td>200</td>
<td>80</td>
<td>40</td>
<td>26</td>
<td>20</td>
</tr>
<tr>
<td>POWER DENSITY W/in²</td>
<td>16.7W/in²</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>V=40 V</th>
<th>R=2 Ohms</th>
<th>R=5</th>
<th>R=10</th>
<th>R=15</th>
<th>R=20</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>I=20 A</td>
<td>I=8 A</td>
<td>I=4 A</td>
<td>I=2.66 A</td>
<td>I=2 A</td>
</tr>
<tr>
<td>I.V WATTS</td>
<td>800</td>
<td>320</td>
<td>160</td>
<td>104</td>
<td>80</td>
</tr>
<tr>
<td>POWER DENSITY W/in²</td>
<td>66.7W/in²</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 6.1: Power density (airfoil area=1600 mm²)

WE EXPECT TO HAVE ABOUT 10 OHMS OF RESISTANCE IN THE CONSTANT HEAT FLUX AREA. AT A MAXIMUM OF 40 VOLTS, THIS WILL REQUIRE 4 AMPS.
(66.7 W/INCH$^2$ and 160 W).

APPENDIX-A
DETAILED RESULTS
ENDWALL COOLANT INJECTION FROM THE WHEELSPACE AND ITS INFLUENCE ON TURBINE MAINSTREAM AERODYNAMICS
Aerodynamic measurements in the stationary frame:

APPENDIX-B
DETAILED RESULTS
ENDWALL COOLANT INJECTION FROM THE WHEELSPACE AND ITS INFLUENCE ON TURBINE MAINSTREAM AERODYNAMICS
Aerodynamic measurements in the rotating frame:

APPENDIX-C
DETAILED RESULTS
TURBINE TIP AERODYNAMICS UNDER THE INFLUENCE OF COOLANT INJECTION FROM A TIP TRENCH
Figure 6.4: Sketch of proposed design showing connector strips
Figure 6.5: Coordinates of blade, mm

Figure 6.6: Enlarged view of fig. 6.5
Figure 6.7: An alternative heater foil boundary that has extensions near the trailing edge.
Bibliography


Attenuation of Hot Streaks and Interaction of Hot Streaks with the Nozzle Guide Vane and Endwall

1 October 2001 to 31 March 2002
Semi-Annual Report
Contract Number 01-01-SR092

D. G. Bogard, PI
Mechanical Engineering Department
University of Texas
Austin, TX  78712

K. A. Thole, Co-PI
Mechanical Engineering Department
Virginia Tech
Blacksburg, VA  24061

March  2002
Attenuation of Hot Streaks and Interaction of Hot Streaks with the Nozzle Guide Vane and Endwall

Semi-Annual Report

David G. Bogard, University of Texas-Austin, Principal Investigator;
Karen A. Thole, Virginia Tech, Co-principal Investigator;
Sean Jenkins and Krishnakumar Varadarajan, University of Texas, Graduate Researchers; and
Severin G. Kempf and Daniel G. Knost, Virginia Tech, Graduate Researchers

Contract Number 01-01-SR092
Clemson University Research Foundation
South Carolina Energy Research and Development Center
Clemson University
Clemson, SC 29634-5702

March 2002

This report was prepared with the support of the US Department of Energy, Morgantown Energy Technology Center, Contract Number 01-01-SR092
Executive Summary

AGTSR ANNUAL REPORT

Turbulence and Turbine VT Experimental and Computational
Cooling Research Lab Convection Lab (VTExCCL)
Department of Mechanical Engineering Mechanical Engineering Department
University of Texas Virginia Tech
Austin, TX  78712 Blacksburg, VA  24061
Phone:  (512)471-3128  (540)231-7192
Fax:     (512)471-8727  (540)231-9100

Project Title: Attenuation of Hot Streaks and Interaction of Hot Streaks with the Nozzle Guide Vane and Endwall

AGTSR Subcontract No.: Contract Number 01-01-SR092
Principal Investigator: David G. Bogard, University of Texas-Austin
Co-Principal Investigator: Karen A. Thole, Virginia Polytechnic Institute and State University
UT Research Students: Sean Jenkins and Krishnakumar Varadarajan
VT Research Students: Severin G. Kempf and Daniel G. Knost

Subcontract Information:
First Year Start Date: March 1, 2001
AGTSR First year contract value: $160,495

STATUS AND RESULTS FOR THE REPORTING PERIOD

Task 1 Initial Planning meeting with industrial partners
This task was completed during the previous reporting period.

Task 2 Benchmark hot streak measurements
Work is still continuing on this task.
Specific Tasks at Virginia Tech

Task 3  Interaction of a hot streak / temperature gradient with the endwall
Initial experiments have been completed whereby coolant, which was nominally 1% of the core flow, was injected through discrete film-cooling holes placed in the combustor liner walls. Adiabatic wall temperatures were measured for this case as a benchmark to compare against slot cooling with the same cooling injection. These preliminary measurements were made with a flat temperature profile upstream of the film-cooling injection. The final measurements will be made by injecting coolant from a two-dimensional slot upstream of the vane.

Task 4  Modify endwall test surface to include film cooling holes
During this period a vendor was identified who will provide the cooling hole cuts with a water jet process for the endwall film-cooling hole plate. Construction materials have been ordered and currently we are awaiting delivery of the endwall plate with the film-cooling holes. In addition, the two-dimensional slot that will be placed upstream of the turbine vane was constructed. Installation of this slot into the wind tunnel is in progress.

Task 5  Endwall film cooling performance tests – with isothermal conditions
No work was done on this task during this reporting period.

Task 6  Endwall film cooling performance tests – with a steep temperature gradient
No work was done on this task during this reporting period.

Task 7  CFD simulation of baseline hot streak/vane interaction
A number of computational simulations have been done during this reporting period, which include the simulation of the temperature profile that was measured downstream of the heater bank, a slot cooled endwall configuration, and a combined slot/film-cooled endwall configuration.

Specific Tasks at the University of Texas at Austin

Task 8  Installation of a hot streak generator and baseline tests
The hot streak generator was installed and preliminary temperature profile measurements were made upstream and downstream of the vane. Modifications still need to be done to induce a uniform total pressure profile downstream of the hot streak generator.

Task 9  Effects of the vane on the hot streak attenuation
No work was done on this task during this reporting period.

Task 10 Effects of the hot streak on vane film cooling performance
No work was done on this task during this reporting period.
Task 11  **Roughness effects on film cooling performance**  
A series of experiments were conducted measuring adiabatic effectiveness performance with roughness section upstream and downstream of a row of cooling holes on the suction side of the vane. Significant decreases in adiabatic effectiveness were found.

**Task 12a  Interaction of the hot streak with a film cooled rough surface**  
No work was done on this task during this reporting period

**Task 12b  Improved hot streak dispersion with modified coolant injection**  
No work was done on this task during this reporting period

**Industrial Contact**

At Virginia Tech, direct industrial interaction has been somewhat limited during this reporting period with the industrial partners, which include Pratt & Whitney, GE and Rolls-Royce. Most of the interaction that has taken place during this reporting period has been with regards to questions that needed to be addressed for the endwall film-cooling hole configuration. At the University of Texas, direct industrial interaction has taken place in several modes. In early February Pratt & Whitney visited the University of Texas to discuss their current contract as well as the DOE work. Details of the effects of surface roughness on film cooling performance were also discussed.
Specific Tasks at Virginia Tech

Task 3 Interaction of a hot streak / temperature gradient with the endwall

Preliminary experiments were conducted in which measurements of temperature, total pressure, and endwall adiabatic effectiveness levels were made with a flat temperature profile and coolant injection from an upstream film-cooled wall. This configuration along with the measurement locations are indicated in Figure 1. Note that these were preliminary measurements in which the experimental conditions, flow uniformity, and instrumentation were all checked in preparation for the final experiments in which coolant will be injected from a two-dimensional slot rather than the film-cooled wall.

Although it is not shown here, the experimental conditions for these tests consisted of a temperature difference between the film cooling and the mainstream of 28°C and the uniformity of the mainstream temperature was within 0.5°C. The thermal field temperatures were measured using a thermocouple rake while the total pressure measurements were made using a Kiel probe rake. The adiabatic effectiveness measurements were calculated based on the infrared camera measurements of the endwall temperatures as well as the coolant and freestream temperatures measured using thermocouples.

Figure 2 shows the non-dimensional temperature and pressure profiles upstream of the turbine vane measured at the locations indicated in Figure 1. As can be seen from these profiles there are large variations in both peak and profile shapes entering the turbine section. These variations are a result of the discrete film-cooling holes. Peak total pressure and minimum temperatures (maximum non-dimensional temperatures) coincide with the centerline of the film-cooling holes whereas between the holes, the values were much different. For example, consider the P1 measurement location which is mid-way between two cooling holes where the total pressure profiled is nearly flat but decreasing towards the wall. This total pressure profile is quite similar to what one would expect for an approaching turbulent boundary layer. For the P3 location, there is a peak total pressure at approximately 2% of the span which is much different than a turbulent boundary layer profile. Similar differences are indicated by the thermal field measurements.

Figure 3 shows preliminary adiabatic effectiveness measurements with 1% of the total cooling flow exiting the upstream rows of film-cooling holes. As can be seen, a 1% coolant injection from these film-cooling holes is not particularly effective in reducing the endwall temperatures in the turbine vane passage. Small fingers of coolant are indicated on the endwall, but for the most part the endwall effectiveness levels are near zero indicating no real cooling benefit. It is expected that the two-dimensional slot will provide better protection than the film-cooling injection since the coolant mix-out will be reduced.

Task 4 Modify endwall test surface to include film cooling holes

During this period a vendor was identified who will provide the cooling hole cuts with a water jet process for the endwall film-cooling hole plate. Construction materials have been ordered and currently we are awaiting delivery of the endwall plate with the film-cooling holes.
In addition, the two-dimensional slot that will be placed upstream of the turbine vane was constructed. Installation of this slot into the wind tunnel is currently in progress.

**Task 7 CFD simulation of baseline hot streak/vane interaction**

A number of computational simulations have been done during this reporting period, which include the simulation of the temperature profile that was measured downstream of the heater bank, a slot cooled endwall configuration, and a film-cooled endwall configuration. These computations were completed using the commercial software by FLUENT Inc. The passage was modeled to midspan and a symmetry condition was applied at midspan. Also, periodic conditions were applied in the stagnation plane and in the trailing plane. Table 1 provides the status of the computations being performed as well as those being planned in the near-term future.

Figures 4a-c show the effectiveness predictions for three different cases illustrating the effects of the coolant flow and of a non-uniform, spanwise (radially) varying temperature profile. Note that these computations were done using the two-dimensional upstream slot configuration. The temperature profile that was placed at the inlet to the turbine section for the predictions shown in Figure 4c was parabolic in the spanwise direction and uniform in the pitchwise (circumferential) direction. This is the same temperature profile that we are simulating in the wind tunnel and was shown in our October 2001 report. It is quite clear from Figure 4 that there is a warm ring around the turbine vane for all of the cases indicating the coolant flow does not exit the slot uniformly. In addition, most of the coolant is swept toward the suction side of the vane. These results do indicate, however, that a flush slot is more effective at cooling the downstream endwall than the upstream film-cooling holes as was discussed for our experimental results.

In comparing Figure 4a and 4b, it is clear that there is an added benefit to doubling the slot flow given the cooler temperatures predicted on the endwall. Although this benefit does exist, the warm ring around the vane still occurs for the doubled slot flow case. Figure 4c indicates the effect of the parabolic temperature profile on the endwall effectiveness levels for the 1% slot flow case. Note that the endwall temperature is much cooler due to the cooler temperatures of the parabolic profile as illustrated in Figure 5. In general the same effectiveness pattern exists in Figure 4a as compared with Figure 4c except there are still relatively cooler effectiveness levels for the profile case in Figure 4c. These cooler temperatures are a result of cooler fluid being present in the near-wall region that is diluting the slot flow as compared to the flat temperature profile results.

Figures 6-9 show the streamline patterns in the near wall region for a number of different cases. Note that for each of these figures the streamlines were seeded over the exact same region and in the exact same location to allow direct comparisons to be made. The pathlines are colored by the non-dimensional spanwise velocity magnitude such that red indicates a flow away from the endwall and blue indicates a flow towards the endwall.

Figure 6 represents the baseline case with no slot and a uniform temperature profile for comparison purposes. As can be seen from the streamlines in case 6, there is a leading edge and
passage vortex that develops through the passage. In comparing Figures 6 and 7, the first noticeable effect of the slot flow is the extraneous streamlines shown in mid-passage in Figure 7. These streamlines originate from the main gas path, are ingested into the two-dimensional slot, and then exit from the slot back into the main gas path. This was also seen for the case with the flat temperature profile (Figure 8). Although the ingestion was not severe, it does provide less cooling capability for the slot flow due to the mix-out with the main gas path fluid. For the doubled coolant flow case shown in Figure 9 there appears to be no ingestion into the slot.

All of the cases shown in Figures 6-9 indicate the presence of secondary flows through the passage. Two effects were thought to minimize the secondary flows in the combustor-turbine junction. First, the upstream contraction has a tendency to flatten out the boundary layer along the approaching endwall, which would also flatten out the total pressure profile. Second, the slot flow present energizes the boundary layer and would also tend to give a flatter total pressure profile at the inlet to the turbine vane. More analysis of the computational predictions needs to be done to determine these various effects on the leading edge vortex. From the pathlines indicated in Figures 6-9 there is an indication, however, that there is an effect on the passage vortex. Figure 8 indicates the least severe passage vortex pattern as compared with the other cases. Although further analysis needs to be done on the CFD results, we believe the reason for the smaller passage vortex is because of the two reasons cited above. Without the blowing, Figure 6 indicates that the contraction is not enough to flatten out the profile. With too much blowing the boundary layer may be energized too much such that a strong peak is formed in the near-wall region (as was shown by the peaks in the film-cooling flow data in Figure 2b). Further work needs to be done to fully understand these effects on the passage vortex.

For the 2% slot flow there appears to be stronger secondary flows than in any of the other cases. At the leading edge, there appears to be a strong flow away from the endwall, which is caused by the stronger injection from the slot. This effect is also indicated in the adiabatic effectiveness contour given in Figure 4b in which some coolant is detected at the leading edge in comparison with Figure 4a in which there is no coolant detected at the leading edge. Further into the passage, the stronger secondary flow patterns are illustrated by the fact that the streamlines have moved further into the passage as can be seen by comparing Figures 8 and 9 for the 1% and 2% injection, respectively. The effectiveness contours for the 1% and 2% coolant flow cases (Figures 4a and 4b) indicate that even though more coolant is present for the latter case, it is still quickly swept off the endwall by the strong secondary flows.

Figures 10a-b show the effect of film-cooling the endwall. Note that these are preliminary results and that further grid refinements still need to be evaluated. As can be seen from these contours, there are hot streaks near the leading edge of the vane which are due to a lack of cooling in this region. As can be seen in Figure 10b, some of the coolant is also convecting up the suction side of the vane. Figures 10c-d show the streamline pattern in which the streamlines were seeded at the same location and over the same region as those in Figures 6-9. Note that Figures 10c-d should be directly compared with Figure 8, which includes 1% slot cooling and a flat temperature profile. In general, the largest differences that can be seen are that there is a stronger upward migration of the streamlines along the suction surface, as further illustrated in Figure 10d. These streamlines in the near-wall region are being pushed away from the endwall by the film-cooling injection.
Specific Tasks at the University of Texas at Austin

Task 8 Installation of a hot streak generator and baseline tests.

