Design, Fabrication and Characterization of a New Wind Tunnel Facility – Linear Cascade with a Wake Simulator

Jean-Philippe Junca-Laplace

Louisiana State University and Agricultural and Mechanical College

Follow this and additional works at: https://digitalcommons.lsu.edu/gradschool_theses

Part of the Mechanical Engineering Commons


https://digitalcommons.lsu.edu/gradschool_theses/2773
DESIGN, FABRICATION, AND CHARACTERIZATION OF A NEW WIND TUNNEL FACILITY - LINEAR CASCADE WITH A WAKE SIMULATOR

A Thesis

Submitted to the Graduate Faculty of the Louisiana State University and Agricultural and Mechanical College in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering

in

The Department of Mechanical Engineering

by

Jean-Philippe Junca-Laplace
Institut Supérieur de l’Aéronautique et de l’Espace, ENSICA, Toulouse, France August 2011
Acknowledgments

I am very grateful to my advisors, Dr. S. M. Guo and Dr. D. E. Nikitopoulos for giving me the chance to work on this project. The support from the Air Force Research Laboratories is greatly appreciated as well.

I would like to thank all the Mechanical Engineering machine shop technicians, Barry Savoy, Don Colvin and Jim Layton who is unfortunately no longer with us. Without their great work and help, this research project would not have been realized in such a manner. Thanks are also due to Paul Rodriguez, Chemical Engineering shop manager for his precious advice and good work quality.

I would also like to thank all the members of the Friday afternoon weekly meetings for their support and guidance: Guillaume Bidan, Lem Wells and Jason W. Bitting. In addition, I would like to thank Chris Foreman for sweating it out with me during the assembling of the wind tunnel.

Thanks to all the friends I made here: Avishek, Guillaume and Joe for our afternoon finger soccer games. Diane, Tony and Sara for sharing the Louisianan culture with me. I would also like to thank Tim, Heather and Bidur for being outstanding roommates. And of course, Farren for all the great times we shared together in Baton Rouge and New Orleans.

Finally, thanks to my parents and my sisters for their support and the delicious packages they have sent from France during these two years.
# Table of Contents

Acknowledgments .................................................................................................................. ii
List of Tables ............................................................................................................................ v
List of Figures .......................................................................................................................... vi
Nomenclature ........................................................................................................................... ix
Abstract ..................................................................................................................................... xi

Chapter 1: Introduction .......................................................................................................... 1
  1.1 Background ....................................................................................................................... 3
  1.2 Film Cooling ...................................................................................................................... 3
  1.3 Motivation ........................................................................................................................ 3
  1.4 Literature Survey ............................................................................................................. 4
  1.5 Objectives ....................................................................................................................... 10
  1.6 Outline of Thesis ............................................................................................................. 10

Chapter 2: Wind Tunnel Design ............................................................................................. 12
  2.1 General Description ......................................................................................................... 13
  2.2 Components .................................................................................................................... 15
    2.2.1 Divergent Cone .......................................................................................................... 15
    2.2.2 Corner Vanes ............................................................................................................. 15
    2.2.3 Settling Chamber ....................................................................................................... 16
    2.2.4 Contraction Cone ...................................................................................................... 22
    2.2.5 Fan ............................................................................................................................ 25
    2.2.6 Fan Pitch Control Device ......................................................................................... 25
    2.2.7 Windows and Safety Screens ................................................................................... 27
    2.2.8 Breather ................................................................................................................... 27
    2.2.9 Supporting Structure ............................................................................................... 28
  2.3 Pressure Loss Estimation in the Wind Tunnel ................................................................. 31
    2.3.1 Straight, Constant Area Sections ........................................................................... 32
    2.3.2 Corners ..................................................................................................................... 34
    2.3.3 Divergent Sections ................................................................................................... 34
    2.3.4 Contraction Cone ...................................................................................................... 35
    2.3.5 Settling Chamber: Screens and Honeycomb ........................................................... 35
    2.3.6 Summary of Pressure Losses .................................................................................. 35
  2.4 Wind Tunnel Energy Ratio and Design Analysis ............................................................ 36

Chapter 3: Cascade with Wake Simulator Design ................................................................. 37
  3.1 Setup Characteristics ...................................................................................................... 38
    3.1.1 Linear Cascade Characteristics .............................................................................. 38
    3.1.2 Wake Simulator Characteristics .............................................................................. 39
    3.1.3 Cascade Inlet Plane Angle ....................................................................................... 39
  3.2 Test Section Design ........................................................................................................ 40
  3.3 Wake Simulator Design .................................................................................................. 42
    3.3.1 Frame ........................................................................................................................ 43
    3.3.2 Belt ............................................................................................................................ 44
List of Tables

Table 2.1. Screen combination in the settling chamber..............................................18
Table 2.2. Turbulence reduction factors and pressure loss coefficients of screens.........20
Table 2.3. Estimate of wind tunnel part weights......................................................28
Table 2.4. Pressure losses estimation.................................................................35
List of Figures

Figure 1.1. Specific fuel consumption versus pressure ratio and turbine inlet temperature (Gas Turbine Engineering Handbook, Third Edition) .......................................................... 1

Figure 1.2. Vane and blade cooling in a high pressure turbine stage (Reproduced from Rolls-Royce plc) ............................................................................................................. 2

Figure 1.3. CAD drawing and schematic of wake generating system used at Ohio State University (Bloxham et al, 2009) ........................................................................... 5

Figure 1.4. Cascade with upstream symmetrical airfoil used by Olson et al. (2011) at the University of Minnesota .......................................................... 6

Figure 1.5. Wake generating mechanism at Texas A&M University ............................................. 9

Figure 1.6. Wake generating rig at the NASA Glenn Research Center ........................................... 9

Figure 2.1. Global view of the wind tunnel .............................................................................. 12

Figure 2.2. 2D drawing of the wind tunnel ............................................................................ 14

Figure 2.3. Wind-tunnel parts ......................................................................................... 14

Figure 2.4. Corner vanes a) Type 1 b) Type 2 .................................................................... 16

Figure 2.5. Settling chamber configuration and screen spacing ............................................. 18

Figure 2.6. (a) Screen with aluminum frame (b) Opened settling chamber image ........... 22

Figure 2.7. Matched cubic polynomials lateral curve of the contraction cone .................. 23

Figure 2.8. Contraction Cone Image ............................................................................... 24

Figure 2.9. Pitch Control Supply Pressure System .............................................................. 26

Figure 2.10. Pitch Control System Images. a) Pressure Supply b) Piston Actuator .............. 27

Figure 2.11. Images of the fan installation with supporting structure .................................. 29

Figure 2.12. Test section support image. a) Overhead support b) Global view ............. 30

Figure 2.13. Overhead support load model ....................................................................... 31
Figure 2.14. Moody diagram (from http://www.engineeringtoolbox.com).......................... 33

Figure 3.1. 3-D view of the test section.................................................................37

Figure 3.2. Linear cascade with wake simulator 2-D sketch.................................38

Figure 3.3. Cascade angles and velocity triangle..................................................40

Figure 3.4. Cascade image..................................................................................41

Figure 3.5. Cascade endwall plate location.........................................................41

Figure 3.6. CAD view and picture of the wake simulator.....................................42

Figure 3.7. Conveyor frames.............................................................................43

Figure 3.8. Intralox Series 800 modular plastic conveyor belt (from Intralox® Conveyor Belting Engineering Manual).............................................................44

Figure 3.9. Conveyor shaft with sprockets image................................................46

Figure 3.10. Image of the inside frame of the conveyor with wearstrips..............47

Figure 3.11. Conveyor stand picture. (a) General view (b) Zoom on clevis rods.....50

Figure 3.12. Views of the wake simulator mechanism integrated to the test section....51

Figure 3.13. Plate assembly image (a) Stand alone (b) Fixed to belt....................52

Figure 3.14. Moving plate assembling process..................................................53

Figure 3.15. Conveyor assembling steps............................................................53

Figure 4.1. Velocity signal through time.............................................................59

Figure 4.2. Schematic of a Constant Temperature Anemometer..........................61

Figure 4.3. Measuring chain components of a CTA system (Jorgensen, 2002)........62

Figure 4.4. TSI 1211-20 standard CTA probe..................................................63

Figure 4.5. CTA experimental setup..................................................................64
Figure 4.6. Image of Pitot tube with handheld manometer

Figure 4.7. Calibration fitting curve

Figure 4.8. Slot locations for the traverse system

Figure 4.9. Variation of static dynamic and total pressure along the wind tunnel

Figure 4.10. Fan operating conditions. a) Volume flow rate versus pitch setting. b) Total pressure rise versus pitch setting

Figure 4.11. Temperature variation through time in the wind tunnel

Figure 4.12. CTA measurement points location over the test section inlet plane ‘S0’

Figure 4.13. Velocity contours over the test section inlet ‘S0’ plane (in m/s)

Figure 4.14. CTA measurement points location over the test section inlet plane ‘S0’ bottom right corner

Figure 4.15. Velocity contours over the plane ‘S0’ bottom right corner (in m/s)

Figure 4.16. Streamwise turbulence intensity contours over the cross-section ‘S0’ in %

Figure 4.17. Streamwise turbulence intensity contours over the plane ‘S0’ bottom right corner (in %)

Figure 4.18. Figure 4.17. Boundary layer profile at the center of the plane ‘S0’ bottom wall

Figure 4.19. Inclined CTA traverse setup

Figure 4.20. Wake velocity profile on a L1A blade at Re=450,000

Figure 4.21. Turbulence intensity profile on a L1A blade at Re=450,000
Nomenclature

c  Axial chord length
C  Contraction ratio, Absolute velocity
C\textsubscript{X}  Axial chord length
D  Local equivalent diameter
E  Young modulus
ER\textsubscript{t}  Wind tunnel energy ratio
f\textsubscript{a}  Axial turbulence reduction factor
f\textsubscript{l}  Lateral turbulence reduction factor
H  Boundary layer shape factor
I  Moment of inertia
k  Absolute roughness
K  Pressure loss coefficient
K\textsubscript{0}  Pressure loss coefficient referred to jet dynamic pressure \( q_0 \)
K\textsubscript{t}  Total pressure loss coefficient
L  Section length
M  Mesh size (number of wires per inch)
P  Pressure
q  Mean flow dynamic pressure
q\textsubscript{0}  Jet dynamic pressure
R\textsubscript{d}  Reynolds number based on wire diameter
R\textsubscript{e}  Reynolds number
U  Rotor blade velocity
W  Relative velocity

Greek Symbols

\( \alpha \)  Divergence equivalent angle
\( \beta \)  Open-area ratio (%)
\( \delta_{\text{max}} \) Maximum deflection length
\( \delta^* \) Boundary layer displacement thickness
\( \lambda \) Skin friction coefficient
\( \theta^* \) Boundary layer momentum thickness

**Acronyms**

2-D Two dimensional
3-D Three dimensional
AC Alternative Current
AFRL Air Force Research Laboratories
AISC American Institute of Steel Construction
CAD Computer-Aided Design
CFD Computational Fluid Dynamics
CFM Cubic Feet per Minute
CNC Computer Numerical Control
CTA Constant Temperature Anemometry
FPS Feet Per Second
HP Horse Power
LPT Low Pressure Turbine
NGV Nozzle Guide Vanes
RPM Rotation Per Minute
SFC Specific Fuel Consumption
TIT Turbine Inlet Temperature
UHMW Ultra High Molecular Weight
VFD Variable Frequency Drive
Abstract

A new wind tunnel has been designed and constructed at the LSU Mechanical Engineering Laboratories. The objective was to design a versatile test facility, suitable for a wide range of experimental measurements on turbine blades. The future study will investigate the impact of unsteady inflow conditions on film cooling performance. More specifically, it will study how the unsteady flow due to the upstream passing wakes coming from the front row vane affects the film cooling performances on the turbine blades.

The test section consists of a four passage linear cascade composed of three full blades and two shaped wall blades. The 2D blade shape profile of the cascade was provided by the Air Force Research Laboratories (AFRL). It is a High Lift Low Pressure Turbine (LPT) blade, ‘L1A’ profile. A conveyor setup was designed and fabricated to simulate the passing wakes upstream of the testing blades. Wakes are generated with thick plates in translation on this conveyor. These moving plates simulate the wake passing of the front row vane. This facility has been designed to enable easy interchanges of different experimental setups. The new test facility was chosen to be a closed-circuit wind tunnel to ensure a controlled return flow and reach low levels of turbulence and unsteadiness in the test section. Preliminary characterization of the experimental apparatus was conducted using a Constant Temperature Anemometry technique coupled to pressure and temperature measurements. The velocity variation over the cascade inlet cross section is found to be less than 2% at a mean velocity of 50 m/s (164 fps) and the freestream turbulence intensity reaches values as low as 0.12% at the cascade inlet cross section.
Chapter 1: Introduction

1.1 Background

Gas turbine manufacturers are continually developing cutting-edge technologies to improve the performance of their engines in order to meet the demand of today’s market. Newly developed aircraft engines have to meet the needs for high efficiency, performance and longevity in addition to the environmental constraints through reduced chemical and noise pollutions. A parameter that is used to define the performance and the efficiency of a gas turbine is the Specific Fuel Consumption (SFC). The SFC is the fuel consumption per unit net work output or per unit thrust for aircraft engines. Figure 1.1 illustrates the variation of SFC with turbine inlet temperature (TIT) and compressor pressure ratio, based on the thermodynamics of the gas turbine cycle. It can be observed that increasing the TIT at a constant pressure ratio through a change in fuel/air ratio causes a reduction in SFC and an increase in efficiency.

Figure 1.1. Specific fuel consumption versus pressure ratio and turbine inlet temperature. 
(Gas Turbine Engineering Handbook, Third Edition)
In modern gas turbine, the turbine inlet temperature has been increased up to 3,000 °F, well above the turbine blade and vane operating temperature of 1,700 °F. This advancement has been partly due to improvements in materials used for turbine blades, but more significantly due to extensive cooling on the blades to protect them from extreme temperatures. A variety of cooling techniques have been investigated but we can specify two main categories usually used in combination:

- **Internal convection cooling**, in which cool air is circulated inside the blade in cast channels and used as a heat sink before being ejected into the main flow, usually at the trailing edge.
- **External cooling**, such as film cooling for which numerous discrete holes across the blade surface discharge coolant air to form a thin film which insulates the blade from the main flow, as illustrated in Figure 1.2.

