P28 Detailed Flow and Thermal Field Measurements on a Scaled-up Stator Vane

Karen A. Thole (thole@engr.wisc.edu; 608-262-0923)
David G. Bogard (dbogard@mail.utexas.edu; 512-471-3128

Karen A. Thole (thole@engr.wisc.edu; 608-262-0923)
Mechanical Engineering Department
University of Wisconsin
1513 University Avenue
Madison, WI 53706-1572

David G. Bogard (dbogard@mail.utexas.edu; 512-471-3128)
Department of Mechanical Engineering
University of Texas
Austin, TX 78712

Introduction

Future developments of power generation turbines require a complete understanding of the complex flows experienced by the turbine blades. CFD simulations of turbine flows have evolved to the point of making predictions for full three-dimensional flows, but the experimental data available for the evaluation of these codes is noticeably inadequate. Because most previous experimental studies of turbine airfoil or film cooling flow fields have purposefully simplified the flow field and only provided surface heat transfer results, these data are not well-suited for evaluating three-dimensional CFD predictions.

The proposed research will provide an experimental data base of a turbine vane flow designed for detailed evaluation of CFD simulations and will provide the physical understanding needed to improve CFD models. The experiments will be performed using airfoil geometry that has been enlarged to nine times engine scale. A series of experiments will be conducted in which the flow field complexity will be incrementally increased to provide multi-levels of difficulty for code validation. The experiments will be conducted using simulated airfoil facilities currently being installed at two locations: The University of Wisconsin-Madison and the University of Texas at Austin. The turbine vane geometry, provided by Pratt & Whitney, is representative of an advanced first stage vane.

Objectives

The objectives for this study are to provide to the gas turbine community detailed measurements of the flow and thermal fields for a dry and film-cooled turbine vane. The experiments that will take place will provide several levels of difficulty for evaluating CFD models. The large-scale airfoil studies would provide the detailed surface, velocity and thermal field information necessary to fully evaluate current CFD simulations. Moreover, the understanding of the physical details of this complex flow field resulting from this experimental program will aid in the development of improved models for the CFD codes.

Problem Approach and Project Description

The approach to providing these detailed measurements is to use a large-scale (factor of nine) vane that will allow spatially well-resolved measurements, particularly for the thin boundary layers along the vane surface and for film flows. For these tests, the Reynolds number and acceleration will be matched to that of the actual turbine vane. Two wind tunnel facilities (one at the University of Wisconsin and the other at the University of Texas) will entail the same overall experimental design that includes a single central airfoil with two leading edges, representing the adjacent vanes, that are each attached to contoured side walls. The contoured walls insures that the same blade Reynolds numbers and acceleration factors that occur for the actual case.

The proposed experiments will range in complexity from a two-dimensional flow over a dry airfoil (non-film cooled) and with low freestream turbulence to a strongly three-dimensional flow using a film cooled airfoil and with very high freestream turbulence levels. Measurements will be made of both the velocity (mean and turbulent) and thermal fields. Velocity field measurements will be made with two-component LDV (laser Doppler velocimeter) systems. Thermal field measurements will include surface temperatures obtained with an infrared camera, and mappings of the thermal fields associated with film cooling flows using thermocouple probes.

Film cooling experiments will involve film cooling injection from leading edge holes and downstream holes (positions and hole geometries to be determined. Large density ratios (DR>1.6) will be achieved using cryogenic cooling of the injectant. Several blowing conditions will be examined in terms of adiabatic effectiveness, but only one blowing rate will be selected for detailed measurements including velocity and thermal fields. The velocity and thermal field measurements will be concentrated primarily in the region of selected film cooling holes.

Results to Date and Future Activities

Prior to the start of this contract, the University of Wisconsin installed the scaled-up Pratt & Whitney Advanced Commercial Turbine vane geometry into the wind tunnel and developed a turbulence generator for the large scale tests. The static pressure distribution along the vane was measured to insure the correct velocity distribution. The turbulence generator was developed to give a range of turbulence levels with a range of turbulence length scales. The turbulent kinetic energy and turbulent length scales are currently being measured along a streamline at the spanwise center of the turbine vane passage to document the turbulent freestream characteristics. Since the start of this contract, we have performed baseline surface heat transfer measurements at various Reynolds numbers and have compared those results against predictions using the two-dimensional boundary layer code, TEXSTAN. High freestream turbulence effects on the vane's surface heat transfer are currently being documented. Future work will entail documenting the boundary layer along the vane's surface.

Prior to the start of this project at the University of Texas at Austin, a new large scale turbine vane cascade facility was designed, and an existing wind tunnel facility was extensively modified to incorporate the turbine vane model. The airfoil is the same vane geometry as installed in the University of Wisconsin facility. The film cooling hole configuration used on this turbine vane was also designed in collaboration with Pratt and Whitney engineers. Standard round holes were used in this design, with six rows of holes around the leading edge, two rows downstream on the pressure surface, and three rows downstream on the suction surface. To better simulate current practice in turbine vane design, an impingement plate configuration has been used in the internal cavities feeding the film cooling holes. This was done to have a better simulation of the inlet flow to the film cooling holes which we believe is critical in correctly simulating the film cooling effectiveness. Testing will begin soon to document the surface adiabatic effectiveness.

Contract Information

Subcontract Number	96-01-SR043
Contractor Name:	Clemson University Research Foundation
Contractor Address and Phone Number:	Clemson University Office for Sponsored Programs Box 345702 300 Brackett Hall Clemson, SC 29634-5702 (864)656-5266 and FAX (864)656-0881
Technical Representative:	Dr. Daniel B.Fant Email:dfant@clemson.edu
Contractor Officer's Representative:	William F. Geer, Jr.
Period of performance	June 13, 1997-June 12, 1998

Detailed Flow and Thermal Field Measurements on a Scaled-Up Stator Vane

ATS Annual Review Meeting
October 1997

Karen A. Thole, PI
Roger Radomsky,
Graduate Student
University of Wisconsin-Madison

David G. Bogard, PI
Marc Polanka and Michael Cutbirth
Graduate Students
University of Texas-Austin

CONVECTIVE HEAT TRANSFER LABORATORY
THE UNIVERSITY OF WISCONSIN-MADISON

Turbulence and Turbine Cooling Research Lab University of Texas

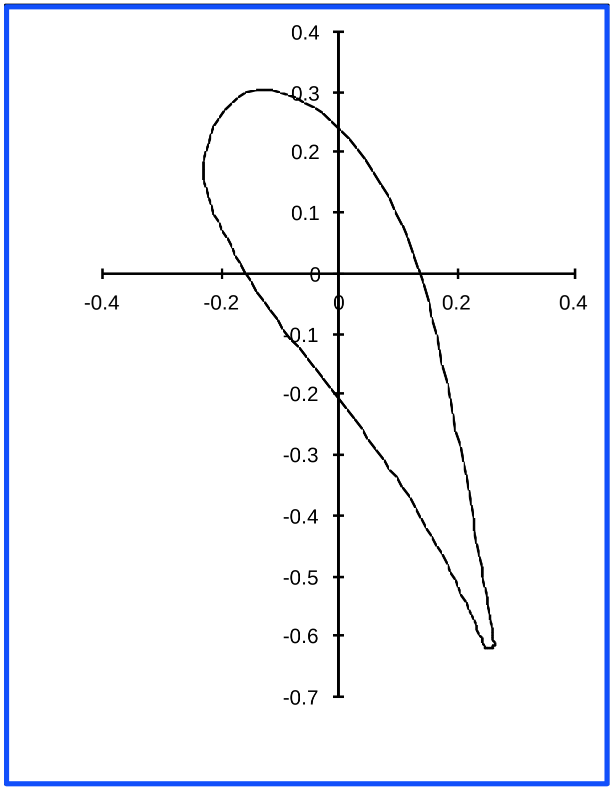
Research Objectives

- Provide vane heat transfer and detailed flow and thermal field data to benchmark computational flow codes for various levels of difficulty
- Provide a better understanding of turbine vane boundary layers through detailed measurements
- Provide a better understanding of freestream turbulence and length scale effects on vane heat transfer