Hot Streak Installation

The electric heater used to produce the hot streak was installed in a tunnel section immediately upstream of the plexiglass test section. As mentioned in the previous report, the electric heater is a nominally 20 x 20 cm square duct heater with a heating capacity of 7kW, which may be run at full and half power. The heating element is contained within the square duct section followed by a transition section leading to an 20 cm diameter round section, i.e. 0.44P, where P is the pitch between airfoils. The tunnel section constructed to contain the hot streak generator has the same dimensions as the entrance to the plexiglass test section with a height of 55 cm and width of 102 cm. The tunnel section is 117 cm long in the streamwise direction and is situated immediately downstream of the converging or nozzle section of the wind tunnel. The duct containing the heater coils is suspended on steel rods, which support the heater at the center of the tunnel section vertically (or spanwise) and allow continuous movement of the hot streak in the horizontal (or pitch) direction. As expected, preliminary measurements indicate a sizeable velocity defect due to the heater coils and supports within the hot streak generator. The velocity reaches a minimum of 0.5*U₀ where U₀ is the mainstream velocity. Future work involves adding resistance to the air flow in the area surrounding the hot streak generator to induce a uniform velocity across the tunnel in both the spanwise and pitch directions.

Measured Temperature Profiles

Temperature profiles were measured at several positions downstream from the heater exit. An initial profile at 0.77 chord lengths downstream from the heater exit indicates a maximum temperature ratio of T/T₀ = 1.18 using only the upstream heater stage with 3.5 kW of heating. For this airfoil, the chord length is 59.4 cm. As seen in Figure 11, the profile is peaked with the maximum value at 0.56*S where S = 53.5 cm is the span. Since there is large velocity defect through the hot streak generator, the temperature rise is much greater than was predicted assuming uniform flow throughout the tunnel. Figure 11 also shows a temperature profile at 1.38 chord lengths downstream of the heater before the flow has reached the geometric leading edge of the vane. This profile was taken about 0.25 chord lengths (or 15 cm) upstream of the leading edge before the vane has affected the streamlines. This temperature profile reaches a maximum temperature ratio of T/T₀ = 1.17 at 0.59*S. Both temperature profiles were taken at a position where the temperature has reached a maximum in the pitch direction and this coincides with the vertical centerline of the hot streak generator. The third profile was taken 18 cm behind the trailing edge of the vane or about 3.0 chord lengths downstream of the heater exit. This profile was taken with the hot streak generator positioned to pass the hot streak through the center of the passage between vanes. Again, the spanwise temperature profile was made where the temperature ratio reaches a maximum in the pitch (horizontal) direction, and reaches a maximum value of T/T₀ = 1.07 at 0.66S. It is believed that buoyancy or secondary flows in the tunnel contribute the change in the position of the hot streak as it moves downstream. Lines indicating the top, centerline, and bottom of the hot streak generator are provided for comparison.
with the temperature profiles. For the profiles upstream of the vane, the hot streak is approximately the same height as the hot streak generator. Behind the trailing edge, the hot streak width in the vertical direction has shrunk significantly. The reduced centerline temperature of the hot streak as it passes through the passage is an indication of a relatively strong dispersion of the hot streak. At this point, it is probably an artifact of the velocity defect within the hot streak, which will cause high turbulence levels. After the facility is modified to produce a uniform velocity profile, we expect that the hot streak will remain much more coherent through the passage.

**Task 11  Roughness effects on film cooling performance.**

A series of experiments were completed to evaluate the effect of surface roughness on the film cooling performance from the first row of holes on the suction side of the simulated vane. As shown schematically in Figure 12, the simulated vane has six rows of coolant holes in the showerhead region, two rows on the pressure side, and three rows on the suction side. The adiabatic film cooling performance for this simulated vane has been documented in numerous previous publications, eg. Polanka et al. (1999) and Ethridge et al. (2001). The focus of this study was the effects of roughness on the adiabatic effectiveness for film cooling from the first row on the suction side of the vane ($x/d = 0$ on Figure 12).

**Experimental conditions**

The array of conical roughness elements used in these experiments were described in the previous semi-annual report (Mar-Aug, 2001). The roughness Reynolds for the array of roughness elements is $Re_k = 60$, which is a “fully rough” condition. Three roughness configurations were studied: roughness upstream of the film cooling row of holes, roughness downstream, and roughness upstream and downstream. The configuration for roughness upstream of the row of holes had a roughness section that started 44 mm upstream of the row of cooling holes, and extended to a point immediately upstream of the holes, i.e. a total streamwise extent of about 11$d$ (where $d$ is the coolant hole diameter). The downstream extent of roughness elements was 85 mm or 21$d$.

All film cooling holes had a diameter of $d = 4.11$ mm. Holes for the first row on the suction side of the vane had an injection angle of 50° with respect to the surface, and were oriented in the streamwise direction. Adiabatic effectiveness tests were generally conducted with a density ratio of $DR = 1.6$, though some tests were done at a lower density ratio of $DR = 1.2$. Low and high mainstream turbulence levels were used, $Tu = 0.5\%$ and $Tu = 20\%$, respectively. Tests were conducted with the showerhead holes taped over to produce a smooth leading edge condition. When the showerhead was blowing, a blowing ratio of $M = 1.6$ was used (based on the average blowing velocity from the showerhead and the approach velocity to the vane). All tests were conducted with an approach velocity of $U_0 = 5.8$ m/s.
Adiabatic Effectiveness Results

As indicated in Table 1, the repeatability of the test results was established by a number of repeated experiments with various operating conditions. The repeatability of the laterally averaged adiabatic effectiveness was generally better than $\delta \bar{\eta} = \pm 0.015$. The following results are presented with a focus on three blowing ratios: $M = 0.7, 1.0, \text{ and } 1.4$. In each case the maximum adiabatic effectiveness was found at the lowest blowing ratio. At the higher blowing ratios the coolant jets are progressively more separated.

Figure 13 shows the effect of having roughness upstream of the cooling holes with low mainstream turbulence and no showerhead blowing. The upstream roughness was found to cause a dramatic decrease in adiabatic effectiveness. This decrease can be attributed to the effect of the surface roughness on the approach conditions of the boundary layer. The roughness probably causes a much thicker boundary layer that is less effective in turning the coolant jet towards the surface. Consequently the coolant jet separates a greater distance from the surface resulting in decreased adiabatic effectiveness.

With a roughness section only downstream of the coolant holes, shown in Figure 14, there is still a decrease in adiabatic effectiveness, but the decrease is much less than for the upstream roughness. The decrease in adiabatic effectiveness with a downstream roughness section may be attributed to an increase dispersion of coolant by increased turbulence levels caused by the surface roughness.

The effect of roughness sections upstream and downstream of the cooling holes is presented in Figure 15. At $M = 0.7$ there was still a significant decrease in adiabatic effectiveness, but not as much as for the upstream alone roughness section. At higher blowing ratios of $M = 1.0$ and 1.4 there was only a slight decrease in adiabatic effectiveness. Apparently the downstream roughness section has a compensating effect so that the combined upstream and downstream roughness causes a smaller decrease in adiabatic effectiveness than upstream roughness alone. Recall that upstream roughness was speculated to cause increased separation of the coolant jets. The downstream roughness could mitigate the effect of jet separation by increasing turbulence and hence transport of coolant back to the surface.

With showerhead blowing, and high mainstream turbulence, the operating conditions best simulate actual turbine airfoil conditions. Experiments were conducted under these conditions with smooth surfaces and with a roughness section upstream of the cooling holes. Adiabatic effectiveness results for these conditions are presented in Figure 16. These results show that the upstream roughness causes a significant decrease in adiabatic effectiveness, approximately 20% to 25%, for all blowing ratios. Of particular interest was the $\Delta \bar{\eta} = 0.05$ decrease for $M = 1.4$ that suggests that the roughness is dispersing the coolant from the showerhead injection as well as disrupting the coolant injection from the suction side holes.
**Goals for Next Reporting Period**

The goals for the work being done at Virginia Tech for the next reporting period are the following: finish the baseline studies for the two-dimensional slot alone with a range of radial temperature profiles, and to install the endwall film-cooling hole plate. Regarding the computational work of this project, simulations for a range of film-cooling flows interacting with the hot streak will be evaluated.

Further modifications of the hot streak generator will be completed to induce a uniform total pressure profile downstream of the generator at the University of Texas. This will eliminate the wake region downstream of the generator and result in a much more coherent hot streak. Baseline tests will then be completed showing the decay of the hot streak positioned to pass through the center of the passage and to impact the vane. The effect of the hot streak on the adiabatic effectiveness performance of the vane film cooling will be measured. Furthermore, the effect of the film cooling on the dispersion of the hot streak will be measured.

**References**


Table 1. Test Cases for CFD Simulations of Endwall Film-Cooling Studies

<table>
<thead>
<tr>
<th>No.</th>
<th>Geometry</th>
<th>Temperature Profile</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Contraction Only</td>
<td>Parabolic</td>
<td>Complete</td>
</tr>
<tr>
<td>2</td>
<td>Contraction with Slot (1% coolant flow)</td>
<td>Flat</td>
<td>Complete</td>
</tr>
<tr>
<td>3</td>
<td>Contraction with Slot (2% coolant flow)</td>
<td>Flat</td>
<td>Complete</td>
</tr>
<tr>
<td>4</td>
<td>Contraction with Slot (1% coolant flow)</td>
<td>Bottom Peaked</td>
<td>Not Complete</td>
</tr>
<tr>
<td>5</td>
<td>Contraction with Slot (1% coolant flow)</td>
<td>Parabolic</td>
<td>Complete</td>
</tr>
<tr>
<td>6</td>
<td>Contraction with Film-Cooling Pattern 1</td>
<td>Flat</td>
<td>Not Complete</td>
</tr>
<tr>
<td>7</td>
<td>Contraction with Slot and Film-Cooling Pattern 1</td>
<td>Flat</td>
<td>Complete</td>
</tr>
<tr>
<td>8</td>
<td>Contraction with Slot and Film-Cooling Pattern 2</td>
<td>Flat</td>
<td>Not Complete</td>
</tr>
<tr>
<td>9</td>
<td>Contraction with Slot and Film-Cooling Pattern 2</td>
<td>Bottom Peaked</td>
<td>Not Complete</td>
</tr>
</tbody>
</table>

Table 2. Test Conditions for Rough Surface Experiments

<table>
<thead>
<tr>
<th>No.</th>
<th>Turbulence</th>
<th>DR</th>
<th>Roughness</th>
<th>Showerhead</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>None</td>
<td>Off</td>
</tr>
<tr>
<td>2</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>None</td>
<td>Off</td>
</tr>
<tr>
<td>3</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>None</td>
<td>Off</td>
</tr>
<tr>
<td>4</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>None</td>
<td>Off</td>
</tr>
<tr>
<td>5</td>
<td>Low - 0.5%</td>
<td>1.2</td>
<td>None</td>
<td>Off</td>
</tr>
<tr>
<td>6</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>None</td>
<td>Off</td>
</tr>
<tr>
<td>7</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>None</td>
<td>On</td>
</tr>
<tr>
<td>8</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>Up &amp; Downstream</td>
<td>Off</td>
</tr>
<tr>
<td>9</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>Downstream</td>
<td>Off</td>
</tr>
<tr>
<td>10</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>Upstream</td>
<td>Off</td>
</tr>
<tr>
<td>11</td>
<td>Low - 0.5%</td>
<td>1.6</td>
<td>Upstream</td>
<td>On</td>
</tr>
<tr>
<td>12</td>
<td>High - 20%</td>
<td>1.6</td>
<td>None</td>
<td>Off</td>
</tr>
<tr>
<td>13</td>
<td>High - 20%</td>
<td>1.6</td>
<td>None</td>
<td>Off</td>
</tr>
<tr>
<td>14</td>
<td>High - 20%</td>
<td>1.6</td>
<td>None</td>
<td>On</td>
</tr>
<tr>
<td>15</td>
<td>High - 20%</td>
<td>1.6</td>
<td>Upstream</td>
<td>On</td>
</tr>
<tr>
<td>16</td>
<td>High - 20%</td>
<td>1.2</td>
<td>Upstream</td>
<td>On</td>
</tr>
</tbody>
</table>
Figure 1. Schematic illustrating measurement locations for the preliminary endwall effectiveness measurements with no upstream slot but rather cooling provided by a number of film-cooling rows.
Figure 2a-b. Non-dimensional temperature (a) and pressure (b) measurements upstream of the turbine vane with cooling from the combustor film-cooling holes.
Figure 3. Measured adiabatic effectiveness levels on the turbine endwall downstream of the combustor liner film-cooling holes for a coolant flow of 1% of the core flow.
Figure 4. Contours of eta for cases (a) 1\% slot cooling, uniform inlet (b) 2\% slot cooling, uniform inlet (c) 1\% slot cooling, inlet profile.
Figure 5. Non-dimensional temperature contours for 1% slot cooling with an inlet temperature profile.
Figure 6. Pathlines colored by non-dimensional spanwise velocity for a case with no slot and a uniform temperature profile.

Figure 7. Pathlines colored by non-dimensional spanwise velocity for a parabolic temperature profile and 1% coolant mass flow from the slot.

Figure 8. Pathlines colored by non-dimensional spanwise velocity for a flat temperature profile and 1% coolant mass flow from the slot.

Figure 9. Pathlines colored by non-dimensional spanwise velocity for a flat temperature profile and 2% coolant mass flow from the slot.
Figure 10a-b. Endwall film-cooling predictions for a case with a flat temperature profile. These predictions include endwall adiabatic effectiveness levels, (a) and vane effectiveness levels along the suction side (b).
Figure 10c-d. Endwall film-cooling predictions for a case with a flat temperature profile and 1% slot cooling. These predictions include streamline pattern in the near-wall region colored by the spanwise velocity component.
Figure 11. Temperature profiles for the hot-streak with a single heater on. Distances measured downstream of the heater. Positions $x/C = 0.77$ and $1.38$ are upstream of the vane, and $x/C = 3.0$ is downstream of the vane.

Figure 12. Schematic the simulated vane test model.
Figure 13. Adiabatic effectiveness with roughness section upstream of the cooling holes. Low mainstream turbulence and no showerhead blowing.
Figure 14. Adiabatic effectiveness with roughness section downstream of the cooling holes. Low mainstream turbulence and no showerhead blowing.
Figure 15. Adiabatic effectiveness with roughness sections upstream and downstream of the cooling holes. Low mainstream turbulence and no showerhead blowing.
OPTIMIZATION OF THE INJECTOR FUEL DISTRIBUTION FOR STABLE, LOW EMISSIONS COMBUSTION IN LEAN PREMIXED GAS TURBINE COMBUSTORS

Semi-Annual Progress Report
(Report Period: 2/01-8/01)

D.A. Santavicca, Principal Investigator
Kwanwoo Kim, Graduate Student
Jongguen Lee, Research Associate

The Pennsylvania State University
University Park, PA 16802

Subcontract No. 01-01-SR090
South Carolina Institute for Energy Studies
Clemson, SC 29634-5712

December 2001

This report was prepared with the support of the U.S. Department of Energy, under Cooperative Agreement No. DE-FC21-92MC29061. However, any opinions, findings, conclusions, or recommendations expressed herein are those of the authors and do not necessarily reflect the views of the DOE.
Research during the first six months of Subcontract No. 01-01-S090 has focused on a study of the effect of the combustor inlet fuel distribution on the stability and emissions characteristics of a laboratory dump combustor. This study was conducted in the optically accessible dump combustor shown schematically in Figure 1. As illustrated, this combustor provides for the fuel to be introduced at one or more of six different locations, thereby allowing for significant variations in the fuel distribution. At location (1) the fuel is introduced well upstream of the choked inlet to the mixing section. Fuel introduced at this location is thoroughly mixed with the air before entering the mixing section and results in a perfectly uniform fuel distribution at the combustor inlet. At location (2) the fuel is introduced through 4 injectors (diameter = 0.81 mm) located in the outer wall of the mixing tube at a position 110 mm upstream of the dump plane. This location is downstream of the choked inlet to the mixing section and upstream of the axial swirlers in the mixing section. At location (3) the fuel is injected into the mixing section through 12 holes (diameter = 0.34 mm) located around the circumference of the centerbody, 25 mm upstream of the dump plane. By varying the fuel split between these three injection locations, the spatial fuel distribution at the inlet to the combustor can be varied. Locations (3), (4) and (5) provide for what is called “targeted” injection and are intended to allow for fuel to be introduced directly into the combustor at locations which are likely to enhance flame stabilization. At location (4) the fuel is injected through 12 holes (diameter = 0.34 mm) equally spaced around the inner circumference of the dump plane. At location (5) the fuel is injected through 12 holes (diameter = 0.34 mm) equally spaced around the outer circumference of the end-face of the centerbody and angled at 45 degrees. At location (6) the fuel is injected through a single 1.18 mm diameter hole on the axis of the centerbody. These six injection locations not only provide a means to vary the spatial fuel distribution, but modulating the fuel flow to any of these injection locations also provides a means for varying the temporal fuel distribution.