![Figure 1.2. Vane and blade cooling in a high pressure turbine stage (Reproduced from Rolls-Royce plc)](image-url)
1.2 Film Cooling

Film cooling is widely used in modern high temperature and high pressure gas turbine engines as an active cooling technique. The air used for cooling is drawn from the outlet of the compressor, before the combustion chamber where the air is relatively cooler, resulting in losses in overall engine efficiency due to loss of work producing capacity. In addition, thermodynamic and aerodynamic losses are introduced when the cooling air is mixing with the hot mainstream airflow and then affects the efficiency of the turbine stage. Nonetheless, the benefits from an increase in permissible TIT to reduce the SFC are still substantial even when the additional losses introduced by the cooling techniques are taken into account (Cohen, 1996).

The objective of film cooling is to provide a relatively uniform and constant material temperature within the material operating temperature limit in order to minimize the thermal stress and maximize the component life. Ongoing research on film-cooling is focusing on minimizing the amount of coolant used while keeping a satisfactory cooling of the blades. Several solutions are explored to improve the cooling efficiency such as optimizing the shape of the cooling holes or using an actively controlled pulsed film-cooling to minimize the necessary coolant mass flow rate and increase the global performance.

1.3 Motivation

Film cooling has been studied for more than four decades in order to increase the cooling efficiency and minimize the induced losses, but very few studies have investigated the impact of unsteady inflow conditions. In real gas turbines, the flow is highly unsteady. Two types of unsteadiness can be specified: the unsteadiness due to high freestream turbulence in the flow and the unsteadiness due to the interaction between vane and blade rows (Womack, 2008). Upstream
wakes coming from vanes cause a deficit of mean velocity coupled to an increased turbulence, altering greatly film cooling performance. Passing wakes can cause a disruption of the film cooling jet and may cause a decrease in film cooling efficiency in some areas and an increase in others. Some studies (see Literature Survey Part 1.4) have shown that an increase in freestream turbulence causes an increase in heat transfer coefficient and reduce the film cooling efficiency. Wake induced turbulence may also reduce the overall performance of film cooling.

An actively controlled film cooling system that takes into consideration the effects of periodic wakes coming from the front row vanes may enhance the overall performance of film cooling on the following blade row. A closed loop active control of pulsed jets could be investigated to mitigate the detrimental effects of passing wakes on film cooling and emphasize positive effects if known. This control of coolant fluid may reduce the necessary coolant mass flow rate for good cooling performance and therefore increase the overall engine efficiency.

1.4 Literature Survey

The impact of unsteady inflow conditions on turbine blades due to upstream passing-wakes has been investigated with different approaches during the last decade. Several wake generating mechanisms have been designed for this purpose and the idea of an active control system to enhance the aerodynamic or heat transfer of the blades has been issued several times.

Bloxham et al. (2009) suggested a synchronization of the unsteady jet disturbance with the unsteady passing wakes for a separation control system. Although this research study is not directly linked with film cooling, it involves active control techniques of the jets on the blades to reduce blade aerodynamic losses. Wakes are generated using a spoked-wheel and passing rods. This wake generator is placed $0.53C_x$ upstream of the cascade inlet plane and made with carbon
fiber rods on a chain sprocket system driven by a variable frequency drive as illustrated in Figure 1.3. An optical sensor gives information about the passing wakes frequency. Then this frequency signal is sent to the pulsing system to coordinate the jet’s pulsing frequency.

In Bloxham’s paper, it is stated that any active flow control scheme (through pulsing or variation of a jet blowing ratio) should be compatible with the unsteady flow environment. Then the objective of their study was to find a synchronization scheme between the unsteady vane wakes and their jet properties (blowing ratio and duration) to optimize the time-averaged reduction of the separation bubble and to reduce aerodynamic losses of their cascade. They concluded that several parameters such as the jet duration and blowing ratio as well as time delay between the wake disturbance and the jet pulse disturbance (synchronization) should be considered to optimize the separation control effectiveness.

Olson et al. (2011) investigated the effect of wakes under quasi-steady cases to heat transfer to a turbine blade, using symmetrical blades to simulate the wakes of a row of stator
blades (see Figure 1.4). They also studied in parallel the effect of the background turbulence level using several turbulence-generating grids. A heat transfer study was conducted using a naphthalene mass transfer technique to determine the local mass transfer coefficient, and then get the heat transfer coefficient according to the heat/mass transfer analogy. Velocity and turbulent intensity profiles were measured at 4 locations downstream the vanes.

![Diagram of a cascade with upstream symmetrical airfoil](image)

**Figure 1.4. Cascade with upstream symmetrical airfoil used by Olson et al. (2011) at the University of Minnesota**

Olson investigated the effect of wakes and turbulence in a two-dimensional cascade on heat transfer to turbine blades (without film-cooling). They studied:

- the effect of the wake-blade (vane blade) position
- the effect of the wake-blade pitch
- the effect of the wake-blade gap (distance between the trailing edge of the vane row and the leading edge of the turbine-blade row). Ratios “gap over axial chord” were changed to various values: 20, 40 and 80%.
They noticed that wakes coming from the vanes caused an earlier start of transition to a turbulent boundary layer on the suction side when the vanes are near the leading edge of the turbine blades, which highly increases the heat transfer coefficient.

As for the effect of the wake-blade pitch, they noticed that when the vane was in the center of the passage, changing the pitch had no or few effect. Though, when the vane trailing edges are directly ahead of the turbine blade leading edge, they observed that a lower pitch caused an earlier start of transition to a turbulent boundary layer on the suction surface and a greater separation on the suction surface. In addition, they pointed out that a wake directly upstream of the leading edge of the turbine blade reduced the separation on the suction surface.

Increasing the gap between the blade rows delays the transition to a turbulent boundary layer on the suction side and reduces the effect of the wake on the separation phenomenon on the pressure side. The further the blade is, the less the effects of the wakes can be observed.

At a higher Reynolds number, the heat transfer coefficient globally increases, and the transition to a turbulent boundary layer on the suction surface moves upstream. On the pressure surface, the separation phenomenon near to the leading edge is reduced. Higher background turbulence can be viewed as dampening the effect of the wakes on heat transfer.

Womack, Volino and Schultz (2008) studied experimentally film cooling flows with periodic wakes as well as the combined effects of wakes and pulsed film cooling. They generated the wakes with a spoked-wheel similar to Bloxham et al (2009) in front of a flat plate. It is stated that over a cycle of wake passing, film cooling jets provide a good coverage and a good film cooling effectiveness during a part of the cycle, but when the wake passes over the film cooling jets, the effectiveness drops and the heat transfer coefficient increases. Then an active control over each cycle may increase the overall film cooling efficiency. It is suggested to control the jet
pulsing favorably with respect to the wake passing event. Activating the jet when the film cooling efficiency is optimal may be an option. Nonetheless, decreasing the jet blowing ratio when the wake passes may cause damages to the blades on the long term. When the wake impinges on the blade, the film cooling efficiency drops, the heat transfer coefficient increases and the blade surface may need to be protected by a better film cooling layer.

Womack et al. studied transient flow behavior with phase averaged flow temperature measurements. It was clearly observed that wakes are disturbing the film cooling jets resulting in what they called an “unsteady effectiveness” during the wake passing, lower than the steady case effectiveness. The jet recovery between wakes depended on the wake Strouhal number (the Strouhal number is proportional the frequency and size of the wake generating body and inversely proportional to the mainstream flow velocity). At low Strouhal number, the recovery of the jets was good but at high Strouhal number, it was insufficient, reducing the cooling effectiveness by 50%.

In a following paper (2008), Womack et al investigated the combined effect of wakes and jet pulsing. After studying the effect of all combination of jet pulsing and wake timing, they concluded there was no clear benefit to imposing pulsation on film cooling on their flat plate with a cooling hole geometry. Though, they provided a better comprehension of how wakes are affecting the film cooling behavior. For instance, at low blowing ratios ($B=0.5$), when the jet stays well attached to the cooled surface, both jet pulsing and passing wakes are detrimental to the film cooling effectiveness and the combination of both effects is even worse. However, for a higher blowing ratio ($B=1.0$), when the cooling jet is lifting off, pulsing was still reducing the film cooling effectiveness but wakes tended to increase the overall cooling effectiveness by forcing the jet coolant fluid closer to the wall.
Other noteworthy wake simulator mechanisms can be pointed out. A facility was designed at Texas A&M University to study unsteady boundary layer transition on the blade surface (Shobeiri, 2007) and later on, the effect of upstream wakes with vortex on blade platform film cooling behavior (Wright, 2009). This system also uses cylindrical rods and their size, their distance upstream from the cascade; their pitch and velocity are chosen so that generated wakes are matched to the wakes shed from vane blades.

![Image](image1.png)

**Figure 1.5. Wake generating mechanism at Texas A&M University (Shobeiri, 2007)**

Another wake generating mechanism was designed at the NASA Glenn Research Center. It is a rotating rig with cylindrical rods in an annular cascade (Figure 1.6). Heidmann (2001) used this facility to investigate the effect of wake-passing on turbine blade film cooling.

![Image](image2.png)

**Figure 1.6. Wake generating rig at the NASA Glenn Research Center (Heidmann, 2001)**
An average reduction of 0.094 due to the effect of wakes on film cooling effectiveness was found. They also observed a better film cooling effectiveness at higher Strouhal numbers on the blade pressure side because wakes force the jet towards the blade surface. An opposite effect was found on the suction side. It is also recommended to have an advance instrumentation capable of resolving high frequency data if transient measurements are to be done.

1.5 Objectives

The primary objective of this study will be to investigate the impact of Nozzle Guide Vanes (NGV) wakes on turbine blades, and, their influence on pulsed film cooling performances. Although several studies have investigated the effect of upstream passing wakes with rods and spoked-wheel, never has a study investigated the transient flow behavior of film cooling with more realistic vane profiles at a high Reynolds number.

The present work will focus on the design and fabrication of a new low turbulence wind tunnel facility with a versatile experimental setup suitable to a wide range of heat transfer and aerodynamic measurements on turbine blades. The new facility will include a linear cascade with a wake simulator mechanism consisting of flat plates moving in translation upstream of the linear cascade. It should enable easy interchanges to study the influence of different trailing and leading edges on these flat plates. These plates should also be easily replaced by blade profiles in the future. An approximate Reynolds number of 500,000 based on the blade axial chord and similar to real engine conditions is targeted.

1.6 Outline of Thesis

The content of this thesis will cover in detail the design and fabrication of all the different components of the new facility. The closed circuit wind tunnel design will be described in the
The design of the experimental apparatus - the linear cascade with the wake simulator - and its operating conditions will be covered in the third chapter. Most of the detailed drawings of the designed parts can be found in the appendices. The fourth chapter will present a first characterization of the wind tunnel performances, with the nominal operating conditions. Constant temperature anemometry (CTA) measurement technique with an automated traverse was utilized for this characterization. A summary and conclusion of this work will be presented in the fifth and last chapter.
Chapter 2: Wind Tunnel Design

A new wind tunnel has been designed and constructed at the LSU Mechanical Engineering Laboratories. The design of this wind tunnel facility with all relevant dimensions and observations of interest are given in this chapter for future works.

The objective was to design a test facility adapted to a wide range of experimental measurements on turbine blades. The test section consists of a 4 passage linear cascade. A conveyor setup was designed and fabricated to simulate passing wakes upstream of the cascade in order to study their impact on the turbine blade performances. This facility has been designed to enable easy interchanges of different experimental setups. All the parts, as well as the complete assembly, have been fully designed using a Computer Aided Design (CAD) software. Figure 2.1 shows a general view of the 3D design.

Figure 2.1. Global view of the wind tunnel
The new test facility was chosen to be a closed-circuit wind tunnel to ensure a controlled return flow and to reach low levels of turbulence and unsteadiness in the test section. In order to achieve a first design of this facility, general design rules and suggestions from Bradshaw P. and Mehta R. (1979) and Rae W. H. and Pope A. (1984) were first followed.

2.1 General Description

The whole wind tunnel is about 33 feet long and 11 feet high. The fan diameter is 38 inches. Velocity distribution in the wind tunnel is shown in Figure 2.2. After the test section, the flow is expanding through a first diverging duct and reaches the first corner vanes (see Figure 2.3). Then 2 consecutive diffusers with equivalent cone angle of $5^\circ$ expand the flow without separation before passing through the fan. The fan is surrounded by 2 flexible ducts to prevent excessive vibration propagation within the wind tunnel. After the fan, the flow passes through a last diffuser and expands into a 38 inches side square section duct. Then the flow goes through 2 more corners with 13 corner vanes in each. A 19 feet long duct with constant section follows before the flow goes into the settling chamber. The settling chamber consists of a 2 inch thick honeycomb to straighten the flow and 5 screens to reduce turbulence levels. Wood spacers are used to hold them in place inside the settling chamber box. The screen spacing depends on the mesh length of each upstream screen in order to optimize the turbulence reduction of the air flow (see Part 2.2.3). Then, the well-conditioned and uniformed flow enters the contraction cone and is accelerated to the test section inlet. The contraction cone consists of 2 matched cubic polynomial curves to guide the flow from a 38 inch square duct to a 19.5 inch x 12 inch rectangle duct, equivalent to a contraction ratio of 6.16.
Figure 2.2. 2D drawing of the wind tunnel

Figure 2.3. Wind-tunnel parts
2.2 Components

The wind tunnel is mainly made of 16 gage galvanized sheet metal except for the test section parts that are mostly made out of acrylic to allow visualizations and non-intrusive measurement techniques. Adapted stiffeners have been added to prevent excessive vibration in some of the large tunnel ducts. Quarter inch thick rubber gaskets are placed between all flanges to reduce vibration and for sealing concerns. Detailed dimensions are given in appendix A.

2.2.1 Divergent Cone

Two divergent cones, or diffusers, expand the flow from the test section to the fan with a total section area ratio of 4.6. One more diffuser is located downstream of the fan to increase the wind tunnel section area and to reach a good contraction ratio to get a better flow quality in the test section (see Contraction Cone part). For best flow steadiness, the equivalent angle of the diffusers, considering a circular cone with the same length and same area ratio, do not exceed 5°. When this condition is satisfied, the boundary layer does not separate and unwanted pressure fluctuations with time in the wind tunnel are prevented (Mehta and Bradshaw, 1979).