Introduction

Heat transfer and film-cooling studies for a commercial vane provided by Pratt & Whitney

Actual chord	6.60 cm
Scaling factor	9
Scaled-up chord	59.4 cm
Chord-to-vane	1.30
Chord-to-span	1.08
Flow inlet angle	0Þ
Flow exit angle	78Þ
Exit Reynolds (Re_{ex})	$1.1x10^{6}$



University of Wisconsin
University of Texas

Experimental approach provides increasing levels of difficulty

Task	Description	Organization
1 & 2	Two-dimensional dry airfoil with and without high turbulence	UW-Madison
3 & 4	Two-dimensional film-cooled airfoil with and without high turbulence	UT-Austin
5 & 6	Three-dimensional dry airfoil with and without high turbulence	UW-Madison
7 & 8	Three-dimensional film-cooled airfoil with and without high turbulence	UT-Austin

University of Wisconsin

University of Texas

Experimental approach is unique for a number of reasons

- Well-resolved flow and thermal fields will be obtained through a scaled-up simulation
- Multi-levels of complexity in the data base
- High freestream turbulence with different length scales will be simulated to determine each effect independently

Measurement Techniques

Pressure Transducers and Thermocouples

National Instruments Data Acquisition System with LabView Software

Two-Component Laser Doppler Velocimetry

Fiber-optic back-scatter system with two focal length lenses (480 mm and 750 mm)

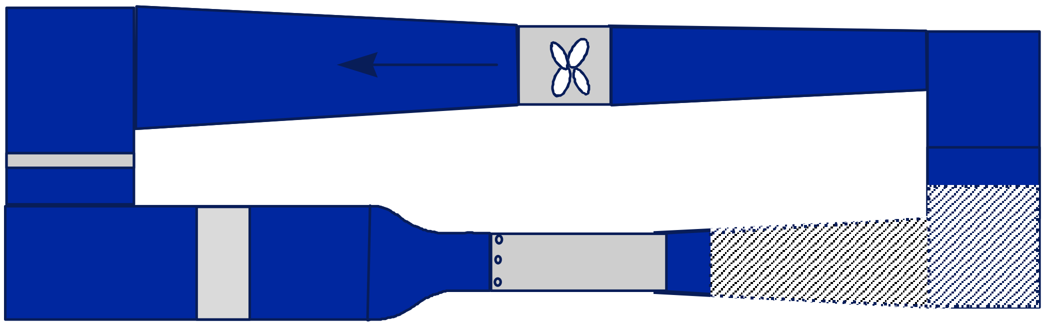
Hot-Wire Anemometry

Collaboration with Industry

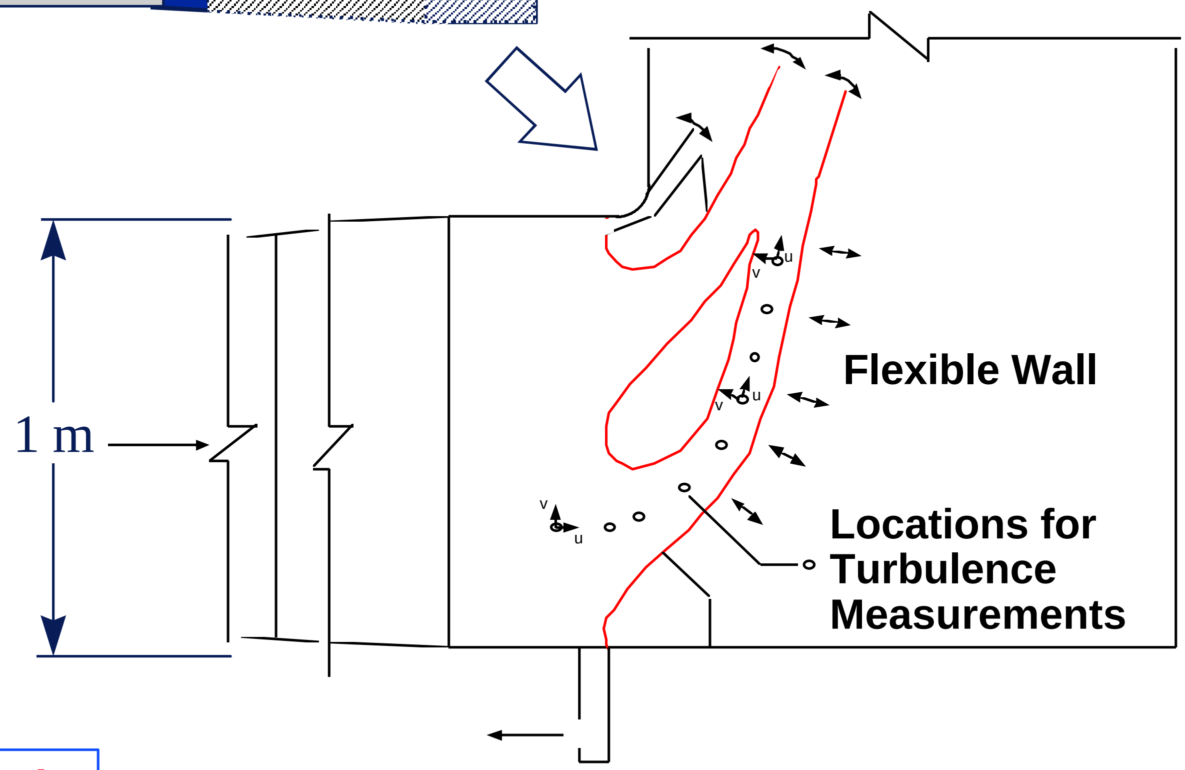
- Baseline and high turbulence results presented to Pratt & Whitney on October 9, 1997
- Student intern Kristina Hermanson from the University of Wisconsin was at Pratt & Whitney during the summer 1997
- High turbulence generator presented at IGTI 1997 with technology transferred to Solar Turbines and United Technologies during the summer 1997

