Experimental investigations on complex vortex flows using advanced flow diagnostic techniques

Zifeng Yang
Iowa State University

Follow this and additional works at: https://lib.dr.iastate.edu/etd
Part of the Aerospace Engineering Commons

Recommended Citation
Yang, Zifeng, "Experimental investigations on complex vortex flows using advanced flow diagnostic techniques" (2009). Graduate Theses and Dissertations. 10979.
https://lib.dr.iastate.edu/etd/10979
Experimental investigations on complex vortex flows using advanced flow diagnostic techniques

by

Zifeng Yang

A dissertation submitted to the graduate faculty in partial fulfillment of the requirements for the degree of DOCTOR OF PHILOSOPHY

Major: Aerospace Engineering

Program of Study Committee:
Hui Hu, Major Professor
Tom Shih
Zhijian Wang
Terry Meyer
Chris Rehmann

Iowa State University
Ames, Iowa

2009

Copyright © Zifeng Yang, 2009. All rights reserved.
DEDICATION

To my mom and uncle Liu Changchun for their love and support throughout my life
# TABLE OF CONTENTS

LIST OF TABLES .......................................................................................... VI

LIST OF FIGURES ...................................................................................... VII

CHAPTER 1. INTRODUCTION TO THE THESIS ........................................ 1

CHAPTER 2. CHARACTERISTICS OF TORNADO-LIKE WINDS ................. 4

  2.1. INTRODUCTION TO CHARACTERISTICS OF TORNADOES ............ 4

  2.2. ISU TORNADO SIMULATOR AND EXPERIMENTAL SETUP .......... 7

  2.3. CHARACTERISTICS OF TORNADO-LIKE FLOW ......................... 10

CHAPTER 3. FLOW AROUND A BUILDING MODEL IN TORNADO-LIKE WINDS ........................................................................................................ 20

  3.1. INTRODUCTION TO WIND LOADS AND FLOW STRUCTURES OVER A BUILDING MODEL IN TORNADO-LIKE WIND ........................................ 20

  3.2. FLOW AROUND A HIGH-RISE BUILDING MODEL IN TORNADO-LIKE WINDS .................................................................................................. 22

    3.2.1. Wind Loads and PIV Measurements on a High-rise Building Model in Straight-line Winds .......................................................... 22

    3.2.2. Wind Loads acting on a High-rise Building Model in Tornado-like Winds .................................................................................. 29

    3.2.3. Flow Structure around a High-rise Building Model in Tornado-like Winds .................................................................................. 34

  3.3. FLOW AROUND A LOW-RISE GABLE-ROOFED BUILDING MODEL IN TORNADO-LIKE WINDS ............................................................................. 41

    3.3.1. Wind Loads and PIV Measurements in Straight-line Winds ........ 41
3.3.2. Wind Loads and Pressure acting on a Gable-roofed Building Model in Tornado-like Winds ................................................................................................................. 48
3.3.3. Flow Structure around a Gable-roofed Building Model in Tornado-like Winds ................................................................................................................................. 55

CHAPTER 4. EXPERIMENTAL STUDY OF TRUBINE BLADE TRAILING EDGE COOLING ................................................................................................................................. 65

4.1. INTRODUCTION TO TRAILING EDGE COOLING .......................................................... 65
4.2. PRESSURE SENSITIVE PAINT TECHNIQUE ................................................................... 70
   4.2.1. Principal Physics of Pressure Sensitive Paint .......................................................... 70
   4.2.2. PSP Calibration ....................................................................................................... 73
4.3. THE COOLING EFFECT MEASUREMENT BY USING PSP ........................................... 77
   4.3.1. Experimental Apparatus ......................................................................................... 77
   4.3.2. Results and Discussion ......................................................................................... 83
   4.3.3. Error Analysis on the Cooling Effectiveness ............................................................ 93
4.4. PIV MEASUREMENTS ON THE TRAILING EDGE COOLING FLOW ...................... 95
   4.4.1. Experimental Apparatus ......................................................................................... 95
   4.4.2. Experimental Uncertainties ................................................................................... 99
   4.4.3. Results and Discussion ......................................................................................... 99

CHAPTER 5. CONCLUSIONS AND FUTURE WORK ......................................................... 120

5.1. THE CHARACTERISTICS OF A TORNADO-LIKE VORTEX ........................................ 120
5.2. FLOW-STRUCTURE INTERACTION FOR A BUILDING MODEL IN A TORNADO-LIKE VORTEX ......................................................................................................................... 121
5.3. FILM COOLING EFFECT ON A TRAILING EDGE ....................................................... 122
5.4. THE CHARACTERISTICS OF THE TRAILING EDGE COOLING FLOW .............. 123
5.5. FUTURE WORK ............................................................................................................. 124

APPENDIX A. THE DESIGN OF CALIBRATION CELL ................................................. 125
APPENDIX B. THE DESIGN OF A TRAILING EDGE MODEL..... 129
APPENDIX C. SYNCHRONIZATION SETUP FOR PIV .................. 131
APPENDIX D. THE PRINCIPLE OF STEREOSCOPIC PIV .......... 133
REFERENCE ......................................................................................... 136
ACKNOWLEDGEMENTS ........................................................................ 145
LIST OF TABLES

TABLE C-1. THE TIMING SETTING FOR PIV MEASUREMENT...................................................... 132
LIST OF FIGURES

FIGURE 2.1. A TYPICAL STRUCTURE OF A TORNADO .......................................................... 5
FIGURE 2.2. SCHEMATIC AND PICTURE OF THE ISU TORNADO SIMULATOR ......................... 8
FIGURE 2.3. EXPERIMENTAL SETUP FOR PIV MEASUREMENTS ..................................... 9
FIGURE 2.4. TYPICAL INSTANTANEOUS STREAMLINE PATTERNS IN THE LOWEST MEASURED
   HORIZONTAL PLANE (Z=4MM) ....................................................................................... 12
FIGURE 2.5. DISTRIBUTION OF INSTANTANEOUS VORTEX CENTERS AT THE LOWEST HORIZONTAL
   PLANE (Z=15MM) ..................................................................................................... 12
FIGURE 2.6. THE FLOW CHARACTERISTICS OF THE TORNADO-LIKE VORTEX .................... 13
FIGURE 2.7. VELOCITY FLOW FIELD WITH STREAMLINES AND VORTICITY FIELD IN HORIZONTAL
   PLANES AT VARIOUS ELEVATIONS (Z=4, 65 AND 150MM ABOVE THE GROUND) ............. 14
FIGURE 2.8. THE FLOW CHARACTERISTICS OF THE TORNADO-LIKE VORTEX .................... 16
FIGURE 2.9. THE SIMULATED TORNADO-LIKE VORTEX VS. MULHALL AND SPENCER TORNADOES .... 18
FIGURE 2.10. A PLANAR VIEW OF A TORNADO-LIKE VORTEX IN EACH HORIZONTAL PLANE .......... 19

FIGURE 3.1. TEST MODEL AND DEFINITION OF PARAMETERS ............................................. 23
FIGURE 3.2. A PICTURE OF THE ROUGHNESS SETUP ....................................................... 24
FIGURE 3.3. VELOCITY AND TURBULENCE INTENSITY PROFILES ....................................... 24
FIGURE 3.4. A PICTURE OF THE TEST MODEL AND JR3 LOAD CELL ..................................... 27
FIGURE 3.5. WIND LOADS ACTING ON THE TEST MODEL VS. ORIENTATION ANGLES .............. 27
FIGURE 3.6. PIV MEASUREMENTS AROUND THE TEST MODEL AT Z=70MM IN STRAIGHT-LINE WIND .... 28
FIGURE 3.7. WIND LOADS VS. THE DISTANCE BETWEEN THE TEST MODEL AND THE TORNADO-LIKE
   VORTEX .................................................................................................................... 30
FIGURE 3.8. THE MEASURED WIND LOADS VS. ORIENTATION ANGLES .............................. 32
FIGURE 3.9. SCHEMATIC OF THE FLOW STREAMS APPROACHING THE TEST MODEL IN STRAIGHT-LINE
   AND TORNADO-LIKE WINDS ..................................................................................... 33
FIGURE 3.10. FLOW FIELD AROUND THE TEST MODEL VS. THE DISTANCE BETWEEN THE CENTERS OF
   THE TEST MODEL AND THE TORNADO-LIKE VORTEX ................................................... 35
FIGURE 3.11. FLOW FIELD AROUND THE TEST MODEL VS. ORIENTATION ANGLE AT R/R₀=1 ....... 38
FIGURE 3.12. FLOW FIELD AROUND THE TEST MODEL AT DIFFERENT ELEVATION LEVELS AT R/R₀=1 40
FIGURE 3.13. THREE-DIMENSIONAL VIEW OF THE WAKE VORTEX STRUCTURES AROUND THE MODEL
   AT R/R₀=1 WITH OA=0 DEG .................................................................................... 41
FIGURE 3.14. TEST MODEL AND DEFINITION OF PARAMETERS .............................................. 43
FIGURE 3.15. PRESSURE TAP DISTRIBUTION ...................................................................... 43
FIGURE 3.16. PIV EXPERIMENTAL SETUP ......................................................................... 43
FIGURE 3.17. FORCE AND MOMENT COEFFICIENTS VS. ORIENTATION ANGLES .......... 44
FIGURE 3.18. PIV MEASUREMENTS ON A VERTICAL PLANE (Y=0MM) IN STRAIGHT-LINE WIND .......................................................... 46
FIGURE 3.19. PIV MEASUREMENTS ON A HORIZONTAL PLANE (Z=15MM) IN STRAIGHT-LINE WIND .................................................. 47
FIGURE 3.20. WIND LOADS VS. THE DISTANCE BETWEEN THE TEST MODEL AND THE TORNADO-LIKE VORTEX .................................................................................................................. 49
FIGURE 3.21. MEASURED TYPICAL PRESSURE DISTRIBUTIONS AROUND THE TEST MODEL ................................................................. 53
FIGURE 3.22. WIND LOADS VS. ORIENTATION ANGLES .............................................................. 55
FIGURE 3.23. FLOW FIELD AROUND THE TEST MODEL VS. THE DISTANCE OF BETWEEN THE CENTERS OF THE TEST MODEL AND THE TORNADO-LIKE VORTEX ................................................. 60
FIGURE 3.24. FLOW FIELD AROUND THE TEST MODEL VS. THE ORIENTATION ANGLES ........................................................................ 61
FIGURE 3.25. THREE-DIMENSIONAL FLOW FIELD WITH STREAMLINES AND SWIRLING STRENGTH CONTOURS IN Y AND Z DIRECTIONS ...................................................................... 63

FIGURE 4.1. SCHEMATIC DRAWING OF A TYPICAL TURBINE BLADE WITH A CUTBACK TRAILING EDGE .............................................................................................................................................. 66
FIGURE 4.2. MODEL OF PRESSURE SENSITIVE PAINT .............................................................. 71
FIGURE 4.3. THE PICTURE AND SCHEMATIC OF THE PSP CALIBRATION SETUP .................. 74
FIGURE 4.4. INTENSITY CONTOURS OBTAINED FROM CALIBRATION: A. INTENSITY FROM REFERENCE IMAGE; B. INTENSITY FROM VACUUM IMAGES P/Pref=0.23; C. INTENSITY RATIO Iref/I; D. ERROR LEVEL % ........................................................................................................... 75
FIGURE 4.5. PSP CALIBRATION CURVE .................................................................................. 76
FIGURE 4.6. ERROR LEVEL OF THE INTENSITY RATIO VS. PRESSURE RATIO ...................... 76
FIGURE 4.7. PSP EXPERIMENTAL SETUP .............................................................................. 77
FIGURE 4.8. DIMENSIONS OF STUDIED TRAILING EDGE COOLING MODEL ...................... 80
FIGURE 4.9. SCHEMATIC OF THE EXPERIMENTAL SETUP .................................................. 81
FIGURE 4.10. THE PICTURE OF THE EXPERIMENTAL SETUP .............................................. 82
FIGURE 4.11. ADIABATIC FILM COOLING EFFECTIVENESS ON THE TRAILING EDGE FOR DIFFERENT BLOW RATIOS M ............................................................................................................ 84
FIGURE 4.12. COOLING EFFECTIVENESS PROFILES FOR THE CASE WITHOUT LANDS .......... 86
FIGURE 4.13. SLOT GEOMETRY FOR THE FILM COOLING OF A SURFACE IN MUKHERJEE’S MODEL .............................................................. 87
FIGURE 4.14. SPANWISE AVERAGED COOLING EFFECTIVENESS PROFILES FOR THE CASE WITHOUT LANDS ......................................................................................................................... 88