2.2.2 Corner Vanes

Corner vanes are generally used as guide vanes, which deflect the flow while avoiding separation of the boundary layer in a bend. There are two different types of corner vanes in this wind tunnel. Both are made out of sheet metal.

The first type of corner vane, used in the corners “Turn 1” and “Turn 2” is the most conventional one (Mehta and Bradshaw, 1979). The design rule of those 90° circular arc vanes is similar to a turbomachine cascade. This cascade consists of 17 vanes with a 10 inch chord and a
gap of 3 inches between each other. The gap-chord ratio of the vanes is less than 1:3, as recommended according to experience. The wake effect from each vane disappears over a shorter distance if many short chord vanes are used instead of fewer large-chord vanes.

For design concerns, the first corner “turn 1” downstream the test section, with a non 90° angle has a different configuration. Larger-chord vanes are used while keeping a similar gap-chord ratio. In addition, the turning vane is extending before and after the bend by 10% of its chord to provide a better guide.

![Figure 2.4. Corner vanes a) Type 1 b)Type 2](image)

2.2.3 Settling Chamber

In order to reduce the turbulence level in the test section and to get a good flow quality, 5 screens and a honeycomb are installed upstream of the contraction cone. The role of the honeycomb is to remove some swirl and lateral velocity variations. Screens mostly reduce streamwise velocity fluctuations.
2.2.3.1 Design and Specifications

The settling chamber consists of a 2 inch thick, 1/4 inch cell honeycomb to straighten the flow and 5 stainless steel wire mesh screens. Different open-area ratio and mesh size screens are combined to optimize the turbulence level reduction. Two of the screens are used before and after the honeycomb to hold it in place. All screens have an open-area ratio greater than 57%. For lower open-area ratios, flow instabilities due to jet coalescence may occur (Morgand, 1960). Turbulence reduction by use of screens was investigated by Tan-Atichat et al (1982) and Groth and Johansson (1988). They found that the turbulence damping ability was improved - for a given open area ratio- when decreasing the mesh size. Groth and Johansson demonstrated that subcritical screens - for Reynolds numbers based on the wire mesh diameter $Re_d$ less than 40 - resulted in a better turbulence reduction but with a large pressure drop. Tan-Atichat observed that for supercritical screens, the length scale of the mesh should be chosen so that the turbulence eddies generated by the screen are smaller than the incoming ones. Groth and Johansson found that the maximum turbulence suppression for a given total pressure loss was obtained with a cascade type combination of supercritical screens with decreasing mesh size in the streamwise direction. In addition, they recommended the furthest screen downstream to have a low supercritical Reynolds number ($Re_d = 50-60$) and the first screen upstream to have a relatively coarse mesh while keeping its wire mesh Reynolds number less than 300. Over a Reynolds number of 300, increasing the wire mesh diameter is not as beneficial in terms of pressure drop reduction.

Our final screen configuration is showed in Table 2.1 below. The Reynolds number has been calculated for a nominal air velocity of 50 m/s (164 fps) in the test-section, corresponding to a velocity of 8.1 m/s (26.6 fps) in the settling chamber.
Table 2.1. Screen combination in the settling chamber

<table>
<thead>
<tr>
<th>Screen</th>
<th>Mesh Size (per in)</th>
<th>Open-area ratio</th>
<th>Wire diameter (in)</th>
<th>Reynolds number ( R_d )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12 M</td>
<td>61.5%</td>
<td>0.018</td>
<td>215</td>
</tr>
<tr>
<td>2</td>
<td>20 M</td>
<td>67.2 %</td>
<td>0.009</td>
<td>107</td>
</tr>
<tr>
<td>3</td>
<td>24 M</td>
<td>67.4%</td>
<td>0.0075</td>
<td>89</td>
</tr>
<tr>
<td>4</td>
<td>42 M</td>
<td>59.1%</td>
<td>0.0055</td>
<td>78.5</td>
</tr>
<tr>
<td>5</td>
<td>56 M</td>
<td>60%</td>
<td>0.004</td>
<td>57</td>
</tr>
</tbody>
</table>

Screen spacing is adapted to the upstream mesh diameter size. Groth and Johansson found an initial turbulence decay region after a screen depending on the mesh size. Within the first 15-25 mesh widths, the turbulence intensities decay rapidly for a single screen. Though, for a combination of screens, they recommended to choose a larger spacing than this initial decay region. Then, the configuration shown in Figure 2.5 was chosen.

Figure 2.5. Settling chamber configuration and screen spacing
2.2.3.2 Pressure Losses and Turbulence Reduction Estimate

Most of the turbulence theories are based on a pressure loss coefficient $K$. This coefficient is defined as the ratio of pressure loss across the screen $\Delta P$ over the mean flow dynamic pressure $q$. DeVahl (1964) found that this pressure loss coefficient was equal to:

$$K = K_0 + \frac{55.2}{R_d} = \frac{\Delta P}{q}$$

With,

$$K_0 = \left(\frac{1 - 0.95\beta}{0.95\beta}\right)^2$$

$$\beta = \frac{Projected\ open\ area}{Total\ area}$$

$R_d = Reynolds\ number\ based\ on\ wire\ diameter$

The turbulence reduction factor $f$ is then calculated from the pressure loss coefficient $K$. This factor $f$ represents the rate of turbulence reduction. DeVahl found some consistent results to determine the turbulence reduction factor based on the pressure loss. Though, he did not observe any trend based on the velocity. His experiment was covering a range of screen Reynolds number based on the diameter from 70 to 300, similar to our case. The average measured values of $f_a$ for axial turbulence reduction and $f_l$ for lateral turbulence reduction were following the equations below in function of the pressure loss coefficient:

$$f_a = \frac{1}{1 + K}$$

$$f_b = \frac{1}{\sqrt{1 + K}}$$
The following table summarizes the different coefficients for each screen:

<table>
<thead>
<tr>
<th>Screen</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\beta)</td>
<td>0.615</td>
<td>0.672</td>
<td>0.674</td>
<td>0.591</td>
<td>0.6</td>
</tr>
<tr>
<td>(R_d)</td>
<td>215</td>
<td>107</td>
<td>89</td>
<td>78.5</td>
<td>57</td>
</tr>
<tr>
<td>(K_0)</td>
<td>0.462</td>
<td>0.286</td>
<td>0.281</td>
<td>0.562</td>
<td>0.522</td>
</tr>
<tr>
<td>(K)</td>
<td>0.718</td>
<td>0.801</td>
<td>0.9</td>
<td>1.26</td>
<td>1.5</td>
</tr>
<tr>
<td>(f_a)</td>
<td>0.582</td>
<td>0.55</td>
<td>0.53</td>
<td>0.44</td>
<td>0.4</td>
</tr>
<tr>
<td>(f_l)</td>
<td>0.76</td>
<td>0.745</td>
<td>0.725</td>
<td>0.665</td>
<td>0.63</td>
</tr>
</tbody>
</table>

The total turbulence reduction factor is the product of each individual screen turbulence reduction factor when multiple screens are used. The total pressure loss is the sum of each individual screen pressure loss, when their spacing between each other is greater than their initial turbulence decay region length. As shown by Groth and Johansson, this region length is less than 30 mesh widths and in our settling chamber, the spacing between 2 screens is always greater than 130 mesh widths. Therefore, we obtain the total axial and lateral turbulence reduction factors \(F_a\) and \(F_l\) and the total pressure loss \(K_t\) due to screens only:

\[
F_a = \prod f_a = 0.029
\]

\[
F_l = \prod f_l = 0.17
\]

\[
K_t = \sum K = 5.17
\]

We can notice that the screens mostly reduce the axial turbulence, reducing it by more than 97%. As for the lateral turbulence, the screens suppress only 83% of the turbulence. That is why we add a honeycomb upstream of the screens to improve the lateral turbulence factor.

According to Loerhke and Nagib (1976), a \(\frac{1}{4}\) inch cell honeycomb at similar velocities as the operating one in our settling chamber has the same axial reduction factor \(f_{a,HC}\) as a 20 mesh
screen but is equivalent to three 20 mesh screens in terms of lateral turbulence reduction, \( f_{l,HC} \). As for its pressure loss coefficient \( K_{HC} \), it is similar to the pressure loss coefficient for a 20 mesh screen. It yields the following coefficients for the honeycomb:

\[
\begin{align*}
    f_{a-HC} &= f_{a-20M} = 0.55 \\
    f_{l-HC} &= (f_{l-20M})^3 = (0.745)^3 = 0.413 \\
    K_{HC} &= K_{20M} = 0.80
\end{align*}
\]

The total axial and lateral turbulence reduction factors \( F_{a-SC} \) and \( F_{l-SC} \), as well as the total pressure loss \( K_{t-SC} \) for the whole settling chamber can now be computed:

\[
\begin{align*}
    F_{a-SC} &= \prod f_a = 0.016 \\
    F_{l-SC} &= \prod f_l = 0.07 \\
    K_{t-SC} &= \sum K = 5.97
\end{align*}
\]

It can be concluded that the settling chamber will reduce the axial turbulence by 98.4% and the lateral turbulence by 93%. The total pressure loss coefficient is 5.97, which means when operating at a nominal velocity of 8.1 m/s (26.6 fps), with a dynamic pressure \( q \) of 0.16 inH\(_2\)O, the settling chamber will introduce a pressure loss of 0.95 inH\(_2\)O.

**2.2.3.3 Fabrication**

The screens are tightened and mounted on aluminum frames. A uniform and good tension is ensured by the means of a \( \frac{1}{4} \) inch aluminum rod located in a channel between opposite panels of the aluminum frame. The screen is tightened when the two panels are clamped together and compress both the rod and the screen into the channel. This ensures a better fastening with a high degree of tension (see Figure 2.6.a).
The five screens and the honeycomb can easily be removed from the settling chamber for cleaning purpose (see Figure 2.6.b). Indeed, screens have a tendency to accumulate dirt and dust and should be cleaned at regular intervals to maintain their pressure drop at a reasonable level. Once the screens and the honeycomb are cleaned, a 16 gage sheet metal plate is used to close the settling chamber. Suitable 1/16 inch thick weather-stripping has been installed to prevent any leaks through the cover.

2.2.4 Contraction Cone

The contraction cone accelerates the flow from the settling chamber to the test section. Its role is to guide the flow smoothly, without degradation of the quality of the flow from a 38 inch square duct to a 19.5 inch x 12 inch rectangle duct, which gives a contraction ratio of 6.16. The axial length of the contraction is 3 feet. Mehta and Bradshaw (1979) pointed out two main advantages of the contraction cone:

- It increases the mean velocity, increasing the Reynolds number in the test section while keeping reasonable pressure losses in the settling chamber where the velocity through the honeycomb and screens are much lower.
• It reduces both lateral and axial turbulence levels to a smaller fraction of the mean velocity.

Two different curves of 2 matched cubic polynomials have been used to generate the 3D contraction profile, one for the lateral contraction and the other for the vertical. Those curves have their first derivatives equal to zero at the entrance and exit of the contraction, giving a good conditioning to the flow. Watmuff (1986) found with both experimental and numerical studies that a matched cubic polynomial curve offered the smallest adverse pressure gradient at the entry and exit of the contraction cone compared to other polynomial curves, and reduces the upstream and downstream influence of the contraction cone. He found the flow at the exit of such a contraction cone to be uniform within +/- 1%. The inflection point of our contraction curve is located at 60% of the axial length. Therefore, the radius of curvature is slightly larger at the wide inlet than at the outlet to avoid a danger of boundary layer separation. The lateral contraction profile is shown Figure 2.8., with the first derivative of the matched cubic polynomial.

Figure 2.7. Matched cubic polynomials lateral curve of the contraction cone
Accelerating the flow through the contraction cone is also beneficial to reduce the turbulence level in the test section. Mehta and Bradshaw stated that a simple analysis due to Prandtl can show that the axial turbulence is reduced by a factor $1/C^2$ while the lateral turbulence level will be increased by $\sqrt{C}$, where $C$ is the contraction ratio.

In our case, the axial turbulence reduction factor of the contraction cone is 0.026, while the lateral turbulence level increases by a factor 2.48. This increase in lateral turbulence is linked to the strong variations in intensity of elementary longitudinal vortex lines through the contraction cone.

As for fabrication concerns, precise curves, point coordinates tables and 3D drawings have been given to the manufacturer to get a good precision in fabrication (Figure 2.9.).
2.2.5 Fan

The facility is powered by a 60 HP axial fan with an output capacity of 408 lb/min (15 kg/s). This fan is a Joy Manufacturing Company (current owner is Howden Buffalo Inc.), AXIVANE series 2000. It has a 38 inch diameter and an output rotational speed of 1770 rpm. The wind tunnel has been dimensioned to be able to run experiments at a Reynolds number of 500,000, based on the axial chord, similar to engine conditions.

The 60 HP wind tunnel fan has been renovated, its 18 blades with the rotor mechanism has been sandblasted and rebalanced for vibration concerns. The electric motor is a Reliance Electric (the current company is Baldor), Duty Master AC, 3 phases, rotating at 1775 rpm. This 60 HP motor has been rewound from 200 V to 460 V so that it can be compatible to the new power drop of 75 A/460 V installed in the LSU laboratory. A simple full-voltage starter is used to start the motor. The flow rate of the fan is controlled by a pitch control device using a pneumatic piston actuator linked to the fan blades (see part 2.2.6).

The fan is located at a constant section area to get straight flow nominal conditions upstream and downstream. It is connected to the surrounded parts of the wind tunnel with flexible cylinder couplings made out of rubber to reduce vibration. In addition, the fan and its stand are mounted on anti-vibration pads. Any metal to metal connection has been suppressed to reduce vibration propagation.

2.2.6 Fan Pitch Control Device

The pitch control device mechanical power is supplied by a Fisher® 480 Series yokeless piston actuator (Fisher® owned by Emerson Company). This actuator requires a pressure loading from a double-acting positioned (the 3570 Series). It can operate at a wide range of supply
pressure from 35 psi to 150 psi. The needed supply pressure to control the pitch of the fan blades is 60 psi, which is equivalent to a thrust force of 1600 lbs according to the manufacturer specification for our actuator size 40. The supply pressure connection is a 1/4 NPT pipe. The input pressure signal range is from 3 psi to 20 psi. Its connection is a 1/8 NPT pressure pipe.

Two pressure regulators are used, the first one is to control the supply pressure and lower it from 290 psi (maximum nominal working pressure coming from the main compressor) to about 60-80 psi and keep a steady pressure supply. The second pressure regulator enables us to control the input pressure signal from 0 to 20 psi.