![Figure 1. Schematic Drawing of Laboratory Dump Combustor Illustrating Various Fuel Injection Locations](image-url)
During this reporting period the effect of changing the inlet fuel distribution, by using various combinations of injection locations (1), (2) and (3), on the stability and emissions characteristics of the combustor was investigated. The operating conditions over which these tests were conducted are summarized in Table 1. The inlet velocity given in Table 1 refers to the bulk velocity in the combustor at the indicated combustor pressure and inlet temperature. The combustor to mixing section area ratio is 13.4, therefore the velocities in the mixing section are 13.4 times greater than the indicated combustor inlet velocities.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>pressure</td>
<td>1 atm</td>
</tr>
<tr>
<td>inlet temperature</td>
<td>300°, 350°C</td>
</tr>
<tr>
<td>inlet velocity</td>
<td>3.5, 5.0, 6.5 and 8.0 m/s</td>
</tr>
<tr>
<td>swirl</td>
<td>30°</td>
</tr>
<tr>
<td>equivalence ratio</td>
<td>LBO to 0.8</td>
</tr>
<tr>
<td>fuel</td>
<td>natural gas</td>
</tr>
<tr>
<td>power</td>
<td>40 - 180 kW</td>
</tr>
<tr>
<td>fuel distribution</td>
<td>see figure 2</td>
</tr>
</tbody>
</table>

Table 1. Operating Conditions

The fuel distribution at the inlet to the combustor, i.e., the annular exit of the mixing section, was varied by changing the fuel split between injection locations (1), (2) and (3). For the tests conducted during this reporting period the 7 fuel split cases listed in Table 2 were used. For each of these cases the actual fuel distribution across the exit of the annular mixing section was measured using laser induced fluorescence where the fuel was doped with a small amount of acetone vapor which served as a fluorescence tracer. Fuel distribution measurements for fuel split cases 1, 2 and 3 are presented in Figure 2 for an inlet velocity of 6.5 m/s, an inlet temperature of 100°C and an equivalence ratio of 0.55. As expected, case 3, where all of the fuel is introduced at location (1), results in a uniform fuel distribution. The results for cases 2 and 1 show that fuel injected through the centerbody at location (2) penetrates to the outer wall of the mixing tube, resulting in a skewed fuel distribution. In case 1, where all of the fuel is injected at location (2), the equivalence ratio appears to vary almost linearly from about 0.25 to about 0.85 from the inner to outer diameter of the mixing section. Figure 3 shows the effect of fuel injected at location (3) on the fuel distribution. As shown, there appears to be little effect on the fuel distribution, even when all of the fuel is injected at location (3), i.e., case 7. Although the fuel distribution is the same for injection locations (1) and (3), there is an important difference between these two injection locations. Since location (1) is upstream of the choked inlet to the mixing section it is isolated from the effect of pressure fluctuations in the mixing section, i.e., so-called feed system coupling. Injection location (3), however, will be affected by pressure fluctuations in the mixing section and therefore susceptible to feed system coupling. This may have a significant effect on the stability characteristics.
### Table 2. Fuel Splits Cases

<table>
<thead>
<tr>
<th>Fuel Distribution</th>
<th>Injection Location (1)</th>
<th>Injection Location (2)</th>
<th>Injection Location (3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0%</td>
<td>100%</td>
<td>0%</td>
</tr>
<tr>
<td>Case 2</td>
<td>50%</td>
<td>50%</td>
<td>0%</td>
</tr>
<tr>
<td>Case 3</td>
<td>100%</td>
<td>0%</td>
<td>0%</td>
</tr>
<tr>
<td>Case 4</td>
<td>75%</td>
<td>0%</td>
<td>25%</td>
</tr>
<tr>
<td>Case 5</td>
<td>50%</td>
<td>0%</td>
<td>50%</td>
</tr>
<tr>
<td>Case 6</td>
<td>25%</td>
<td>0%</td>
<td>75%</td>
</tr>
<tr>
<td>Case 7</td>
<td>0%</td>
<td>0%</td>
<td>100%</td>
</tr>
</tbody>
</table>

**Figure 2.** Laser Induced Fluorescence Fuel Distribution Measurements for Fuel Split Cases 1, 2 and 3.

**Figure 3.** Laser Induced Fluorescence Fuel Distribution Measurements for Fuel Split Cases 3, 5 and 7.
At each test condition the pressures in the mixing section, the fuel line and at several locations in the combustor were measured with high frequency response pressure transducers. In addition, the exhaust gases were sampled and analyzed with a chemiluminescence NO\textsubscript{x} analyzer. The results are then plotted in terms of stability and emissions maps, where stability is expressed in terms of the peak-to-peak pressure fluctuation and emissions are expressed in terms of the NO\textsubscript{x} concentration.

The stability map, based on the combustor pressure measured at the dump plane, for one operating condition (6.0 m/s inlet velocity, 350\textdegree C and 30\textdegree swirl), is shown in Figure 4. Fuel split case 3 can be considered the reference case. In case 3 all of the fuel is injected and mixed with the air upstream of the choked inlet to the mixing section, which as shown in Figure 2 results in a uniform fuel distribution. In this case the combustor is stable for equivalence ratios between 0.6 and 0.75, but becomes unstable at equivalence ratios below 0.6.

The remaining cases can be separated into two groups, those where the fuel is split between injection locations (1) and (2) and those where the fuel is split between injection locations (1) and (3). When 50\% of the fuel is injected at location (2) the stability characteristics are observed to improve (case 2), however, when all of the fuel is injected at location (2) the combustor is unstable at all equivalence ratios (case 1). Since fuel injection at location (2) results in fuel lean conditions along the centerbody (see Figure 2) where the flame is stabilized, it is not surprising that in case 1 the combustor is more unstable. The improved stability in case 2, however, requires further study.

In cases 4 through 7, increasing amounts of fuel are injected at location (3). As shown in Figure 3, the fuel distribution is the same in cases 3 through 7, however, the stability characteristics change as the fraction of fuel injected at location (3) is increased.
In particular, the combustor becomes very unstable when 50% or more of the fuel is injected at location (3). One possible explanation for this is feed system coupling since injection location (3) is downstream of the choked inlet to the mixing section and therefore can be affected by pressure oscillations in the mixing section. This will be investigated in more detail during the next reporting period.

In summary, the stability results shown in Figure 4 clearly demonstrate the significant effect that the fuel distribution can have on the stability characteristics. These results suggest that by changing the fuel distribution one may actually be able to avoid instabilities. Stabilities are obviously not the only concern in lean premixed combustion systems. One also must understand the effect of the fuel distribution on emissions. Figure 5 shows the NO\textsubscript{x} emissions at the same operating conditions for which the stability data were presented. As expected there is a significant decrease in NO\textsubscript{x} emissions as the equivalence ratio decreases, but what is of greater interest is the effect of the fuel distribution among the seven fuel distribution cases. If one associates incomplete mixing with increased NO\textsubscript{x} emissions, one would expect the cases with a uniform fuel distribution, i.e., cases 3 through 7, to have the lowest NO\textsubscript{x} emissions for a given equivalence ratio. Figure 5 certainly shows that as the fuel distribution becomes less uniform, as it does in going from case 3 to case 2 to case 1, that the NO\textsubscript{x} emissions increase. The somewhat surprising result in Figure 5 is the fact that even though cases 3 through 7 all have a uniform fuel distribution, the NO\textsubscript{x} emissions first decrease in going from case 3 to 4 and then increase in going from cases 4 to 7, as an increasing amount of fuel is injected at location (3). This is likely to be related to the fact that the combustor becomes more unstable as a greater fraction of the fuel is introduced at location (3). This behavior will be studied more closely in the next reporting period.

Figure 5. NO\textsubscript{x} Emissions Map for a 5 m/s Inlet Velocity, 350\textdegree C Inlet Temperature, 30\textdegree Swirl Operating Condition and Fuel Split Cases 1 through 7.
Future Work

During the next reporting period work will continue to determine the effect of fuel distribution on the stability and emissions characteristics of the dump combustor illustrated in Figure 1. These tests will be extended to a pressure of 2.5 atm, inlet temperatures of 400°C and 450°C, swirl angles of 0° and 60°, and one or two new fuel distributions. In addition, targeted injection will be investigated, i.e., using locations (4), (5) and (6). Once the effects of fuel distribution and fuel targeting on stability and emissions are thoroughly documented the focus of the research will shift to attempting to understand the causes of the observed instabilities and the relationship between this behavior and the fuel distribution/targeting. This phase of the study will likely begin during the second year of this project and will be based on the use of phase-synchronized chemiluminescence imaging measurements and IR absorption equivalence ratio fluctuation measurements.
Status/Accomplishments for the Report Period

The goal of the ATS program is to reduce the emission of air pollutants while maintaining the high levels of efficiency of the combustion process achieved by the gas turbine industry. The research under way here at UC Berkeley has the goal of providing the gas turbine industry with information and tools to help them make informed decisions on combustion designs and strategies. Our research project has four tasks, largely directed at the Technical Objective of Combustion Area – 1, Sensors and Diagnostics, part b) which states “Diagnostic instrumentation of use in perfecting premixing fuel preparation of systems are of interest.” The progress to date on each of these four tasks will be discussed in the following sections of this report.

Task 1 – IR-LED:

Visible light emitting diodes (LEDs) are very common, but they have only recently become available in the infrared. These IR LEDs are available in various infrared bands including a band centered at a wavelength of 3.3 microns. UC Berkeley has had success in quantifying hydrocarbon concentration by using a 3.39 micron HeNe laser as an optical probe. It may be possible to replace the laser with a solid state 3.3 micron LED. The advantages of LEDs over use of the HeNe laser are lower cost (currently LEDs are about $400 each vs. $1500 each for the HeNe Laser), increased ruggedness and reduced size.

Status: Experiments, both at vacuum and high pressures, have been performed using a 3.3 µm LED. These experiments incorporated a 24” pathlength cell at varied pressures to determine the effect of pressure on overall transmission of light from the LED, through various concentrations of methane in nitrogen. The absorption cell was vacuum-sealed with sapphire windows on each end through which the LED light was passed. The cell had a relatively long pathlength to assure significant (≈ 50%) attenuation of the LED light occurred. In order to
measure LED radiation with such a long pathlength, a chopper wheel was used with a phase-lock amplifier. In this way, the very low levels of radiation from the LED are measured (the phase-lock amplifier filters out all components of the detector signal that are not near the frequency of the chopper wheel, and amplifies components of the signal that are close in frequency to that of the chopper wheel; in this way, very small signals can be extracted in spite of large levels of noise).

At the same output power, the LED has a weaker signal at the detector due to the wide divergence of the LED light, compared to a laser. Thus, unlike the laser, most of the light emitted by the LED is not received at the detector.

Figure 1, below, is a schematic of the test setup used for characterizing the transmission of the LED vs. pressure for a given concentration of methane. Focusing mirrors were also used to reduce the effect of distance on measured LED radiation. The absorption cell shown in Figure 1 was a vacuum-tested cylinder with sapphire windows on either end. The sample gases were passed through the absorption cell at controlled pressures.

Figure 1: Schematic of IR LED Test Stand. The Visible Laser Was Made To Be Collinear with the LED and Was Used for Alignment.
The results of high pressure transmission (Intensity $I$ divided by baseline intensity $I_0$) vs. cell pressure are given in Figure 2. As can be seen, for 5% methane the absorption coefficient for the IR LED is about an order of magnitude smaller than that for a 3.39 $\mu$m He-Ne laser. This demonstrates that fuel concentration is measurable with this instrument at pressures and air-fuel ratios found in lean premixed gas turbine combustors. This is an important and significant result, in that measurements of the fuel concentration can be performed at pressures typical of lean premixed gas turbines (ranging from 6 to 15 bars). Transmission from a 3.39 $\mu$m He-Ne laser would be too low for accurate concentration measurements for methane at these typical gas turbine concentrations and pressures for any pathlength of around 1 cm or greater. It was also shown that the amount of absorption (absorption is one minus transmission) increases with pressure for a given fuel concentration.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{Absorption_Coefficient_vs_Pressure_IR_LED_and_3.392_\mu m_Laser.png}
\caption{Absorption Coefficient vs. Pressure, IR LED and 3.392 $\mu$m Laser}
\end{figure}

**Figure 2: Absorption Coefficient of IR LED vs. Pressure**

As part of this work, we are talking with Solar Turbines, Inc. (San Diego, CA) about performing tests with the LED in one of their pressurized test cells, to characterize the mixing performance of a production lean premixed fuel injector. These tests will make use of a pressurized optical access test rig that Solar Turbines has recently installed. With our past experimental results showing that the LED absorption by methane is improved at higher pressures, it is thought that performing tests at these high pressures would be ideal for the infrared LED.
Task 2 – Tomographic Reconstruction:

We are confident that the IR-HeNe laser can be used to rapidly determine (within say ten seconds) the mean concentration of fuel at, for example, the entire exit of a premixer; by using tomographic reconstruction. This is not new. However, what has not been reported, but should be feasible, is to use tomographic reconstruction for determination, spatially resolved, of the RMS of fuel concentration at, for example, the entire exit of a premixer. The assumption of axisymmetry would not be needed.

Status: Work is currently underway to adapt a genetic algorithm program to calculate concentration profiles and concentration rms profiles of a coaxial pipe flow from line of sight measurements at multiple angles. Results of this work have been reported in a paper submitted to the 29th International Symposium on Combustion to be held in Sapporo, Japan, July 21-26, 2002. This paper, giving details about the tomographic reconstruction and the LES results, is attached as Appendix A.

Task 3 – Droplets:

We have previously developed a low cost IR-HeNe laser probe that measures the spatial and temporal fuel concentration in premixers. In this task, we wish to extend the probe to two-phase flows so that we can measure the appearance of droplets as well as determine the gas phase fuel concentration. We will do this by superposing the beam from a red HeNe laser with the beam of the IR HeNe laser. Droplets will diminish both beam intensities while fuel vapor will diminish only the IR laser beam.

Status: Significant results on this Task were given in a previous semiannual report (covering June 1 to December 30 2000), and in a paper submitted to the 2nd Joint Meeting of the U.S. Sections of the Combustion Institute as Paper Number WSS/CI 2001-246. This paper was attached to a previous semiannual report (covering July 1 to December 31 2001) as an appendix.

Task 4 – CFD of Transient 3-D Mixing Process:

The mixing quality is strongly influenced by the temporal as well as spatial distribution. Current CFD largely ignores transient phenomena and often the three-dimensional effects are not computed in most Reynolds Averaged Navier-Stokes (RANS) applications.