Facility and Development

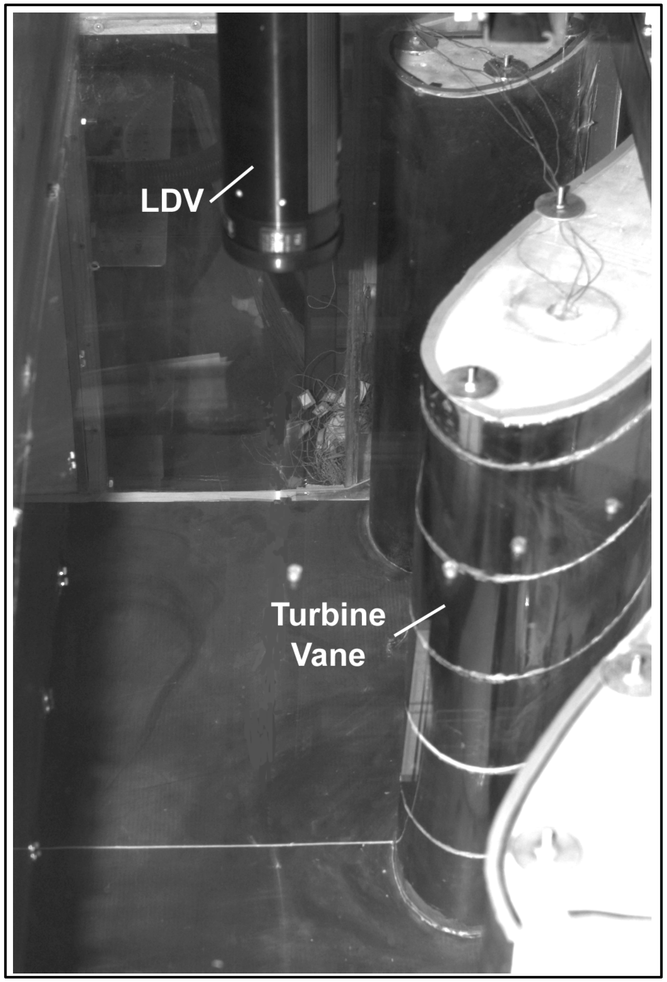
Stator Vane Test Section Geometry



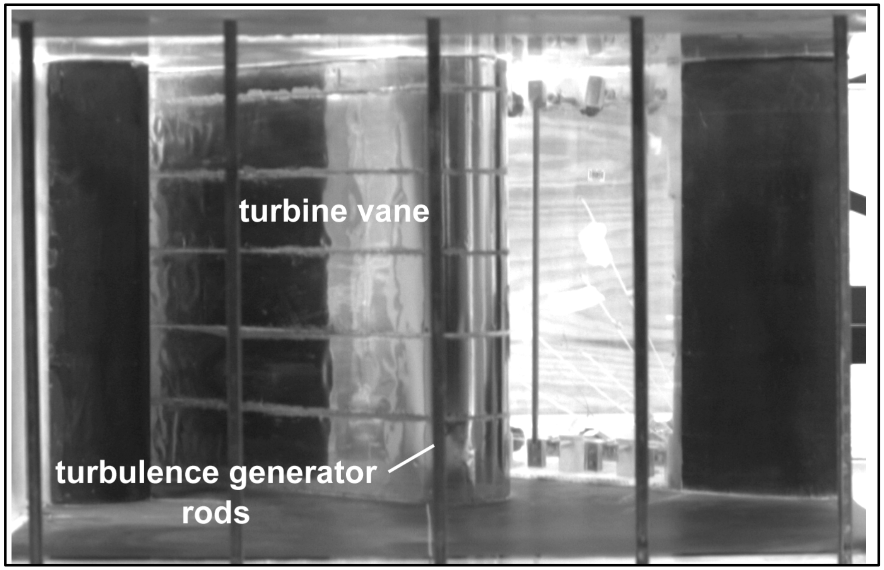
Wind Tunnel Corner Section



Photgraph of Test Section

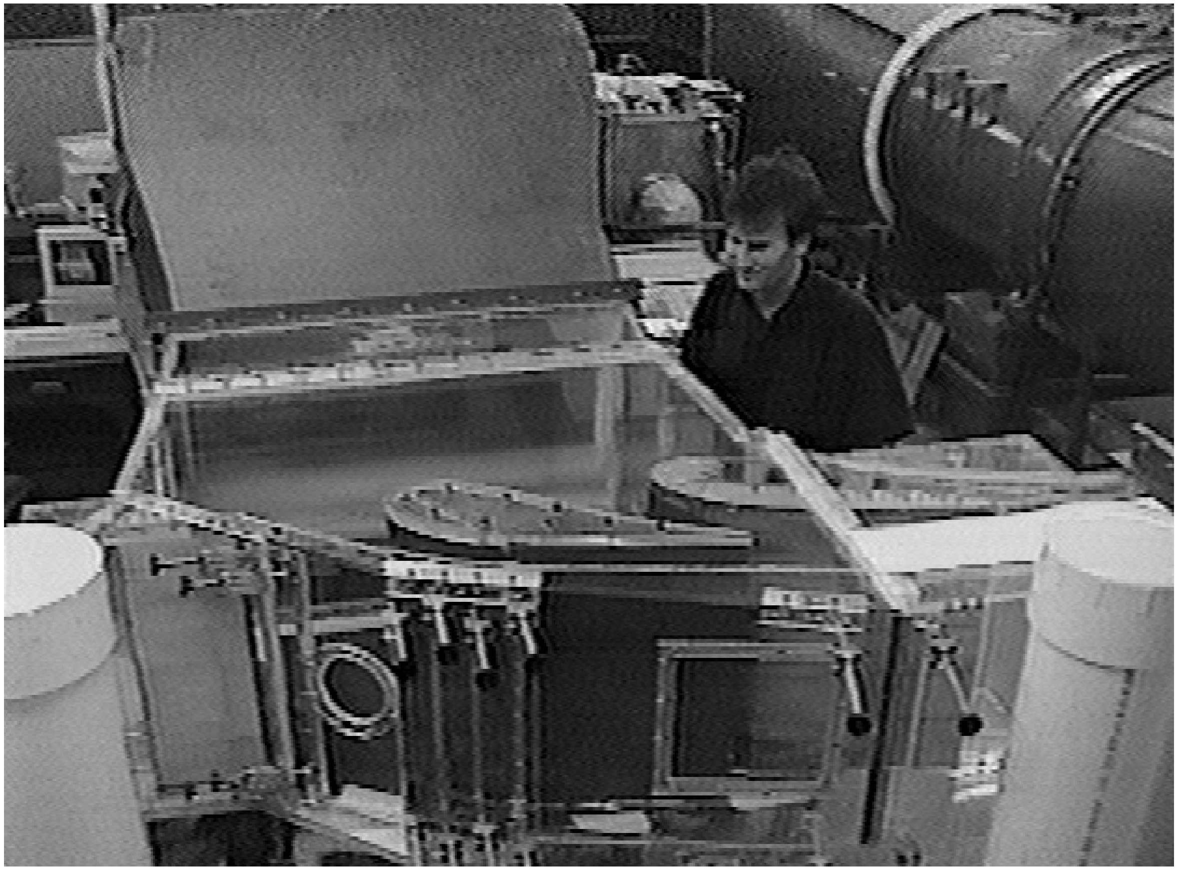