FIGURE 4.15. ADIABATIC FILM COOLING EFFECTIVENESS (C.E.) ON THE TRAILING EDGE CUTBACK FOR DIFFERENT BLOW RATIOS M ........................................................................................................ 89

FIGURE 4.16. COOLING EFFECTIVENESS PROFILES FOR THE CASE WITH LANDS ................................................................................................. 90

FIGURE 4.17. SPANWISE AVERAGED COOLING EFFECTIVENESS PROFILES FOR THE CASE WITH LANDS .......................................................................................................................................... 91

FIGURE 4.18. COMPARISON OF COOLING EFFECTIVENESS PROFILES ........................................................................................................................... 93

FIGURE 4.19. SCHEMATIC OF THE CAMERA VIEWS ........................................................................................................................... 94

FIGURE 4.20. THE SCHEMATIC OF THE PIV SETUP FOR FLOW MEASUREMENT ........................................................................................................................... 98

FIGURE 4.21. PIV MEASUREMENT RESULTS AT BLOW RATIO OF M =0.43 ........................................................................................................................... 101

FIGURE 4.22. PIV MEASUREMENT RESULTS AT BLOW RATIO OF M =0.64 ........................................................................................................................... 102

FIGURE 4.23. PIV MEASUREMENT RESULTS AT BLOW RATIO OF M =0.76 ........................................................................................................................... 102

FIGURE 4.24. PIV MEASUREMENT RESULTS AT BLOW RATIO OF M =1.1 ........................................................................................................................... 103

FIGURE 4.25. PIV MEASUREMENT RESULTS AT BLOW RATIO OF M =1.6 ........................................................................................................................... 103

FIGURE 4.26. STREAMWISE VELOCITY PROFILES ON THE CENTER VERTICAL PLANE WITH VARIOUS BLOW RATIOS AT DIFFERENT DISTANCES AFTER THE EXITS ........................................................................................................................... 104

FIGURE 4.27. THREE-DIMENSIONAL VELOCITY VECTOR AND AXIAL VELOCITY CONTOUR AT BLOW RATIO OF M=0.43 AT FIVE DIFFERENT DISTANCES AFTER THE SLOT EXIT: A. WITHOUT LANDS

B. WITH LANDS .................................................................................................................................................. 107

FIGURE 4.28. THREE-DIMENSIONAL VELOCITY VECTOR AND AXIAL VELOCITY CONTOUR AT BLOW RATIO OF M=0.64 AT FIVE DIFFERENT DISTANCES AFTER THE SLOT EXIT: A. WITHOUT LANDS

B. WITH LANDS .................................................................................................................................................. 108

FIGURE 4.29. THREE-DIMENSIONAL VELOCITY VECTOR AND AXIAL VELOCITY CONTOUR AT BLOW RATIO OF M=0.76 AT FIVE DIFFERENT DISTANCES AFTER THE SLOT EXIT: A. WITHOUT LANDS

B. WITH LANDS .................................................................................................................................................. 109

FIGURE 4.30. THREE-DIMENSIONAL VELOCITY VECTOR AND AXIAL VELOCITY CONTOUR AT BLOW RATIO OF M=1.1 AT FIVE DIFFERENT DISTANCES AFTER THE SLOT EXIT: A. WITHOUT LANDS

B. WITH LANDS .................................................................................................................................................. 110

FIGURE 4.31. THREE-DIMENSIONAL VELOCITY VECTOR AND AXIAL VELOCITY CONTOUR AT BLOW RATIO OF M=1.6 AT FIVE DIFFERENT DISTANCES AFTER THE SLOT EXIT: A. WITHOUT LANDS

B. WITH LANDS .................................................................................................................................................. 111
FIGURE 4.32. AVERAGED STREAMWISE VORTICITY CONTOUR WITH VARIED BLOW RATIOS AT
THREE DIFFERENT DISTANCES AFTER THE EXITS FOR THE CASES WITH LANDS ....................... 113
FIGURE 4.33. FLOW FIELD AT DIFFERENT CROSS PLANES FOR M=0.43 AND M=1.6 ...................... 114
FIGURE 4.34. FLOW FIELD COMPARISONS AT DIFFERENT DISTANCE DOWNSTREAM .......................... 116
FIGURE 4.35. FLOW FIELD COMPARISONS AT DIFFERENT DISTANCE DOWNSTREAM ...................... 117
FIGURE 4.36. FLOW STRUCTURES IN THE TRAILING EDGE OF TURBINE BLADE ............................ 118
FIGURE 4.37. AVERAGED 3-D TURBULENT KINETIC ENERGY AT X/H=8 AT BLOW RATIO OF M=1.6. 119

FIGURE B-1. A SCHEMATIC OF THE TRAILING EDGE OF A TURBINE BLADE ................................. 129
FIGURE B-2. THE UPPER PART OF THE TRAILING EDGE MODEL .................................................. 129
FIGURE B-3. THE LOWER PLATE OF THE TRAILING EDGE MODEL .............................................. 130
FIGURE B-4. THE LAND OF THE TRAILING EDGE MODEL ............................................................. 130

FIGURE C-1. THE SYNCHRONIZATION SETUP FOR LASER AND CAMERA ................................. 131

FIGURE D-1. SCHEIMPFLÜG CONFIGURATION FOR STEREOSCOPIC PIV ................................. 133
FIGURE D-2. IMAGE PERSPECTIVE EFFECT DUE TO CAMERA TILT IN SCHEIMPFLÜG ARRANGEMENT 135
CHAPTER 1. INTRODUCTION TO THE THESIS

Everyone has noted at some time swirling leaves on a windy autumn day. Circle eddies on the surface of a river can be observed when one plays ducks and drakes. What is the attraction of this phenomenon? Is there an unknown force behind the fast swirling movement of matter? Until 19th century, humans have thought that supernatural forces were the causes of vortices. Again and again the danger and mystery of vortical motions, such as tornadoes, have excited fantasy and superstition. Today, however, no student of science believes in supernatural causes on the cryptic meaning of vortices, but both danger and mystery have remained characteristic of many vortex flows. Hurricanes and tornadoes still occur all around the world, and many people die when these disasters happen. Obviously vortices play much more powerful roles in nature than merely whirling leaves or eddies in a river suggest. Thus it is not surprising that the vortex concept is of central importance in the history of science and philosophy.

Modern science has extended human understanding of vortices and their role in nature or aerodynamic field in two ways: It has clarified the basic significance of vortical motion in mechanics, and it has extended the spatial horizon of humans so that they can observe both large-scale and micro-scale vortices and interpret them. Today one can follow the movements of hurricanes on satellite photographs and can visualize vortical flow structure in micro-scale by using advanced diagnostic techniques, such as Particle Image Velocimetry (PIV). Therefore, the vortex flow problems are not that mysterious today.

Using advanced diagnostic techniques, two complex vortex flow topics were addressed in the present thesis. First, the characteristics of flow structures around building models in a
tornado-like vortex were studied by using a high-resolution Particle Image Velocimetry (PIV) system. Second, the 3D vortical structures of the film cooling flows in the trailing edge of a turbine blade were studied by using a stereoscopic PIV system. The flow measurement results were correlated to the film cooling effectiveness measured by the application of a relatively new technique, pressure sensitive paint (PSP).

In the study of tornado-like vortex, by using the world-largest tornado simulator of Iowa State University, a comprehensive PIV study on the flow structures around a high-rise building model and a gable-roofed building model, as well as the surface pressure measurements and force measurements, was conducted to elucidate the underlying physics. The ultimate objective of the present study is to quantify the surface winds generated by tornadoes and flow-structure interactions between tornadoes and built environments to assess wind-induced damage with the purpose of mitigating damage and improving public safety. The characteristics of tornado like flow will be demonstrated and discussed in Chapter 2. The flow features around a building model and flow-structure interactions will be discussed in Chapter 3.

In the study of vortex flow in the trailing edge of a turbine blade, detailed distributions of film cooling effectiveness measurements were obtained in the cutback region of trailing edge by using the PSP technique. Before the application of PSP, a self-designed calibration facility was developed to study the characteristics and to complete the calibration of the PSP. Corresponding to the film cooling effectiveness measurements, an experimental study was conducted to quantify the characteristics of coolant flows in the cutback region at the trailing edge of a turbine blade. A high-resolution stereoscopic PIV system were used to conduct detailed flow measurements to quantitatively visualize the evolution of the unsteady vortex
and turbulent flow structures in coolant jet streams, and to quantify the mixing process between the wall jet cooling streams and main streams. The film cooling effectiveness and flow characteristics in the cutback region at the trailing edge of a turbine blade will be discussed in Chapter 4.
CHAPTER 2. CHARACTERISTICS OF TORNADO-LIKE WINDS

2.1. Introduction to Characteristics of Tornadoes

Tornadoes are violently rotating columns of air and are considered nature’s most violent storms. Tornadoes may occur wherever conditions favor the development of strong thunderstorms. Essential conditions for such storms are the presence of cool, dry air at middle levels in the troposphere, overlying a layer of moist, conditionally unstable air near the surface of the Earth. Conditional instability occurs when a saturated air parcel continues to rise once set in motion. When conditionally unstable air rises, it becomes warmer owing to the condensation of water vapor. As the water condenses, heat is released, further warming the air and fueling its rise. This convective action (i.e., the circulation of air as a result of heat transfer) produces the huge clouds commonly associated with thunderstorms and tornadoes. Convection can be initiated when the sun heats a localized area of the ground, destabilizing the near-surface air.

Most tornadoes are formed when a strong updraft, such as those described above, acts to concentrate atmospheric rotation, or spin, into a swirling column of air, as shown in Figure 2.1. Spin is a natural occurrence in air because horizontal winds almost always experience both an increase in speed and a veering in direction with increasing height above the ground. The increase of wind speed with height (called vertical speed shear) produces “crosswise spin,” that is rotation about a horizontal axis crosswise to the direction of wind flow. When air containing crosswise spin flows into an updraft, the spin is drawn upward, producing rotation about a vertical axis. The veering of wind direction with height (vertical direction shear) is another source of horizontal spin, this time oriented in the same direction as the
wind flow and known as “streamwise spin.” When air containing streamwise spin is drawn into an updraft, it too is tilted upward and rotates about a vertical axis. Although crosswise spin and streamwise spin are oriented at right angles to each other, both rotations exist in the horizontal plane, and both types have been revealed by Doppler radar observations to contribute to the evolution of a rotating updraft.

Figure 2.1. A typical structure of a tornado

In an average year, 800 ~ 1000 tornadoes occur in the U.S. alone, and cause about 80 deaths (on average), over 1500 injuries, and $850 million worth of property damage [1]. Tornados are inherently destructive and complicated in appearance, including variety of shapes (such as thin ropes, cylindrical columns, funnels etc.), sizes and configurations under a wide variety of circumstances. All have a central core (i.e. tornado eye) of organized and concentrated vorticity. Therefore, the flow field in a tornado is much different from the straight-line, boundary-layer wind. Tornado flows have been extensively studied by conducting field observations [2-4], numerical simulation and experimental simulation using
a vortex chamber or tornado simulator. Many laboratory simulator designs have been based on the pioneering work of Ward [5]. For laboratory tornado simulators, the common flow parameter has historically been the swirl ratio. Essentially a measure of the relative amount of angular to radial momentum in the vortex, the swirl ratio was expressed by Ward [5] and Church et al. [6] as:

$$ S = \frac{r_1 \Gamma}{2Q} = \frac{V_\theta}{2V_r a} $$

(1)

where $\Gamma$ is the circulation, $r_1$ is the radius of the domain, $Q$ is the inflow rate, $V_\theta$ is the maximum tangential velocity, $V_r$ is radial velocity at $r_1$ and $a$ is the aspect ratio defined as:

$$ a = \frac{h}{r_1} $$

(2)

where $h$ is the inflow depth. The swirl ratio is a measure of the amount of rotational energy in the vortex relative to the convective energy in the vortex and correlates well with vortex structure. Typical value of swirl ratio ranges from 0.1~6.