Status: Significant progress on LES was reported as a Poster at the Nov 1999 DoE ATS meeting held in Pittsburgh, PA. Graphic details are available in the Proceedings or from the above Author directly. Since then, we have modified the boundary conditions of the LES model at UC Berkeley to simulate the turbulent coaxial pipe flow setup briefly described under Task 2, above. The setup attempts to obtain fully developed flow for both the fuel flow and the surrounding air flow. Comparisons between the experimental results from this setup and the LES model results have been performed. These comparisons have been submitted as a paper to the 29th International Symposium on Combustion to be held in Sapporo, Japan, July 21-26, 2002. This paper, giving details about the tomographic reconstruction and the LES results, is attached as Appendix A.
APPENDIX A

Fuel-Air Mixing in a Turbulent Coannular Pipe Flow Measured with Laser Absorption with Genetic Algorithm-Based Tomographic Reconstruction and Modelled with LES

(Submitted to the 29th International Symposium on Combustion, Sapporo, Japan, July 21-26, 2002)
Fuel-Air Mixing
In A Turbulent Coannular Pipe Flow
Measured With Laser Absorption
With Genetic Algorithm-Based Tomographic Reconstruction
And Modelled With LES

JAMES W. GIRARD, J-Y CHEN, ROBERT W. DIBBLE, JOHANNES JANICKA\textsuperscript{1} and RYOJI HOMMA\textsuperscript{2}

\textit{University of California, Berkeley}
50B Hesse Hall
Berkeley, CA 94720, USA

\textsuperscript{1}\textit{Institute of Energy- and Powerplant-Technology}
\textit{Technical University of Darmstadt}
Darmstadt, Germany

\textsuperscript{2}\textit{Tokyo Gas Co., Ltd.}
16-25 Shibaura, 1-Chome, Minato-ku, Tokyo, Japan, 105-0023

Colloquium on Turbulent Premixed Combustion
Twenty-Ninth International Symposium on Combustion

Corresponding Author: James W. Girard
Combustion Lab
50B Hesse Hall
Univ. of California, Berkeley
Berkeley, CA 94720, USA
Tel: (510) 642-9158
Fax: (510) 642-1850
e-mail: girardj@newton.berkeley.edu

Total Word Count (Counted by MS Word; Figures, Tables Counted According to Submission Guidelines): 3254+1400 (figs.)+400(tables)+198(eqn.s)+238(ref.s)=5490
Fuel-Air Mixing In A Turbulent Coannular Pipe Flow
Measured With Laser Absorption With Genetic Algorithm-Based
Tomographic Reconstruction And Modelled With LES

JAMES W. GIRARD, J-Y CHEN, ROBERT W. DIBBLE,
JOHANNES JANICKA\textsuperscript{1} and RYOJI HOMMA\textsuperscript{2}

University of California, Berkeley
50B Hesse Hall
Berkeley, CA 94720, USA

\textsuperscript{1}Institute of Energy- and Powerplant-Technology
Technical University of Darmstadt
Darmstadt, Germany

\textsuperscript{2}Tokyo Gas Co., Ltd.
16-25 Shibaura, 1-Chome, Minato-ku
Tokyo, Japan, 105-0023

ABSTRACT

This research advances the use of 3-D LES to simulate the mixing of fuel with air. LES predictions, including the mean and RMS of the fuel concentration are compared to measurements of the mean and RMS of fuel concentration. These time resolved measurements are tomographically reconstructed from time resolved LOS laser absorption measurements. The measurements were made at several locations downstream of where a central round pipe discharged fuel into a surrounding co-flow of air.

The LES model predicts the extent of mixing quite well.

A novel aspect of this research is the use of tomography to generate not only the mean concentration, but also the RMS of concentration. A genetic-algorithm was developed for this reconstruction.
INTRODUCTION

Many practical combustion devices premix fuel and air to lean conditions prior to combustion. Such devices include spark ignited piston engines and dry-low-NO\textsubscript{x} (DLN) premixed gas turbine combustors. By using excess air (or EGR) the flame temperature is reduced. The lower temperatures greatly reduce the rate of chemical reactions that produce the toxic pollutant nitric oxide. For example from the gas turbine industry, steam injected nonpremixed combustors have achieved 25ppm while the DLN lean premixed combustors have been able to achieve NO\textsubscript{x} levels from 15 to 10 ppm at 15% O\textsubscript{2} [1,2]. In order for a lean premixed combustor to effectively reduce NO\textsubscript{x} levels, the air and fuel should be well mixed prior to the combustion event. A well mixed system is characterized by a mean fuel concentration, that is lean, with a small RMS about the mean. Inadequate fuel-air mixing can have the same mean, but has a larger RMS such that, occasionally, combustion may take place at near-stoichiometric air-fuel ratios at some time, while at other times the mixture is too lean to burn. The sporadic combustion at near-stoichiometric air-fuel ratios has higher than average temperatures. These high temperatures lead to very high NO\textsubscript{x} levels due to the well known exponential temperature dependence of the production rates of oxides of nitrogen (NO\textsubscript{x}) [3]. In addition, the incomplete combustion at temporally lean air-fuel ratios leads to high levels of hydrocarbon (HC) and carbon monoxide (CO) emissions. Additionally, lean premixed combustors are prone to high pressure oscillations, and the extent of mixing of the fuel and air has been shown to correlate with these oscillations in combustor pressure [4,5]. Thus, in the design of a lean premixed gas turbine, it is essential that the mixing of fuel and air is well characterized; minimally one needs a mean and RMS of fuel concentration.
The goals of the current research are two-fold. First, to study the performance of a Large Eddy Simulation (LES) for a simplified geometry as a step toward application of LES modeling to an actual lean premixed combustor. Simply stated, does the LES predict the mean and RMS fuel concentration that is measured. The second goal is to study the ability of generating both the mean and RMS of fuel concentration, at the exit of a premixer, from measurements obtained using a robust easy-to-use line of sight (LOS) laser absorption technique.

For the current research, this allows us to use experimental measurements of mean and RMS to characterize the performance of the LES model. As a future goal, we imagine this simple, but powerful diagnostic can be easily used for studies of gas turbine premixers. Reconstruction of a 2-dimensional field from a single (assuming axis-symmetry) or multiple 1-dimensional measurements (i.e. tomography) is very common. Some applications other than concentration measurements include reconstruction of density of human head tissue from x-ray absorption measurements [6], and reconstruction of particle speed distributions from imaging measurements in the reaction product imaging technique [7]. What is important about the current research is that the method has the potential to reconstruct an asymmetric concentration field with reasonable accuracy from a limited number of experimental measurements, and that we apply the technique to the RMS of the concentration profile. The research described in this paper consists of developing a genetic algorithm (GA) for tomography, applying it to some numerical “test cases” for validation, then using it on data from a turbulent coannular pipe flow to assess the ability of an LES model to resolve the fuel-air mixing spatially and temporally.

**EXPERIMENTAL SETUP/PROCEDURE**

In most combustion models, such as Reynolds Averaged Navier-Stokes (RANS), time resolved details are not predicted. At the other end of the spectrum, direct numerical simulation (DNS) is
a model that provides many details of the flow, resolved both temporally and spatially. However, due to the vast computational effort involved with DNS, it is usually only applied to simple flows. LES models are truncated DNS models that use subgrid models for small-scale turbulence to reduce the computational costs (see, for example Branley and Jones, [8]). LES gives time resolution in a 3-dimensional simulation, unlike most combustion models, but requires much more computer resources. Table 1 gives the input and grid parameters of current LES and details can be found [9].

In order to evaluate the performance of the LES mixing model, a coannular pipe flow setup was constructed, consisting of a center pipe flow of fuel, surrounded by a pipe flow of air. The diameter of the outer pipe was $D = 7.6$ cm, while that of the inner pipe was $d = 6.4$ mm. Thus the diameter ratio of the pipe flow setup was $d/D = 0.084$. The center fuel pipe was 3.3 meters in length, while the air pipe length varied from 3.3 to 3.5 meters. The long fuel pipe meant that there was a nearly fully developed coannular pipe flow. Figure 1 gives a schematic of the setup.

Laser absorption measurements were made at the exit of the outer (air) pipe. In this way measurements at different axial distances from the center (fuel) pipe exit were accomplished by changing the length of the outer pipe. Axial distance from the center pipe exit will be referred to as small $x$, and $x/d$ will refer to this axial distance normalized by the diameter of the center pipe. Measurements of the fuel concentration were obtained by the use of a Helium-Neon laser at a wavelength of $3.392 \, \mu\text{m}$. Hydrocarbons absorb radiation at this wavelength, and Beer’s law can be used to relate the amount of absorption to the concentration of fuel (see, e.g. Lee et al. [10], Yoshiyama et al.[11], Perrin and Hartmann, [12], Mongia [13], or Ebert et al. [14]). At standard
temperature and pressure, the absorption of laser light by a concentration of molecules follows Beer’s Law:

$$\frac{dI}{d\ell} = -I \cdot \alpha \cdot P_{abs} \cdot X_{CH_4}(\ell),$$  

(1)

where $I$ denotes intensity of laser radiation, $\alpha$ the absorption coefficient for given fuel ($atm^{-1}cm^{-1}$), $\ell$ the coordinate along laser beam path ($cm$), $P_{abs}$ the absolute pressure ($atm$), and $X_{CH4}$ the fuel mole fraction.

Integration of Eqn. 1 yields

$$\int \frac{dI}{I} = -\alpha \cdot P_{abs} \cdot \int X_{CH_4}(\ell) \cdot d\ell,$$  

(2)

$$\ln(I/I_o) = -\alpha \cdot P_{abs} \cdot \int X_{CH_4}(\ell) \cdot d\ell,$$  

(3)

where $I_o$ is the initial (un-attenuated) laser radiation intensity. If the concentration is assumed constant across the measurement length, one obtains

$$X_{CH_4} = -\frac{\ln(I/I_o)}{L(\alpha \cdot P_{abs})},$$  

(4)

where $L$ is the pathlength of absorption ($cm$). The absorption coefficient $\alpha$ is dependent on temperature, pressure, and the type of fuel being characterized. The absorption coefficient for methane was measured by Perrin and Hartman [12] to be approximately $\alpha = 10 \ atm^{-1}cm^{-1}$ at standard temperature and pressure. The absorption coefficient for ethane is taken from the results of Tsuboi et al. [15] to be approximately $5 \ atm^{-1}cm^{-1}$. As can be seen from Eqn. 3, laser absorption measurements can only give an integrated fuel concentration over the path of the laser beam. In our application, the laser beam was passed across the pipe exit, at the same orientation, at various cords. This is shown in Fig. 1. In order to generate a radial profile of concentration
from the laser beam absorption data, a computer aided tomography (CAT) program using a genetic algorithm was employed. The reconstruction program will be described in detail in the following section.

After demonstrating that the experimental measurement system was reasonably accurate, radial concentration profiles were generated at various downstream locations for comparison with the LES model. Table 2 describes the operating conditions at each of the sampling points. The flow rates shown in Table 2 were set with the goal of making the Reynolds number as high as possible, and of matching the average velocities of the inner and outer pipe flows. For the experiments described in the table, the nominal velocities for the inner and outer pipe flows were 26 to 27 m/s (making the friction velocities for the inner and outer pipe flows around 1.6 and 1.2 m/s, respectively).

GENETIC ALGORITHM FOR TOMOGRAPHY

As discussed above, the experimental data consisted of LOS absorption measurements. In order to compare the experiments with the LES model results, one can find radial concentration profiles from these line-of-sight measurements (or one could determine what LOS absorptions would be obtained given the LES generated concentration profiles). For absorption of laser light with a concentration that is not constant across the beam, Eqn. 3 applies. For a radial concentration profile, the concentration is dependent on the radius from the center r for the given segment (see Fig. 1 for orientation of z and r). This is an integral equation of the first kind [16].

\[
g(z) = \int_{z}^{R} K(z,r) \cdot f(r) dr ,
\]

where R is radius of the outer (air) pipe, z the distance of a given LOS measurement from the center of the pipe, g is \(\ln(I/I_o)\), the natural log of the intensity ratio of a given cord (a function of
$K(z,r) = 2r/(r^2-z^2)^{1/2}$ the Parameter relating optical pathlength to radial location, $r$ is the radial distance from the center-point, and $f(r) = -\alpha \cdot P_{abs} \cdot X_{CH4}$.

The solution to this equation is not trivial. Typically, this equation is solved by reconstruction techniques such as an Abel inversion or Fourier deconvolution [17,18]. Looking at Eqn. 3 it is seen that if we knew the concentration profile it would be relatively easy to calculate the LOS intensity ratios that would be generated from it. This is accomplished by breaking the LOS measurement into segments at a small enough scale that assuming the concentration across a segment is constant would result in little error. Eqn. 6 shows an example for a measurement line:

$$\ln(I/I_0) = -\alpha \cdot P_{abs} \cdot \sum_{i=1}^{n} X_{CH4,i} \cdot \Delta \ell_i,$$

where $\Delta \ell_i$ is the distance across segment $i$ and $X_{CH4,i}$ is the averaged concentration over segment $i$.

For this research we have come up with a GA to reconstruct the time averaged concentration profile of an axis-symmetric jet from LOS measurements. Additionally, we modified the program to allow reconstruction of the RMS concentration profile from RMS values of the intensity ratios, and further modified the code for an asymmetric jet. Kihm et al. [19] have applied a genetic algorithm based program to the case of tomographic reconstruction of a time-averaged concentration field and found that it performed well with a small number of measurements. In the interest of brevity, we refer the reader to Powel and Skolnick [20] and Kihm et al. [19] for a discussion of the principles of GAs.

The reconstruction of radial concentration profiles (mean and RMS) from our experimental line of sight measurements is desired. In order to test the ability of the GA to
reconstruct the time-averaged and RMS concentration profiles, two sets of data were used where the correct answer was known a priori. The first set of test data assumed that the fuel concentration varied as a function of radial position with the form of an inverted parabola. The equation for this concentration profile was known, and thus the solution to Eqn. 1 can be found analytically. For these cases, the intensity ratios for the absorption lines can be found for the concentration profile and the results of the GA can then be compared to the known answer. The results of this comparison are shown in Fig. 2. As can be seen from the figure, the genetic algorithm performed satisfactorily at reconstructing the concentration field for a parabolic input.

The GA program was modified to reconstruct the RMS of the radial concentration profile based on the RMS of the measured LOS intensity ratios. The theory involved in this procedure is that of propagation of error. Eqn. 6 describes the relation between the natural log of the intensity ratio for a given absorption line and the concentration field. Thus the natural log of the intensity ratio, ln(I/I₀), is a linear function of the time-dependent concentrations of each radial position. The RMS of ln(I/I₀) then follows Equation 7 (adapted from Beckwith et al. [21]).

\[
RMS_{\ln(I/I_0)} = \left\{ \sum_{k=1}^{K} \left( \frac{\partial (\ln(I/I_0))}{\partial X_k} \times RMS_{\ln(X_k)} \right)^2 \right\}^{1/2}
\]  

Equation 7 assumes that the time-varying concentration at each radial position is independent of the time-varying concentration at every other radial position. If this independence does not exist, there will be correlation terms, and the relation between RMS of ln(I/I₀) will no longer be as in eqn. 7. We can apply the same type of program to Eqn. 7 to reconstruct the RMS concentration profile given the RMS values of ln(I/I₀). Because our sampling rate for the LOS measurements was high enough to see the time-dependent features of
the flow (sampling rate was 10 kHz), we can determine the RMS values of \( \ln(I/I_o) \) based on fluctuations whose periods are 200 \( \mu \)seconds or greater.

We modified the parabolic concentration profile to include a sinusoidal fluctuation in order to test the capability of the genetic algorithm scheme for reconstruction of the RMS of the concentration field. The equation for the time-dependent concentration field is then of the form:

\[
X(r,t) = V_A(r) \sin^2(\omega t),
\]

where \( X(r,t) \) is the time-dependent concentration, \( V_A \) the Amplitude (previous, non-time-dependent concentration field) and \( \omega \) the frequency. The square of the sinusoidal function was used so the concentration would not take on negative values. The RMS of \( X(r,t) \) is then found analytically to be

\[
RMS \ of \ X(r,t) = V_A(r) \cdot (3/8)^{1/2}.
\]

We therefore know the RMS of the concentration field analytically, and it is also easy to find the RMS of the intensity ratio for the given absorption lines for this case, because if \( X(r,t) \) follows the form of Eqn. 8, one can integrate Eqn. 3. Thus, we can check the performance of the GA for reconstructing the RMS of the concentration profile for this case. These results are also shown in Fig. 2.

It can be seen from Fig. 2 that the GA successfully reproduces the right overall trend for the profile of the concentration RMS. However, small departures from the actual RMS values are noticeable. This departure is due to the fact that the time-dependent concentration value at a given radial position is correlated to all other positions, so that there will be some error in applying Eqn. 7. This was done intentionally as a test of how well this reconstruction scheme would handle correlated data. Our analytical test showed that the correlated data gave roughly the same answer as uncorrelated data to within 30%. With the current treatment of the problem
of reconstructing the signal RMS, we find no direct way of overcoming error due to the
interdependence of the concentration values. A method that would remove this problem would
be to use the time dependent data, reconstructing the concentration field at each time step from
the experimental results at that time step, and repeating this for around 200 time steps.