![Diagram of the pressure control system](image)

**Figure 2.9. Pitch Control Supply Pressure System**

The principle of operation of the actuator is based on a pressure unbalance that moves the piston. The translational motion is then transmitted to the pitch control mechanism that transforms the translational motion into a rotational motion through independent crank-connecting rod systems for each blade and change their pitches simultaneously.
2.2.7 Windows and Safety Screen

A safety screen is located upstream of the fan between the parts “div2a” and “div2b” (see Figure 2.3) to protect the fan from potential objects blown out of the test section. Three transparent ¼ inch thick acrylic windows are located upstream of the safety screen, between the corners “Turn 1” and “Turn 2” and downstream of the last one. It allows the user to visualize unexpected objects blocked in the turning vanes or safety screen and get an access to remove them if necessary.

2.2.8 Breather

A “breather”, slot on the top and bottom wall of the duct, is located after the test section at the beginning of the first diffuser, in order to increase the test section static pressure close to atmospheric pressure and reduce leaks into the test section. This opening also prevents unwanted pressure oscillations in the wind tunnel, as well as an internal pressure increase as the air heats up during a run. A 24 mesh metal screen around the perimeter enables to dampen the noise and
secures the opening. The breather opening area is about 25% of the cross section area of the test section outlet. This opening ratio can easily be reduced by adding some spacers over the breather.

### 2.2.9 Supporting Structure

A steel supporting structure has been designed to support all different parts of the wind tunnel. This one being a vertical closed-loop type, loads have been carefully estimated before designing a proper supporting structure. Table 2.3 presents an estimate of the weights of the different parts composing the wind tunnel:

**Table 2.3. Estimate of wind tunnel part weights**

<table>
<thead>
<tr>
<th>PARTS</th>
<th>WEIGHTS (in lb)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Upper parts</strong></td>
<td></td>
</tr>
<tr>
<td>Turn 1</td>
<td>66</td>
</tr>
<tr>
<td>Divergent 2a</td>
<td>242</td>
</tr>
<tr>
<td>Divergent 2b</td>
<td>264</td>
</tr>
<tr>
<td>Pre Fan</td>
<td>99</td>
</tr>
<tr>
<td>Fan</td>
<td>2200</td>
</tr>
<tr>
<td>Divergent 3</td>
<td>220</td>
</tr>
<tr>
<td>Turn N2</td>
<td>264</td>
</tr>
<tr>
<td>TOTAL UP</td>
<td>3355</td>
</tr>
<tr>
<td><strong>Bottom parts</strong></td>
<td></td>
</tr>
<tr>
<td>Turn N3</td>
<td>264</td>
</tr>
<tr>
<td>Main duct 1</td>
<td>308</td>
</tr>
<tr>
<td>Main duct 2</td>
<td>308</td>
</tr>
<tr>
<td>Main duct 3</td>
<td>264</td>
</tr>
<tr>
<td>Settling duct</td>
<td>220</td>
</tr>
<tr>
<td>Contraction cone</td>
<td>154</td>
</tr>
<tr>
<td>TOTAL ALL</td>
<td>5027</td>
</tr>
</tbody>
</table>

The method of “Load and Resistance Factor Design” from the American Institute of Steel Construction (AISC) manual, 2nd edition (1994), has been used to select the appropriate beams and bracings.
2.2.9.1 Top supporting structure

The top structure has been designed considering its flexural strength. Two main I-beams are resting on cantilever beams which are welded to existing pillars in the room. Part legs are bolted to the I-beams and anti-vibration pads have been placed to reduce vibration. Bending moments have been computed to make sure it was not exceeding its critical value according to the method of Load & Factor Resistance Design manual (LFRD) specification of the AISC (part 4, Beam and Girder Design). Maximum beam deflections have also been verified. Beams have been chosen in function of their nominal flexural strength. Cantilever beams are W4x13 shape. Cross beams and cantilever bracings are S3x5.7 shape (see appendix A for detailed structures).

2.2.9.2 Fan Support

The fan support is made out of two different structures to be able to lift it and put it in place with a forklift as shown on the Figure 2.12 (see appendix B for detailed drawings). The main columns, axially loaded, have been sized to withstand the weight of the fan (more 2,000 lbs) according to the LFRD manual of the AISC (part 3, Column Design, Column Load Tables, p 3-30).

Figure 2.11. Images of the fan installation with supporting structure
2.2.9.3 Bottom Supporting Structure

Each bottom part is supported with a different stands made out of 2 inch angle beams. Those stands were also fabricated in the LSU Mechanical Engineering machine shop. Their height is adaptable thanks to leveling feet. Rubber pads are used to reduce vibration and avoid any metal to metal contact.

2.2.9.4 Test Section Support

A particular attention has been given to the test section support to leave as much free space as possible for future measurement apparatuses. An overhead support has been designed for that purpose as shown in Figure 2.13. The total weight of all test section parts is estimated to be 400 lbs. The different supporting elements have been designed to withstand this load. A detailed description of all test section supporting structures can be found in Appendix B.

![Figure 2.12. Test section support image. a) Overhead support b) Global view](image)

A simple calculation example is shown below to find the estimated deflection of the overhead support. A cantilever beam model with a uniform load per unit length $w$ is used.
The distributed load on each cantilever beam is estimated at \( w = 10 \text{ lbs/in} \) over a length \( l = 24 \text{ inches} \). Beams are 2 inch square tubes with a \( \frac{1}{4} \) inch wall thickness. The Young’s modulus for hot-rolled steel is \( E = 30 \times 10^6 \text{ psi} \). The moment of inertia of this square tube is \( I = 0.745 \text{ in}^4 \). Then, the estimated maximum deflection is \( \delta_{\text{max}} \) such that:

\[
\delta_{\text{max}} = \frac{wl^4}{8EI}
\]

Numerically, we find a reasonable value of \( \delta_{\text{max}} = 0.017 \text{ inches} \).

### 2.3 Pressure Loss Estimation in the Wind Tunnel

In this closed-loop wind tunnel, the axial fan produces a rise in static pressure, which compensate for the total pressure losses in the rest of the tunnel. To estimate pressure losses through the different parts of the wind tunnel, the pressure loss coefficient \( K \) is used. This coefficient is defined as the ratio of pressure drop in static pressure \( \Delta p \) over the mean flow dynamic pressure \( q \).

Wattendorf (1938) developed a logical approach to calculate the different pressure losses in a closed circuit wind tunnel. This approach was to divide the wind tunnel into four different kinds of parts: (1) straight, constant area sections, (2) corners, (3) divergent sections, (4)
contracting sections. Then, he referred all local losses to the “jet” dynamic pressure $q_0$, greatest dynamic pressure in the wind tunnel. In our wind tunnel, this dynamic pressure $q_0$ occurs at the exit of the cascade. Then the coefficient of loss becomes:

$$K_0 = \frac{\Delta p}{q_0} = K \frac{q}{q_0} = K \frac{D_0^4}{D^4}$$

Where: $D_0 = \text{jet equivalent diameter}$, $D = \text{local tunnel equivalent diameter}$

Then, the wind tunnel energy ratio $ER_t$ can be defined as:

$$ER_t = \frac{\text{Jet energy}}{\sum \text{Circuit losses}} = \frac{1}{\sum K_0}$$

Below is an estimation of the loss coefficients $K_0$ for each wind tunnel components, as suggested by Wattendorf:

2.3.1 Straight, Constant Area Sections

$$K_0 = \lambda \left( \frac{L}{D} \right) \left( \frac{D_0^4}{D^4} \right)$$

Where $\lambda = \text{skin friction coefficient}$, $L = \text{section length}$.

The local friction coefficient $\lambda$ is determined in function of two parameters: the local Reynolds number based on the equivalent diameter $Re$, and the relative roughness of the local duct wall $k/D$ (where $k$ is the absolute roughness of the wall material and $D$ is the equivalent diameter). From this two parameters and solving the Colebrooke equation, valid in turbulent flow condition, which is true in our case, we obtain the local friction coefficient.

$$\frac{1}{\lambda^{1/2}} = -2. \cdot \log \left[ \frac{2.51}{Re \cdot \lambda^{1/2} + \frac{k}{3.72D}} \right]$$
A graphical method can be used to solve the Colebrooke equation, called the Moody diagram shown Figure 2.14.

![Moody diagram](http://www.engineeringtoolbox.com)

**Figure 2.14. Moody diagram (from http://www.engineeringtoolbox.com)**

For example, the main part of constant section area duct is made of galvanized sheet metal over a total length of 30 feet with an equivalent diameter of 38 inches. The nominal air velocity is 8.1 m/s (26.6 fps) for a 50 m/s (164 fps) velocity at the cascade inlet, and yields to a Reynolds number based on the equivalent diameter of 580,000. For galvanized sheet metal, the roughness coefficient is \( k = 1.5 \times 10^{-4} \) ft, which gives a relative roughness coefficient \( k/D = 1.39 \times 10^{-4} \). Reporting this values on the Moody diagram, we obtain the friction coefficient for this part of the duct \( \lambda = 0.017 \).
2.3.2 Corners

Raw and Pope (1984) related that for corners of the type presented Part 2.2.2., one third of pressure losses are caused by friction in the guide vanes while rotation losses account for about two-thirds. Then, they suggested a partly empirical relation to estimate pressure losses through this type of corners:

\[
K_0 = \left(0.1 + \frac{4.55}{\log (Re)^{2.58}}\right)\frac{D_0^4}{D^4}
\]

Pressure losses through the cascade turbine blade are more delicate to estimate. In Garmoe’s thesis (2005), from the Air Force Institute of Technology, we can find some experimental data on a cascade of pack-B turbine blades, similar to the high lift low pressure turbine profiles we are studying. He reported an integrated total pressure loss coefficient through the cascade of about 0.10 at a Reynolds number of 100,000. This loss only considers the integrated pressure profile at mid-span of the 2D profile. It does not take fully into consideration all the secondary losses such as endwall effects. Though, our operating Reynolds number is 500,000 - similar to engine conditions- and closer to the optimal operating condition of this blade profile. Therefore, it should reduce the pressure loss coefficient of the cascade. Thus, we will assume as a first guess a total pressure loss coefficient of the cascade \( K = 0.1 \).

2.3.3 Divergent Sections

In divergent section, expansion losses are added to the friction losses. Wattendorf found the combined losses give the following relation where \( K_0 \) is expressed as a function of the divergence equivalent angle \( \alpha \):
\[ K_0 = \left( \frac{\lambda}{8 \tan \left( \frac{\alpha}{2} \right)} + 0.6 \tan \left( \frac{\alpha}{2} \right) \right) \left( 1 - \frac{D_4}{D_2} \right) \frac{D_0^4}{D_1^4} \]

Where \( D_1 \) = smaller equivalent diameter and \( D_2 \) = larger equivalent diameter.

From this relationship, it can be found that the most efficient divergent angle is about 5°.

### 2.3.4 Contraction Cone

In the contraction cone, losses are caused by friction only and:

\[ \Delta p = \int_0^{L_C} \frac{\lambda}{2} \frac{\rho}{\mu} V^2 \frac{dL}{D} \]

Where \( L_C \) = length of contraction cone

### 2.3.5 Settling Chamber: Screens and Honeycomb

Pressure losses in the settling chamber are studied Part 2.2.3.2. The total pressure drop \( K \) was found as \( K = 5.97 \). This yields to a loss coefficient \( K_0 = K \frac{D_0^4}{D_4^4} = 0.0615 \)

### 2.3.6 Summary of Pressure Losses

<table>
<thead>
<tr>
<th>Part</th>
<th>( K_0 )</th>
<th>Total Losses (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test section inlet</td>
<td>0.021</td>
<td>6.86</td>
</tr>
<tr>
<td>Cascade</td>
<td>0.0384</td>
<td>12.55</td>
</tr>
<tr>
<td>Test section outlet</td>
<td>0.0348</td>
<td>11.38</td>
</tr>
<tr>
<td>Divergent 1</td>
<td>0.0886</td>
<td>28.96</td>
</tr>
<tr>
<td>Corner 1</td>
<td>0.0377</td>
<td>12.32</td>
</tr>
<tr>
<td>Divergent 2 &amp; 3</td>
<td>0.0187</td>
<td>6.11</td>
</tr>
<tr>
<td>Main duct</td>
<td>0.0017</td>
<td>0.56</td>
</tr>
<tr>
<td>Corner 2</td>
<td>0.0015</td>
<td>0.49</td>
</tr>
<tr>
<td>Corner 3</td>
<td>0.0015</td>
<td>0.49</td>
</tr>
<tr>
<td>Settling chamber</td>
<td>0.0615</td>
<td>20.10</td>
</tr>
<tr>
<td>Contraction cone</td>
<td>0.0005</td>
<td>0.16</td>
</tr>
<tr>
<td><strong>TOTAL</strong></td>
<td><strong>0.3059</strong></td>
<td><strong>100.00</strong></td>
</tr>
</tbody>
</table>
### 2.4 Wind Tunnel Energy Ratio and Design Analysis

The energy ratio can now be calculated:

\[
ER_t = \frac{\text{Jet energy}}{\sum \text{Circuit losses}} = \frac{1}{\sum K_0} = 3.27
\]

For our nominal velocity conditions, 50 m/s (164 fps) velocity at the cascade inlet and 100 m/s (328 fps) at the cascade outlet, equivalent to a 24 in H\textsubscript{2}O of jet dynamic pressure \(q_0\), the estimated total pressure loss of the wind tunnel is 7.34 in H\textsubscript{2}O. An energy ratio of 3.27 is within the range of most closed circuit wind tunnels (from 3 to 7 according to Raw and Pope). Some changes that could have improved this factor as well as their downsides can be pointed out:

- Extending the length of the part “Divergent 1” to reduce its divergence angle from 19° to 5° would have reduced the divergence losses of this part by 20% and the total losses by 5.7%. Though, such an extension would have extended the height or the length of the wind tunnel by 10 feet and would have, increasing the cost and complexity of fabrication.

- Choose a smaller fan, closer to the test section size to lower the divergent sections length. Though, we already owned this fan before designing the wind tunnel, and investing in a new fan would have been an extra expense.

- Increase the size of the cascade blades to have a larger size closer to the fan size and limit the divergent section length. Yet, increasing the size of the cascade would have greatly increased the fabrication cost of the test section. Blades are made with 3D printers and their fabrication material and process are somewhat expensive, with a price proportional to the volume of material used.