Subsequent efforts—based on the Ward model—at Purdue University [6], the University of Oklahoma [7] and that of Davies-Jones [8] employed various means to improve the similarity between laboratory simulations and full-scale tornado events. These laboratory simulations were aimed at obtaining a greater understanding of the tornado vortex itself. However, numerical simulation has overtaken physical simulation as the tool of choice for tornado vortex studies—both because of cost and because of versatility. While both laboratory and numerical simulation efforts have revealed a great deal about tornado structure [9], the problem of numerically simulating a domain large enough to accommodate a tornado vortex with ground-based structures remains computationally impractical.
From the lessons learned from testing several simulator concepts and from some insights into tordadogenesis, a new approach was developed and tested [1]. The new design employs a “rotaing forced downdraft” to loosely match observations from natural tornadoes. A central fan produces an updraft, and the incoming flow is directed downward through an annular duct. Vanes in the annular duct impart rotation to the flow which leads to the technique being named a “rotating forced downdraft”. The rotating downdraft exits the duct near the ground plane and diverges with a sizeable portion of the flow moving inward beneath the fan. This flow feeds the updraft as a circular symmetric inflow. The vorticity present in the inflow is stretched beneath the updraft, forming a tornado that travels along the ground plane as the entire fan/downdraft-producing mechanism translates.

In the present study, a high resolution PIV system was utilized to conduct whole field measurements to quantify the characteristics of the tornado-like vortex generated by the ISU tornado simulator before the test building models were mounted on the ground plane.

2.2. ISU Tornado Simulator and Experimental Setup

Figure 2.2 shows the schematic and a photo depicting the flow circuit and dimensions of the ISU tornado simulator used in the present study. A circular duct of 5.49m in diameter and 3.35m in height is suspended from a heavy duty overhead crane. A 1.83m diameter fan (maximum flow rate is 59.0 m$^3$/s, 125,000 cfm) is mounted concentrically inside the circular duct to generate a strong updraft. The flow from the fan is redirected downward in a 0.30m wide annular duct to simulate the rear flank downdraft (RFD) encirclement found in natural tornadoes [10]. Swirling is imparted to the airflow in the duct by adjusting the angle of the vanes at the top of the tornado simulator. The downdraft air diverges upon hitting the ground
with most of the flow moving inward toward the fan. The fan updraft stretches the low-level vorticity into a tornado-like vortex. A unique feature of the ISU tornado simulator is that the tornado-like vortex can travel along the ground plane as the entire fan/downdraft-producing mechanism translates. This translation, along with the fact that there is a adjustable clearance between the translating duct and the ground plane, allows a wide range of building models to be placed in the path of the tornado-like vortex for testing. The ISU tornado simulator can generate a tornado-like vortex with a maximum diameter of 1.2 m and maximum tangential velocity of 14.5 m/s. The maximum swirl ratio achieved is 1.14, and the translation speed of the tornado-like vortex can reach up to 0.8 m/s. The vortex height can vary from 1.2 m to 2.4 m by adjusting the ground plane upward or downward. Further information about the design, construction, performance and the quantitative comparisons of the tornado-like vortex generated by the ISU tornado simulator with tornadoes found in nature can be found in Haan et al. [1].

![Schematic diagram and picture of the ISU tornado simulator](image)

Figure 2.2. Schematic and picture of the ISU tornado simulator

In the present study, the ground floor was fixed at 0.457 m below the exit of the outer duct, and the fan speed was fixed at 20 Hz (1/3rd of the maximum speed). The radius of the
A tornado-like vortex core, $R_O$, was found to be 0.165m ($R_O = 0.165m$), where the maximum tangential speed ($V_o = 7.0 \text{ m/s}$) was observed. Based on the definition of Church et al. [6], the swirl ratio of the tornado-like vortex (i.e., the measure of rotating momentum to the radial inflow momentum) used in the present study was about 0.1, and the aspect ratio of the tornado-like vortex was about 3.6.

Figure 2.3. Experimental setup for PIV measurements

As shown in Fig. 2.3, a digital Particle Image Velocimetry (PIV) system was used to conduct detailed flow field measurements to quantify the evolution of the unsteady vortex and turbulent flow structures around the transparent test model. The flow was seeded with ~1 μm oil droplets by using a droplet generator. Illumination was provided by a double-pulsed Nd:YAG laser (NewWave Gemini 200) adjusted to the second harmonic and emitting two pulses of 200 mJ at a wavelength of 532 nm with a repetition rate of 10 Hz. The laser beam
was shaped into a sheet by a series of spherical and cylindrical lenses. The sheet was then adjusted to the plane of interest with a mirror. The thickness of the laser sheet in the measurement region was about 1.0mm. A high resolution 12-bit CCD camera (Pixelfly, CookeCorp) was used for PIV image acquisition with the axis of the camera perpendicular to the laser sheet. The CCD camera and the double-pulsed Nd:YAG lasers were connected to a workstation (host computer) via a Digital Delay Generator (Berkeley Nucleonics, Model 565), which controlled the timing of the laser illumination and the image acquisition. More detail about the times series setup can be found in Appendix C. Instantaneous PIV velocity vectors were obtained from a frame to frame cross-correlation technique involving successive frames of patterns of particle images in an interrogation window of 32×32 pixels. An effective overlap of 50% of the interrogation windows was employed in PIV image processing. The time-averaged velocity \((U, V)\) distributions were obtained from a cinema sequence of 498 frames of measured instantaneous velocity fields. The measurement uncertainty level for the velocity vectors was estimated to be within 1.0%.

2.3. Characteristics of Tornado-like Flow

During the experiments, PIV measurements were first conducted in the flow field without building model to quantify the characteristics of the tornado-like vortex generated by the ISU tornado simulator. The cinema sequence of instantaneous PIV measurement results revealed clearly that tornado-like vortices are highly turbulent with varying vortex sizes and vortex center locations from one frame to another. Fig. 2.4 demonstrates a typical instantaneous flow field with streamlines in the lowest measured horizontal plane \((Z=4\text{mm})\). Images were taken from below the generated tornado, but all flow field images are shown from above for
convenience. A single counter-clockwise vortex is apparently shown in the streamlines. Even though the streamline are not smooth spiral circles, the vortex center was found to be located at the origin center (X=0, Y=0).

Vortex wandering is the seemingly random motion around the mean position of the vortex center, which was firstly mentioned in Lund and Snow [11]. This feature causes difficulty in determining the vortex center. This is an important issue because the vortex center is the reference for computation of all the physical quantities and interpretation of vortex structure. To some extent, vortex wandering may be an inherent feature of a vortex core, closely related to turbulence in the incoming flow and the flow separation. Church and Snow [6] modified their vortex chamber configuration and made efforts to minimize vortex wandering, but they found it impossible to avoid completely. For point measurement techniques, such as LDV, big problems in setting up the measurement system and interpreting the obtained results occur due to vortex wandering. Fortunately, the vortex center can be tracked using PIV technique and the mean vortex center can be obtained by ensemble-averaging sufficiently large amounts of data samples. Here vortex wandering was observed in the lowest horizontal plane as shown in Figure 2.5. The instantaneous vortex centers were scattering distributed without evident orientation priority. Almost all vortex centers recorded from the 498 PIV images were randomly distributed in a circle with a radius of $2R_0/3$, where $R_0$ is the tornado core radius where the maximum tangential velocity is found. Since the vortex center is always travelling in a certain range, the most appropriate approach to determine the location of the vortex center and corresponding flow fields is ensemble-averaging the flow fields, provided enough samples are collected. The vortex center did not vary much when it was
compared for 300 PIV images and 498 images. Therefore 498 images can be said enough for the averaged measurement results.

Figure 2.4. Typical instantaneous streamline patterns in the lowest measured horizontal plane (Z=4mm)

Figure 2.5. Distribution of instantaneous vortex centers at the lowest horizontal plane (Z=15mm)
Figure 2.6. The flow characteristics of the tornado-like vortex

The time-averaged PIV measurement results are presented to characterize the time-averaged behavior of the tornado-like vortex. Figure 2.6 shows the time-averaged PIV measurement results (i.e., velocity distributions and the corresponding streamlines) to reveal the 3-D flow structures of the tornado-like vortex. In the figure, axisymmetric flow patterns in the form of a well-defined single clockwise vortex can be seen clearly in the horizontal planes. The streamlines in the vertical plane passing the time-averaged vortex center reveal clearly that flow streams near the ground and far away from the vortex center would move towards the vortex core and turn upward abruptly before reaching the vortex center. It indicates a radial and upward vertical flow appearing in the region outside the vortex core, as expected. An interesting flow feature is seen in the vortex core region, where flow is found to be a downdraft jet impacting the ground. As the downdraft jet approaches the ground in the
vortex center, it move outwards and contacts the radial inflow from outside. Both branches will join and move upward. As a result, a “circulation bubble” structure was found to form near the ground at the interface between the upward outside flow stream and the downdraft jet in the center of the tornado-like vortex. The downdraft jet flow in the center of a tornado-like vortex, revealed from the PIV measurements, was found to be consistent with the findings of Wurman & Gill [4], who conducted Doppler Radar measurements of the Dimmitt, Texas (2 June 1995) tornado.

![Figure 2.7. Velocity flow field with streamlines and vorticity field in horizontal planes at various elevations (Z=4, 65 and 150mm above the ground)](image)

Fig. 2.7 shows the mean horizontal velocity fields with streamlines at three elevations (Z=4, 65 and 150mm). The color contour in Fig. 2.7 (a) represents the horizontal velocity magnitude ($MV = \sqrt{V_x^2 + V_y^2}$). The color contour in Fig. 2.7 (b) indicates the vorticity magnitude in the Z-direction. Since ensemble-averaging process filtered out details of the instantaneous unsteady small scale vortices, if any, as shown in Fig. 2.4, streamlines in the averaged flow are much more smooth and circular. It was found that the flow pattern in the lowest horizontal plane (Z=15mm) is more spiral than those found in higher levels. The
velocity magnitude, at the same radius, is found to be bigger in lower levels than those in higher levels. That is because all the incoming flow moves into the tornado core from outside at a lower level, which can be seen in Fig. 2.6. The maximum vorticity region is found in the lowest level due to the high rotation rate. Even though the vorticity field is not very uniform, the vorticity contour is almost axisymmetric with respect to the vortex center.

In order to reveal the flow structure of the tornado-like vortex more clearly, Figure 2.8 shows the time-averaged PIV measurement results in a typical horizontal plane with the center of the tornado-like vortex moving away from the center of the measurement window (i.e., the position where the building models would be mounted). For the PIV measurements shown in Fig. 2.8, the measurement plane was selected as the horizontal plane (X-Y plane) at 70mm above the ground plane (Z=70mm), which is also the middle plane of the high-rise building model if mounted on the test plate.

Fig. 2.8(a) and Fig. 2.8(b) show the measurement results in the core region of the tornado-like vortex with the measurement window centered at $R/R_O \approx 0.0$ and $R/R_O \approx 0.5$. As shown in the plots, the streamlines of the flow field in the core region of the tornado-like vortex were found to be a group of concentric circles. The magnitude of the flow velocity was found to increase almost linearly with the increasing radial distance away from the center of the vortex. The flow velocity vectors in the core region of the tornado-like vortex were found to be tangential to the circular streamlines, with the radial components of the flow velocity vectors being almost zero. Such measurement results indicate that the flow in the core region of the tornado-like vortex is very much like that of a potential vortex with the tangential velocity vectors and the negligible radial components.
Figure 2.8. The flow characteristics of the tornado-like vortex.
Fig. 2.8(c) shows the time-averaged flow velocity distribution and corresponding streamlines with the center of the measurement window located at $R/R_O \approx 1.0$. While the streamlines of the flow field in the inner region of the tornado-like vortex core (i.e., region with $R/R_O < 1.0$) were found to be concentric circles, the streamlines in the outside region of the tornado-like vortex core (i.e. the region with $R/R_O > 1.0$) revealed a spiral shape.