Top View

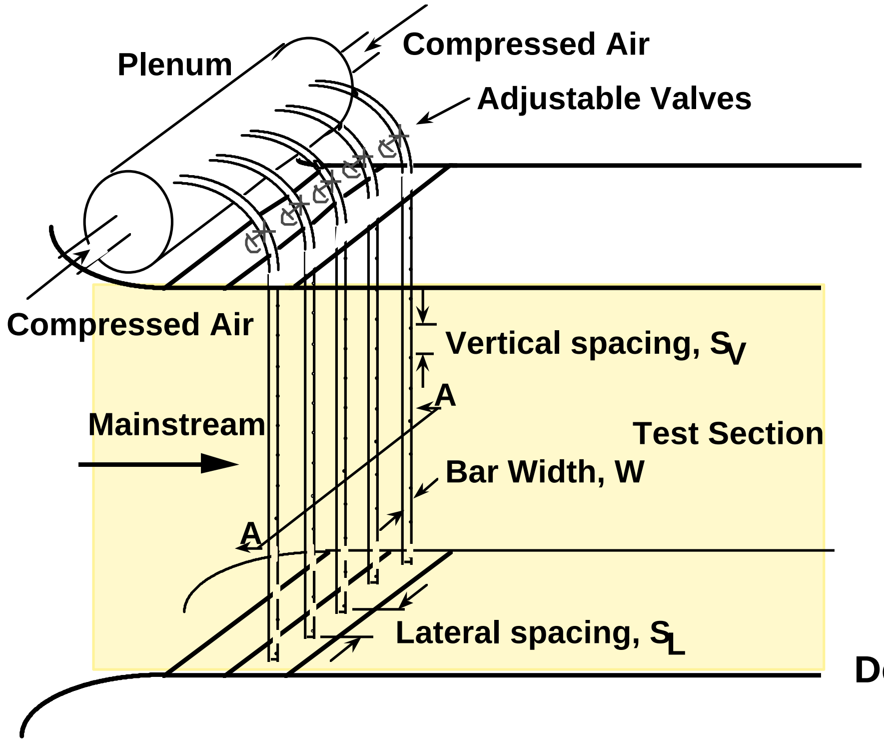


Front View

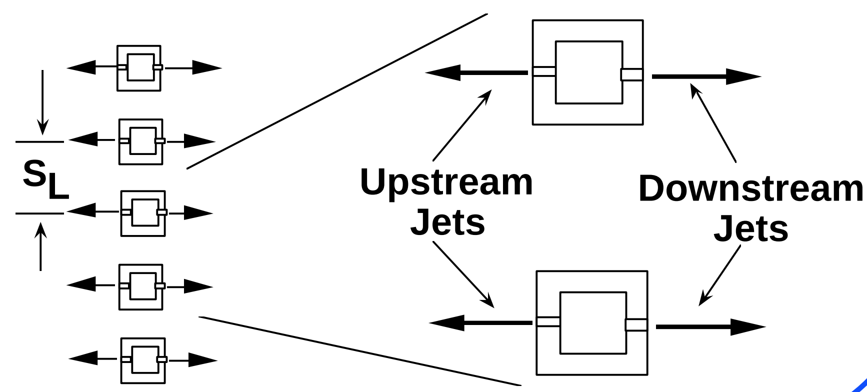
University of Texas Facility



Turbulence Generator

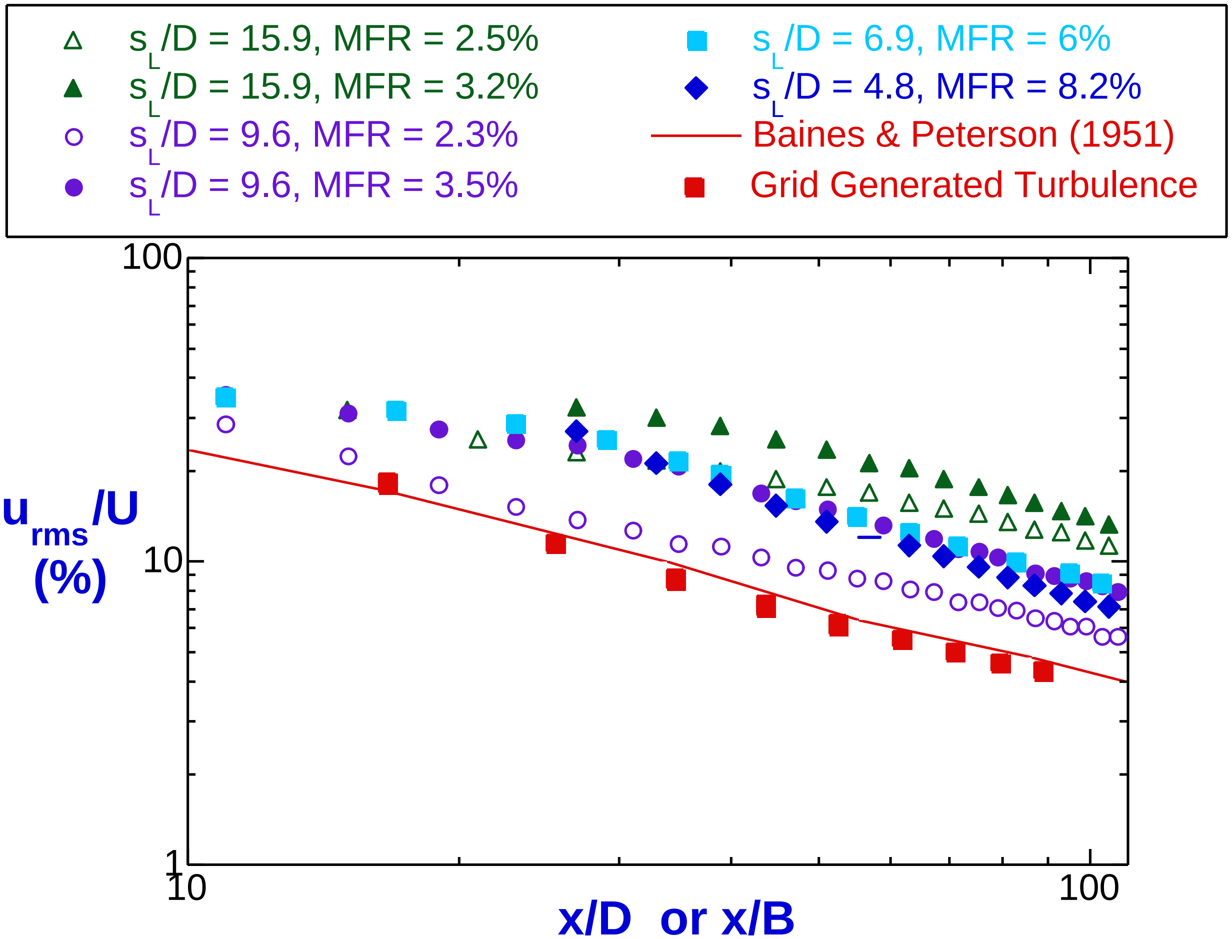


Detail of Section A-A