A second set of numerical tests was done to further explore the capabilities of the GA
program to reconstruct the concentration field. These tests consisted of generating a set of time-
resolved LES results, and calculating the RMS and time-averaged fuel concentration profile.
From these, the intensity ratios that would give the concentration profile were found at each time
step using Eqn. 6, then the RMS of the intensity ratio was calculated from these values. 200 time
steps of LES output data were used. The intensity ratios for the time-averaged concentration
profile were used as an input to the genetic algorithm, and the results are given in Fig. 3. From
Fig. 3 we see that the GA once again performs satisfactorily at reconstructing the radial profile of
time-averaged concentration, with the center-point concentration slightly underpredicted.

Fig. 3 also gives the reconstructed profile of the concentration RMS. Once again, the
general trends are represented and the values are correct to within 30% of the actual values.
Again the departures are felt to be due to the fact that the time-varying concentration values at
each radial position are not completely independent. The results of the GA tend to give adequate
agreement for the time-averaged profiles and modest agreement for the RMS profiles (Figs. 2
and 3). Therefore the GA technique was employed for quantitative evaluation of the LES model
for time-averaged data, and for semi-quantitative evaluation of the LES for the time-dependent
results.

We then took the additional step of modifying the program to enable reconstruction of an
asymmetric concentration field. This would of course require more measurements - taken at
more than one angular position at the pipe exit. A Gaussian function with a peak at 0.13 was used for the concentration field, but with the peak of the Gaussian shifted off-center by 0.5 cm. The 2-D version of the code uses multiple angles of data (6 angles in the present state) corresponding to 60 input measurements to optimize. The results of using line of sight data that would correspond to a shifted Gaussian curve (not shown) demonstrated that given enough data points, this method can successfully be used to reconstruct an asymmetric concentration profile. Future work may apply this program to 2-dimensional sets of measurement data.

**LES MODEL VS. EXPERIMENT**

Velocity measurements were performed and compared with the LES model results. One set of velocity profiles are presented in Fig. 4. The velocity profile predicted by the model agrees approximately with the measured velocity profiles. Fig. 5 presents the results of the LES prediction vs. experiment for several axial locations (x/d, where x is axial distance and d is the diameter of the inner pipe) with methane as the fuel. From this figure, one can see that the LES model results agree with experiment reasonably well. Comparisons were also done with Ethane as the fuel at one mixing length (x/d=24) in order to assure that the effects of buoyancy on the system are negligible. As the Froude number is much greater than one for both the methane and ethane experiments (15,800 for the methane jet vs. 11,500 for ethane), momentum rather than buoyancy dominates the flow [22], so that little difference in the mixing would be expected upon changing the fuel. The results (not shown), both experimentally and from the LES model, showed insignificant effects of buoyancy on the concentration profile.

Figure 6 shows a comparison of concentration RMS vs. radial position for the GA program based on experimental line-of-sight intensity ratios, and for the LES model. As can be seen from the figure, the RMS values obtained by the two methods are in reasonable agreement.
for the most part (given the error in finding the RMS concentration profile with the GA program, seen earlier in Figs. 2 and 3). It can be seen that the reconstructed concentration profile tends to underpredict the RMS concentration. The finite diameter of the laser beam will contribute to this. The Rayleigh range for the given laser is approximately 500 mm, and with the optical setup used, the beam diameter is 2.2 mm - defined as the diameter of the beam containing 86% of the laser power [23]. Thus, some smearing of the actual fluctuations of fuel concentration would occur, causing the measured signal RMS to be lower than what is really there (thus while temporal resolution was adequate, spatial resolution was marginal). By using an optical system that focuses the beam at the measurement point [23], the resolution of the measurements for finding the RMS of the concentration would be improved.

SUMMARY CONCLUSIONS

1.) The GA-based tomography program is successful at calculating the time-averaged radial concentration profile from line of sight absorption measurements.

2.) The GA reconstructions of concentration RMS are quantitatively correct, giving error at some points up to 30% of the actual value for known test cases, but reproducing trends well.

3.) The LES model predicts the extent of mixing vs. downstream location quite well compared to the reconstructed experimental results, for both methane and ethane fuel.

ACKNOWLEDGEMENTS

The authors wish to thank the Department of Energy’s Advanced Gas Turbine Systems Research Program for providing the funding for this research.
REFERENCES


Figure Captions

Fig. 1 Experimental Setup for Coannular Pipe Flow Mixing Experiment

Fig. 2 RMS and Average Concentration vs. Radial Location, Time-Varying Parabolic Profile and Genetic Algorithm Results

Fig. 3 RMS and Average Concentration vs. Radial Location, LES Output and Genetic Algorithm Results Given LES Data

Fig. 4 Time-Averaged Radial Velocity Profiles, x/d = 24

Fig. 5 Time-Averaged Results of LES Model vs. Experiment for Various Mixing Lengths for Methane

Fig. 6 RMS of Methane Concentration, LES vs. GA (Experimental) Results, x/d = 24
Table 1
Details of Large Eddy Simulation Software Model Used

<table>
<thead>
<tr>
<th>Grid System</th>
<th>Non-uniform in radial direction, clustered grids near central pipe</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid Size</td>
<td>128x64x47</td>
</tr>
<tr>
<td>(x=2D_{outer}, Θ=360°, R=D_{outer}/2)</td>
<td></td>
</tr>
<tr>
<td>Accuracy</td>
<td>Fourth-Order in Space, Third-Order in Time</td>
</tr>
<tr>
<td>Subgrid Turbulence Model</td>
<td>Germano Dynamics</td>
</tr>
<tr>
<td>Inlet Flow Boundary Conditions</td>
<td>Stored time serial results from a separate 3-D LES for fully developed coannular pipe flow</td>
</tr>
<tr>
<td>Wall Treatment</td>
<td>Wall Function</td>
</tr>
<tr>
<td>Steps for Statistically Stationary State</td>
<td>50,000</td>
</tr>
<tr>
<td>Typical CPU Time</td>
<td>~ 3 days on Alpha Dec 500au</td>
</tr>
</tbody>
</table>
Table 2

Test Matrix for Methane and Ethane Experiments$^a$

<table>
<thead>
<tr>
<th>Axial Distance (x/d)$^b$</th>
<th>Azimuthal Orientation (Arbitrary 0° Ref.)</th>
<th>Reynolds Number (Outer Flow)</th>
<th>Fuel Used</th>
</tr>
</thead>
<tbody>
<tr>
<td>24</td>
<td>0, 90, 180 &amp; 270</td>
<td>$1.37 \times 10^5$</td>
<td>Methane</td>
</tr>
<tr>
<td>12</td>
<td>90 &amp; 270</td>
<td>$1.37 \times 10^5$</td>
<td>Methane</td>
</tr>
<tr>
<td>6</td>
<td>90 &amp; 270</td>
<td>$1.37 \times 10^5$</td>
<td>Methane</td>
</tr>
<tr>
<td>24</td>
<td>90 &amp; 270</td>
<td>$1.37 \times 10^5$</td>
<td>Ethane</td>
</tr>
</tbody>
</table>

$^a$ Nominal air flow rate was 7460 slm, nominal fuel flow rate was 53 slm.

$^b$ x/d is the axial distance x from the end of the center (fuel) pipe divided by the center pipe diameter d.
Figure 1
Figure 2

RMS of Fuel Concentration (RMS volume fraction, %)

Radial Location (cm)

Fuel Concentration (% Volume)

Input Data (Steady)
Input Data (RMS)

GA Results (Steady)
GA Results (RMS)
Figure 3

Figure 3: Graph showing the RMS of Fuel Concentration (RMS volume fraction, %) as a function of Radial Location (cm). The graph compares LES Data (RMS) and GA Results (RMS), LES Data (Avg.), and GA Results (Avg.).
Figure 4
Figure 5 (Counts as 400 Words)
Figure 6

CH$_4$ Concentration RMS (% Vol. RMS) vs Radial Position (cm)

- LES
- GA Tomography

Inner Pipe Wall
A SCIENCE BASED APPROACH TO ENHANCED ZIRCONIA-BASED THERMAL BARRIER COATINGS FOR ADVANCED GAS TURBINE APPLICATIONS

Semi-annual report covering the period
1 August 2001 – 31 January, 2002
Contract No: 01-01-SR093 (DOE-SCERDC)

to
Advanced Gas Turbine Systems Research Program
South Carolina Institute for Energy Studies
Clemson, South Carolina 29634-5180
Program Manager: Dr. Richard A. Wenglarz

Principal Investigators:
D.R. Clarke, C.G. Levi, and A.G. Evans

The support of the Department of Energy through the AGTSR program, administered by the South Carolina Institute for Energy Studies, is gratefully acknowledged. The research described in this report has benefited from the active support and advice of the industrial members of the AGTSR consortium, notably Siemens-Westinghouse, Solar Turbines, General Electric CRD/Aircraft Engines, UTRC/Pratt & Whitney, and Howmet.

February 2002
EXECUTIVE SUMMARY

This report describes progress during the just completed second six-month period of the first year of the program. The previously submitted report describes the results of the initial, first six-month period of the program. Work has initiated in the major areas of study aimed towards developing a science-based approach to enhancing the properties of zirconia-based materials for advanced thermal barrier applications in gas turbine engines. This includes: a) the development of refined models for thermal cycling induced damage in plasma sprayed coatings; b) investigation of alternate stabilizers of the tetragonal-prime (t’) phase for higher temperature operation; c) the effect of different dopants on the optical absorption properties of zirconia in the infra-red; d) the chemical compatibility between monazite and sodium vanadate and sodium sulfate and the suitability of monazite as a superficial, protective layer against vanadate and sulfate attack; e) the effect of TBC sintering on life.

Substantial progress has been made in both modeling thermal cycling induced damage in plasma sprayed coatings and in the development of phase equilibria for alternate rare-earth element stabilizers and this is described in the attached, more detailed technical summaries. In the area of the effects of different dopants on infra-red absorption, series of samples with different potential absorber ions have been prepared by sintering. Initial characterization has largely been completed but more detailed characterization awaits the (delayed) delivery of a solid-state detector which has high-sensitivity from 1 µm to 6 µm, the infra-red range of interest for TBCs exposed to the high-temperatures in current and future engines.

Systematic experiments to quantify the extent of sintering of thermal barrier coatings, attached to a bond-coated superalloy, as a function of time and temperature are underway. For comparison, the sintering behavior of thermal barrier coatings deposited on sapphire is also being investigated. As described in the following more detailed technical summary, as sintering occurs locally between adjoining columns, “mud-cracks” appear in the coating because of the constraint imposed on the overall TBC sintering by the underlying substrate, whether it be a bond-coated superalloy or a sapphire substrate. A number of distinct sintering mechanisms have been identified, some specific to morphological changes occurring in the underlying bond-coat. With the identification of what are believed to be the principal sintering mechanisms, the mechanics issues related to predicting “mud-cracking” will be pursued in the coming contract period with the objective of establishing quantitatively the development of “mud-cracking”, their spacing and depth as a function of time at temperature.

Our experiments have revealed that thin-section parts distort upon cyclic oxidation. To guide further experimentation, an analysis of the stresses and thermal strains has been undertaken through a thermal cycle. New insight has been gained from this combined analytical and finite element analysis concerning the conditions for shape distortion on cycling. The new insight is that the fundamental re-
quirement for distortion upon cyclic oxidation is that both the bond-coat and the TGO must experience yielding during at least one of the thermal stages in each thermal cycle.

Studies on the potential use of alternate stabilizers have been initiated, seeking improved understanding of the effects of gradually substituting rare-earth cations for Y. Work during this period has focused on three main issues: (1) Establishing the basic features of the ZrO$_2$-REO phase equilibrium for the systems and regions of interest, with an emphasis on Gd additions as specified in the proposal; (2) Exploring the relative stabilizing efficiency of Y and Gd, and combinations of them; (3) Developing a methodology for growing TBCs with mixtures of stabilizers by EB-PVD. This work, described in the following technical summary, indicates that there is considerable flexibility – within the fundamental constraints afforded by thermodynamic equilibrium – for the use of alternate stabilizers. This important area will be continued in the second year of the program.

Experiments to investigate thermal cycling induced “rumpling” of CoNiCrAlY bond coats that will be used in support of the models of thermal cycling have still not begun. We await delivery of a series of samples promised by our industrial partners. They themselves, however, are awaiting the requested materials from their vendors. Despite the delays, we prefer to study the bond coats promised rather than make up our own so that the results will be of direct value to our partners.

The education and training aspects of the program continue. Three graduate students are now working exclusively on the program, one having begun five months ago. An undergraduate student, who worked on the program during the summer, is also continuing to be heavily involved in the program and is actively involved in several of the experiments. The weekly meetings in which issues related to TBC are discussed, students present up-dates of their research and faculty present tutorials on specific topics are continuing.
SUMMARY OF PRINCIPAL TECHNICAL PROGRESS

TBC Sintering

In addition to having a low thermal conductivity, a TBC must maintain strain compliance in the plane of the coating to accommodate the thermal expansion mismatch with the underlying superalloy. With increasing turbine temperatures, there is the natural concern that TBC sintering will decrease the strain compliance and promote failure. Furthermore, there is concern that at higher temperatures the tetragonal-prime phase will transform to the monoclinic phase. Previous investigations have studied TBC sintering by first removing the TBC from the superalloy and then studying its sintering behavior in a traditional ceramics manner, for instance, by measuring length changes or physical density as a function of time and temperature. This approach neglects the effects of the constraint provided by the super-alloy and possibly hides important microstructural changes that can affect TBC failure. This is especially so since TBCs in service are exposed to a temperature gradient.

To investigate the effects of constraint on sintering, EB-PVD TBC coatings on two types of substrates were provided by Howmet Castings and heated isothermally at high temperatures. One substrate type was a N5 superalloy with a standard platinum-modified nickel-aluminide bond-coat. The other substrate type was a sapphire single crystal. These two were selected since the N5 superalloy has a larger thermal expansion coefficient than zirconia whereas sapphire has a smaller coefficient. Thus, at temperatures above the TBC deposition temperature, the differential thermal expansion with the N5 should serve to separate the individual TBC columns whereas the differential expansion with the sapphire should act to bring the columns together. The former would be expected to delay sintering whereas the latter should promote sintering.

Sintering was initially performed at 1200°C; higher temperatures were not used with the PtNiAl coated N5 substrate so as to avoid incipient melting between the bond-coat and the superalloy. Higher temperature sintering will be undertaken on the sapphire substrates in future. Microscopy observations of the coatings at different times indicate that column sintering on both the N5 and sapphire substrates occurs by the same microscopic mechanism. The mechanism involves surface diffusion, the formation of necks between adjoining columns by a Rayleigh instability process and the sintering together of the columns as the necks grow. This process is shown in figure 1 below.

The response of the coating to the sintering process is the formation of a network of cracks, reminiscent of “mud-cracking”, as illustrated in figure 2. The cracks, more strictly “gaps”, form as a result of the competition between local, in-plane densification caused by inter-columnar sintering, and the constraint of the underlying substrate that fixes the lateral size of the coating. Studies are underway to quantify the spacing of the cracks and their depth as a function of time and temperature. On a flat substrate, such as the sapphire, the onset of cracking has not yet been identified. However, on rougher substrates or on a substrate that undergoes rumpling with thermal cycling the length scale of the cracking appears to correlate with the underlying roughness. This is illustrated by the micrographs in figure 3.
where the cracking is associated with the grain boundaries of the bond-coat. It appears that the TBC columns formed on the remains of the grain boundary ridges (left after grit-blasting) are canted and so are slightly closer together along the grain boundaries than away from the boundaries. Consequently, they sinter together faster and a gap opens up across the lines delineating the bond-coat grain boundaries.

At this stage of the work it is not possible to anticipate all the consequences of the sintering processes. However, there are two obvious consequences. One is the “gaps” provide a ready access path deep into the coating for the ingress of corrosive species, such as vanadates and sulphates. The second is that cracks can deflect parallel to the substrate, promoting coating loss, under a thermal gradient.