The spiral flow streams in the outside region of the tornado-like vortex core were elucidated more clearly from the measurement results shown in Fig. 2.8(d), Fig. 2.8(e) and Fig. 2.8(f) with the center of the tornado-like vortex moving farther away from the measurement windows ($R/R_O \approx 2.3$, $R/R_O \approx 4.5$ and $R/R_O \approx 6.2$, respectively). As revealed from streamlines shown in the figures, the flow velocity vectors in the outside region of the tornado-like vortex were found to have significant radial components in addition to tangential components. As a result, a strong spiral flow was generated in the outside region of the tornado-like vortex core, which made the surrounding air flow towards the core of the vortex.

Figure 2.9 shows the measured tangential velocity profile of the tornado-like vortex compared to those of the field measurement data of two tornados found in nature to demonstrate the similarity between the tornado-like vortex generated by using the ISU tornado simulator and the tornados found in nature. The field measurement data of the tornados used for the comparison were made available to ISU researchers by Mr. J. Wurman under a subcontract of a NSF-sponsored project, which was also published in Wurman & Alexander [3]. The field data was acquired using Doppler Radar on Wheels observations from the Spencer, South Dakota tornado of May 30, 1998 and the Mulhall, Oklahoma tornado of May 3, 1999. It can be seen clearly that, even though the tornado-like vortex generated by the ISU tornado simulator and the two tornados found in nature have significant differences in their core diameters (0.33 m for the tornado-like
vortex generated by the ISU tornado simulator vs. approximately 400m and 800m for Spencer and Mulhall tornados, respectively), the overall flow structures scale with each other reasonably well.

It should also be noted that the measured radial velocity profile given in Fig. 2.9 reveals quantitatively that the radial components of flow velocity vectors in the core region of the tornado-like vortex (i.e. $R/R_o < 1.0$) are very small; almost negligible. However, in the outside region of the tornado-like vortex ($R/R_o > 1.0$), the radial component of flow velocity was found to increase rapidly with increasing radial distance from the center of the tornado-like vortex.

Based on the PIV measurements described above, the flow structures of the tornado-like vortex in horizontal planes was reconstructed, which is shown schematically in Figure 2.10. As revealed in the figure, the tornado-like vortex in each horizontal plane can be divided into two regions: an inner core region and an outer region. In the inner core region, air streams rotate concentrically with the wind speed increasing linearly with increasing radial distance.
from the rotation center. The flow inside the tornado-like vortex core rotates like a cylindrical column. The flow velocity vectors in the vortex core region were found to be almost tangential with negligible radial components. Wind speed reaches a peak value at the outer boundary of the tornado-like vortex core, and then begins to decrease with increasing distance away from the center of the tornado-like vortex in the outer region. The flow velocity vectors in the outer region of the tornado-like vortex have significant radial components flowing towards the center of the tornado-like vortex. As a result, a strong spiral motion is generated, which induces surrounding airstreams to flow toward the core of the tornado-like vortex. It should be noted, as revealed from the PIV measurement results in vertical planes shown in Fig. 2.6 that there also exists a strong upward flow in the outer region and downdraft jet flow in the vortex core making the tornado-like vortex a very complex, turbulent, three-dimensional vortex flow.

Figure 2.10. A planar view of a tornado-like vortex in each horizontal plane
CHAPTER 3. FLOW AROUND A BUILDING MODEL IN TORNADO-LIKE WINDS

3.1. Introduction to Wind Loads and Flow Structures over a Building Model in Tornado-like Wind

Statistics show that 90% of all recorded tornados are rated F2 or less [12] on the Fujita Scale—that is, they involve wind speeds less than 160 mph (3-sec gust). It may be economically feasible to design structures to resist F2 tornados. It might be argued that certain essential facilities such as power plants, hospitals, and airports should be designed for tornados of F3 intensity or higher. The design work for built structures requires a good understanding about the nature of tornados and accurate information about the tornado-induced wind loads and wind field information around the civil structures due to the presence of tornados.

With the consideration of buildings as surface-mounted obstacles, numerous experimental and numerical studies have been carried out to investigate wind loads and flow structures around surface-mounted obstacles. Castro & Robins [13] and Hunt et al. [14] investigated the vortex structure and topology of the flow around cuboids for a boundary-layer-type of approaching flow. Martinuzzi & Tropea [15] investigated the turbulence characteristics of flows around prismatic obstacles mounted in a plane channel. Schofield & Logan [16] aimed their research at the characterization of the recovery of the turbulent boundary layer downstream of prismatic obstacles. The effect of flow angle of attack was contemplated in the investigations conducted by Natarajan & Chyu [17]. Shah & Ferziger [18] conducted a numerical study to investigate the flow passing a cubic obstacle by using the Large Eddy Simulation (LES) method. Yakhot et al. [19] conducted a direct numerical
simulation of the turbulent flow around a wall-mounted cube to investigate the spatial and temporal evolution of large-scale vortex structures around the cube. Besides the research on general prismatic obstacles, on one hand, several studies have been recently conducted to consider more realistic high-rise building models [20-24]. On the other hand, several studies have also been conducted to consider more realistic low-rise gable-roofed building models with various roof shapes [25-28]. It has been found that the roof shape has a strong influence on the near-field flow characteristics. For example, Sousa and Pereira [29] demonstrated that the suction distribution over the walls of the building model and the dispersive characteristics of the flow in the vicinity of the obstacle changes significantly as a result of the existence of gable roofs.

While those experimental and numerical efforts have revealed a great deal about the wind loads and flow structures around surface-mounted obstacles or building models, all the studies mentioned above were conducted with straight-line boundary layer flows or straight-line winds. Tornadoes are strong vortices with significant tangential component, radial inflow/outflow, and vertical updraft/downdraft. Tornado-induced wind loads and flow structures around surface-mounted obstacles in tornado-like winds would be quite different from that in straight-line boundary-layer flows [10, 30].

Surprisingly, very few studies can be found in literature that specifically address the wind loads and flow structures around building models in swirling, tornado-like winds. Jischke & Light [7] and Bienkiewicz & Dudhia [31] conducted comparative studies to measure wind loads and surface pressure on small building models in swirling, tornado-like winds and straight-line winds. They found that the wind loads acting on the tested models are significantly higher (3-5 times) in swirling, tornado-like winds, and surface pressure
distributions on the tested building models are also quite different compared to those in straight-line winds. This suggests that it is incorrect, at least incomplete, to use a conventional straight-line wind tunnel running with maximum tornado wind speed to estimate tornado-induced wind loads on built structures. It should also be noted that almost all the previous work on building models in tornado-like winds were conducted by measuring wind loads and surface pressure distributions only [32-35]. No study has been conducted so far to provide detailed flow measurements to investigate the characteristics of wake vortices and turbulent structures around building models in tornado-like winds.

By using the world-largest tornado simulator of Iowa State University, a comprehensive study on the flow structure around building models, as well as the surface pressure measurements and force measurements, was conducted to elucidate the underlying physics. The ultimate objective of the present study is to quantify the surface winds generated by tornadoes and flow-structure interactions between tornadoes and built environments to assess wind-induced forces with the purpose of mitigating damage and improving public safety.

### 3.2. Flow around a High-rise Building Model in Tornado-like Winds

#### 3.2.1. Wind Loads and PIV Measurements on a High-rise Building Model in Straight-line Winds

In order to compare the wind loads acting on a high-rise building model in tornado-like wind with those in straight-line wind, the same test model was mounted in an Aerodynamic/Atmospheric Boundary Layer (AABL) straight-line wind tunnel [30] to conduct the force and PIV measurements. The AABL wind and gust tunnel is located in the ISU WiST Lab at Iowa State University. This wind tunnel is a closed circuit wind tunnel with a capacity of generating a maximum wind speed of 53m/s in the test section of 1.95m ×
2.29m. There is a rotating turntable in the test section where the models can be mounted to test for different wind directions.

3.2.1.1. Studied models

Two high-rise building models were made to conduct the present study. These two test models have the same dimension, a square cross section of 34.4×34.4mm and a height of 140mm, shown in Fig. 3.1(a). The first one was made of wood for wind load measurements. The second one was made of transparent Plexiglas, which was used for PIV measurements of flow fields around the tested building model. Fig. 3.1(b) shows the relative positions of the tornado-like vortex center with respect to the building model and the orientation angle (OA) of the building model, where $R$ is the distance between the center of the tornado-like vortex and the center of the building model. It also shows the definition of an orientation angle in straight-line wind.

![Diagram of test model and definitions of parameters](image)

Figure 3.1. Test model and definition of parameters
A suburban boundary layer was used for this study. The roughness elements used to generate the suburban boundary layer can be seen in Fig. 3.2. The elements used for generating the suburban boundary layer consist of spires, blocks, and chains. Spires are
triangle-shaped protrusions in the flow which cause more perturbation at the bottom than the top. There were 4 spires located at 15.24m upstream of the model center. The spires are 0.23m wide at the bottom and 1.37m high, and the tips of each spire are spaced 0.5m apart from the wall and each other. The blocks are 0.076m tall with a square cross section (0.038 × 0.038m) and there is a 0.38m space between blocks in all directions. The blocks were arranged in 30 rows starting 12.89m away from the center of the model and ending 1.46m from the center of the model. The odd rows of blocks started 0.076m from the side of the tunnel. The even rows of blocks started 0.28m from the side of the tunnel. The chains were used to maintain the turbulent flow in the flow in the region between the model and the last row of blocks. The chain is 0.0125m thick. It was laid in rows starting 1m from the center of the model and ending 0.24m from the center of the model. The chain was arranged in three rows spaced 0.38m apart.

The mean velocity and the turbulence intensity profiles at the center line of the wind tunnel where the model was located were measured using a Cobra probe and are shown in Fig. 3.3. The horizontal axis represents non-dimensional averaged wind velocity \( U/U_r \) in Fig. 3.3 (a), where \( U_r \) is the reference velocity \( U_r = 14.3 \text{m/s} \) at a height of 1m. In Fig. 3.3 (b), the horizontal axis represents the turbulence intensity \( U_{RMS}/U \). The wind speed increases in an atmospheric boundary layer with height above the ground. It is known that the streamwise wind profile in nature follows a power law as given below.

\[
\frac{U(z_1)}{U(z_2)} = \left( \frac{z_1}{z_2} \right)^\alpha
\]  

(3.1)

The power law exponent \( \alpha \) for a suburban boundary layer can range from approximately 0.2 to 0.3. The power law exponent of the fit curve for the suburban boundary layer used in
this study was 0.22. The roughness length ($z_0$) of the suburban boundary layer used in the experiment was 0.00215 ft. above the ground plane. The turbulence intensity was 26% at 4 in (33 ft. in full-scale equivalent height by using a length scale ratio of 1:100).