Thus, this design can be seen as a good balance between flow quality, cost, and complexity of fabrication.
Chapter 3: Cascade with Wake Simulator Design

As stated before, the primary objective was to design a test section adapted to a wide range of experimental measurements on turbine blades. The test section consists of a 4-passage linear cascade composed of 3 full blades and 2 shaped wall blades (inner and outer blades). A conveyor setup was designed and fabricated to simulate passing wakes upstream of the cascade. Wakes are generated with thick plates in translation on the conveyor. These moving plates simulate the wake passing of the front row vane in order to study their impact on the cascade test blades and, later on, on pulsed film cooling performance. Aerodynamic and heat transfer experimental tests will be conducted in a relative frame with fixed rotor turbine blades while the nozzle guide vanes are rotating. This facility has been designed to enable easy interchanges of different experimental setups.

Figure 3.1. 3-D view of the test section
3.1 Setup Characteristics

3.1.1 Linear Cascade Characteristics

The 2D blade shape profile of the cascade was provided by the Air Force Research Laboratories (AFRL). It is a High Lift Low Pressure Turbine (LPT) blade ‘L1A’ profile with a 1.34 incompressible Zweifel coefficient. A 6-inch axial chord and 1 foot span were chosen. The span is chosen to be two times the axial chord to make sure we have a nominally 2D flow at the mid span. The solidity of the cascade, ratio of the axial chord to the spacing, is 1. The blade inlet air angle is 35 degrees from axial, and the exit angle is -60 degrees from axial, resulting in a 95 degrees total turning. The linear cascade inlet plane is 1 ft high and 2 ft wide. The nominal inlet velocity is 50 m/s (164 fps) and was chosen as a reference, with the axial chord to determine the Reynolds number. Two inlet bleeds and two tailboards are placed upstream and downstream of the cascade to adjust the flow and obtain periodic cascade performances as specified by Eldrege and Bons (2004).

Figure 3.2. Linear cascade with wake simulator 2-D sketch
3.1.2 Wake Simulator Characteristics

Wakes are generated with thick plates in translation on a conveyor setup while staying parallel to the mainstream airflow. Those plates are 3/8 inch thick and are designed to be in size similar to the cascade test blades. The conveyor with the 17 moving plates will be enclosed in an airtight shell. The design speed is 1 m/s (3.28 fps) but can be adjusted to study its influence. The length of the plates was chosen to be equal to the axial chord of the cascade blade, 6 inches. Their pitch is the same as the pitch of the cascade blades, 6 inches. This configuration will markedly simplify future CFD models. The distance between the trailing edges of the plate and the cascade inlet plane is 2.4 inches, which is 40% of their chord, and similar to engine stator-rotor stage spacing. Trailing and leading edges of the plates are interchangeable to study different configurations. In addition, plates are removable and can be substituted with more realistic airfoil profiles if wanted in the future.

3.1.3 Cascade Inlet Plane Angle

The cascade inlet plane angle was adjusted from 35°, flow inlet angle of the cascade blades to 36° to take into consideration the deviation of the mainstream airflow due to the wake simulator. The tangential velocity component induced by the wake generator can be estimated at 1 m/s (3.28 fps), which is the nominal operating velocity of the moving plates. Considering a 50 m/s (164 fps) nominal freestream velocity, a simple vector decomposition gives a velocity triangle with about 1° angle of deviation of the mainstream flow as shown Figure 3.3. The designation used is the same as turbomachinery theory, the absolute velocity is \( C \), the relative velocity is \( W = 50 \text{ m/s} \) and \( U \) is the rotor blade speed 1 m/s. Then, the total turning angle of the cascade with wake simulator is 96°, which is 1° more than the deviation angle of the test blades.
3.2 Test Section Design

The test section parts are mostly in acrylic to allow visualizations and use non-intrusive measurement techniques such as Particle Image Velocimetry in the future. Aluminum angle bars are used to hold acrylic walls together. They are attached to the acrylic panels flat screws in countersunk holes to keep a flat surface on the inside walls. Two different cascade configurations were designed: with or without the wake simulator. Both configurations can be easily interchanged. Several mechanisms are used in this experimental setup to keep an adjustable cascade such as inlet bleeds or tailboards as summarized on Figure 3.4.
All blades were fabricated out of plastic using a 3-D printer. Blades are fixed to the endwall with bolts inserted into helicoils. Different blade setups for various experiments can be interchanged quickly by removing and replacing the cascade endwall plate with the three inner blades fixed on it as shown Figure 3.5. Some detailed drawings can be found in Appendix C.
3.3 Wake Simulator Design

A conveyor mechanism has been specially designed and fabricated for this purpose. The conveyor belt and standard conveyor components such that spockets, wearstrips or shafts have been purchased from a company specialized in modular plastic conveyor belts, Intralox L.L.C. Main design guidelines and belt selection instructions have been followed based on the Intralox® Conveyor Belting Engineering manual. Besides, the frames, the stand, the cover or plate assemblies have been designed using a CAD software. All drawings have been submitted to the LSU Mechanical and Chemical Engineering machine shops where parts have been fabricated. The conveyor shell has been made separately by an external contractor, specialized in sheet metal work (BMR Metal Works LLC, Watson, LA).

The design of this rotating mechanism with all relevant dimensions and observations of interest for future works is given below in this chapter.

Figure 3.6. CAD view and picture of the wake simulator
3.3.1 Frame

The frame is composed of three main elements. The inside frame which is carrying and
guiding the belt (green part on Figure 3.7) and two outer frames which are structural supports for
the bearings, shafts or motor (blue part on Figure 3.7) and allow us to mount the conveyor on the
stand and adapt its angle and height.

The inside frame is in aluminum to reduce the total weight of the mechanism while the
outer frames are mainly made of 1”1/2 carbon steel angle (1/4 inch wall thickness) to increase
their strength. Welding has been avoided as much as possible not to have any deformation and to
keep a straight frame. Precise drawings of the conveyor frames can be found in Appendix D.

Two take-up bearings holding the idler shaft are used as tensioners for the belt. Their role
is to properly accommodate the increase (or decrease) in the length of the belt while operating.
Control of the belt length is crucial to make sure the belt engages properly on the sprockets.

Figure 3.7. Conveyor frames
3.3.2 Belt

3.3.2.1 Characteristics

The belt, sprockets, carryways and both shafts have been ordered at Intralox L.L.C, corporation specialized in modular plastic conveyor belts. The belt is made out of polypropylene, with a specific gravity of 0.90. The belt is a Series 800 Intralox® flat top type. It is 6 inches wide and consists of 2 inch long plastic modules linked together with polypropylene rods as shown in Figure 3.8. The belt strength is 1000 lb/ft.

![Figure 3.8. Intralox Series 800 modular plastic conveyor belt (from Intralox® Conveyor Belting Engineering Manual).](image)

This belt has been chosen for its strength and its dimension. One of the criteria was to get a module length compatible with the required moving plate pitch of 6 inches. With 2 inch belt modules, a plate assembly can be attached every three modules to obtain a 6 inch pitch.

3.3.2.2 Estimation of Belt Elongation

Several factors can cause a variation of the belt length (Intralox Engineering Manual, 2010):

- Temperature variation: The belt being made of plastic, any significant change in temperature will result in a contraction or elongation of the belt due to thermal elongation.
• Elongation (strain) under load: Any material will elongate if tension is applied. This strain should not exceed 2.5 % of the conveyor length for our conveyor belt.

• Elongation due to break in and wear: New belts usually experience an elongation in the first hours of operation as the belt modules and linking rods “seat” themselves while older belts will experience elongation due to wear.

The first type of elongation is due to thermal expansion. The change in the dimension of the conveyor belt $\Delta L$ can be written as:

$$\Delta L = L1 \times (T2 - T1) \times e$$

Where: $L1 = \text{total belt length (ft.)}$, $T2 = \text{Operating temperature}$, $T1 = \text{initial temperature}$, $e = \text{Coefficient of thermal expansion (in/ft/°F)}$.

For our case, with a polypropylene belt $e = 0.001 \text{ in/ft/°F}$ and $L = 9 \text{ ft}$, an initial room temperature $T1 = 70 \text{ °F}$ and an operating temperature (wind tunnel airflow when not cooled) $T2 = 115 \text{ °F}$, the thermal expansion will be $\Delta L = 0.4 \text{ in}$. The increase in shaft to shaft length would then be about 0.2 inch.

The second type of elongation is due to strain under load and wear. This elongation should not exceed 2.5 % of the conveyor length according to the belt manufacturer. For our belt, the elongation due to strain and wear would then be about 2.5 inches, which gives an increase of about 1.25 inch of the shaft to shaft length.

Therefore, the total maximum shaft to shaft elongation is estimated at 1.45 inch. The tensioning mechanisms with take-up bearings, as well as the conveyor shell have been designed to meet this requirement.
3.3.3 Shafts

1.5 Inch Square stainless steel 303 shafts are used to drive the belt. The motor end of the drive shaft is 1.25 inch diameter bore with a 1/4 inch width keyway for motor transmission. All other shaft ends are 1 inch diameter bore rotating into ball bearings. Precise dimension can be found in Appendix D.

3.3.4 Sprockets

Sprockets are made out of acetal and are used to transmit the torque from the square shaft to the belt. The largest available sprockets have been selected to minimize the centrifugal force on the plate assembly when turning, with a 10.3 inch diameter. Three sprockets are used on each shaft. They are laterally retained with two retainer rings, as shown on Figure 3.9. Some lateral play is also left to allow the sprockets to move laterally and accommodate for possible lateral thermal expansion of the belt.

![Figure 3.9. Conveyor shaft with sprockets image](image)
3.3.5 Wearstrips

Wearstrips are plastic strips that are added to the conveyor frame to carry the belt and reduce sliding friction forces. They are made of Ultra High Molecular Weight Polyethylene (UHMW), plastic material with a very low friction coefficient, good abrasion resistance and self-lubricating properties. Two different kinds of wearstrips are used: carryway wearstrips and side wearstrips that are attached to the frame to guide the belt into a U-channel (see Figure 3.10.).

![Image of the inside frame of the conveyor with wearstrips](image)

**Figure 3.10. Image of the inside frame of the conveyor with wearstrips**

3.3.6 Gearmotor

3.3.6.1 Specifications

The conveyor gearmotor is a combination of an AC electric motor, a gear unit to reduce the rotational speed and a Variable Frequency Drive (VFD) to control the output speed of the motor. This is a SEW-EURODRIVE gearmotor, model # KAF37DRE80M4 / MM11.
The AC electric motor is a DRE80M4 model, 3-phase with a 1.5 HP power. It is rated at 2.4 A, 460 V, 100 Hz for a 2900 RPM output speed. The gear unit is a KAF37 model: it is a helical-bevel gear unit, flange-mounted with a hollow shaft. Its speed reduction ratio is 15.31. Coupled to a MM11 VFD, the output speed range of this gearmotor can vary from 19 to 189 RPM, with a 486 in-lb nominal torque at 189 RPM. This RPM range corresponds to a belt velocity range from 0.28 m/s (0.92 fps) to 2.76 m/s (9.05 fps).

The speed of the motor can be controlled either directly on the VFD box using the keypad or with an external computer with the SEW MOVITOOLS -Motion Studio software under manual operation. The output speed of the motor can also be recorded with this software. The operating output speed range can be adjusted through potentiometers and switches of the VFD controller (called MOVIMOT®). Two main parameters can be set with the available switches: Minimum VFD frequency and acceleration of the drive (also called ramp time). The potentiometer is used to set the maximum frequency. More information can be found in the MOVIMOT® Operating Instructions Manual.

**3.3.6.2 Required Power Estimation**

The required 1.5 HP motor power has been calculated based on the Intralox Engineering Manual design guidelines. This calculation is based on several parameters such as the belt loading, the belt speed, width and weight, the length of the conveyor or its elevation change. This calculation also determines if the chosen belt is suitable for our application. The main lines of this calculation are:

- Belt tension load estimated at 30 lb/ft
• Adjust the belt tension load to specific service conditions. In our case, we have a relatively high speed of the belt. This increases the number rotations per time unit and decreases the belt life. Combined with the fact this is an elevating mechanism with frequent starts under load, a Service Factor of 3 is taken and we adjust the estimated belt tension load to about 100 lb/ft.

• The next step is to calculate the belt strength. In our case, we have a very high ratio belt speed over belt length due to the short length of the mechanism. As a result, the belt strength drops by a factor 5 to reach 200 lb/ft. Though, the adjusted belt strength still exceeds the adjusted belt tension load. Then, the belt is strong enough for our application.

• The last step is to determine the required torque and power to drive the belt. These values vary with the belt tension load. The torque depends on the sprocket diameter while the power depends on the velocity of the belt.

The required torque was estimated to 300 in.lb and the required power to about 0.75 HP. Therefore, a 1.5 HP motor, 486 in-lb drive has been chosen to be able to reach higher speeds if needed.

3.3.7 Stand

The conveyor stand is a steel structure mainly made of 1” 1/2 - 1/4 inch wall thickness square tubes. Some additional bracings are used to reinforce the motor side. The angle and height of the conveyor can be adjusted to the cascade plane at 36° from horizontal with two pairs of clevis rod ends that can be screwed or unscrewed inside the structure. The bottom link between the conveyor and the stand clevis rods is a revolute pair while the upper link is a prismatic pair.
In addition, some vibration damping leveling mounts are used to reduce mechanical vibrations and as a second way to adjust the height and angle of the mechanism.

![Conveyor stand picture. (a) General view (b) Zoom on clevis rods](a) ![Conveyor stand picture. (a) General view (b) Zoom on clevis rods](b)

**Figure 3.11. Conveyor stand picture. (a) General view (b) Zoom on clevis rods**

### 3.3.8 Shell

A shell was designed and fabricated to keep the wind tunnel mainstream flow contained, and to suppress any additional leakage when moving plates are entering or leaving the wind tunnel test section. This airtight shell will keep the whole apparatus in an enclosed area with a constant static pressure. In fact, the pressure difference between the inside of the test section and the room atmospheric pressure can be quite important, and the mainstream flow may be considerably altered when moving blades are entering the test section if not contained in an enclosed space. Therefore, the incoming cross flow will be well conditioned inside the shell before reaching the mainstream test section flow upstream of the cascade (see Figure 3.12).
This shell is made out of 16 gage sheet metal and some appropriate stiffeners have been designed. Detailed drawings can be found in Appendix D. Some soft foam is covering the inside walls of the shell in order to get a better flow quality but also to reduce small vibrations.