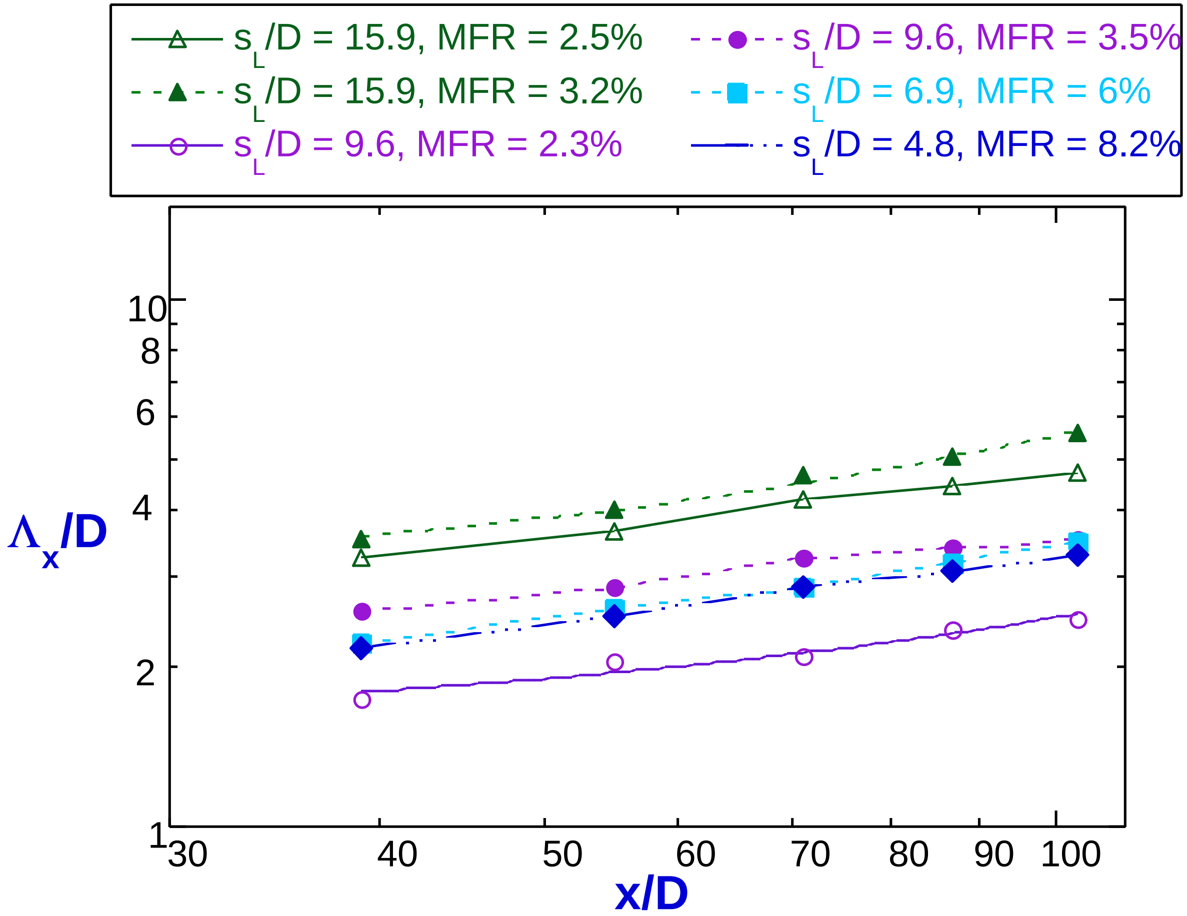
University of Wisconsin

The turbulence generator was developed to achieve various levels



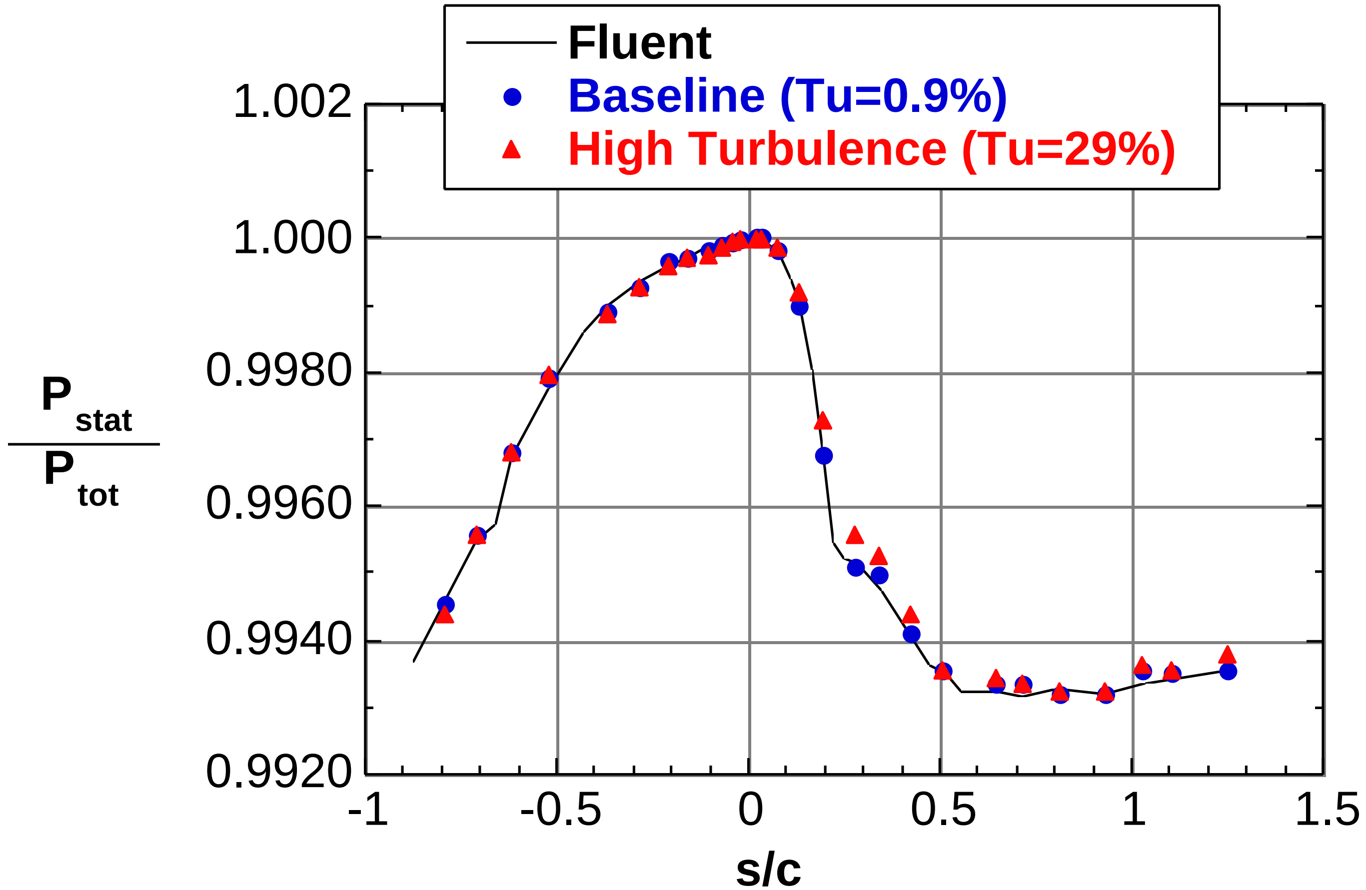
University of Wisconsin

Different length scales could be achieved by varying the rod spacing