3.2.1.2. Force measurements in straight-line winds

During the experiments, the aerodynamic force coefficients in all three direction

\[ CF_x = F_x \left( \frac{1}{2} \rho V_0^2 A_1 \right), \quad CF_y = F_y \left( \frac{1}{2} \rho V_0^2 A_1 \right), \quad \text{and} \quad CF_z = F_z \left( \frac{1}{2} \rho V_0^2 A_1 \right) \]

and moment coefficients

\[ CM_x = M_x \left( \frac{1}{2} \rho V_0^2 A_2 H \right), \quad CM_y = M_y \left( \frac{1}{2} \rho V_0^2 A_2 H \right), \quad \text{and} \quad CM_z = M_z \left( \frac{1}{2} \rho V_0^2 A_2 H \right) \]

corresponding to the forces ($F_x$, $F_y$, $F_z$) and moments ($M_x$, $M_y$, $M_z$) acting on the building model were measured by using a highly sensitive force-moment sensor (JR3, model 30E12A-I40) as shown in Fig. 3.4. In the calculation of force and moment coefficients, $A_1$ represents the area of the side wall (i.e. $A_1=34.4\text{mm} \times 140\text{mm}$); $A_2$ represents the area of the square cross section (i.e. $A_2=34.4\text{mm} \times 34.4\text{mm}$); $H$ represents the height of the test model; $V_0$ represents the streamwise velocity selected at the half height of the test models in straight-line winds, and represents the maximum tangential velocity at the same height in tornado-line winds. The JR3 load cell is composed of foil strain gage bridges, which are capable of measuring the forces on three orthogonal axes, and the moment (torque) about each axis. The load it can measure in X and Y directions ranges from 0–100 lb, and 0–400 lb in Z direction. The moment can be measured in X and Y directions with a range of 0–250 in-lb, and in the Z – direction with a range of 0–200 in-lb. The precision of the force-moment sensor cell for force measurements is ±1.0 g. The force and moment data was scanned at 1000 Hz for 60 seconds.
Figure 3.4. A picture of the test model and JR3 load cell

Figure 3.5. Wind loads acting on the test model vs. orientation angles

Figure 3.5 (a, b) shows the averaged wind loads (aerodynamic force and moment coefficients) as functions of the orientation angle in straight-line wind ($CF_{x,y,z}(S)$, $CM_{x,y,z}(S)$, $S$ represents straight line wind). When the test model was placed in straight-line winds, the maximum wind load in the flow direction (X-direction) was found to be reached at the orientation angle of 45°, as was expected. The curve of the moment $CM_y$ corresponding to the
force $F_x$ with varied orientation angles showed a similar trend as that $CF_x$. The maximum $CM_y$ was found at OA=45°. The maximum wind load in the Y-direction was found at about 22.5° and 67.5° with opposite signs. As expected, the $CF_y$ reverses the sign of value when passing the orientation angle of 45° because of the symmetric geometry of the test model. Compared to the curve of $CF_y$, the moment in the X – direction showed a reversed trend symmetric about the turning point of OA = 45°. Wind lift coefficients in the Z – direction have an average value of about 1.0 with very little variation. The torque moment coefficients in the Z – direction was very small value and can be neglected in present study.

3.2.1.3. PIV measurements around a high-rise building model in straight-line winds

The same high-resolution PIV system as the one used in the study of tornado-like wind was utilized to measure the flow field around the high-rise building model in a horizontal plane ($Z=70\text{mm}$), which is the middle-height plane of the test model. The PIV setup is very similar to the one shown in Fig. 2.2. The thickness of the laser sheet in the measurement region was about 1.5mm.

![Figure 3.6. PIV measurements around the test model at Z=70mm in straight-line wind](image-url)
As shown in Fig. 3.6, the symmetric flow pattern is clearly demonstrated both in the instantaneous and ensemble-averaged flow field. From the instantaneous flow field, the velocity vectors along with colorful vorticity contours reveal that the air flow separates at the two windward corners of the test model. This separation developed along the flow direction and finally formed two relatively big circulation vortex regions at the leeward side of the test model. This flow pattern was also captured by the ensemble-averaged streamlines with streamwise velocity depicted as colorful contours. The low streamwise velocity regions were symmetrically shown in the wake region in the leeward side.

3.2.2. Wind Loads acting on a High-rise Building Model in Tornado-like Winds

3.2.2.1. Effects of distance between the tornado-like vortex and the building model

Figure 3.7 shows the profiles of the measured wind loads (both force and moment coefficients) acting on the tested high-rise building model as a function of the distance between the centers of the tornado-like vortex and the test model. During the experiments, the orientation angle (OA) of the building model related to the tornado-like vortex was set to 0.0 deg. (OA=0.0 deg.). As revealed in Fig. 3.7, since the X, Y and Z components of the measured aerodynamic forces are positive, it indicates that the tornado-like wind tends to push the test model tangentially, pull the test model towards the vortex core, and lift the model up from the test ground. Such motions are the most common damage patterns of the damaged buildings observed in the aftermath of tornado attacks in nature [12].
Figure 3.7. Wind loads vs. the distance between the test model and the tornado-like vortex

The force measurement results given in Fig. 3.7 also reveal that all the components of the aerodynamic force acting on the building model increase rapidly as the vortex core moves away from the center of the building model and reach their peak values at the location of \( R/R_O \approx 1.0 \). This indicates that the building model would experience maximum aerodynamic forces if it is mounted at the outer boundary of the core of the tornado-like vortex, where the velocity of the flow approaching the building model was found to be the maximum. As revealed from the PIV measurement results given in Fig. 2.5 and Fig. 2.7 in Chapter 2, since the tangential components of the flow velocity vectors are dominant in the inner core region of the tornado-like vortex, the X-component of the aerodynamic force, which is along the tangential direction of the flow streams approaching the test model, was found to be much more significant compared to the other two components. The maximum value of the X-component of the force is about 4.0 times greater than the other two components (Y and Z components) at \( R/R_O \approx 1.0 \). The roof uplift component (Z-component) was found to be slightly bigger than the inward pushing force (Y-component) in the region close to the core of the tornado-like vortex (\( R/R_O < 3.5 \)).
When the building model was mounted in the outer region of the tornado-like vortex core (i.e., $R/R_O > 1.0$), all three components of the aerodynamic force acting on the test model decreased with increasing radial distance from the center of the vortex. The Y-component of the aerodynamic force (the component pushing the test model towards the vortex core) increased slightly with increasing distance when $R/R_O > 2.5$. The slight increase of the inward pushing force acting on the building model is believed to be closely related to the spiral motion of the flow field in the outer region of the tornado-like vortex. As revealed from the PIV measurements shown in Fig. 2.7 and Fig. 2.8 in Chapter 2, the radial components of the flow velocity vectors increased rapidly in the outer region away from the tornado-like vortex core, which resulted in the generation of a strong spiral motion sucking surrounding air flow towards the vortex core. As a result, the inward pushing component of the aerodynamic force (Y-component) increased slightly with increasing distance from the vortex core and became dominant in the region farther away from the vortex core ($R/R_O > 4.5$).

The negative sign of the X-component of the measured moment coefficient, $CM_X$, indicates that the building model would likely bend towards the core due to the wind loads induced by tornado-like winds. The positive sign of the Y-component of the measured moment coefficient, $CM_Y$, indicates that the building model would bend towards the tangential flow direction, as expected. Such collapsed patterns are consistent with those of the damaged buildings in the aftermath of tornado attacks. The magnitude of the Z-component of the moment coefficient, $CM_Z$, was much smaller than other two components, and is considered negligible. Similar to the measured force coefficients, the measured moment coefficients reached their peaks at the location of $R/R_O \approx 1.0$. Such measurements indicate that buildings
are most likely to fail when they are located near the outer boundary of the tornado core (i.e. \( R/R_O \approx 1.0 \)), where the winds reach maximum speed.

### 3.2.2.2 The effects of the orientation angle of the test model in tornado-like winds

The effects of the orientation on the characteristics of the wake vortices and flow structures around the test model as well as the resultant wind loads (force and moment) were also investigated in the present study. The definition of the orientation angle (OA) of the model related to the tornado-like vortex is given schematically in Fig. 3.1. During the experiments, the tested building model was mounted at the location of \( R/R_O \approx 1.0 \).

![Figure 3.8. The measured wind loads vs. orientation angles](image)

(a) force coefficient  
(b) Moment coefficient

Figure 3.8 shows the measured wind loads (aerodynamic force and moment coefficients) as the functions of the orientation angles in tornado-like wind compared with the wind loads in straight-line winds. For the same test model used in the present study, when it was placed in a straight-line wind, the maximum wind force in the flow direction, \( CF_{xmax} \), and corresponding moment coefficient, \( CM_{ymax} \), was found at the orientation angle OA= 45 degrees. This is explained by the maximum blockage of the test model in the flow direction. However, when the square-shaped test model was mounted in tornado-like wind, the wind
load measurement data shown in Fig. 3.8 revealed that the maximum wind loads (both force and moment) acting on the test model were found to be at OA < 45.0 degrees, instead of OA = 45.0 degrees obtained from the case in straight-line winds.

We believe that the reason why the maximum wind loads (i.e. $CF_x$ and $CM_y$) acting on the model in tornado-like wind occurring at OA < 45.0° instead of OA = 45.0° is closely related to the spiral motion of a tornado-like vortex. As shown schematically in Fig. 3.9, with the test model mounted in tornado-like vortex, the velocity vectors of flow streams approaching the test model also have inward radial components besides the tangential velocity components along the X-axis direction. As a result, the direction of the resultant flow velocity vector approaching the test model would be tilted slightly inwards to the vortex core center. As it is shown in straight-line wind, maximum wind loads (aerodynamic forces and moments) acting on the test model has been reached when the test model blocks a maximum approaching flow. Therefore, the maximum wind loads acting on the test model in tornado-like winds were at OA ≈ 30° - 45°, instead of OA = 45.0°.
3.2.3. Flow Structure around a High-rise Building Model in Tornado-like Winds

3.2.3.1. Effects of distance between the tornado-like vortex and the building model

With the findings derived from the force and moment measurements in mind, PIV measurements were carried out to visualize the evolution of the wake vortex and flow structures around the high-rise building model in order to elucidate underlying physics to improve our understanding about the characteristics of the wind loads acting on the test model in tornado-like winds. Fig. 3.10 shows the time-averaged results of the PIV measurements (the velocity distributions and the corresponding streamlines around the test model) with the tornado-like vortex positioned at different distances relative to the center of the test model. For the PIV measurements given in Fig. 3.10, the measurement plane was selected as an X-Y plane at 70mm above the ground plane (i.e. Z=70mm), which is also the middle plane of the high-rise building model.

Compared to those in straight-line winds, as shown in Fig. 3.6, the wake vortices and flow structures around the building model became much more asymmetrical and complicated in the tornado-like wind. As visualized in Fig. 3.10(a), while a large wake vortex formed at the leeward side of the building model, the streamlines outside the wake region were still found to be concentric circles when the tested building model was mounted near the center of tornado-like vortex core ($R/R_o \approx 0.0$). Such distribution of streamlines suggests that the wake structures behind the building model would stay locally within the core of the vortex. Since the local wind speed in the core of the tornado-like vortex is low, the aerodynamic forces and moments acting on the building model would be relatively small when the model is located in the core region of the tornado-like vortex, and is confirmed by the wind load measurement results shown in Fig. 3.7.
Figure 3.10. Flow field around the test model vs. the distance between the centers of the test model and the tornado-like vortex

Fig. 3.10 (b) shows the wake vortex and flow structures around the test model when the test model was moved a distance of $R/R_O \approx 0.5$ away from the center of the tornado-like
vortex (the building model is still within the core region of the tornado-like vortex). Two strong wake vortex structures were found to be generated at the leeward side of the model with the outer vortex being stronger than the inner one. Concentric circular streamlines can still be found in the regions slightly away from the model. Since the local wind speed becomes much higher than that at the center of the tornado-like vortex core, aerodynamic forces and moments acting on the building model increase rapidly, which were also confirmed by aerodynamic force and moment coefficients data shown in Figure 3.7.

Fig. 3.10(c) shows the streamlines of the flow field around the tested building model when the test model was mounted at the outer boundary of the tornado-like vortex core ($R/R_O \approx 1.0$). As described above, this is the position where maximum wind speed was observed. It is also the location where maximum wind loads acting on the test building model were measured. As visualized by the streamlines, the wake vortex structures in the leeward side of the model became much larger and stronger because of the high local wind speed, which is different from the symmetric wake vortex structure as found in straight-line wind, shown in Fig. 3.6. The wake vortex structures and the streamlines at the outer side of the tested building model leaned slightly towards the inner core of the tornado-like vortex due to the spiral motion in the outer region of the tornado-like vortex.

Fig. 3.10 (d), Fig. 3.10 (e) and Fig. 3.10(f) show the time-averaged flow field around the tested high-rise building model when it was mounted in the outer region of the tornado-like vortex ($R/R_O \approx 2.3$, $R/R_O \approx 4.5$ and $R/R_O \approx 6.2$). Depicted in the figures, the wake vortex structures at the leeward side of the model were elongated significantly. Because of the spiral motion in the outer region of the tornado-like vortex, all the streamlines tilted towards the inner core of tornado-like vortex. It should be noted that, although the magnitude of local
wind speed became smaller and smaller as the radial distance increased from the center of the tornado-like vortex, the Y-components (inward components) of the velocity vectors became more and more significant. As a result, the Y-components of the aerodynamic forces and resultant moments (X-component of the moment) increased slightly with increasing distance away from the tornado-like vortex core, as revealed from the wind load measurement results shown in Fig. 3.7.