A detailed description of these findings will be written up as a complete manuscript in the near future.

Figure 1. TBC columns sintering together by neck growth at 1150°C. Left, after 100 hrs. Right, after 850 hrs. Note the surface smoothing of the columns.
Figure 2. Surface of EB-PVD after 100 one-hour thermal cycles at 1150°C. Higher magnification images reveal columns are beginning to sinter together and small gaps are beginning to appear.

Figure 3. Same coating but after 850 one-hour cycles. Periodic gaps, having the appearance of “mud-cracking”, are forming as columns are locally sintering together constrained by the fixed size of the underlying superalloy.
Mechanisms Governing The Distortion of Alumina-forming Alloys Upon Cyclic Oxidation

1. Synopsis

The problem is addressed by adapting an approach developed for understanding TGO displacement instabilities. The following material properties are incorporated. The FeCrAlY is assigned a yield strength that varies with temperature. It is constant, $\sigma_{Y_{bc}}$, between temperature $T_1$ and the maximum, $T_{max}$. Below $T_1$ the strength increases rapidly with decreasing temperature. The TGO is allowed to yield at the peak temperature, with yield strength, $\sigma_{Y_{tgo}}$, but otherwise is elastic with modulus, $E_{tgo}$. It has a thermal expansion coefficient lower than the FeCrAlY, with misfit, $\Delta \alpha$.

The strains generated by the TGO are basic. One component is caused by thermal expansion misfit and the other by TGO growth. The growth strains require some explanation. An element of the FeCrAlY changes composition with a net diminution of Al. This Al reacts with ingressing O to form $\text{Al}_2\text{O}_3$ with an associated increase in volume. The new TGO forms primarily on the interface. Since this interface is incoherent, on a planar segment, the volume increase (thickening) is accommodated by an upward, rigid body displacement of the TGO, obviating the development of stress. However, some of the new TGO resides on the internal grain boundaries causing it to elongate when the constraint from the bond coat is relaxed. This strain induces in-plane compression and causes the observed curvature changes.

Based on prior understanding of TGO induced displacements, acquired through an analytic model with spherical symmetry, the following assessment provides a framework for performing simulations and analysis. To obtain a permanent curvature, the convex TGO layer must exhibit plastic elongation and therefore, must yield at the temperature maximum, during the growth step. Correspondingly, the FeCrAlY must exhibit plastic bending and elongation, requiring that through-thickness yielding occur upon cooling, motivated by growth and thermal expansion misfit. Accordingly, the fundamental requirement for cyclic curvature is that both the bond coat and the TGO must experience yielding during at least one of the thermal stages within every cycle. A consequence is that curvature change occurs only within a defined window of properties and thickness. For the TGO to yield at the temperature maximum, the substrate thickness, $H$, must be thick enough to satisfy:

$$H > \frac{\sigma_{Y_{tgo}}}{\sigma_{Y_{bc}}} (h_1 + h_2) \quad (1a)$$

where $H$ is the thickness of the FeCrAlY, $h_1$ the thickness of the convex TGO and $h_2$ the thickness of the concave TGO. For the bond coat to yield on cooling, its strength must satisfy,

$$\sigma_{Y_{bc}} < \frac{(h_1 + h_2)}{H} \left[ \sigma_{Y_{tgo}} + E_{tgo} \Delta \alpha \Delta T / (1 - \nu_{tgo}) \right] \quad (1b)$$

where $\Delta T = T_{max} - T_1$. To realize large-scale curvature changes, both inequalities must be satisfied simultaneously. When only one is satisfied (or neither), the system can elongate, but the curvature change will be small.
2. Numerical Results

Finite element simulations have been performed using the ABAQUS code. The approach has been described elsewhere. The first variable to explore has been the elongation strain. Initially, to differentiate the roles of this strain and the thickening, any relationship between them has been neglected. Instead, the TGO thickness is held constant and the elongation strain, $\Delta e_g$, is introduced in each cycle. It is a parameter in the simulations. The calculations are performed for different TGO thickness. Once the trends have been identified, a full simulation is performed in accordance with parabolic thickening and corresponding elongation. Coincidence with the measurements will establish the magnitude of $\Delta e_g$ relative to the thickening.

Preliminary calculations probe the property ranges that result in curvature changes, with insights from (1), starting with the final TGO thickness found experimentally ($h_1 = 2.8 \mu m, h_2 = 1.8 \mu m, H = 700 \mu m$). The yield strength range for the TGO is assessed from growth stress measurements: $500 \text{ MPa} \leq \sigma_{ytgo} \leq 2 \text{ GPa}$. Then, (1) is used to establish the range of bond coat yield strengths that might be expected to produce curvature change. This range is: $5 \text{ MPa} \leq \sigma_{yb} \leq 20 \text{ MPa}$. Within these property ranges, preliminary results for the stresses affirm that the response is fully-elastic at temperatures below $T_1$ ($T < 750\text{C}$), enabling further calculations to be confined to higher temperatures, between $T_1$ and $T_{max}$. In this range, the bond coat responds as follows. Upon initial cooling, the displacements are elastic. It then reaches yield and the stress remains essentially constant down to $T_1$. Reheating beyond $T_1$ elastically unloads the alloy and induces compression. In some cases, reverse yielding occurs. The stress in the TGO layers is always compressive. At the start of a cycle, typically it is below the yield strength. Imposing the elongation strain during growth increases the compression until the yield strength is reached. Thereafter, the stress remains constant and there is no further elongation, only thickening. The initial stress is much closer to yield on the concave than the convex TGO layer, allowing a larger component of elongation strain. This difference is the basis for the permanent change in curvature.

Calculations of the curvature made for a wide property range affirm that, for the chosen thickness, large scale curvature changes occur only when the TGO yield strength is in the range, $1.5 \text{ GPa} \geq \sigma_{ytgo} \geq 0.7 \text{ GPa}$ with an alloy having strength in the range, 5 to 10MPa.

3. Analytical Approximation

An analytical solution can be obtained by using the insights gained from the numerical results, particularly the responses that apply when (1) is satisfied. After the first few cycles, the TGO on the concave side acquires a stress at the peak temperature about equal to its yield strength. There are two main consequences. (a) The stress in this layer cycles in a linear elastic manner. (b) At the peak temperature, the layer yields immediately upon application of the growth strain, such that it thickens without elongation. Conversely, on the convex side, the TGO is below yield upon reheating to $T_{max}$, allowing some elongation strain to be added during the growth step. On cooling, the bond coat yields throughout and attains a state of uniform tension equal to its high temperature yield strength at $T_1$. On reheating, there is no reverse yielding. The system is elastic below $T_1$. 
Since the concave TGO is at its yield strength at $T_{\text{max}}$ and otherwise, behaves in an elastic manner, then at instant A in the reheat cycle (coincident with $T_1$), the stress in this layer is:

$$\sigma_{\text{TGO},A}^{(2)} = -\frac{E_{\text{TGO}} \Delta \alpha \Delta T}{1 - \nu_{\text{TGO}}} - \sigma_y^{\text{TGO}}$$  (2)

The stress in the bond coat at this instant is,

$$\sigma_{\text{bc}} = \sigma_y^{\text{bc}}$$  (3)

Hence, for force equilibrium, the stress in the convex TGO at A is;

$$\sigma_{\text{TGO},A}^{(1)} = \frac{h_2}{h_1} \left[ \frac{E_{\text{TGO}} \Delta \alpha \Delta T}{1 - \nu_{\text{TGO}}} + \sigma_y^{\text{TGO}} \right] - \frac{H}{h_1} \sigma_y^{\text{bc}}$$  (4)

This result only applies once the TGO is thick enough to assure that the bond coat yields on cooling.

At instant B, at the end of the reheat step, coincident with $T_{\text{max}}$ since there is no reverse yielding, the stress in the convex TGO becomes:

$$\sigma_{\text{TGO},B}^{(1)} = \frac{h_2}{h_1} \left[ \frac{E_{\text{TGO}} \Delta \alpha \Delta T}{1 - \nu_{\text{TGO}}} + \sigma_y^{\text{TGO}} \right] - \frac{H}{h_1} \sigma_y^{\text{bc}} + \frac{E_{\text{TGO}} \Delta \alpha \Delta T}{1 - \nu_{\text{TGO}}}$$  (5)

The difference between the stress in the convex TGO at instant B and its yield strength dictates the increment in elongation strain as

$$\Delta \varepsilon_g^* \equiv \frac{\left( \sigma_y^{\text{TGO}} - \sigma_{\text{TGO},B}^{(1)} \right) (1 - \nu_{\text{TGO}})}{E_{\text{TGO}}}$$  (6)

This result applies when $\Delta \varepsilon_g^* < \Delta \varepsilon_g$. This is the strain that dictates the change in radius of curvature on a cycle-by-cycle basis, discussed next.
The elongation strain introduced into the convex TGO during each cycle dictates the change in curvature. The stresses are essentially the same at the beginning and the end of each cycle. The only difference is that attributed to the small change in TGO thickness. During growth, the elasticity of the bond coat restricts the strain that can be realized. Then, upon cooling, once the bond coat yields, the associated plastic strains accommodate the TGO elongation. This is why the change in curvature develops primarily upon cooling. The change in elongation strain in the convex TGO layer at this stage relates to the change in curvature, $\Delta \kappa$, by:

$$\Delta \varepsilon^{(2)} = \Delta \varepsilon^{(1)} + H \Delta \kappa$$  \hspace{1cm} (7)

Since it is elastic, the change in the strain in the concave TGO layer, $\Delta \varepsilon^{(1)} \approx 0$. Accordingly, the change in curvature per cycle becomes:

$$\Delta \kappa \approx \Delta \varepsilon^{(2)} / H$$  \hspace{1cm} (8)

The basic implication is that a change in curvature upon thermal cycling is enabled by a high TGO yield strength, $\sigma_{Y,tgo}$ and relatively low bond coat strength, $\sigma_{Y,bc}$. An inversion occurs when $\sigma_{Y,bc}$ becomes too small (<10MPa) because reverse yielding upon reheating negates the plastic displacements on cooling.

Alternate Stabilizers.

Phase Equilibria.

The additions of interest are rare earth oxides to conventional YSZ, and the initial emphasis is on Gd. The fundamental thermodynamic question is the position of the $T_0$ surfaces for the cubic/tetragonal and tetragonal/monoclinic transformation as these determine the viable composition range for the metastable $t^*$ phase. Little is known, however, about the relevant phase equilibria and associated free energy functions for the relevant phases. The ZrO$_2$-YO$_{1.5}$ system has been extensively studied and assessed, but some regions are still under debate, particularly the lower temperature equilibrium on the ZrO$_2$-rich end. There are also partial and/or tentative diagrams derived experimentally for many of the ZrO$_2$-REO$_{1.5}$ systems, but the uncertainty on the ZrO$_2$-rich end tends to be generally greater than for the ZrO$_2$-YO$_{1.5}$. It has been suggested, however, that the lanthanide systems exhibit regular trends with ionic size in their thermodynamic behavior as solutes within ZrO$_2$. The calculated stability range of the fluorite solid solution is reported to increase systematically in composition and temperature with decreasing size of the dopant cation (La→Lu). Concomitantly, the stability of the equilibrium pyrochlore zirconate decreases with decreasing ionic size from La to Gd, beyond which is replaced by the $\delta$ structure whose stability increases with further reduction in the ionic size (Ho→Lu). The calculated phase diagrams in the literature also suggest an increase in the stability of the tetragonal phase with decreasing ionic size, but no significant discussion of this trend is provided.

In view of the limited information on the ZrO$_2$-GdO$_{1.5}$ system, an effort was initiated to develop an understanding of the general features of the ZrO$_2$-rich end of the phase diagram, as well as the extension of the phase fields into the ternary system ZrO$_2$-YO$_{1.5}$-GdO$_{1.5}$. First, binary samples with
different concentrations of GdO$_{1.5}$ or YO$_{1.5}$ were prepared from mixed precursor solutions. (The YO$_{1.5}$ samples were intended to ascertain the reliability of the method and provide comparison with GdO$_{1.5}$ materials of similar composition.) The solutions were flash-dried by spraying them onto a hot (300°C) Teflon-coated surface to minimize segregation. The resulting powders were pyrolyzed at 900°C, pelletized, heated to 1600°C for 24 h, and then to 1200°C for an additional 168 h (one week), and in selected cases for up to 504 h (three weeks). Phase composition after these treatments was undertaken by X-ray diffractometry. Some samples were also examined by Raman spectroscopy, which is more sensitive than XRD for detecting small amounts of tetragonal or monoclinic phases.

The results are summarized in Figure 4, which depicts a tentative 1200°C isothermal section in the ZrO$_2$-YO$_{1.5}$-GdO$_{1.5}$ ternary and the corresponding ZrO$_2$-REO binaries. The results for the ZrO$_2$-YO$_{1.5}$ are in remarkable agreement with the thermodynamic assessment in the literature, except for the δ-zirconate which retained the disordered fluorite structure even after the relatively lengthy heat treatment. The Gd samples, however, revealed some significant differences relative to the calculated phase diagram. Notably, the fluorite field at 1200°C is substantially wider than that predicted by the thermodynamic model, and hence the eutectoid reaction $F \rightarrow Py + m$ is probably located at lower temperatures. These changes have been incorporated into Figure 4. The limited results so far suggest that the ZrO$_2$-rich end of the ZrO$_2$-GdO$_{1.5}$ system is probably closer to the ZrO$_2$-YO$_{1.5}$ than initially anticipated. Further work, however, is necessary to confirm the proposed diagram and outline the Gd-rich end, but the present results provide sufficient insight to proceed with the exploration of the ternary, which is now ongoing.

**Relative stabilizing efficiency**

An investigation has been undertaken on the relative stabilizing efficiency of candidate cations to be added to YSZ. Compositions of initial interest are shown on Figure 4. They include different combinations of Y and Gd at a constant level of total stabilizer addition (7.6 mole% MO$_{1.5}$, equivalent to 7 wt.% Y$_2$O$_3$), as well as additions of GdO$_{1.5}$ at a fixed level of YO$_{1.5}$. Preliminary results for the 7.6% YO$_{1.5}$ and GdO$_{1.5}$ are shown by the X-ray diffraction data in Figure 5, together with results for LaO$_{1.5}$, which is also of interest as an alternative stabilizer. Compositions are prepared by precursor methods, which allow the low temperature synthesis of chemically homogeneous t' without the need for long heat treatments in the fluorite field, which obviously involves very high temperatures. The compositions are heat treated as powders, to avoid possible constraints to the transformations introduced in a compact. The initial condition corresponded to the product of the 900°C pyrolysis treatment. The powders were subsequently heated for 24 h periods and characterized after each stage. The treatment consisted of 4 cycles at 1200°C, followed by single cycles at 1250, 1300, 1350, 1400 and 1450°C.

All samples were found to be single phase tetragonal (t') after pyrolysis. The least stable material was that containing La, which exhibited partitioning into t + pyrochlore after 24 h at 1200°C, and subsequent transformation of the t phase into monoclinic. The material stabilized with Gd was still t' after the treatment at 1300°C, but exhibited detectable monoclinic phase after 1350°C. In contrast, the Y bearing material showed only a trace of monoclinic after the 1400°C cycle and clear presence of monoclinic after 1450°C. The above results, albeit preliminary, suggest that direct substitution of Y in
YSZ may be detrimental to the phase stability of the coating. The preferred strategy is then to retain Y as the main stabilizer, with the additional dopant addition tailored for optimum effect on the optical properties.

Synthesis of Compositions with Mixed Rare Earth Oxide

A methodology has been developed to deposit a TBC with simultaneous addition of Y and Gd. In principle, the compositions could simply be ordered from a commercial supplier, but that approach is expensive and limits the flexibility to try multiple new compositions. The strategy selected involved using standard YSZ ingots, which are then infiltrated with a solution of Gd nitrate precursor, dried and pyrolyzed at 1200°C to incorporate the Gd₂O₃ into the structure. To prevent segregation during drying the precursor is gelled immediately after impregnation. The amount of Gd added can be tailored by controlling the concentration of the solution and the number of impregnation cycles.