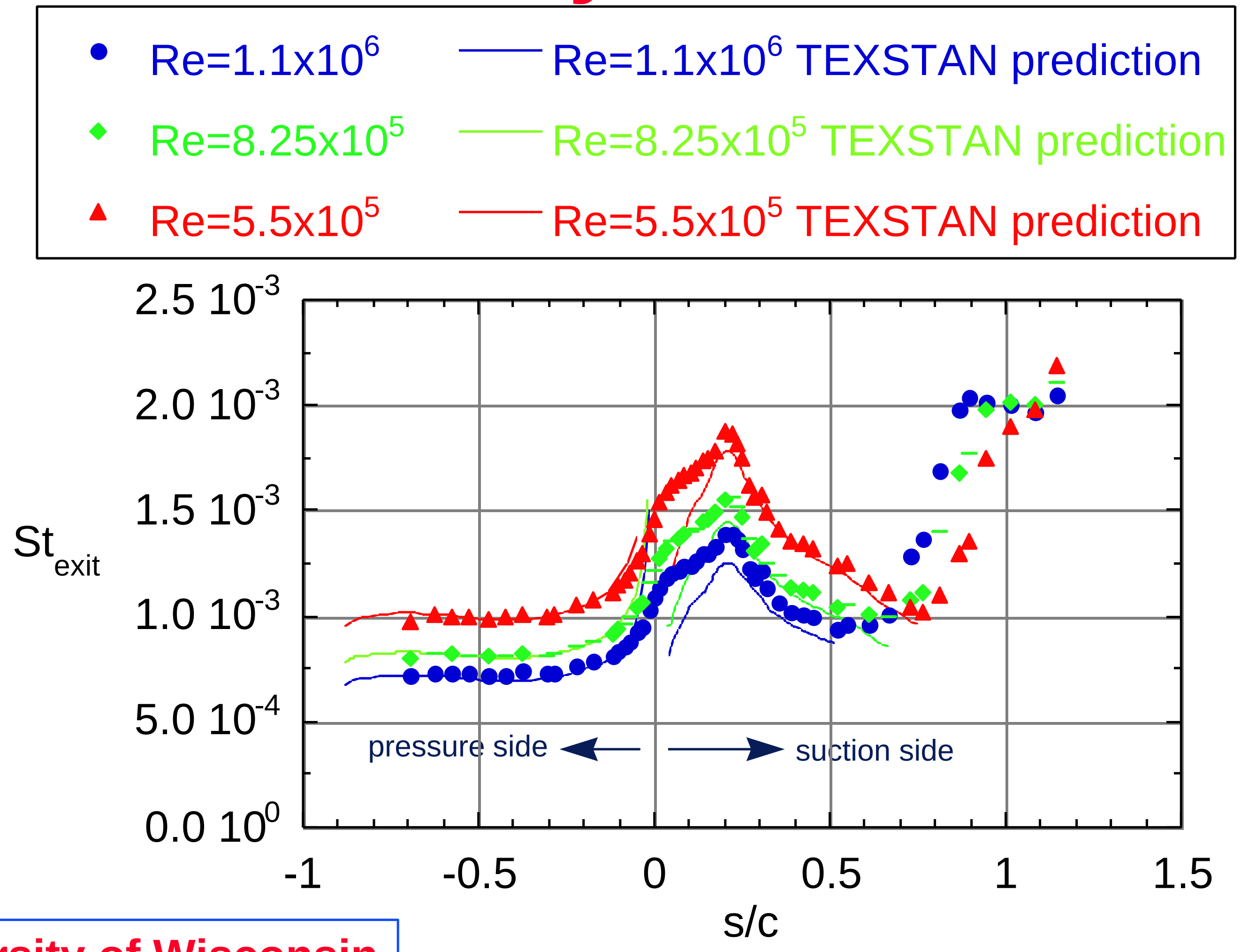


University of Wisconsin

The vane test rig was designed using CFD simulations from FLUENT/UNS



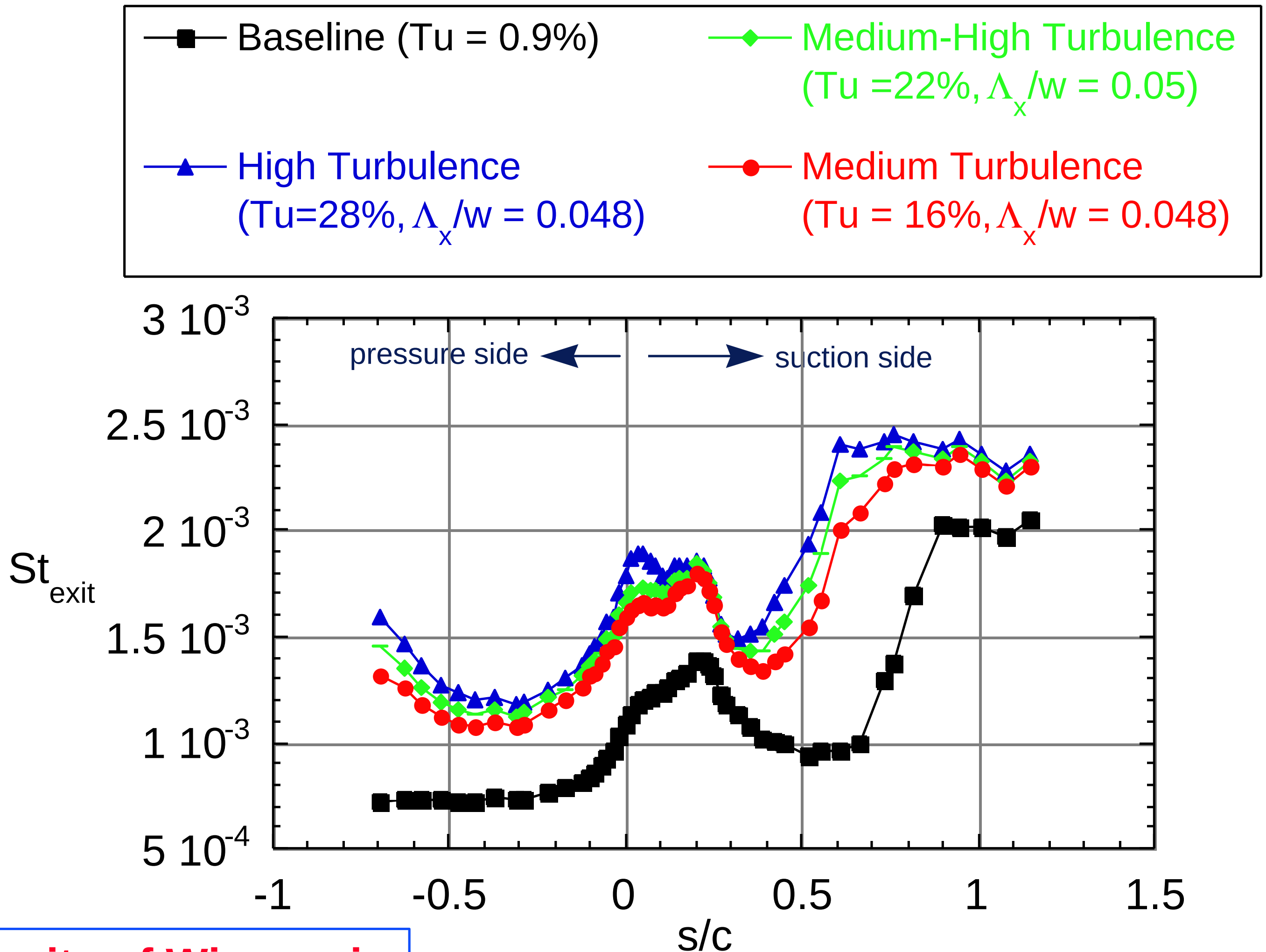
Baseline Vane Heat Transfer For Different Reynolds Number