3.2.3.2. Effects of the orientation angle of the test model in tornado-like winds

The PIV measurement results given in Fig. 3.11 elucidate more details about the variations of the wake vortex and flow structures around the test model with the orientation angles of the test model relative to the tornado-like vortex, which might be used to explain the characteristics of the wind loads induced by tornado-like winds. As visualized clearly from the flow velocity distributions around the test model, a pair of wake vortex structures formed in the leeward side of the test model. The size of the wake region at the leeward side of the test model increased with an increasing orientation angle (OA) until it reached a maximum value at $\text{OA} \approx 30^\circ \sim 45^\circ$. Such flow patterns in the wake of the test model indicate increasing aerodynamic forces acting on the test model, which was confirmed by the wind load measurement data shown in Fig. 3.8. The wake region at the leeward side of the test model decreased with increasing orientation angle when the orientation angle exceeded $45^\circ$ (i.e., $\text{OA} > 45^\circ$). This would indicate decreasing wind loads acting on the test model and was confirmed from the wind load measurement results shown in Fig. 3.8.
Figure 3.11. Flow field around the test model vs. orientation angle at $R/R_0=1$
3.2.3.3. The vortex and flow structures around the test model at different elevations

During the experiments, PIV measurements were also conducted at different horizontal cross planes at different elevation heights in order to show the flow structure more three-dimensionally. Figure 3.12 shows ensemble averaged PIV measurement results (based on 330 pairs of PIV images) with streamlines and wind speed contours at different elevation heights with the test model mounted at $R/R_O = 1.0$. During the experiments, the orientation angle (OA) of the building model relative to the tornado-like vortex was set fixed at OA=0º.

As revealed clearly from the PIV measurements results, for the vortex structures in the leeward side of the model, the wake vortex on the inner side (the one closer to the tornado vortex core) seems to be dominant when the measurement planes were near the bottom of the test model (closer to the ground plane). As the elevation height increases, the wake vortex structures on the outer side (the one farther away from the tornado vortex core) grew rapidly and became dominant. The measurement results also show that the streamlines downstream of the recirculation region in the wake of the test model were tilted more toward the center of the tornado-like vortex as the elevation height of the measurement plane increased. This may indicate that the strength of the spiral motion of the tornado vortex becomes stronger and stronger with increasing elevations.
Figure 3.12. Flow field around the test model at different elevation levels at $R/R_0=1$

Based on the PIV measurements at different elevation planes, the flow structures around the test model in tornado-like winds were reconstructed. Fig. 3.13 shows a stereoscopic view
of the reconstructed wake vortex and flow structures around the test model in tornado-like winds in terms of iso-surfaces of local swirling strength [28, 36]. The 3-D features of the wake vortex and flow structures around the test model in the tornado-like vortex can be visualized clearly from the 3-D view of the reconstructed flow field.

![Figure 3.13](image)

Figure 3.13. Three-dimensional view of the wake vortex structures around the model at \( R/R_0 = 1 \) with \( OA = 0 \) deg

### 3.3. Flow around a Low-rise Gable-roofed Building Model in Tornado-like Winds

#### 3.3.1. Wind Loads and PIV Measurements in Straight-line Winds

In order to compare the wind loads acting on the gable-roofed building model in tornado-like wind with that in straight-line wind, the same test model was mounted in a AABL straight-line wind tunnel (as mentioned in section 3.2) to conduct the force and PIV measurements.
3.3.1.1. Studied models and experimental setups

Figure 3.14 shows the schematic of the gable-roofed house models used in the present study. This 1/100 scaled building structure is modeled from a general low-rise building prototype. Two identical models were made for the present study. The first house model is equipped with 89 pressure taps all around the surface. The locations of the pressure taps are indicated in Figure 3.15. There are some port numbers beginning with A and B. This labeling implies that the pressure values at those locations were obtained by using interpolation. The 89 pressure taps were connected by plastic tubing to 89 channels of a pressure acquisition system (Scanvalve zoc33/64 Px electronic pressure scanner). Each ZOC33 module incorporates 64 individual silicon pressure sensors, a calibration unit, a high speed multiplexer (50 kHz), and an instrumentation amplifier. The precision of the pressure acquisition system is ± 0.15% of the full scale (± 10 inch H₂O). During the experiment, each pressure transducer input was scanned at 430 Hz for 48 seconds. The pressure coefficient was calculated as $C_p = \frac{p - p_\infty}{\frac{1}{2} \rho V_0^2}$, where $p$ is the measured pressure from each tap, $p_\infty$ is the atmospheric pressure in the laboratory, $V_0$ is the maximum velocity found at $R/R_0$. The force and moment coefficients acting on the house model were measured by using a force-moment sensor cell, as mentioned in section 3.2. Fig. 3.14(b) shows the relative positions of the tornado-like vortex center with respect to the center of building model and the building orientation angle (OA), where $R$ is the distance between the center of the tornado-like vortex and the center of the building model. It also shows the orientation angle definition in straight-line wind.
Figure 3.14. Test model and definition of parameters

Figure 3.15. Pressure tap distribution

The second low-rise gable-roofed building model, which is made of transparent Plexiglas and has identical dimensions, was used to conduct PIV measurements. Figure 3.16 shows the schematic of the experimental setup used for the PIV measurement. During the experiment, the transparent house model was installed on a transparent ground plate. A high-resolution PIV system was used to make detailed flow field measurements at different horizontal (X – Y
plane) and vertical measurement planes (X – Z plane) crossing the house model. The description of the PIV system can be found in Chapter 2.

![PIV experimental setup diagram]

**Figure 3.16.** PIV experimental setup

### 3.3.1.2. Force Measurements in straight-line winds

During the experiments, the aerodynamic force coefficients in all three directions

\[
(CF_x = F_x / (\frac{1}{2} \rho V_o^2 A_1), \quad CF_y = F_y / (\frac{1}{2} \rho V_o^2 A_1) \quad \text{and} \quad CF_z = F_z / (\frac{1}{2} \rho V_o^2 A_1))
\]

and moment coefficients

\[
(CM_x = M_x / (\frac{1}{2} \rho V_o^2 A_1 H), \quad CM_y = M_y / (\frac{1}{2} \rho V_o^2 A_1 H) \quad \text{and} \quad CM_z = M_z / (\frac{1}{2} \rho V_o^2 A_1 H))
\]

corresponding to the forces \((F_x, F_y, F_z)\) and moments \((M_x, M_y, M_z)\) acting on the building model were measured by using the same high-sensitivity force-moment sensor as mentioned before. In the calculation of the force and moment coefficients, the area parameter is fixed in order to compare forces. \(A_1\) is the area of the lateral wall perpendicular to the Y axis as shown in Fig.
3.14 (a) (i.e. \(A_1=90\text{mm} \times 30\text{mm} + 0.5 \times 90\text{mm} \times 30\text{mm}\)); \(A_2\) is the area of the square cross section (i.e. \(A_2=90\text{mm} \times 90\text{mm}\)); \(H=90\text{mm}\) is the width of the test model.

As shown in Fig. 3.17, the measured wind loads (i.e., aerodynamic force and moment coefficients) were plotted as the functions of the orientation angle in straight-line wind \((C_{Fx,y,z}(S), C_{My}(S))\). When the test model was placed in the straight-line wind with the velocity profile and turbulence intensity profile shown in Fig. 3.16, the maximum wind load in the flow direction (X-direction) was found to be reached at the orientation angle \(OA=52.5^\circ\), which differs from the \(OA=45^\circ\) for the high-rise building model with a uniform square cross section as studied in section 3.2. The curve of the moment \(C_{My}\) corresponding to the force \(F_x\) with varied orientation angles showed a similar trend as that of \(C_{Fx}\). The maximum \(C_{My}\) was found at \(OA=52.5^\circ\). The maximum wind load value in the Y- direction was found at about \(30^\circ\) and \(75^\circ\) with opposite signs. As seen from the curve, \(C_{Fy}\) reverses the sign of value when passing the orientation angle of \(52.5^\circ\) where the maximum wind load in the X – direction appeared. Correspondingly, the curve of moment \(C_{Mx}\) showed a reversed trend compared to that of \(C_{Fy}\) with the same turning point at \(OA=52.5^\circ\). The wind lift coefficient in the Z – direction did not vary much when \(OA\) is less than \(22.5^\circ\), but it decreased drastically.
with increased orientation angles. As demonstrated in Fig. 3.14(b), a smaller orientation angle corresponds to the more roof surface area facing the incoming flow. It indicates that the existence of roof has a significant effect on the generation of wind lift. The torque moment coefficient in the Z – direction, with a pretty small value, was neglected in present study.

3.3.1.3. PIV Measurements around a Gable-roofed Building Model in Straight-line Winds

The high-resolution PIV system was the same system used in previous study and was utilized to measure the flow field around the gable-roofed building model in a horizontal plane (Z=15mm), which is the middle-height plane of the short wall, and a central vertical plane (Y=0mm). The PIV system setup has been shown in Fig. 3.16. The thickness of the laser sheet in the measurement region was about 1.5mm.

As revealed from the PIV measurements results on the central vertical plane in figure 3.18, the vortex structures evolution was captured in both instantaneous flow and ensemble-averaged flow measurements. The flow separation starts at the top roof edge and forms a big circulation flow structure at the leeward side of the test model. Although it is not very clear,
the other vortex flow structure shown in Fig. 3.18 (a) was found to be formed at the windward side of the model near ground, which is commonly known as the horseshoe vortex. Since the vertical wall in the windward side is relatively short and the turbulence intensity is high (about 30%), the horseshoe vortex is not very apparent in the averaged streamlines.

As shown in Fig. 3.19, the symmetric flow pattern was clearly demonstrated both in a typical instantaneous and the ensemble-averaged flow field. From the instantaneous flow field, the velocity vectors with contour of vorticity showed two separations at the two windward corners of the test model. The separation developed along the flow direction and finally formed two relatively big circulation vortex regions at the leeward side of the test model. This flow pattern was also captured by the ensemble-averaged streamlines with streamwise velocity as contours. The low streamwise velocity was symmetrically shown in the wake region in the leeward side.

![Figure 3.19. PIV measurements on a horizontal plane (Z=15mm) in straight-line wind](image)

a. Instantaneous flow field  
b. Averaged flow field with streamlines

Figure 3.19. PIV measurements on a horizontal plane (Z=15mm) in straight-line wind
3.3.2. Wind Loads and Pressure acting on a Gable-roofed Building Model in Tornado-like Winds

3.3.2.1. Effects of distance between the tornado-like vortex and the building model

Figure 3.20 shows the profiles of the measured wind loads (both force and moment coefficients) acting on the tested high-rise building model as a function of the distance between the centers of the tornado-like vortex and the test model. During the experiments, the orientation angle (OA) of the building model related to the tornado-like vortex was set to 0.0 deg. (i.e., OA=0.0 deg.). As revealed in Fig. 3.20, the X, Y and Z components of the measured aerodynamic forces are positive, which indicates the same wind load patterns as those found in study of high-rise building model.

The force measurement results given in Fig. 3.20 also revealed that all the components of the aerodynamic forces acting on the building model would increase rapidly as the vortex core moves away from the center of the building model and reach their peak values at the location of \( \frac{R}{R_O} \approx 1.0 \), except the uplift force in the Z – direction surprisingly. It has been discussed in section 3.2 that the building model would experience maximum aerodynamic forces if it is mounted at the outer boundary of the core of the tornado-like vortex, where the velocity of the flow approaching the building model was found to be a maximum. However, the maximum uplift force coefficient was found at about \( \frac{R}{R_O} \approx 0.5 \) for the gable-roofed building model. This might be attributed to the combination of the impacts of relatively high-speed wind and the pressure gradient near the core in the vertical direction. As shown in Fig. 2.6, the air flow turning abruptly upward before it reaches the core indicates that there must be a significant pressure gradient in the vertical direction. Although the maximum velocity was reached at \( \frac{R}{R_O} \approx 1.0 \), which is accounted for the maximum value for X – component
and Y-component of wind forces, the z-component of wind force does not necessarily reach its maximum value at the same location because the pressure gradient might contribute less than the one at $R/R_O \approx 0.5$ in the final generation of the wind load. This is also confirmed by the pressure measurements given in Fig. 3.21.