An ingot produced in this manner was used to deposit (by electron beam deposition at UCSB) a ~120 µm thick coating of ZrO₂-7.6YO₁.₅-7.6GdO₁.₅ on a FeCrAlY substrate, previously polished and pre-oxidized. The resulting TBC is shown in cross-section in Figure 6. Energy dispersive spectral (EDS) measurements revealed the Gd composition to be uniform throughout the thickness. The columns are well developed, with the typical appearance of YSZ deposited in the same manner. A cursory examination suggests that the feathery structure within the columns is more pronounced than in YSZ, but this issue needs to be studied more carefully. Samples of this type will be used for subsequent studies of morphological evolution during heat treatment, as well as conductivity measurements.
Figure 4. Tentative ternary section for the ZrO$_2$-GdO$_{1.5}$-YO$_{1.5}$ at 1200°C, and the corresponding binaries. The circles represent experimental compositions and the heat treatments to which they were subjected. Note that the 7.6% composition exhibits partitioning because of a previous 24 h exposure at 1600°C. The empty circles in the ternary represent the compositions involved in the phase stability study.
Figure 5. Stability of the t’ phase for three different cations at the same level of addition. The asterisks indicate the position of the monoclinic peaks, and the legends on the left the heat treatment corresponding to the XRD scan.

Figure 6. ZrO$_2$-7.6Y$_{0.5}$-7.6GdO$_{1.5}$ thermal barrier coating deposited by EB-PVD from an ingot prepared by impregnation of a Gd precursor into a YSZ matrix.
AGTSR Subcontract Number 01-01-SR091
First Semi-Annual Report
August 1, 2001 to February 1, 2002

Maurice Gell and Eric Jordan, Principal Investigators
School of Engineering
University of Connecticut, Storrs, CT

In collaboration with

Gerry Meier and Fred Pettit
Materials Science & Engineering Department
University of Pittsburgh, Pittsburgh, PA
And
Yongho Sohn
Adv. Materials Processing & Analysis Center
University of Central Florida, Orlando, FL.
Executive Summary

Advanced industrial gas turbine engines require the use of reliable and highly durable thermal barrier coatings (TBCs) and metallic, stand-alone coatings to meet performance and durability goals. Current TBCs and stand-alone metallic coatings lack the necessary durability and reliability. Much progress has been made in understanding the mechanisms of damage initiation and progression in current TBCs and metallic coatings that ultimately leads to spallation. This understanding indicates that significantly improvements in TBC and metallic coating life and reliability can be achieved by focusing on coating composition and processing. The University of Connecticut (UConn), the University of Pittsburgh (UPitt), and the University of Central Florida (UCF) are partnering in this research program that has the potential of improving electron beam physical vapor deposition (EB-PVD) TBC and metallic coating life by more than a factor of 3X.

Research conducted by the UCONN and Pitt, including previous AGTSR programs, indicates that spallation in electron beam physical vapor deposited TBCs depends strongly on (1) the perfection of the initial, thermally grown oxide (TGO), (2) the magnitude of the localized out-of-plane tensile stress at the TGO to bond coat interface, and (3) the adherence of the TGO during thermal cycles. In this proposed program, the salient bond coat composition and processing features will be systematically investigated in order to demonstrate the optimum combination of features that will provide at least a 3X durability improvement compared to current TBCs and stand-alone metallic coatings. The following features will be assessed individually and in combination for platinum aluminide (Pt-Al) and MCrAlY coatings used as both bond coats for TBCs and as stand-alone coatings:

I. **TGO Perfection**: (a) presence or absence of metastable/transient oxides versus the desirable stable alpha alumina oxide.

II. **TGO Stress**: (a) surface roughness, (b) presence or absence of bond coat surface defects.

III. **TGO Adherence**: (a) presence or absence of active elements (silicon and hafnium) contributing to improved TGO adherence.

Most of the first reporting period has been devoted to (1) obtaining coated specimens, (2) evaluating commercially viable surface finishing treatments and (3) conducting initial oxidation trials.

All metallic and TBC coatings are being provided by Howmet’s Technology Center and Thermatech Division. Initial Pt-Al bond coated CMSX-4 specimens have been provided and NiCoCrAlY bond coats with and without silicon and hafnium are on order.
Toward the end of the current reporting period the remaining 130 of 150 samples of three different types were received. Experiments on the effectiveness of media finishing as a method of improving the surface finish of samples were carried out in the previous reporting period on the PtAl bond coated samples and were promising enough to select this method as a surface improvement method for all sample types. In addition further preoxidation have been carried out on the PtAl bondcoats. Unexpected void formation under the oxide was found under the oxide for these samples in the previous reporting period. Through a series of experiments a heat treatment that eliminated the unwanted void formation was found. As a result the preoxidation heat treatment for PtAl samples has be initiated which will be followed by EB-PVD application of the TBC. For the VPS MCrAlY samples shot peening has been done and was found to yield an unexpectedly rough surface. Surface smoothing experiments are under way to remedy the surface finish problems. It is expected that media finishing will be chosen as one of the surface conditions for all MCrAlY samples and laboratory metallographic finishing will be chosen as the second contrasting surface finish. All samples will be sent for application of the TBC by April 15. Thus the first phase of the program is rapidly coming to an end having received all samples, selected surface finish and preoxidation treatments. The next phase will involve comparison of the performance of the different samples in cyclic furnace tests.

2.0 INTRODUCTION

Both TBCs and stand-alone metallic coatings, that protect turbine blades and vanes from the hot gas stream, are required to meet the stringent performance, durability, reliability and environmental goals for ATS and Next Generation Gas Turbine industrial engines. This program seeks to develop metallic coatings, to be employed as a stand-alone coating or a bond coat for TBCs, with improved durability and reliability with at least 3X improvement in cyclic oxidation and spallation life. Specifically, the effects of heat treatment, surface finish and alloying for the metallic coatings will be addressed to demonstrate that modified processing techniques and compositions can significantly improve the resistance to thermal cycling damage of metallic coatings. Understanding from this program can be used to develop industrial processing routes and design optimum coating compositions to improve the overall performance of both stand-alone metallic coatings, and bond coats for EB-PVD TBCs.
2.1 Background

TBCs are complex engineering material systems consisting of 4 layers (Figure 1): substrate, bond coat, TGO and ceramic layer.\cite{1-4} The composition and processing of each layer affect the durability of the TBC. A comprehensive review of the state-of-the-art has been carried out in preparation for this program. It has been concluded that significant improvement in TBC reliability and durability can be achieved by focusing on the bond coat and the TGO layers. Significant bond coat and TGO factors that determine TBC reliability and durability include:

- Bond coat composition\cite{5-7}
- Bond coat deposition method\cite{3,4}
- Post-bond coat processing\cite{8,9}
- Bond coat creep strength\cite{3,10}
- Bond coat defects\cite{11,12}
- Surface roughness\cite{13,14}
- TGO-bond coat residual stress\cite{11,12,15-17}
- Composition / crystal structure of TGO\cite{8,17-20}
- TGO to bond coat adherence (active elements effects)\cite{21-34}

Of these bond coat and TGO factors, it is concluded that significant improvements in TGO durability and reliability can be achieved by focusing on (a) TGO perfection, (b) TGO stress and (c) TGO adherence in the program.

![Figure 1. A schematic illustration of the EB-PVD TBC system.](image-url)
Phase constituents of TGO, especially during early stages of oxidation, have been identified as critical factors influencing the adhesion at TGO/coating interface.\[8,17-20\] Specifically, the formation of the transient $\theta$-Al$_2$O$_3$ and its conversion to the stable $\alpha$-Al$_2$O$_3$ in the protective oxide scale has been reported to have a profound effect on the structural integrity of TGO/coating interface during thermal cycles. An image\[19\] of $\alpha$-Al$_2$O$_3$ islands nucleated within a $\theta$-Al$_2$O$_3$ TGO formed by oxidation of NiAl for 1 hour at 1000°C is presented in Figure 2. It has been proposed by Clarke and coworkers\[18,19\] that the transformation from $\theta$-to-$\alpha$ Al$_2$O$_3$ is responsible for additional residual stress from the volumetric constraint in the TGO scale and nucleation of sub-critical cracks, eventually leading to the spallation of $\alpha$-Al$_2$O$_3$ TGO scale. Thus, the formation of a “perfect” oxide that consist only of stable $\alpha$-Al$_2$O$_3$ prior to deposition of ceramic layer and thermal cyclic oxidation can lead to improved oxidation resistance, durability and reliability of both stand-alone metallic coatings and TBC bond coats.

![Photoluminescence image of $\alpha$-Al$_2$O$_3$ islands nucleated within a $\theta$-Al$_2$O$_3$ TGO formed by oxidation of NiAl for 1 hour at 1000°C.\[19\]](image)

Extensive microstructural examination of TBCs by Gell and co-workers\[11,12\] has identified bond coat surface features/defects as damage initiation sites. Figure 3 shows these surface features/defects to be: “ridges” associated with platinum aluminide (Pt-Al) coatings and “entrapped” oxides associated with shot-peening of MCrAlY coatings. Also presented in Figure 3 are the oxide cavities and accelerated growth of oxide scale resulting from the cyclic plasticity and oxidation of these features/defects during thermal cycling. It has been demonstrated that the removal of the ridges by fine polishing can improve the TBC lifetime by 3X.\[35\]
Figure 3. Surface features/defects: (a) “ridges” and (b) oxide-filled cavities in platinum aluminide (Pt-Al) EB-PVD TBCs; (c) “entrapped” oxides and (d) oxide-filled cavities associated with MCrAlY EB-PVD TBCs. These features/defects are present due to the processing of the coatings before thermal cycling and evolve into oxide-filled cavities during thermal cycling.\cite{11,12}

In addition, surface roughness has a significant effect on the level of in-plane and out-of-plane tensile stress in the TGO\cite{13,14} as illustrated in Figure 4. In-plane tensile stresses crack the TGO, allowing molecular oxygen to reach the bond coat surface, and oxidation is accelerated. Out-of-plane tensile stress eventually leads to TGO and TBC spallation. Thus, a reliable industrial processing technique that ensures optimum surface roughness and consistent removal of the undesirable surface features/defects would provide improved durability and reliability for both stand-alone and TBC bond coats.
Under the current UCONN AGTSR contract, AGTSR 99-01-SR073, residual stress in the TGO is being measured using a laser fluorescence technique as a function of thermal cycles for various commercial TBC systems. This technique, pioneered by Clarke,[11,12,15-19,36] measures the residual stress in the TGO by examining the shifts in wave-number of Cr$^{3+}$ photoluminescence in α-Al$_2$O$_3$ TGO scale. The shift in photoluminescence can be translated into a biaxial residual stress in TGO through piezo-spectroscopic coefficients. This technique has been applied successfully for both laboratory scale TBC specimens, and thermal barrier coated engine components.[37]

During this study, researchers at UCONN and UC-SB (subcontractor) have found that the laser fluorescence can readily provide information regarding (1) the formation and transformation of transient phases in TGO, specifically regarding the θ-to-α Al$_2$O$_3$ transformation[17-19] (2) changes in the TGO stress due to surface roughness/defects and (3) presence of microscopic spallation of TGO from metallic coatings. Figure 5 shows a typical spectrum collected from a TGO that contain both θ-Al$_2$O$_3$ and α-Al$_2$O$_3$. Preliminary results also reveal that the surface preparation of bond coat can significantly influence the residual stress of TGO. In addition, the laser fluorescence technique can conveniently detect the presence of microcracks that are associated with spallation (i.e., relief of residual stress) of TGO scale as shown in Figure 6.

![Figure 4. Tensile stress calculated from finite element analysis for out-of-plane (vertical) and in-plane (horizontal) stress in the TGO along the line A-B-C near the ridge of the Pt-Al bond coat surface.[14](Image)](image-url)
Figure 5. Laser fluorescence spectrum collected from a TGO scale that contains both $\theta$-$\text{Al}_2\text{O}_3$ and $\alpha$-$\text{Al}_2\text{O}_3$.

Figure 6. Bi-modal laser fluorescence spectrum containing stressed and unstressed regions.
TGO Adherence

Considerable literature\textsuperscript{[5,21-33]} describes the improved oxidation resistance associated with alloying additions for MCrAlY, nickel aluminide (NiAl) coatings and superalloys. Figure 7 shows the improvement in oxidation and spallation resistance of MCrAlY coated specimens and turbine blades.\textsuperscript{[21]} However, controversy exists concerning the mechanisms associated with the improvement, e.g., pegging, neutralization of sulfur segregation, reduced oxidation rate, microstructural enhancement of TGO, inhibition of interdiffusion. Pint and coworkers\textsuperscript{[5,24]} at Oak Ridge National Laboratory (ORNL) have shown that the addition of Hf improved the oxidation resistance of Pt-Al coatings. Preliminary results indicate that the addition of both Si and Hf can significantly improve oxidation resistance of the Pt-Al coatings.\textsuperscript{[34]} This work will study for the first time the synergistic benefit of Si and Hf to Pt-Al coatings for both stand-alone and TBC coatings.

Figure 7. (a) Single crystal turbine blades showing superior coating durability of the Si,Hf added MCrAlY coating (left) vs. the failed MCrAlY coating (right) after 2500 endurance engine cycles. (b) This superiority can be attributed to the synergetic benefits of Si and Hf shown by the progressive failure stages described on the y-axis.\textsuperscript{[21]}

34. K. Murphy, Private Communications, 2000.
2.2 Objectives

The overall objective of this program is to produce oxidation resistant metallic coatings, with improved durability and reliability, to be employed as both the stand-alone coatings and bond coats for TBCs. Specific objectives include:

1. Generate a detailed processing sequence to optimize the initial formation of thermally grown oxide (TGO) based on environmental control (e.g., temperature and partial oxygen pressure) of pre-oxidation for surface finished Pt-Al, MCrAlY, and MCRAIY with Si + Hf coatings to be used as both stand-alone coatings and TBC bond coats.

2. Demonstrate by testing specimens with various surface finishes, that the presence or absence of surface defects on coatings significantly influence the durability and reliability of coatings and demonstrate a production process to remove defects and provide the optimum surface finish for improved durability and reliability of coatings.

3. Demonstrate and define the mechanism(s) for improved oxidation resistance resulting adding Si and Hf to Pt-Al coatings.

4. Define cost-effective processing techniques for producing a perfect, initial TGO and an optimum surface finish free of defects for Pt-Al, Pt-Al (Si,Hf) and MCRAIY coatings. The process defined will be applicable to complex shaped engine components such as turbine blades.

5. Demonstrate that these TGO optimization techniques provide a 3X improvement in oxidation resistance from stand-alone metallic coatings and a 3X improvement in spallation resistance for 3 TBCs in the program.

6. Transfer the attained understanding to industrial partners for the rapid implementation of both the improved stand-alone coatings and TBC bond coats.
3.0 Program Organization, Plan and Schedule

University-Industry Partnership

**University of Connecticut**
Maurice Gell
Eric Jordan

**University of Pittsburgh**
Gerry Meier
Fred Pettit

**Univ. of Central Florida**
Yongho Sohn

**Engine Developers**
- GE Power Systems
- Pratt & Whitney
- Rolls-Royce
- Siemens Westinghouse
- Solar Turbines

**Coating Supplier**
Howmet Technology

**Figure 8 Program Organization**

- **Task I**
  - Heat Treatment and Characterization of Stand-alone Metallic Coatings
  - Heat Treatment and Characterization
  - Process Selection for Task II and III

- **Task II**
  - Oxidation Testing of Stand-alone Metallic Coatings
  - Isothermal & Cyclic Oxidation/TGA
  - Evaluation and Characterization

- **Task III**
  - Testing and Evaluation of TBCs
  - Cyclic Oxidation
  - Evaluation/Characterization

- **Task V**
  - Reports/Industrial Briefings/Technology Transfer

**Figure 9 shows a bar-chart schedule with the main tasks.**
**TASK I**  
Heat Treatment of Stand-Alone Metallic Coatings and Characterization of Initial TGO Scale

Select Heat Treatments That Give:
- Complete $\alpha$-$\text{Al}_2\text{O}_3$
- Mixture of $\alpha$-$\text{Al}_2\text{O}_3$ and $\theta$-$\text{Al}_2\text{O}_3$
- Maximum Amount of $\theta$-$\text{Al}_2\text{O}_3$

**TASK II**  
Isothermal and Cyclic Oxidation of Stand-Alone Metallic Coatings

**TASK III**  
Cyclic Oxidation Testing of Thermal Barrier Coatings

Evaluation and Characterization of Stand-Alone Metallic Coatings and TBCs as Functions of:
- Perfection of Initial TGO
- Surface Defects/Features
- Surface Roughness
- Active Elements (Si, Hf)

Demonstrate Cost-Effective Processing Provides 3X Improvement in Life for TBCs and Stand-Alone Metallic Coatings

Figure 10 shows the work flow in the program.
4.0 Experimental Program

Task I: Heat Treatment and Characterization of Stand-alone Metallic Coatings

Objective: To heat treat and subsequently characterize the TGO scale formed on stand-alone metallic coatings as a function of temperature and oxygen partial pressure.