![Figure 3.12. Views of the wake simulator mechanism integrated to the test section](image)

### 3.3.9 Plate Assembly

The moving plate assembly was designed to allow easy part replacements or interchanges for future works. Plates can be easily replaced by more realistic airfoil profiles as well as leading and trailing edges that can be changed to study different configurations. Its weight was a critical parameter not to overload the belt, especially when a plate is turning; the centrifugal force added to the plate weight generates high tension load in the 3/8 inch thick nylon belt.

Each plate is fixed to the plastic belt using an intermediate aluminum plate, ensuring a better material strength (see Figure 3.13). A set of three 6-32 screws and two 3/16 inch dowel pins is used to tie the plate to the aluminum plates. Then, the aluminum plate is attached to the
plastic belt thanks to four 10-32 flat screws. Hexagonal press fit inserts are used in the plastic belt to get extra strength and pull-out resistance. Detailed drawings are given in Appendix D.

![Plate assembly image](image.png)

(a) Stand alone (b) Fixed to belt

**Figure 3.13. Plate assembly image (a) Stand alone (b) Fixed to belt**

The plate is 3/8 inch thick and made out of black MDS-filled nylon 6/6, a plastic material with good wear-resistance and easy-machining properties. To reduce the total weight of the assembly, the plate has a hollow structure while keeping a structural skeleton to keep a good strength. This structure was designed to be able to machine the 17 blades with a Computer Numerical Control (CNC) machine. A 1/16 inch thick aluminum plate covers the hollow plate. This aluminum cover is fixed to the plate with 4-40 flat head machine screws to keep a flat and smooth surface.

The trailing and leading edges are made of acrylic and fixed to the plate with 6-32 nylon screws still to optimize the weight of the assembly. The Figure 3.14 shows the principle steps of the assembling. Detailed drawings can be found in Appendix D.
3.3.10 Final Conveyor Assembly

The figure below summarizes the different steps of the conveyor assembling procedure:

![Figure 3.15. Conveyor assembling steps](image-url)

**Figure 3.14. Moving plate assembling process**

1) Four 10-32 inserts in Intralox belt module
2) Intermediate aluminum plate
3) Cover in aluminum over hollow plate
4) Flat plate on aluminum plate
5) Fix leading and trailing edges
6) Final plate assembly

**Figure 3.15. Conveyor assembling steps**
Step 1: Fix first structural frame to the stand. Adjust height and angle and fix the motor and the first take-up bearing to the frame.

Step 2: Place the main shell on the first frame.

Step 3: Attach the inside frame to the frame and tighten them together with the shell in between.

Step 4: Place shafts and sprockets. The key is to be inserted into the drive shaft key way.

Step 5: Attach the second structural frame to the inside frame without any bearings on it. A special aluminum cover plate is to be placed between the two parts.

Step 6: Fix bearings on the second frame and insert the belt.

Step 7 & 8: Attach both of the top covers to close the shell.

Once the conveyor mechanism is assembled, the moving plates can be removed or added without disassembling any other components. As explained before, four inserts are located every three modules to attach these moving plate assemblies as shown on Figure 3.13.b. Nevertheless, it can be noticed that to disassemble the belt modules, it is necessary to cut polypropylene rods that link the modules together. Those rods need to be replaced afterwards. Then, it is preferable to disconnect these modules three by three when necessary.

3.3.11 Vibration Reduction

Some optimizations have been done to reduce vibration levels of the mechanism. The belt height was adjusted so that the belt stays in contact with both the carryways underneath and the U channel on top of it and is tightly guided at all times. Side wearstrips and carryways were chamfered to get a gentle insertion of the belt into the guides. Light foam placed on the inside
walls of the shell also reduce vibration by friction. In addition, the shaft to shaft length of the mechanism needs to be adjusted with the take-up bearings so that the belt modules and the sprockets are fitting together at each rotation.

3.3.12 Acoustic Resonance

The wake simulator mechanism may introduce some acoustic resonance noise and vibration concerns due to the high velocity airflow in the test section. Between each intermediate aluminum plate attached on the conveyor belt, there is a 1/4 inch deep - 1/8 of an inch wide gap. This gap is linking the test section with a high flow rate to the wake simulator cavity inside the belt loop where the airflow is at rest. Although the plastic belt is separating the test section from this cavity, the belt is not completely airtight and these gaps could introduce some flow-induced vibrations and noise due to a Helmholtz resonator type structure. The frequency of this resonance can be determined by a simple, flow velocity independent equation:

\[ f = \frac{c}{2\pi} \sqrt{\frac{S}{VL}} \]

Where \( S \) is the total surface of the gaps, \( L \) is its depth, \( V \) is the volume of the cavity and \( c \) is the speed of sound. In our case: \( S = 4 \times S_{\text{gap}} = 4 \times 1/8 \times 5 = 2.5 \text{ in}^2; \ V \approx 2,300 \text{ in}^3 \) and \( L = 1/4 \text{ in.} \)

Hence, the Helmholtz resonance frequency of the system is estimated to be: \( f \approx 3.5 \text{ Hz} \). The frequency of the moving plate passing event is 6.6 Hz, which is two times the Helmholtz resonance frequency. Then, both frequencies may interact together, amplifying their noise and vibration levels. This induced noise and vibration can become a concern if the amplitude becomes too high. In this case, the addition of flexible rubber flaps covering gaps between each aluminum plates is recommended. Making the aluminum plates wider to reduce the gaps (6
inches wide instead of 5.875 inches previously) can also be a solution. Another solution to modify the Helmholtz resonance frequency is to fill the existing resonating cavity with some material such as foam to change its volume and then, change the resonance frequency.

Another source of vibration may be the formation of a cavity when a moving plate is leaving the test section and moves into the shell. The limit case of this configuration is the presence of a 6 inch deep slanted cavity, with an opening area of 5 by 12 inches. A shear layer is started at the upstream edge of the cavity opening and creates a resonator. This configuration was investigated by U. Ingard (2008) as a side-branch resonator in a duct. In this case, if $U$ is the velocity of the mainstream airflow, a flow perturbation of the shear layer is carried by the shear layer with a speed $U' = \beta U$ and makes a “roundtrip” in the cavity. This “roundtrip” defines a characteristic frequency for the shear layer: $f_U = (\beta U/D)/(1 + \beta M)$, where $M = U/c$ is the Mach number, the coefficient $\beta$ is approximately 0.5 and $D$ is the depth of the cavity. In addition, the frequency of the $n$th mode of the cavity resonator is $f_n = (2n - 1)c/4L'$, where the acoustic length of the resonator is estimated to be $L' = L + 0.43D$ from experimental results. In our case, $U = 50$ m/s (164 fps), $M = 0.147$, $D = 6$ inches yields to $f_U = 155$ Hz and $f_1 = 404$ Hz, $f_2 = 1212$ Hz, etc. The $m$th mode frequency of the flow $mf_U$ may interact with the $n$th acoustic mode of the resonator and generate instabilities when the condition $mf_U = f_n$ is satisfied. For example, in our case, $8f_U = 1240 \approx f_2$. Then, an acoustic mode around 1240 Hz may be amplified. However, these frequencies are far away from the wake passing event frequency of the wake simulator mechanism of 6.6 Hz. There are then little chances of vibration amplification due to interactions with this cavity resonator. Furthermore, U. Ingard (2008) stated that when the cavity is slanted with the same flow direction configuration as in our case, “the flow is striking the blunt downstream edge and no acoustic tone is produced”.
It can also be relevant to investigate the natural frequencies of the wind tunnel structure to evaluate possible interactions between the wake simulator mechanism and the wind tunnel acoustic vibrations. These frequencies depend on the duct geometry and are of the form $f \approx c/(2L)$ where $c$ is the speed of sound and $L$ is a characteristic length of the wind tunnel section. It is known that a wave can propagate in a rectangular duct only if its frequency exceeds the cut-off frequency of the duct which is:

$$f_{m,n} = \frac{c}{2\pi} \sqrt{\left(\frac{mn}{Ly}\right)^2 + \left(\frac{nn}{Lz}\right)^2}$$

Where: $c$ is the speed of sound; $m,n$ are positive integers; $Ly$ and $Lz$ are respectively the height and the width of the duct.

In our case, the lowest mode of this cut-off frequency is $f_{1,0} = f_{0,1} = (c/2L)$ (except for the plane wave which is the fundamental mode for which $m = n = 0$). It corresponds to the wind tunnel largest duct with a 38 inch side square section. Numerically, this frequency is $f_{1,0} = f_{0,1} = 157$ Hz, and is much higher than the frequency of the passing plates of the wake simulator mechanism of 6.6 Hz. Then, acoustic waves originating from the wake simulator will be evanescent and will not propagate in the wind tunnel duct. Conversely, there are few chances to get any interaction between the wind tunnel acoustic frequencies and the wake simulator mechanism.
Chapter 4: Characterization of the Wind Tunnel

Experimental studies in wind tunnels generally aim at measuring various parameters of the airflow around aerodynamic bodies. In general, three main parameters are measured: velocity, pressure and temperature. These data measured over an area and through time can yield to more practical characteristics such as heat transfer or aerodynamic measurements, turbulence level or boundary layer profiles.

In order to perform reliable measurements on different quantities in a wind tunnel, several steps are necessary: 1- Selection of equipment, 2- Experiment planning, 3- System configuration and installation, 4- Calibration, 5- Data acquisition, 6- Data reduction. Finally, data can be analyzed and some corrections can be applied if necessary. All these steps will be detailed in the following chapter.

4.1 Characterization Parameters

Several parameters are to be measured to characterize a wind tunnel:

- The static and total pressure distribution along the closed-loop wind tunnel is to be known, especially in order to know what is the working point of the fan, defined by a specific rise in static pressure for a given flow rate.

- The temperature variation through time in the test section: All the energy supplied to the motor driving a wind tunnel is finally dissipated into heat and results in an increase of temperature of the tunnel air until the heat losses finally reach an equilibrium point with the input of energy.

- The tunnel cross section flow can be described with the following parameters:
  - Mean velocity component: Distribution and variation through time
• Stream wise turbulence intensity distribution
• Total pressure measurements
• Static pressure measurements
• Temperature distribution and variation through time

4.1.1 Turbulence Intensity

This is one of the most important aspects of the flow quality in the test section. Much work during the design phase of a wind tunnel is devoted to reduce the turbulence levels. As developed in Part 2.2, screens and honeycomb in the settling chamber before the contraction cone are carefully designed, but also diverging and contracting ducts must follow some precise rules to avoid separations or recirculation areas in the flow and keep low turbulence levels.

Figure 4.1. Velocity signal through time

Below are useful formulas to estimate the Turbulence Intensity from a velocity signal:

Mean Velocity: \[ U_{mean} = \frac{1}{N} \sum_{i=1}^{N} U_i \]

Standard deviation of velocity: \[ U_{rms} = \left( \frac{1}{N-1} \sum_{i=1}^{N} (U_i - U_{mean})^2 \right)^{0.5} \]

Turbulence intensity: \[ Tu = \frac{U_{rms}}{U_{mean}} \]

If the flow is well conditioned throughout the wind tunnel, turbulence intensity levels as low as 0.1% can be reached. When it is needed, proper mesh frames can be placed right in front of the test section, to simulate the real turbulence level in a gas turbine.
4.1.2 Boundary Layer Profile

The boundary layer plays a major role in the quality of a test section. If the test section of a wind tunnel is too long, the boundary layer will develop too much and lead to span wise detrimental effect on the cascade blades. The boundary layer thickness $\delta_{99\%}$, the displacement thickness $\delta^*$, the momentum thickness $\theta^*$ and the shape factor $H$ can be computed as:

$$U(z = \delta_{99\%}) = 0.99 \times U_\infty$$

$$\delta^* = \int_0^\infty \left(1 - \frac{U}{U_\infty}\right) dz$$

$$\theta^* = \int_0^\infty \frac{U}{U_\infty} \left(1 - \frac{U}{U_\infty}\right) dz$$

$$H = \frac{\delta^*}{\theta^*}$$

4.2 Measurement Techniques

Different instrumentations were utilized to characterize the wind tunnel airflow. Most velocity measurements were done with a Constant Temperature Anemometry (CTA) system (also called Hotwire Anemometry). Static and total pressures at various locations in the wind tunnel were measured with Pitot tubes. Temperature probes were used to get temperature variation measurements and a basic sound level meter was used to measure noise levels next to the wind tunnel test section and fan.

4.2.1 Constant Temperature Anemometry (CTA)

4.2.1.1 Principle

Fundamentally, the CTA makes use of the principle of heat transfer from a heated wire being dependent upon the flow conditions passing over it. An electrical current maintains the wire at a constant temperature and the velocity is determined from the current needed to keep the
hot wire at a constant temperature. To do so, a feedback circuit coupled to a Wheatstone bridge is used as shown Figure 4.2.

![Figure 4.2. Schematic of a Constant Temperature Anemometer](image)

The probe used for our study consists of one thin platinum film sensor of 1mm length by 50 μm width. The heat transfer $q$ due to a normal flow past the film is a function of the flow velocity $U$, the film temperature and the fluid properties as illustrated by the following equation (G. Bidan MSc thesis, 2008):

$$q = (T_{film} - T_{flow}) A_{film} h \cong a + b U^n, with n \cong 0.5 and a, b are constants$$

The higher the flow velocity, the higher the current will be needed to maintain the film at a constant elevated temperature. The voltage, proportional to the current through the Wheatstone bridge is then measured with a simple Ohm’s law.