Unlike the wind load measurement results on the high-rise building model, the uplift acting on the gable-roofed building model was much more significant compared to other two components. Its maximum value at $R/R_O \approx 0.5$ was about 2.0 times greater than the maximum values of the other two components (X and Y components) at $R/R_O \approx 1.0$. The X-component in the tangential flow direction was slightly bigger than the inward pushing force (i.e., Y-component) in the region close to the core region of the tornado-like vortex ($R/R_O < 3.5$). As the building model was mounted in the outer region of the tornado-like vortex core (i.e., $R/R_O > 1.0$), all three components of the aerodynamic force acting on the test model decreased rapidly with the increasing radial distance away from the center of the vortex. The wind loads became insignificant three times the core radius, $R_0$, away from the tornado-like vortex center.
The negative sign of the X-component of the measured moment coefficient, $CM_X$, indicates that the building model would likely bend towards the tornado-like vortex core due to the wind loads induced by tornado-like winds. The positive sign of the Y-component of the measured moment coefficient, $CM_Y$, indicates that the building model would bend towards the tangential flow direction, as expected. Such collapsed patterns are found to agree with those of the damaged buildings observed in the aftermath of tornado attacks in nature. The magnitude of the Z-component of the moment coefficient, $CM_Z$, was found to be much smaller than the other two components, and is considered negligible. Similar to the measured force coefficients in the X and Y directions, the measured moment coefficients were also found to reach their peaks at the location of $R/R_O \approx 1.0$. Such measurements indicate that buildings would most likely fail when they are located near the outer boundary of the tornado core (i.e. $R/R_O \approx 1.0$), where the wind reaches its maximum speed.

For a gable-roof building model placed in a straight line wind, Holmes [37] reported a symmetric pattern for the surface pressure distribution around the gable-roof building model, which differs from the present measurement results as shown in Figure 3.21 significantly. From the measured surface pressure coefficient distributions given in Fig. 3.21, it can be clearly seen that the surface pressure distribution around the gable-roof building model is not symmetric about the central plane ($Z=0$ cross plane) of the building model due to the swirling motion of the approaching tornado-like wind. The measured surface pressures on the wall and roof at the windward side were found to be higher than those on the wall and roof at the leeward side, which results in the pushing force which acts on the test model toward flow direction. The lower pressures on the roof and the wall leeward are due to the occurrence of the large-scale flow separation in these areas, which can be seen clearly in the PIV
measurements shown in Fig. 3.23. The pressure distributions on the walls and roofs in the outer-side regions (i.e., regions farther away from the tornado-like vortex core) were found to be high compared to those in the inward side region (corresponding to the vortex center). The unbalanced pressure distribution on the inward side and the outer-side wall results in a net force acting on the building model, which pulls the building model towards the core of the tornado-like vortex, as it is found in the force measurements. All pressure values on the roofs were negative. Such distribution on the roofs is consistent with the well-known fact of that there exists a significant low-pressure suction region near the core of a tornado-like vortex.

Figure 3.21(a) shows the pressure distribution on the surface of the test model at $R/R_O \approx 0.5$. The maximum pressure coefficient value of -1.6 was found at the wall on the windward side because the incoming flow impacts on the wall directly. In a straight-line flow, the pressure at the stagnation point on the wall on the windward side should be positive, but it is not the case in tornado-like wind. Negative pressure values were found on the windward side, because there was considerable negative pressure distribution in core tornado-like vortex. A minimum pressure coefficient value of -4.2 was found at the roof in the leeward side due to the significant separation vortex near the roof at leeward side.
(a) $R/R_o \approx 0.5$

(b) $R/R_o \approx 1.0$
Figure 3.21. Measured typical pressure distributions around the test model

Compared to the pressure distribution at $R/R_O \approx 0.5$, as shown in Figure 3.21 (b), the pressure coefficients rose up at all corresponding positions on the surface of the test model when the model was mounted at $R/R_O \approx 1.0$. The maximum pressure coefficient value of -0.2 was still found at the wall on the windward side. A minimum pressure coefficient value of -3.4 was found at both the roof and lateral wall on the leeward side.

Figure 3.21(c) shows the pressure distribution on the surface of the test model at $R/R_O \approx 2.0$, which is the outside of the tornado core region. Compared to the two pressure distributions above, the pressure coefficient value became much larger. Although the pattern about the positions where high or low pressure appeared did not change, the pressure difference between the high-pressure and the low-pressure regions tended to be trivial with the increasing distance between the tornado-like vortex center and the test model.
3.3.2.2. The effects of the orientation angle of the test model in tornado-like winds

The effects of the orientation angle of the test building model related to the tornado-like vortex on wind loads (force and moment) acting on the test model were investigated in the present study. The definition of the orientation angle (OA) of the model related to the tornado-like vortex is given schematically in Fig. 3.14. During the experiments, the test model was mounted at the location of $R/R_O \approx 1.0$.

Figure 3.22 (a, b) shows the measured wind loads (i.e., aerodynamic force and moment coefficients) as functions of the orientation angle in tornado like wind. The maximum wind load in the tangential flow direction (X-direction) and the corresponding moment $CM_y$ were found to be at OA $\approx 30.0^\circ$~$45.0^\circ$, which is less than OA $= 52.5^\circ$ found in straight-line wind. This discrepancy agrees with results found in the study of the high-rise building that the maximum wind loads ($CF_x$ and $CM_y$) acting on the high-rise building model occurs at OA $< 45^\circ$ in tornado-like winds instead of OA $= 45^\circ$ in straight-line winds, which has been attributed to the spiral motion of the tornado-like vortex. More details about this explanation can be found in section 3.2.

The force in the Y-direction is always positive due to the radial velocity component in tornado-like wind. The maximum force in the Y-direction was found at OA $= 0$~$15$ degrees, which differs from that in straight-line wind. Correspondingly, the maximum moment coefficient in the X-direction was found at $0$~$15$ degrees. The uplift coefficients in tornado-like wind are about three times the corresponding values in straight-line winds, which agree with the previous results found by Jischke & Light [7] and Bienkiewicz & Dudhia [31]. The maximum uplift coefficient in tornado like wind appears at OA $= 15$~$30$ degrees. The lowest
force coefficients appeared at around 75 degrees in all three directions. The torsion moment in the Z-direction was a small and did not vary much with different orientation angles.

![Graph showing force and moment coefficients with orientation angles](image)

(a) force coefficient  (b) Moment coefficient

Figure 3.22. Wind loads vs. orientation angles

3.3.3. Flow Structure around a Gable-roofed Building Model in Tornado-like Winds

3.3.3.1. Effects of distance between the tornado-like vortex and the building model

With findings derived from the pressure and wind load measurements in mind, PIV measurements were carried out to visualize the evolution of the wake vortex and flow structures around the high-rise building model to elucidate underlying physics to improve our understanding about the characteristics of the wind loads acting on the test model in tornado-like winds. Fig. 3.23 shows the time-averaged results of PIV measurement results (i.e., the velocity distributions and the corresponding streamlines around the test model) with the tornado-like vortex positioned at different distances related to the center of the test model.

For these PIV measurements the vertical plane was chosen at Y=0, which is the central plane, and the horizontal plane was selected at Z=15mm, which is half the height of the wall excluding the roof.
Compared to those in straight-line winds [30], the wake vortices and flow structures around the building model were found to become much more complicated in the tornado-like wind. In the vertical plane, as shown in Fig. 3.23 (a1), two large vortices were found to be formed above the roofs of the test model, which was located in the core region of tornado-like vortex ($R/R_0 \approx 0$). Even though the velocity is pretty low, it is still visualized that flow streams near the ground move towards the model and turn upward abruptly before reaching the walls. Two large vortices were formed in the central region near the roofs when the downdraft jet met with the incoming flow from outer region. At the same location, an almost axisymmetric flow pattern on the horizontal plane ($Z=15\text{mm}$) was visualized in Fig. 3.23 (a2). Only one little vortex was found near the lateral wall at the positive-$Y$ side, which might be due to the non-perfect axisymmetric structure of the tornado simulator. Streamlines outside the wake region were still found to be a spiral flow pattern. Since the local wind speed in the core of the tornado-like vortex was low, the aerodynamic forces and moments acting on the building model were relatively small when the model was located in the core region of the tornado-like vortex, and is confirmed by the wind load measurement results shown in Fig. 3.20. The lift, however, is not small because of the corresponding low pressure in the core region and two wake vortices formed near the roofs.

Fig. 3.23 (b1, b2) show the wake vortex and flow structures in the vertical central plane and horizontal middle plane crossing the test model when the model was moved away from the center of the tornado-like vortex by a distance of $R/R_0 \approx 0.5$ (i.e., the building model is still within the core region of the tornado-like vortex). One wake vortex structure was found to be generated in the vertical plane close to the leeward roof of the model. The upward component of velocity tended to lead the streamlines in the whole field to tilt up. Two wake
vortex structures were generated in the horizontal plane near the wall at inward side (corresponding to the vortex center) and the wall at the leeward side with the inward vortex being stronger than the leeward one. Since the local wind speed becomes much higher than that at the center of the tornado-like vortex core, aerodynamic forces and moments acting on the building model increase rapidly, and is confirmed by the measured pressure distribution shown in Fig. 3.21 as well as the aerodynamic force and moment coefficient data shown in Fig. 3.20. The maximum lift value was found at about this distance.

Fig. 3.23 (c1, c2) show the streamlines of the flow field around the tested building model when the test model was mounted at the outer boundary of the tornado-like vortex core (i.e., \( R/R_0 \approx 1.0 \)). As described above, this is the position where maximum wind speed was observed at the center of the model. The maximum wind loads, except the uplift, were also found at this location. Correspondingly, the large high-speed region appeared above the top of the test model in the vertical measurement plane. Two vortices were found in the vertical measurement plane. One was generated near the leeward side of the roof due to the separation there, which corresponded to a low pressure region at the leeward roof, and is confirmed by the pressure measurement shown in Fig. 3.21. The other was generated at the far leeward side near ground. The streamlines on the horizontal plane demonstrated only one large vortex structure instead of two, as one might expect to see at the leeward side of the test model. This is because the wind speed at the outer side is much higher than the inward side. Air flow at inward side has no chance to turn at the corner into the leeward region. The incoming streamlines are almost perpendicular to the windward wall of the test model and then turn a right angle toward the inward side after passing the model induced by the spiral flow feature of the tornado-like vortex.
Figure 3.23 (d1, d2) shows the time-averaged flow field around the test building model when it was mounted in the outer region of the tornado-like vortex (i.e., $R/R_O \approx 2.0$). Similar to the case where $R/R_O = 1$ in the vertical measurement plane, one vortex structure was found to be formed at the leeward roof. The other one forming at the far leeward side was not that apparent. In the horizontal plane, with the increased radial component of velocity, the streamlines show more spiral pattern (i.e. streamlines tilt up). As a result, two vortex structures were formed at the wall leeward and the wall inward respectively.