Technical Approach: Specimen (1 inch diameter by 1/8 inch thickness) coated with stand-alone metallic coatings, PtAl, Si,Hf modified MCrAlY and MCrAIY all on CMSX-4 superalloy substrate which has been supplied by Howmet International and was fabricated with current commercial processing practices that include various surface finishes: as-coated, grit blasted and media finished. In addition a very smooth laboratory polish will be used to justify improved commercial processing if appropriate. The surface morphology and roughness of the specimens has been analyzed using optical and electron microscopy and ZYGO™ surface profilometry, capable of measuring roughness in the nano-scale. Emphasis will be given to the relationship between processing technique, surface roughness and the presence of surface defects/features. The specimens have been cut into small pieces and heat treated as a function of temperature (T = 900°, 1000°, 1100°C) and oxygen partial pressure (P_{O2} = 0.01, 0.2, 1.0 atm) for an hour as described in the first semiannual report. The TGO scale formed during the heat treatment has been characterized with respect to phase constituents, residual stress and morphology by x-ray diffraction, optical and electron microscopy, and by using the laser fluorescence technique. Details of specimen descriptions and heat treatment are presented in Table I.

Based on the characterization of the TGO scale formed on stand-alone metallic coatings that were heat treated, three conditions (3 out of 9 identified in Figure 11) have been selected. These three conditions will be selected so that the specimens will contain three different types of TGO scale. These three conditions are selected to demonstrate the importance of homogeneous stable α-Al_{2}O_{3} scale compared to other oxide conditions. In the case of the tow variants of MCrAlY a Howment process for the removal of residual oxide is being employed on half of the samples instead of having a third surface finish or third preoxidation treatment. The selected heat treatments and the corresponding TGO scale formation will be employed for the thermal cycling tests of stand-alone metallic coatings (Task II) and TBCs (Task III).
Table I. Specimen Descriptions and Evaluation for Task I and II.

<table>
<thead>
<tr>
<th>Metallic Coating</th>
<th>Surface Finish</th>
<th>Surface Defect/Features</th>
<th>Heat Treatment</th>
<th>Total Number of Specimen Required</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pt-Al</td>
<td>As-coated</td>
<td>Ridges at full height</td>
<td>As identified in Figure 8</td>
<td>3 for Heat Treatment, 3 for Oxidation Testing</td>
</tr>
<tr>
<td></td>
<td>Grit-blasted</td>
<td>Reduced ridge height</td>
<td>As identified in Figure 8</td>
<td>3 for Heat Treatment, 3 for Oxidation Testing</td>
</tr>
<tr>
<td></td>
<td>Media-finished</td>
<td>No ridges Smooth surface</td>
<td>As identified in Figure 8</td>
<td>3 for Heat Treatment, 3 for Oxidation Testing</td>
</tr>
<tr>
<td>Pt-Al (Si,Hf)</td>
<td>As-coated</td>
<td>Ridges at full height</td>
<td>As identified in Figure 8</td>
<td>3 for Heat Treatment, 3 for Oxidation Testing</td>
</tr>
<tr>
<td></td>
<td>Grit-blasted</td>
<td>Reduced ridge height</td>
<td>As identified in Figure 8</td>
<td>3 for Heat Treatment, 3 for Oxidation Testing</td>
</tr>
<tr>
<td></td>
<td>Media-finished</td>
<td>No ridges Smooth surface</td>
<td>As identified in Figure 8</td>
<td>3 for Heat Treatment, 3 for Oxidation Testing</td>
</tr>
<tr>
<td>MCrAIY</td>
<td>Shot-peened</td>
<td>Rough surface</td>
<td>As identified in Figure 8</td>
<td>3 for Heat Treatment, 3 for Oxidation Testing</td>
</tr>
<tr>
<td></td>
<td>Media-finished</td>
<td>Smooth surface</td>
<td>As identified in Figure 8</td>
<td>3 for Heat Treatment, 3 for Oxidation Testing</td>
</tr>
</tbody>
</table>

Total number of specimen required: 54 stand-alone metallic coatings

Task II: Oxidation Testing of Stand-alone Metallic Coatings

Objective: To test and evaluate selected stand-alone metallic coatings using isothermal and cyclic oxidation testing.

Technical Approach: After the selected heat treatments, the stand-alone coatings will be subjected to isothermal and cyclic oxidation testing. One forth of a full disk will be used allowing more different conditions to be considered. The isothermal oxidation will be carried out at 1100°C with thermal gravimetric analysis (TGA) to examine the oxidation kinetics of the coatings. The cyclic oxidation will be carried out in CM Rapid Temperature Cyclic Furnace. The thermal cycle will consist of a 10-minute heat-up, 40-minute hold at 1100°C and 10-minute cooling. Microstructural evaluation of oxidized coatings will be carried out by x-ray diffraction, optical and electron microscopy as well as by laser fluorescence piezo-spectroscopy. Emphasis will be given to the oxidation kinetics, TGO phases, stress and adherence/spallation as a function of initial oxide phase constituents, surface roughness, surface defects/features and bond coat compositions. Mechanisms associated with spallation of TGO scale will also be assessed. Early results of this type are described in the section on preoxidation.

Task III: Testing and Evaluation of TBCs
Objective: To procure bond coated TBC specimens with selected heat treatments and to test and evaluate them as a function of thermal cycling.

Technical Approach: In accordance to the specified heat treatment process, bond coated TBCs with CMSX-4 superalloy substrate will be prepared in April by Howmet International. Thermal barrier (ZrO$_2$-7wt.%Y$_2$O$_3$) layer will be deposited on heat-treated bond coats by EB-PVD. The TBC specimens will be subjected to cyclic oxidation testing using a CM Rapid Temperature Cyclic Furnace. The thermal cycle will consist of a 10-minute heat-up, 40-minute hold at 1100°C and 10-minute cooling. During thermal cycling, phase constituents and residual stress of the TGO will be monitored as a function of thermal cycle by laser fluorescence and specimens will be visually inspected frequently for any sign of spallation in order to accurately assess the lifetime of stand-alone metallic coatings and TBCs. Microstructural characterization of TBC specimens will be carried out prior to failure as well as after the failure to address failure mechanisms associated with various heat treatments, surface preparation and bond coat composition. Specimen descriptions and testing plans are presented in Table II.

Table II. Specimen Descriptions and Evaluation for Task III.

<table>
<thead>
<tr>
<th>Metallic or bond coating</th>
<th>Surface finish</th>
<th>Surface Defect/Features</th>
<th>Heat treatment</th>
<th>Total number of specimen required</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pt-Al</td>
<td>As-coated</td>
<td>Ridges at full height</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td></td>
<td>Grit-blasted</td>
<td>Reduced ridge height</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Rough surface</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Media-finished</td>
<td>No ridges</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Smooth surface</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pt-Al (Si,Hf)</td>
<td>As-coated</td>
<td>Ridges at full height</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td></td>
<td>Grit-blasted</td>
<td>Reduced ridge height</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Rough surface</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td></td>
<td>Media-finished</td>
<td>No ridges</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Smooth surface</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td>MCrAlY</td>
<td>Shot-peened</td>
<td>Rough surface</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
<tr>
<td></td>
<td>Media-finished</td>
<td>Smooth surface</td>
<td>3 Selected</td>
<td>9 TBCs</td>
</tr>
</tbody>
</table>

Total number of specimen required: 72 TBCs

5.0 Experimental Results

Task 0: Specimen Procurement

The required specimens described in Task 4.0, in the form of 2.54 cm diameter x 0.32 cm disks are being provided by Howmet Corporation. The disks are bond coated on all surfaces. The original plan was to have 3 metallic/bond coats consisting of a Pt-Al, a Pt-Al with Si+Hf and a NiCoCrAlY coating. Howmet’s CVD chamber for the Pt-Al with Si+Hf is not operational, so the decision has been made to study the Hf+Si effect using...
the NiCoCrAlY coating. Thus the 3 metallic/bond coats will be Pt-Al, NiCoCrAlY, and NiCoCrAlY with Si+Hf.

Of the 3 coating systems, Howmet has delivered 20 specimens with a Pt-Al coating were delivered in the first reporting period. The remainder of the coatings in the as sprayed condition were received near the end of this reporting period. Fifty samples of the three types listed below have been recently received.

<table>
<thead>
<tr>
<th>Type</th>
<th>Base Alloy</th>
<th>Bondcoat</th>
<th>TBC type</th>
</tr>
</thead>
<tbody>
<tr>
<td>XI</td>
<td>CS-CMSX-4</td>
<td>PtAl</td>
<td>EB-PVD</td>
</tr>
<tr>
<td>XII</td>
<td>CS-CMSX-4</td>
<td>MCrAlY+Si+Hf</td>
<td>EB-PVD</td>
</tr>
<tr>
<td>XIII</td>
<td>CS-CMSX-4</td>
<td>MCrAlY+Si+Hf</td>
<td>EB-PVD</td>
</tr>
</tbody>
</table>

**Task IA: Coating Heat Treatments**

**Introduction**

The University of Pittsburgh is collaborating with the University of Connecticut to develop thermal barrier coatings and metallic coatings with superior reliability and durability. In order to determine pretreatments of platinum aluminide bond coats that may be useful to improve TBC performances, platinum aluminide coatings on the substrate CMSX4 with different surface conditions are being exposed to a number of different preoxidation treatments. The surfaces of the exposed specimens are being examined by using optical microscopy and scanning electron microscopy. In the previous semi-annual report it was shown that as processed platinum aluminide coatings developed voids at oxide-coating interfaces for 2 hour exposures at 900°, 1000°, 1100° and 1200°C in air, pure oxygen and in an argon-hydrogen gas mixture that established an oxygen pressure of about $10^{-8}$ atm. In the current report results will be presented for platinum aluminide coatings exposure to additional surface modification and preoxidation treatments.

**Results**

A platinum aluminide coating in the as processed condition was given a special pretreatment by Howmet. Specimens of this coating were then exposed in dry air for: A) 2 hours at 1080°C(1975°F), B) 16 hours at 1080°C(1975°F), C) 16 hours at 1093°C(2000°F), D) 16 hours at 1121°C(2050°F). Scanning micrographs showing the surfaces of these exposed specimens are presented in Figure 11. No voids are evident. Some rosette shaped oxide is evident, Figures 1A and 1B.

Grit blasted platinum aluminide bond coats were exposed for 12 hours at 1100°C in the argon-hydrogen gas mixture and in dry air. Scanning micrographs of the exposed surfaces are presented in Figure 12. No voids are evident in these specimens.
The surfaces of platinum aluminide coatings have also been given a media finish where the times in this finishing treatment were 40 and 120 minutes. The resulting specimens were then exposed for 2 hours at 1100°C in the argon-hydrogen gas mixture and in air. The surfaces of the exposed specimens are shown in Figure 13. Some voids were evident in the specimens exposed in air, but were fewer voids evident beneath the oxide scales on specimens exposed in the argon-gas mixture.

The results that have been obtained show that by using different surface preparations and heat treatments the formation of voids beneath the alumina scales during preoxidation can be inhibited and in some cases prevented. The cause of this void formation is still not understood.

Future Work

At present the following surface preparations and pretreatments are proposed for the platinum aluminide bond coats prior to TBC deposition.

<table>
<thead>
<tr>
<th>Pretreatment Conditions</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heavy media finish + 2hrs @ 1100°C in air</td>
<td>5</td>
</tr>
<tr>
<td>Heavy media finish + 2hrs @ 1100°C in Ar-H₂</td>
<td>5</td>
</tr>
<tr>
<td>Light media finish + 2hrs @ 1100°C in air</td>
<td>5</td>
</tr>
<tr>
<td>Light media finish + 2hrs @ 1100°C in Ar-H₂</td>
<td>5</td>
</tr>
<tr>
<td>Hand Polish + 2hrs @ 1100°C in Ar-H₂</td>
<td>5</td>
</tr>
<tr>
<td>Hand polish</td>
<td>5</td>
</tr>
<tr>
<td>Heavy media finish + 2hrs @ 900°C in dry air</td>
<td>5</td>
</tr>
</tbody>
</table>

All of the above samples would have TBCs deposited. The following specimens would be tested without TBCs.

<table>
<thead>
<tr>
<th>Pretreatment Conditions</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heavy media finish + 2hrs @ 1100°C in dry air</td>
<td>2</td>
</tr>
<tr>
<td>Heavy media finish + 2hrs @ 1100°C in Ar-H₂</td>
<td>2</td>
</tr>
</tbody>
</table>
Figure 11. Sample LG87 (special treatment by Howmet). (A.) 2 hours at 1080°C. (B.) 16 hours at 1080°C. (C.) 16 hours at 1093°C. (D) 16 hours at 1121°C. All exposures done in dry air.
Figure 12. Grit Blast sample LI3 exposed at 1100°C for 2 hours in (A) Ar-H and (B) dry air environments.
Figure 13. Sample LG96, media finished for 40 minutes, was exposed for 2 hours at 1100°C in (A) Ar-H and (B) dry air. Sample LG92, 120 minute media finish, was exposed for 2 hours at 1100°C in (C) Ar-H and (D) dry air.

Note (B) was also taken at 1600x

Task IB: Media Finishing and Surface Roughness Reduction

In the previous reporting period media finishing trials were carried out on as-coated Pt-Al disk specimens by a media finishing company approved for work on turbine airfoils. Laser surface profilometry was used to quantitatively determine the surface geometry on as-coated samples and the changes that occurred after 20, 40 60 and 120 minutes of media finishing. Data was obtained for changes in (a)ridge height, (b) peak to valley distances, and average roughness. It was concluded that 40 and 120 minutes would be used in the experiments as the two conditions for the experimental program. In the current period trial shot peening was done on the MCrAlY samples and it was found that
such surfaces are not suitable as is for EB-PVD coating. Note that the range in height is from 14-27µ. Optical profilometry images and roughness measurements on these samples are given in the figure and table below:

<table>
<thead>
<tr>
<th>Sample #</th>
<th>KOH cleaned</th>
<th>Shot Peened</th>
<th>Py (µ)</th>
<th>Rms(µ)</th>
<th>Ra(µ)</th>
</tr>
</thead>
<tbody>
<tr>
<td>LF95</td>
<td>Yes</td>
<td>Yes</td>
<td>15.330</td>
<td>2.455</td>
<td>1.969</td>
</tr>
<tr>
<td>LH21</td>
<td>No</td>
<td>Yes</td>
<td>27.743</td>
<td>3.425</td>
<td>2.788</td>
</tr>
</tbody>
</table>

**Shot Peened Surface Geometry**

**LF95- Deoxidized**

![Figure 14. Rough surface of deoxidized shot peened surface of MCrAlY bondcoat.](image)

Figure 14. Rough surface of deoxidized shot peened surface of MCrAlY bondcoat.
Figure 15. Rough surface of heat treated shot peened surface of an MCrAlY bondcoat.

As a result media finish trials are under way on these samples along with laboratory polishing trials.