Constant Temperature Anemometry has various advantages:

- Its ability to measure very rapid changes in velocity thanks to its high frequency response due to the use of very fine sensing elements. Frequency responses can be as low as 10 μs.
- Its high spatial resolution is a clear benefit: Directly related to the size of the sensor of 1mm, it allows us to make measurements very close to a wall for instance.
- Its wide operation and measurement range. Probes used in our experiments are calibrated for velocities from 0 to 50 m/s (164 fps) and for temperatures up to 150°C (302 °F).
However, some drawbacks can be pointed out:

- CTA is very sensitive to flow temperature change. A change of 1°K in temperature of the flow can lead to an error of about 2% in the estimated flow velocity. Correcting factors need to be applied to calibrated values, considering the actual flow temperature.
- This is an intrusive method.
- Its sensitivity to velocity orthogonal components

### 4.2.1.2 CTA System Components

The measuring chain components of a CTA system are:

1. Probe, probe support and cabling, and a thermocouple to get the flow temperature
2. Anemometer with signal conditioner (model TSI IFA 300), containing the Wheatstone bridge which reads the voltage given by the probe.
3. An Analog/Digital converter to transfer the data to the computer. A detailed schematic of the cabling between the different components can be found in G. F. Bidan MSc thesis (2008).
4. Computer equipped with the TSI Thermal Pro software. This software has features to calibrate probes, acquire measurements and get a basic data post-treatment.

![Figure 4.3. Measuring chain components of a CTA system (Jorgensen, 2002)]
4.2.1.3 Probe Selection

Probes are selected on the basis of various parameters such as the medium, number of velocity components measured, velocity range or the wanted resolution. In other words, depending on the parameters measured and on the spatial distribution wanted, we can choose the probe.

For our case, we used a single wire probe, which is a probe with only one sensor and designed to measure only one component of a flow. With this probe we can measure stream wise properties of the flow such as mean velocities and get stream wise turbulence intensities. There are different kinds of single wire probes: straight, boundary layer, bent. A TSI probe type 1211-20 was chosen. This sensor (see Figure 4.4) - platinum film type - is good for cross flow measurements, probe interference being reduced by mounting it parallel to the probe body.

Figure 4.4. TSI 1211-20 standard CTA probe

4.2.1.4 Traverse System

The traverse system used for our measurements is a 2 axis VELMEX traverse with a NF90 controller which controls two stepping motors (one for each axis) with a 0.05 mm positioning precision. It will be used to measure the cross section flow quality. The traverse is controlled through the provided TSI Thermal Pro software.

A probe holder stiffener was fabricated to suppress any vibration during the measurement process. Indeed, velocities as high as 50 m/s (164 fps) induce non-negligible forces on the probe.
In order to characterize the test section inlet cross section flow and the cascade outlet flow, various slots will be located at different positions in one of the test section walls so that the CTA probe can remain in the flow while moving on the traverse system (see part 4.3.2).

![Figure 4.5. CTA experimental setup](image)

### 4.2.2 Pressure Measurements

In order to carry out pressure measurements, various instrumentations are available. First, a classic Pitot-tube with a handheld digital manometer will be utilized (Figure 4.6). The Pitot tube consists of a tube pointing directly into the airflow. Then, the moving fluid is brought to rest at the tip of the probe, we reach a stagnation point and we obtain the total pressure. A pressure transducer transforms the pressure value in an electrical signal and then to the handheld digital manometer uses an A/D converter to display the pressure value. The handheld manometer is a HHP4252 model from Omega® with a range from 0 to 55 inches H₂O and an accuracy of +/- 0.3% FS, which is 0.165 inch H₂O.
Another common technique is the use of pressure taps over the body surface to characterize a pressure field. This technique will be used in the future to characterize the linear cascade pressure distribution. The center blade and the two surrounding blades will be instrumented with pressure taps. The center blade will have pressure taps on both suction and pressure side while the two surrounding ones will have only taps on one of their pressure/suction side. On the center blade, pressure taps will be located at half span location as well as at other span wise location (with a lesser concentration) to evaluate the span wise 2-D assumption.

### 4.2.3 Temperature Measurements

An ungrounded T-type thermocouple, with a stainless steel 1/8 inch diameter, 6 inch long tube was selected. It is located in the settling chamber where flow velocities are relatively low the signal received and is directly linked to the TSI CTA system. In addition a handheld J input thermocouple thermometer, model HH802W from Omega® was used to measure the flow temperature at different wind tunnel locations. Its accuracy is +/- 0.05% rdg +0.3 °C (0.54 °F), which gives 0.32 °C (0.58 °F) at 40 °C (104 °F) and its temperature range is from - 50 °C (-58 °F) to 1370 °C (2498 °F).
4.3 Experimental Procedures

4.3.1 CTA Probe Calibration

Calibration plays an important role in the accuracy and quality of the measurements. CTA calibrations are performed in a dedicated calibrator TSI model 1129 with a low turbulence free jet, whose velocity is calculated on basis of the pressure drop at its exit. The calibration process consists in recording the voltage output corresponding to an effective velocity input. The accuracy of this method is about 0.5%. The objective is quite simple: Find which velocity corresponds to an output voltage value. To do so, in addition to the CTA setup, a pressure transducer and a calibrator are necessary. A bridge voltage output value is acquired for each different velocity for 17 points from 0 to 50 m/s (164 fps). Then, we compute the fitting 4th order polynomial coefficients. The result of a calibration is the following parameters: the gain of the probe, the offset, the temperature of the calibration and the polynomial regression coefficients. The obtained points with the fitting curve are illustrated Figure 4.7.

![Calibration Curve Fit](image)

**Figure 4.7. Calibration fitting curve**
4.3.2 CTA System Location

The test section is designed so that CTA measurements can be done easily. The probe should be able to cover any plane we are interested in with the traverse system. Therefore, one of the side walls of the test section is made out of wood and removable to enable easy machining and interchange.

The traverse can be oriented at two different axis angles,

- First axis orthogonal to the incoming airflow (θ=0°) upstream of the cascade to characterize the inlet flow quality (see figure 4.8). The slot S0 is located 18 inches downstream of the contraction cone outlet.
- Second axis parallel to the cascade inlet plane (θ=36°). The slot S1 is located 50% of the axial chord downstream of the cascade outlet plane to characterize the cascade blade wake.

![Figure 4.8. Slot locations for the traverse system](image-url)
4.3.3 Data Acquisition and Reduction

Data acquisition was automated and executed using the TSI Thermal Pro Software. The sampling frequency and acquisition time for CTA measurements needs to be chosen to be relevant. The integral time scale was computed using the autocorrelation method over a first exploratory survey of velocity data with a large number of points. The necessary sampling frequency to get uncorrelated data was found to be 20 Hz or less in the middle of the test section inlet plane. Then, the acquisition time was picked to get a reasonable amount of points and therefore, a reasonable uncertainty. Generally, an acquisition time of 2 seconds, which gives 40 data points, was taken.

Experiments generally generate a high quantity of raw data, from voltage files recorded into text files to velocity files and post-processed data. This raw data need to be processed and data reduction was mostly done with the Mathwork® software Matlab, and programs such as Tecplot360 were also utilized to create and analyze plots.

4.3.4 Uncertainty Analysis

For each measurement -velocity, pressure, temperature or turbulence intensity- an appropriate uncertainty estimation method is to be used. Generally, a 95% confidence interval, or “two-sigma error” (1.96σ) was chosen in this study to estimate the uncertainty. The statistical method depends on the analyzed parameter and on the number of data points. For raw data with a number of points less than 30, the student’s t-distribution law is applied. If the number of points is greater than 30, a normal distribution is appropriate. For data based on the standard deviation such as the turbulence intensity, a $\chi^2$-distribution law is applied (Beckwith, 1993).
4.3.5 Data Analysis

In this part, some potential reasons for abnormal pressure, temperature variation or high turbulence levels in a wind-tunnel will be discussed. The most important factors concern:

- Mean flow variations in time and space over the test section cross section area:
  
  A mean flow variation in time would indicate problems with the fan or with static pressure changes. This kind of problems can be eliminated by changing fan blade angles or by increasing the flow through the breathe slit downstream the test section.

- The turbulence intensities fluctuation:
  
  Mean flow non-uniformities can originate from separation on some of the corner guide-vanes, diffuser separation or by flow blockage due to inefficient screens or honeycomb. If these deficiencies are small the problems can be taken care of by the contraction but larger deficiencies and problems generated in the contraction are difficult to eliminate. High turbulence levels often originate from small separations or deficient screen and honeycomb design.

- Temperature variations in time and over the same cross section area:
  
  Large temperature variations in time suggest an insufficient control system and large variations in space that the heat exchanger is located too close to the test section or that the water flow rate is not large enough (Lindgren, 2002).

4.4 Results and Discussion

4.4.1 Pressure Distribution along the Wind Tunnel

Static and dynamic pressures were measured with a Pitot tube linked to a handheld digital manometer at different locations along the flow path in the closed circuit wind tunnel. Figure 4.9
represents the variation of static, dynamic and total pressure along the wind tunnel for a mean velocity in the test section inlet of 53 m/s (174 fps). The graph starts at the test section inlet (x=0 foot) and ends at the contraction cone outlet (x= 70 feet) to close the loop. Uncertainty bars, based on the student’s t-distribution law (see Part 4.3.4) are represented for each measurement.

As expected, a permanent total pressure decrease along the wind tunnel occurs due to pressure losses. Static and dynamic pressures vary symmetrically, in function of the section areas. As predicted Part 2.3, most pressure losses occur around the cascade, where the dynamic pressure is the highest. Static pressure rises across the fan by 7 inches H₂O at this particular wind tunnel working point with a calculated flow rate of around 17,000 CFM.
4.4.2. Fan Pitch Setting and Operating Conditions

The Figure 4.10 shows the variation of volume flow rate through the fan versus the pitch setting. As explained Part 2.2.6, the pitch of the fan blades can be changed through a pitch control device. The input pressure signal from 0 to 15 psi can be changed using a pressure regulator (see Figure 2.9). Then, we can control the operating conditions of the wind tunnel and the velocity in the test section. A flow rate of 15,900 CFM and a velocity of 50 m/s in the test section (164 fps) are reached for a pitch setting of 8.1. It can as well be observed the fan starts stalling when the pitch is greater than 9.5.

![Figure 4.10. Fan operating conditions. a) Volume flow rate versus pitch setting. b) Total pressure rise versus pitch setting](image)

4.4.3 Noise Measurements

Noise measurements were performed along the wind tunnel using a simple sound meter. It was found that the sound level was the highest next to the cascade where velocities can reach
values as high as 80 m/s (262.4 fps). The noise level next to the test section at a nominal cascade inlet velocity of 50 m/s (164 fps) is 94.5 db with a +/- 0.8 db uncertainty using a t-student law distribution and taking a 95% confidence interval. For the same operating point, the noise level measured from the ground next to the fan is 91 db. At the opposite end from the cascade, when velocities are lower (8 m/s or 164 fps), the noise level was found to be 89.5 db. Therefore, when working next to the test section during a long time, it is recommended to wear ear protectors.

4.4.4 Temperature Variation through Time

Temperature was measured in the settling chamber where flow velocities are relatively low so that we obtain directly the static temperature. A K-type thermocouple was used (see Part 4.2.3 for details). All the energy supplied to the motor driving a wind tunnel is finally dissipated into heat and results in an increase of temperature of the tunnel air until the heat losses finally reach an equilibrium point with the input of energy. The temperature rise in the wind tunnel is described by the curve Figure 4.11.

![Figure 4.11. Temperature variation through time at the test section inlet](image)
This measurement was taken for an operating velocity at the test section inlet of 50 m/s (164 fps). We can see that the temperature reaches a first plateau at 44.1 °C (111.4 °F) from t = 65 min to t = 100 min. Then the temperature varies again to reach a second plateau at 45 °C (113 °F) after t=123 min. This variation from 44 to 45 °C is due to temperature change in the room. Though, an AC ventilation system keeps the room within a temperature range of 2° C (3.6 °F), and therefore, temperature variations stay within a reasonable range after the temperature reaches a first plateau.

4.4.5 Velocity Distribution over the Test Section Inlet Cross-Section

To measure the velocity distribution over the test section inlet cross-section, a CTA system with an automated traverse were employed as explained Part 4.3. Measurement point locations are represented Figure 4.12. The plane is located at the slot (S0) as shown Figure 4.8. Its width is L = 12 inches and its height is H = 19.5 inches. The axis values are adimensional.

![Figure 4.12. CTA measurement points location over the test section inlet plane ‘S0’](image-url)
After acquiring mean velocity values for each measurement points at a 20 Hz sampling frequency over 2 seconds, the velocity contour distribution illustrated Figure 4.13 was obtained.

![Velocity contour distribution](image)

**Figure 4.13. Velocity contours over the test section inlet ‘S0’ plane (in m/s)**

Mean velocity contour values, outside the boundary layer, are varying from 50 m/s (164 fps) to 50.8 m/s (167 fps). The velocity variation over the section is less than 2%. The uncertainty was computed using the Normal-distribution law and was found to be +/- 0.02 m/s (+/- 0.06 fps) with a “95% confidence interval”.

A refined measurement was done over a corner of the same plane ‘S0’ to get more information about the flow near from the walls and in the corners. The measurement points were distributed as shown Figure 4.14, with a finer mesh near the walls.
Results are shown Figure 4.15. The estimated uncertainty based on the Student’s t-distribution law is +/- 0.06 m/s (0.20 fps) in the outer boundary region flow, and +/- 1.5 m/s (4.92 fps) in the corner flow for a 95% confidence interval. A first estimate of the boundary layer thickness of about 0.05L was obtained.

Figure 4.14. CTA measurement points location over the test section inlet plane ‘S0’ bottom right corner

Figure 4.15. Velocity contours over the plane ‘S0’ bottom right corner (in m/s)
4.4.6 Streamwise Turbulence Intensity over the Test Section Inlet Cross-Section

Turbulence intensity was computed with the data acquired for the mean velocity values in the previous part. The measurement point locations are therefore the same. The result is also represented with contours on Figure 4.16.