University of Wisconsin

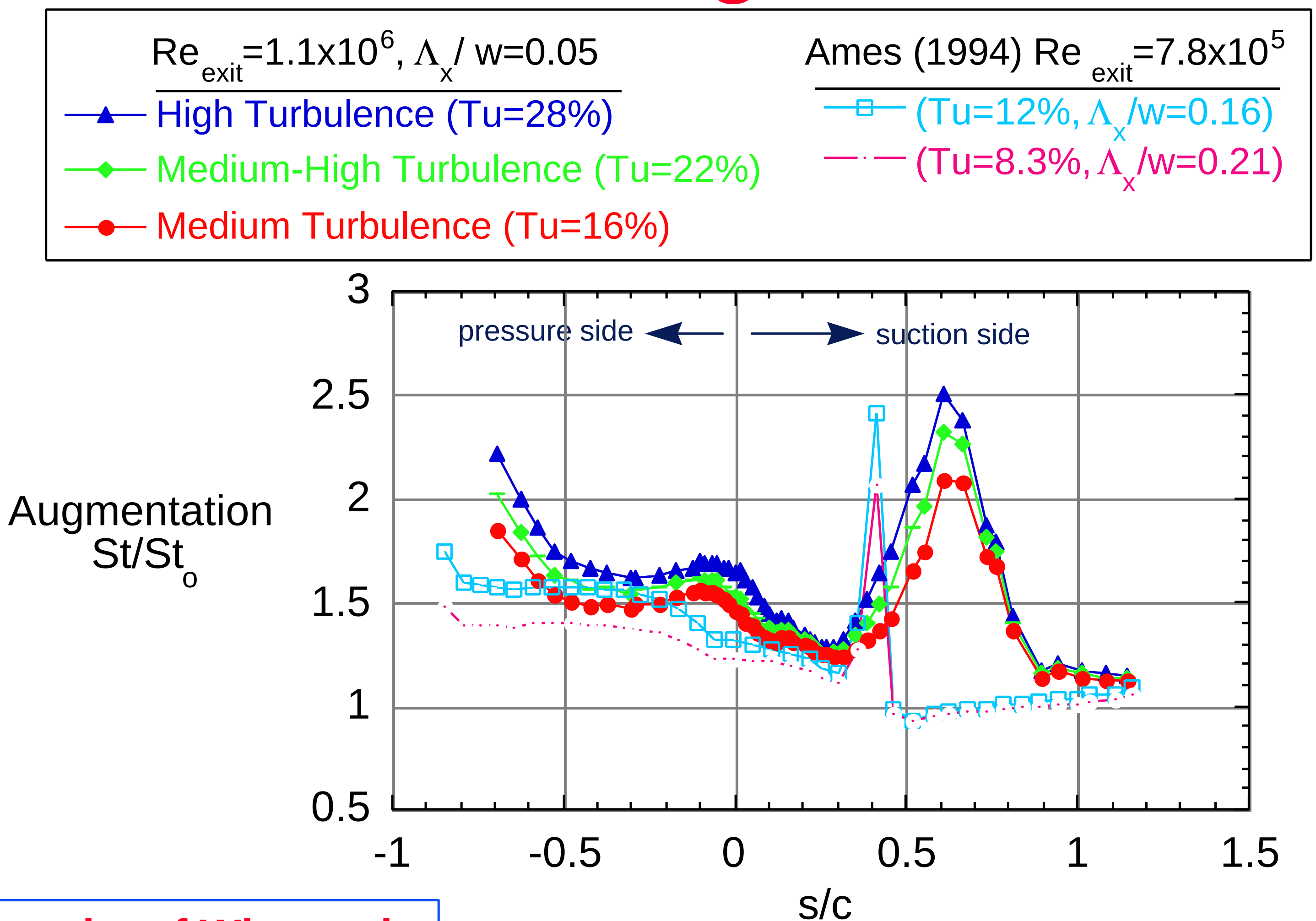
Results To Date

Influence of Turbulence on Vane Heat Transfer at $Re_{exit}=1.1x10^6$



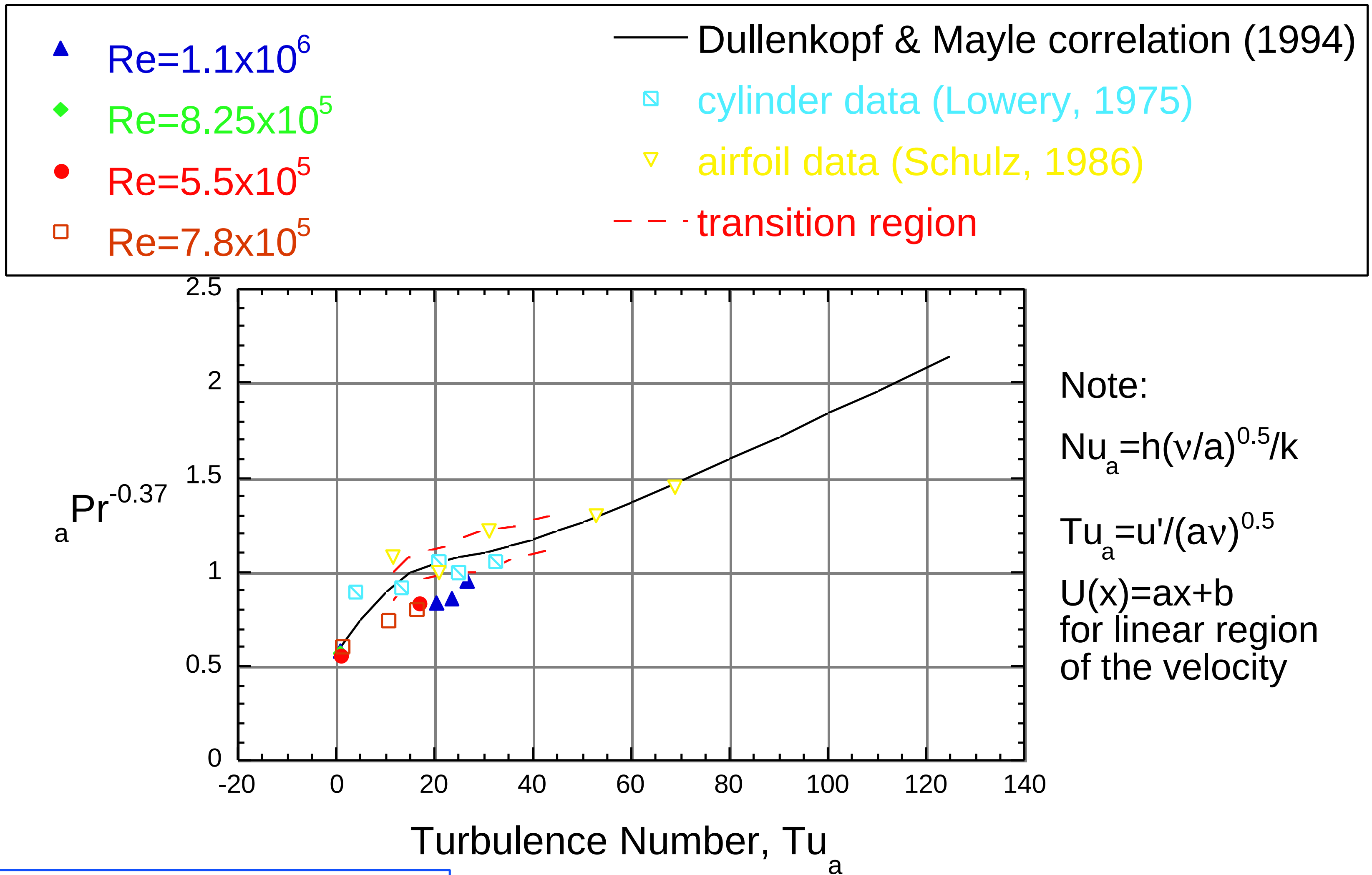
University of Wisconsin

Influence of Turbulence on Heat Transfer Augmentation



University of Wisconsin

Stagnation Point Heat Transfer Correlation



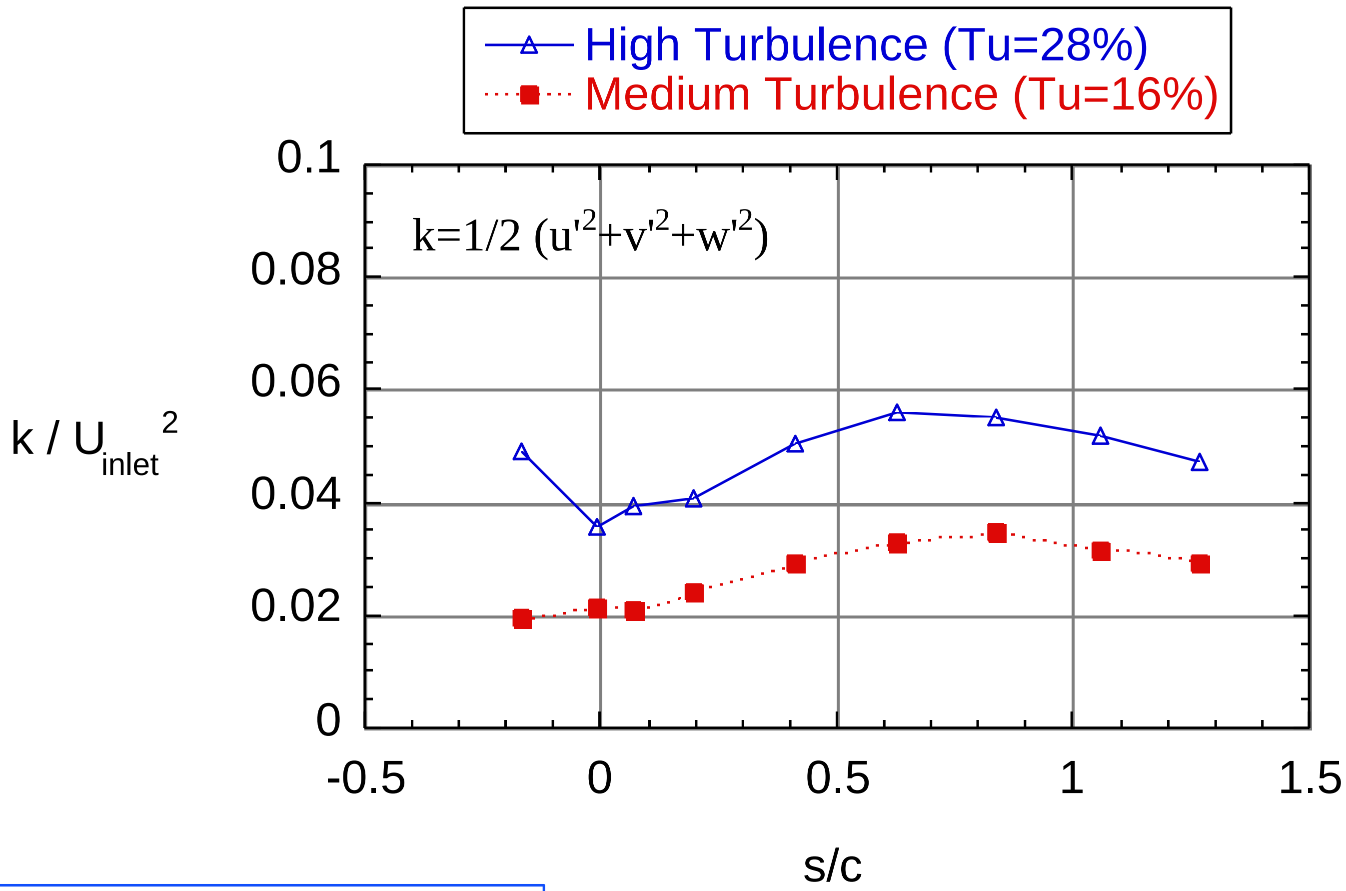
Note:

$Nu_a = h(\nu/a)^{0.5}/k$

$Tu_a = u'/(a\nu)^{0.5}$

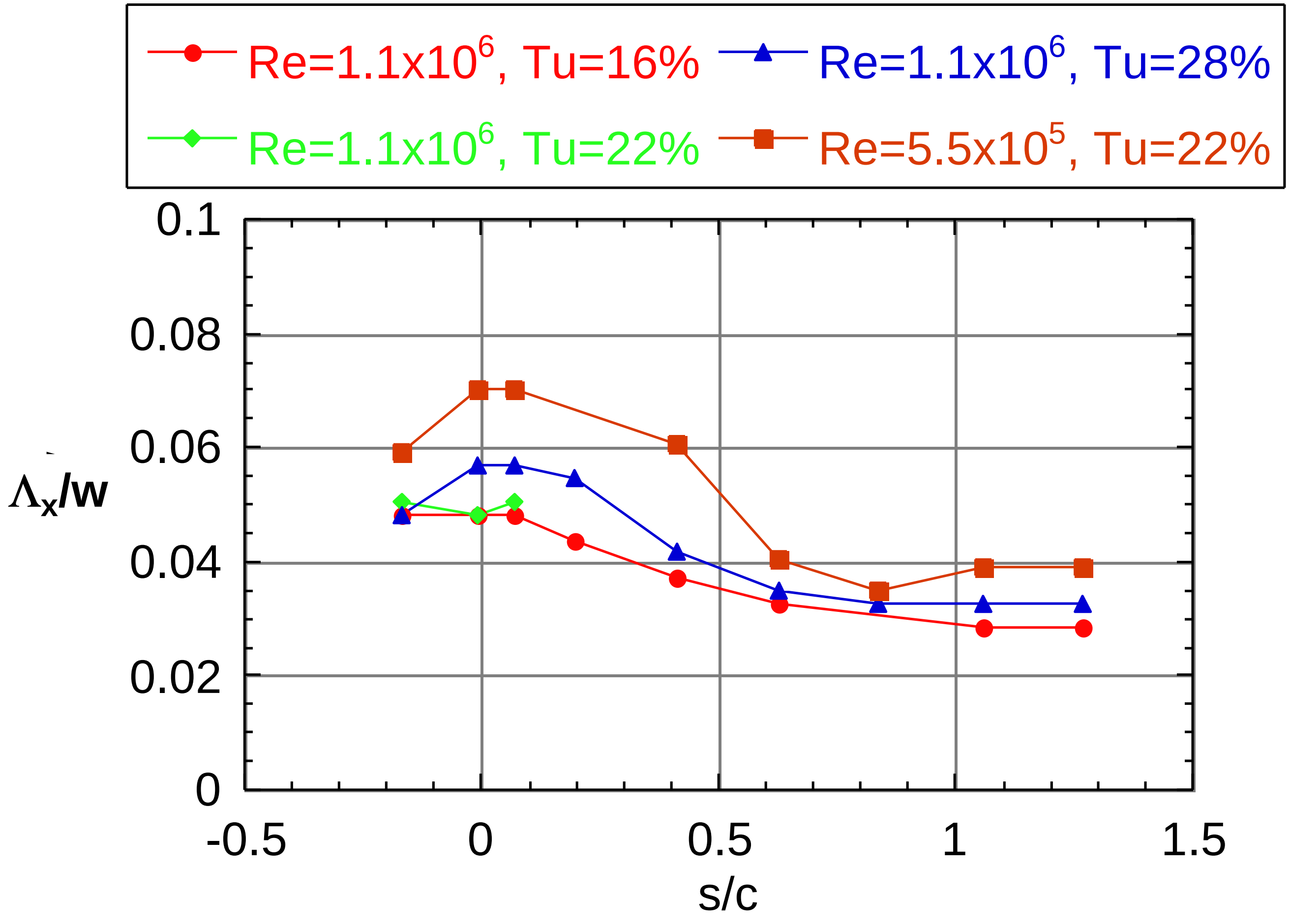
$U(x) = ax + b$
for linear region
of the velocity

Turbulent Kinetic Energy is Relatively High in the Passage

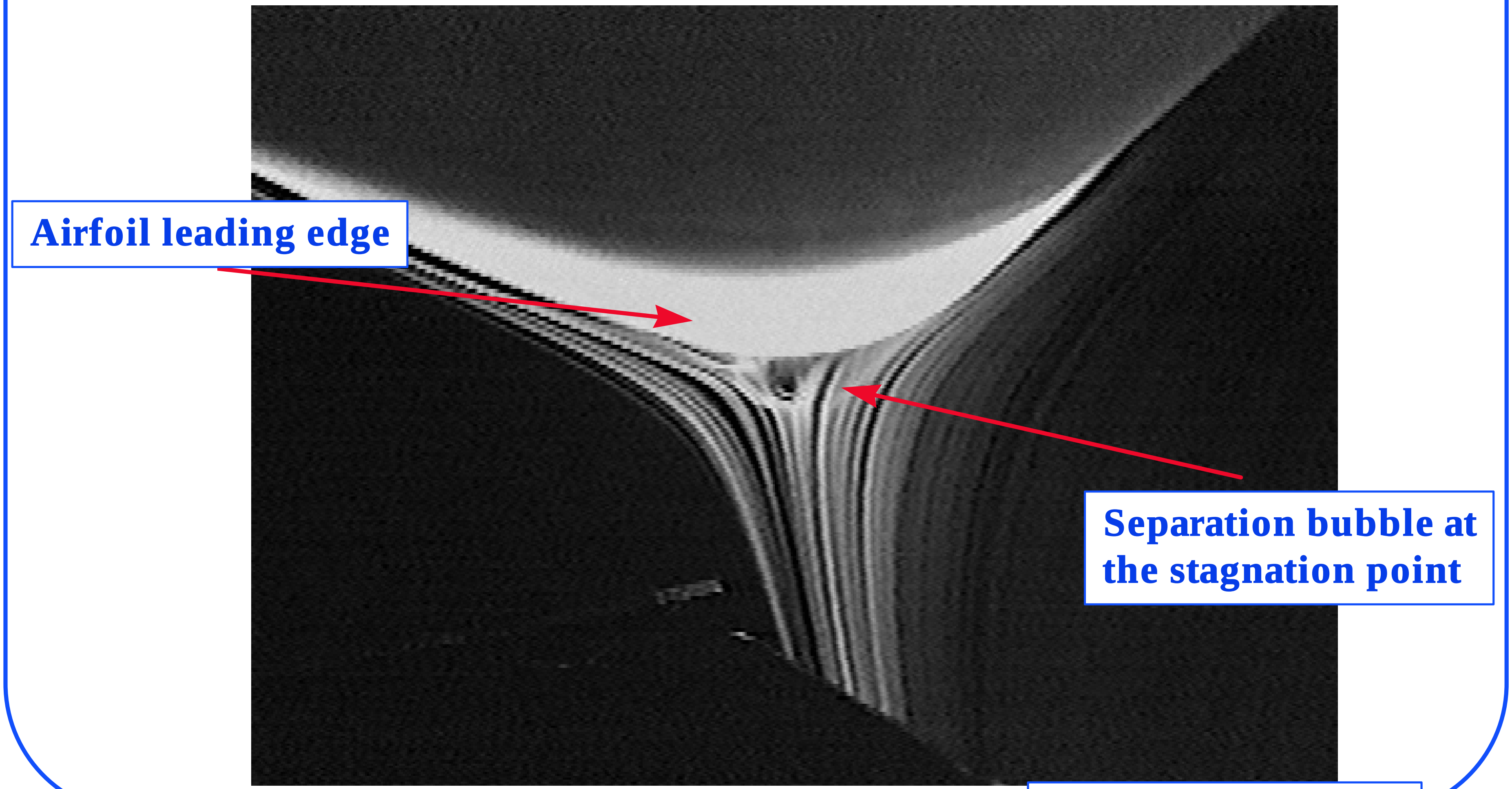


University of Wisconsin

Turbulent Length Scales Measured in the Vane Passage



Smoke wire flow visualization of streamlines around the leading edge of the airfoil



Near Future Activities

- Document the effect of turbulent length scales at high turbulence levels on the surface heat transfer

- Flow and thermal field boundary layer measurements for high and low turbulence cases