![Streamwise turbulence intensity contours over the cross-section ‘S0’ in %](image)

**Figure 4.16. Streamwise turbulence intensity contours over the cross-section ‘S0’ in %**

The turbulence intensity reaches levels as low as 0.12% in the center of the cross-section, with an uncertainty for a 95% confidence interval of +/-0.03%, based on a $\chi^2$-distribution law. In the corners, the streamwise turbulence intensity reaches higher levels, up to 7 %, as illustrated Figure 4.17. This result is based on the refined measuring mesh shown Figure 4.14.
4.4.7 Boundary Layer Profile

The boundary layer profile was characterized using the CTA system and the automated traverse. The mainstream flow velocity was 50 m/s (164 fps). Velocities were measured every millimeter starting from as close to the wall as possible, which is 1/8 inch from the wall (see Figure 4.18). The resulting profile curve was extrapolated to reach a zero velocity at the wall in order to characterize the boundary layer profile and compute the different parameters defining it.
This profile yields to the following values:

- Boundary layer thickness at 99%: \( \delta_{99\%} = 0.14L \approx 1.68 \text{ inch} \)
- Displacement thickness: \( \delta^* = 0.0064L \approx 0.077 \text{ inch} \)
- Momentum thickness: \( \theta^* = 0.0048L \approx 0.058 \text{ inch} \)
- Shape factor: \( H \approx 1.33 \)

The shape factor is close to the one found with the classic 1/7 Power Law used for turbulent boundary layer profile over a flat plate (\( H=1.28 \)). The velocity overlap is characteristic of the flow distribution at the outlet of a finite length contraction. This concave velocity distribution was observed by Morel (1975) in a paper about the design of axisymmetric wind tunnel contractions. This velocity overlap of 1.2% may also be found for turbulent boundary layers. This characteristic of turbulent boundary layer has been discussed over the last two decades in order to redefine the classical boundary layer theory, and find a new analytical description of the turbulent boundary layer. Osterlund (2000) claimed that this overlap region becomes significant when the Reynolds number based on the momentum thickness is greater than 6,000. In our case, the Reynolds number based on the momentum thickness is 4,334.

### 4.4.8 Cascade Blade Wake

A first characterization of the cascade wake profile was performed using the Constant Temperature Anemometry system. The traverse was inclined at 36° to be aligned with the cascade outlet plane as illustrated Figure 4.19. Then the probe is moving along the axis parallel to the slot S1 as illustrated Figure 4.8.
The mean velocity values were measured at mid-span, 50% of the axial chord downstream of the cascade outlet plane at the slot S1 (refer Figure 4.8). The inlet mean velocity was 50 m/s (164 fps), which gives a Reynolds number based on the blade axial chord of 450,000. The wake profile of the center blade of the cascade is shown Figure 4.20. The pressure side is on the right side of the graph and the suction side is on the left.
The turbulence intensity profile was then computed and is illustrated Figure 4.21.

Figure 4.21. Turbulence intensity profile 50% of the axial chord downstream of the L1A cascade center blade at mid-span at Re=450,000

It can be noticed the minimum velocity is 85% of the freestream. It was found the freestream velocity was about 77 m/s (253 fps) for a 50 m/s (164 fps) inlet velocity. In addition, it can be observed this wake is not symmetrical: the wake velocity is lower on the pressure side than on the suction side. This characteristic was also obtained in Garmoe’s experimental study on similar low pressure turbine blades in a cascade (2005) at a Reynolds number of 100,000. It can also be observed that the turbulence intensity is higher on the pressure side than on the suction side near from the center of the wake and reaches a local minimum at the center of the wake. The turbulence intensity reaches levels as low as 0.15% in the outer region of the wake. It can be observed that the velocity profile is not periodic in the cascade with the current configuration. A 2.5% velocity difference between the pitchwise locations +0.5 and -0.5 can be
observed in Figure 4.20. The cascade flow can be adjusted and periodic flow conditions should be obtained by adjusting movable elements such as the tailboards or the inlet bleeds as illustrated Figure 3.2. In addition, a better attention should be given when installing the blades in the cascade in future studies to make sure they are evenly spaced over the whole span.
Chapter 5: Conclusion

This new experimental facility was designed to investigate how the unsteady flow due to the upstream passing wakes coming from the front row vane affects the film cooling performances on turbine blades. The facility can reach velocities as high as 50 m/s (164 fps) as expected, with turbulence intensity values as low as 0.12 %. The velocity variation over the cascade inlet section is less than 2 % of the mean value.

The test section, made out of acrylic walls, will enable the users to carry out Particle Image Velocimetry (PIV) measurements in the future. Blades with pressure taps on their surface are currently being fabricated to obtain information about test blade surface pressure distribution and later on, about the cascade flow periodicity. The airflow in the cascade can be adjusted for periodicity by the use of the tailboards and inlet bleeds in order to “balance” the flow.

Although the experimental studies about film cooling are not part of this thesis, a wake simulator mechanism to simulate passing-wakes upstream of a 4 passage linear cascade was designed. A special attention in the design of the components has been given to make them easily interchangeable. The cascade test blades can be easily replaced. Wake generating blades can be quickly changed to study different wake properties. In addition, trailing and leading edge of the designed flat plates can be replaced with new profiles. This experimental setup is the first wake simulator mechanism upstream of a linear cascade which can use realistic vane profiles.

Future works will also investigate heat-transfer performance of film cooling. In parallel with this work, a student team was preparing the experimental apparatus to study transient heat transfer behavior of film cooling. Furthermore, a heat exchanger has been recently designed to be able to control to wind tunnel flow temperature and facilitate future studies.
Eventually, a list of technical recommendations for complementary work with the new facility in a close future is suggested below:

- Heat exchanger installation: A heat exchanger is currently being installed in the wind tunnel duct, introducing new pressure losses. A new survey of the wind tunnel characteristics should be performed after installation, in order to know the new operating velocity range and the influence of the heat exchanger coil on turbulence intensities values. In addition, a special attention should be given to possible humidity and water drops introduced by the heat exchanger coil in the wind tunnel duct.

- Wind tunnel sheet metal parts: the wind tunnel 16 gage sheet metal walls are not stiff enough for large surfaces and introduce extra vibration when the wind tunnel is operating, due to substantial pressure forces on those walls. A quote has been processed with a local contractor to add stiffeners on the larger sheet metal panels. This stiffening work should be processed after installing the heat exchanger.

- Cascade operation: the objective of the cascade is to obtain a periodic airflow. Then a particular care should be given to the fabrication and installation of the 2-D blades so that they are precisely located, straight (2-D assumption) and evenly spaced. The use of blades with pressure taps should facilitate the characterization of the cascade. The adjustment of the movable elements such as the “inlet bleeds” and the “tailboards” to obtain a periodic airflow. The influence of such elements should be investigated.

- Wake simulator mechanism operation: the wake simulator was designed to be compatible with the test section and the cascade. Though, the addition of this new element to the test section apparatus may introduce some additional leaks. A special care to keep the whole mechanism airtight should be given. The use of caulk for permanent links is well adapted.
The use of aluminum tape can be a convenient means in some cases. As described in Part 3.3, the wake simulator height and angle can be adjusted in order to adjust the whole mechanism to the cascade. A preliminary vibration analysis was performed in this thesis but some tests need to be performed in situ to make sure that vibration due acoustic resonance (Helmholtz resonator) or due to the rotating parts will not disturb future cascade measurements. Excessive vibrations of the moving plates may also become a concern. In this case, lowering the operating speed of the belt can be a good solution. Increasing the width of the moving plate and make them stiffer could also allow the user to operate the mechanism at a higher velocity without having vibration concerns if needed. For additional recommendations about the assembly of the mechanism, the reader can refer to the Part 3.3 of this thesis.
References


Bidan G. F. (2008), Parametric Study of a Pulsed Jet in Cross Flow, Louisiana State University, MSc thesis


TSI Incorporated (2008), *TSI Thermal Anemometry Catalog*


86

Appendix A: Wind Tunnel Parts - Detailed Drawings

1) Turn 1

2) Divergent 2a
Rectangle to square

- Acrylic window (transparent) to have access to a safety mesh between parts div 2a and div 2b

- 2 legs to support the part (1/8 in thick sheet metal)

Window #1
Material: Acrylic

3/16 thick, 2" wide sheet metal (steel) used as a frame
Window edges dimensions (slats):
• 1” 1/2 width
• 1/2 inch thick

Bases to fix the part on the supporting structure: 3” width

3) Divergent 2b
Square to circle

Outlet diameter 38”

Bases to fix the part on the supporting structure: 3” width
5) Divergent 3
Circle to rectangle

6/8) Turn 2 and Turn 3 (same part)
7) Lateral Duct

Cross section

Window edges dimensions (slats):
• 1” 1/2 width
• 1/2 inch thick

3’2”

3’ = Length

Side view

❖ Acrylic window to have access to the corner vanes.

❖ Need thicker edges around the opening in order to fix the acrylic window.

9) Main Duct part 1

❖ Acrylic window to have access to the corner vanes.

❖ Need thicker edges around the opening in order to fix the acrylic window.
12) Contraction cone

Importance to respect the given curves as best you can to have a very regular and quality flow at the outlet.

The following documents will be given:
- **3D** design compatible with Autocad (format .igs or .stp)
- Precise curves with plots and points coordinates

2 different curves (curve 1 and curve 2): one for the vertical side, the other one for the horizontal side
13a) Settling Duct

Particular design due to a removable frame (with screens to improve the airflow) we will place into it. Dimensions are larger by 1” 1/2 on every sides (with respect to the preceding duct).

Settling duct

Top view

Front view
Appendix B: Supporting Structure - Detailed Drawings and Specifications

I) Main support: Upper parts

- Based on 3 existing large pillars
- 3 cantilevers attached on them
**Pillar 1:**

Existing large pillar 7" side square section

Cantilever beam **W4x13** shape

Transversal beam **S3x5.7** shape, 3'8" length

10'

9'9"

12'

3'10"

Underside view

2 cross beams **S3x5.7** shape

4'4"

4'1"

1'11"

**Pillar 2 and 3: (1)**

Existing large pillar 7" side square section

Cantilever beam **W4x13** shape

Transversal beam **S3x5.7** shape, 3'6" length

12'

9'9"

10'

Isometric view
Pillar 2 and 3: (2)

Front view

Top view

2 cross beams S3x5.7 shape

II) Fan support

In 2 separate parts to lift the fan up with a forklift.
Fan support: part 1

- Used to lift the fan up with the forklift
- Fixed on the 2nd part

![Diagram of fan support part 1]

3' 5” on one side
4' on the other side

2'5”
7”

Horizontal beams:
\textbf{S4x9.5} shape

Column: \textbf{W4x13} shape

Bracing: \textbf{L, T} or \textbf{I beam} < 3x3”

Fan support: part 2

![Diagram of fan support part 2]

6'9”

Columns: \textbf{W4x13} shape

Bracing: \textbf{L, T} or \textbf{I beam} < 3x3”

Bracing: \textbf{L, T} or \textbf{I beam} < 3x3”

Side without any strengthening beam
III) Bottom parts supports

Bottom parts support specification

- All these parts can be made with steel angles beams of 2” sides.

A) Turn and 3 main parts supports dimensions:

All these supporting structures have the same side profile - height and width.

Their respective lengths are:

1’7”  3’8”  3’8”  2’10”
1) Overhead support

Use:
- 2” (1/4” wall thickness) square tubes for cross beams
- 1 1/2 (1/4” wall thickness) square tubes for bracing
Location: Bolts overhead structure.  
(Dimensions in inches)

Holes are threaded 1/2”-20, through beam

2) Cantilever Support (existing support extension)

Two ½” plates with threaded holes to place leveling mounts into

All beams are 1” 1/2 (1/4”) wall thickness) square tubes
3) 3rd support

2" (1/4" wall thickness) square tubes

1 1/2 (1/4" wall thickness) square tubes

2" (1/4" wall thickness) square tubes

Plates (3/16" thick)
Top plate with threaded holes

7 small threaded holes
in 1/4 in plate, Hole
size 1/4"-20
BLUE: New Holes
BLACK: Existing Holes

4 big holes through 1" square beam (only) are .625" diameter
BLUE: New Holes
BLACK: Existing holes
Appendix C: Test Section Parts - Detailed Drawings

1) Overview

Test Section Outlet – Part 2

Test Section Outlet – Part 1

Tailboard

Cascade Plate

Test Section Inlet

Contraction outlet
Permanent frame

2) Cascade outlet - Permanent frame

Inside wall in ½ inch plywood
3) Test Section Inlet (case without conveyor)

Acrylic panels:

Flange 1 (blue):
4) Cascade Endwall Plate

Permanent plate (1/2 inch thick aluminum):
5) Tailboards (1/2 inch thick acrylic)

Outer tailboard:

Inner tailboard:

Acrylic 1 (short)
1/2 in thick acrylic.
Holes are countersunk, through for 10-32 screws (except the one in the center)
6) Test Section Outlet - Part 1

Acrylic panel:
7) Test Section Outlet - Part 2

½ inch plywood panels:  

Bottom aluminum frame:
Top aluminum frame:

8) Test Section Cover

Made out of ½ inch plywood. Can be made out of acrylic for visualizations.
Appendix D: Wake Simulator Mechanism - Detailed Drawings

1) Inside Frame

Front

Top view

Side view
2) Outside Frame, Drive Side

Side view

Front view

Top view

Isometric view
3) Outside Frame, Other Side

Front view

Side view

Top view

Isometric view
4) Conveyor Stand

Top view

Side view

Front view

Isometric view
5) **Conveyor Shell**

**Shell Assembly**

- 16 gage sheet metal
- About 6 ft x 3 ft x 8 in
- Main “shell” + 2 covers

### 5.1. Main Shell drawing

[Isometric view]

[Front view]
5.2. Top Cover Drawings
5.3. Bottom Cover Drawings

Top view

Isometric view

Front view

Right view
6) Shaft Specifications
7) Moving Plate Assembly

i. Intralox Plastic Belt Module

ii. Intermediate Aluminum Plate
iii. Flat Plate

iv. Cover Plate

v. Leading and Trailing Edges
Jean-Philippe Junca-Laplace, the son of Bernadette and Jean-Bernard Junca-Laplace, brother of Camille and Lauriane Junca-Laplace, was born in November, 1986 in Tours, France. He quickly left this city after six months to settle in Bordeaux. After obtaining his Baccalauréat with honors in 2004, he joined the “Classes Préparatoires aux Grandes Ecoles” for three years at Lycée Michel Montaigne in Bordeaux, with a physics and engineering sciences specialization.

He entered the Institut Supérieur de l’Aéronautique et de l’Espace, ENSICA program in 2007 and started a master’s program in aerospace engineering. He then joined the Department of Mechanical Engineering at Louisiana State University in 2009 to complete a Master of Science in Mechanical Engineering with a specialization in thermal sciences. He submitted his research about the design, fabrication and characterization of a new wind tunnel facility under the supervision of Dr. Shengmin Guo and Dr. Dimitris E. Nikitopoulos in June 2011. He is expecting to receive both master’s degrees in August 2011.

Jean-Philippe continues to play regularly the piano and the guitar. He is thinking about composing music. Also, during his free time, he enjoys swimming, biking and running; one day he hopes to compete in a triathlon or an adventure race.