The time-averaged flow fields around the test model for $R/R_O \approx 4.6$ are shown in Fig. 3.23 (e1, e2). As visualized in the streamlines in the vertical measurement plane, the separation vortex structure for this case differed from other cases. This discrepancy may be attributed to low Reynolds number effects. For a low Reynolds number flow, the air flow is more readily to separate when passing over the roof of test model. The separation region, with low speed, became larger at leeward side. In the horizontal plane, as visualized in the figures, the wake vortex structures at the leeward side of the model were found to be elongated significantly because of the spiral motion in the outer region of the tornado-like vortex. As the magnitude of the local wind speed was found to become smaller as the radial distance increased from the center of the tornado-like vortex, the aerodynamic forces and resultant moments were found to decrease slightly with the increasing distance away from the tornado-like vortex core, and were revealed from the wind load measurement results shown in Fig. 3.20.
a1.  
Streamwise Velocity m/s  
1:100 Gable-roofed building model  

a2.  
Streamwise Velocity m/s  
1:100 Gable-roofed building model  

b1.  
Streamwise Velocity m/s  
1:100 Gable-roofed building model  

b2.  
Streamwise Velocity m/s  
1:100 Gable-roofed building model  

c1.  
Streamwise Velocity m/s  
1:100 Gable-roofed building model  

c2.  
Streamwise Velocity m/s  
1:100 Gable-roofed building model  

d1.  
Streamwise Velocity m/s  
1:100 Gable-roofed building model  

d2.  
Streamwise Velocity m/s  
1:100 Gable-roofed building model  

3.3.3.2. The effects of the orientation angle of the test model in tornado-like winds

The PIV measurement results given in Figure 3.24 elucidate more details about the variations of the wake vortex and flow structures around the test model with different orientation angles, which might be used to explain the characteristics of the wind loads induced by tornado-like winds. It can be assumed that all wind loads were induced by the impacting of air flow on the test model. Taking the flow field boundary as a control volume boundary in a 2-D plane, the momentum difference in the Y direction would result in the force in the Y direction. Assuming that the Y-direction flow comes in from the bottom boundary with the same velocity distribution, then the velocity of the flow going out from the top boundary could indicate the momentum change to some extent. For OA=0deg, the velocity of the flow leaving the control volume from the top boundary is relatively low compared with other orientation angles, which means this momentum change would lead to a large force in the Y-direction acting on the test model. This analysis was confirmed with the force measurements in Fig. 3.22 (a). For OA = 15deg, the largest wake region was shown near the sidewall at the inward side. Correspondingly, the maximum force coefficients in the Y and Z directions and the momentum coefficient in the X-direction were also found at this angle.
Figure 3.24. Flow field around the test model vs. the orientation angles
Even though the relationship between wind load measurements and flow measurements in one plane is not straightforward, one can still get some clues to understand the wind loads acting on the test model. The wake pattern for OA = 30deg, OA = 45deg, OA = 60deg and OA = 75deg are very similar. The area of wake region for OA=45 deg was the smallest one and corresponded to the largest force in the X-direction as displayed in Fig. 3.22. Since the flow structure would be drastically complicated due to the existence of the roof, the averaged velocity field in only one horizontal plane cannot give a complete explanation of the force behavior acting on the test model. More comprehensive and detailed studies would be needed to elucidate the underlying physics of the orientation effect.

3.3.3.3. The vortex and flow structures around the test model

During the experiments, PIV measurements were also conducted at different cross planes parallel to the ground at different elevations and different vertical cross planes parallel to the lateral walls. Figure 3.25(a) shows typical PIV measurement results at different elevations and the central vertical plane with the test model mounted at $R/R_o = 1.0$ with the zero orientation angle (OA=0). As revealed clearly from the PIV measurements results, two main vortex flow structures were observed at the leeward roof and the leeward wall of the model, respectively. In the vertical plane, after the flow passed the roof top, the streamlines surrounding the vortex at the leeward roof sank down abruptly into the vortex near the leeward wall. It seems the flow retrieved in that vortex and blew downstream. The maximum velocity region in the vertical plane appeared above the top of the test model. In the horizontal plane, the streamlines surrounding the vortex structure turned a right angle toward the tornado core with a high speed after passing the leeward wall.
Figure 3.25. Three-dimensional flow field with streamlines and swirling strength contours in Y and Z directions
Based on the PIV measurements taken at different elevation planes and vertical planes, the flow structures around the test model in tornado-like winds were reconstructed. Fig. 3.25 (b, c) show the stereoscopic views of the reconstructed wake vortex and flow structures around the test model in tornado-like winds in the terms of iso-surfaces of local swirling strength [28, 36] in the Y and Z directions. The imaginary part of the complex eigenvalue pair of the velocity gradient tensor matrix was referred to as the local swirling strength of the vortex. The 3-D features of the wake vortex and the flow structures around the test model in the tornado-like vortex can be clearly visualized from the 3-D view of the reconstructed flow field.
CHAPTER 4. EXPERIMENTAL STUDY OF TRUBINE BLADE TRAILING EDGE COOLING

4.1. Introduction to Trailing Edge Cooling

Thermodynamic analysis reveals that thermal efficiency and power output of a gas turbine can be increased with higher turbine inlet temperatures. Advanced gas turbines are operated at peak turbine inlet temperature, which are well beyond the maximum endurable temperature for the blade material. As a result, hot gas-contacting turbine blades have to be cooled intensively by using various cooling techniques, such as internal convective cooling and film cooling on the blade exterior, in order to increase the fatigue lifespan of the turbine blades.

One of the most difficult regions where cooling techniques can be applied is the trailing edge, as shown in Figure 4.1, because of a combination of geometric constraints and aerodynamics demands. From an aerodynamic point of view the trailing edge should be designed as thin as possible in order to reduce aerodynamic pressure losses. This conflicts with the structural integrity of the blade and cooling design requirements as enough coolant cannot be channeled into the thin trailing edge. Most of the catastrophic turbine blade failures commonly originate at the edge – trailing edges, tips and roots. The thin trailing edges of turbine blades are prone to thermal damage. Common modes of failure include cracking, erosion, or simply melting. Oddly enough, even though trailing edge failure will increase maintenance load and severely restrict operating temperatures, trailing edge cooling has received little attention from the research community. In this study, we are particularly concerned with the trailing edge cooling of turbine blades.
One state-of-the-art cooling concept frequently used by turbine designers for trailing edge cooling is using pressure side cutback, where the pressure side wall of the trailing edge is shortened with respect to suction side [38]. Cooling air is ejected through spanwise slots in the trailing edge of turbine blade with slot jetting onto the cutback surface to provide thin film cooling. Overheating is generally a consequence of failure of the cooling streams to protect the blade surfaces. At a basic level, overheating is due to hot gas being mixed next to the blade surface. Holloway et al. [39, 40] found that trailing edge cooling is not as effective as they anticipated. Their studies showed that hot gas reaches the surface more readily than expected.

For instance, numerical simulations based on steady, Reynolds-averaged analysis approaches predicted notably better cooling effectiveness than those observed in the
experimental studies conducted for blade design. More recently, numerical simulations of Medic and Durbin [41] have revealed that three-dimensional unsteadiness occurring in cooling slot jets is a primary cause of poor trailing edge protection. This suggests a path to redesign the coolant streams to alleviate the heat load. While Medic and Durbin [41] showed that cooling jet streams contain unsteady, three-dimensional vortical components that have a large effect on transporting heat and contaminants to the surface due to the coherent, three-dimensional, vortex shedding from the upper lip of the breakout slot, the computational evidence has not been verified yet experimentally.

While several experimental investigations have already been conducted in recent years to investigate trailing edge cooling of turbine blades, the majority of those studies were mainly based on the pressure loss measurements and the quantification of adiabatic film cooling effectiveness on the wall downstream of the breakout [42-45]. Very few experimental studies [46] can be found in the literature to quantify the detailed flow characteristics of cooling wall jets in the cutback region.

The trailing cooling effectiveness has been experimentally measured for many years. Just as more effective film cooling designs have been developed, techniques to determine the film cooling effectiveness have also progressed to give more detailed cooling effectiveness analysis. In heat transfer experiments, thermocouples are traditionally used to measure both fluid and surface temperatures. When used to measure surface temperatures, thermocouples provide accurate measurements at discrete points [47, 48]. Therefore, it is difficult to observe detailed temperature distribution using thermocouples.

Liquid crystal thermography has been utilized in both transient and steady state heat transfer experiments to provide detailed surface temperature distributions. Liquid crystals
reflect different colors when they are exposed to temperature changes by reflecting a single
wave length of light. The steady state technique, as described by Han et al. [49], records the
color of the liquid crystals with an RGB camera. Every pixel on the test surface is converted
into a hue, saturation, and intensity. From a calibration of hue-temperature, the hue at each
pixel can be converted to a local temperature. Although detailed temperature can be acquired
by using this method, it is often difficult to use because it is limited by the temperature range
of the liquid crystals. Besides, the test surface must be painted black with a layer of liquid
crystals over the black paint. The liquid crystal alters the roughness of the surface and can
affect the surface temperatures. Liquid crystal thermography also requires an in situ
calibration to account for the lighting and camera angle; any change in lighting or camera
position will change the color the camera records, resulting in inaccurate temperature
readings.

A more popular experimental method to obtain detailed surface temperature distributions
is infrared (IR) thermography [50, 51]. This method has been applied over a wide range of
temperatures. Traditionally with IR measurements, the test surface is altered by applying
black paint in order to raise the emissivity of the surface (approaching unity). Similar to the
application of liquid crystals to the test surface, the black paint can change the surface
roughness. However, if an in situ calibration is performed (calibration surface is identical to
the test surface) the need to alter the surface with black paint is eliminated. Since IR cameras
can provide detailed film cooling effectiveness distributions over a wide range of
temperatures without altering the test surface, they become an attractive experimental
technique for film cooling measurements. The film cooling effectiveness based temperature
measurements is defined as
\[ \eta = \frac{T_{aw} - T_\infty}{T_c - T_\infty} \]  

(4.1)

where \( T_{aw} \) is adiabatic wall temperature, \( T_\infty \) is the mainstream temperature and \( T_c \) is the temperature of coolant.

Pressure sensitive paints (PSP) have recently been applied to film cooling applications [52-55]. PSP was first used in aerospace applications to obtain detailed surface pressure distributions [56]. PSP is photoluminescence paint, and the intensity of light emitted by the PSP depends on the partial pressure of oxygen in contact with the paint due to a process called oxygen quenching [57]; decreased oxygen pressure increases the emission intensity.

When applied to film cooling effectiveness, a mass transfer analogy must be utilized. With air as the mainstream gas and the pure nitrogen as the film coolant, the film effectiveness can be expressed in terms of the oxygen concentrations that are measured by the PSP light intensity [55] as

\[ \eta = \frac{C_{mix} - C_\infty}{C_c - C_\infty} \]  

(4.2)

The concentrations of oxygen are used in this equation instead of temperatures used in Equation (4.1). \( C_{mix} \) is the oxygen concentration of the mixing flow approaching the test surface (between 0 and 21%); \( C_\infty \) is the oxygen concentrations of the mainstream flow (near 21%); \( C_c \) is the oxygen concentration of coolant flow. Since 100% N\(_2\) is used as coolant, \( C_c \) is zero in the present study. Hence equation (4.2) can be rewritten as

\[ \eta = \frac{C_\infty - C_{mix}}{C_\infty} \]  

(4.3)

As a result, the film effectiveness will be between 0% far downstream of the coolant slots and 100% inside the slots. The method can only be applied to measurements of the film
cooling effectiveness, but not the heat transfer coefficients. However, one can take advantage of this technique to measure the mass transfer rather than the heat transfer. With the mainstream and coolant flows the same temperature, conduction and heat losses are not a problem for measuring the adiabatic film cooling effectiveness. Therefore, detailed measurements can be obtained near the exits of slots.

In the present study, experiments were conducted to quantify the cooling effectiveness on the cutback region as well as the flow characteristics of wall jets pertinent to trailing edge cooling of turbine blades. A relatively new technique, PSP, was utilized to measure the film cooling effectiveness in the cutback region of a trailing edge model. A high-resolution stereoscopic PIV system was used to conduct detailed flow field measurements to quantitatively visualize the evolution of unsteady vortices and turbulent flow structures in cooling wall jet streams and to quantify the dynamic mixing process between the cooling wall jet streams and the mainstream flows. The detailed flow field measurements were correlated with the adiabatic film cooling effectiveness maps to elucidate underlying physics and provide further insight to explore new trailing edge cooling strategies for protecting the critical portions of turbine blades from the harsh ambient conditions.

4.2. Pressure Sensitive Paint Technique

4.2.1. Principal Physics of Pressure Sensitive Paint

Pressure sensitive paint (PSP) has been quickly gaining recognition as an important experimental tool for optically and quantitatively measuring pressure of oxygen or concentration of oxygen based on luminescence quenching. Luminescent molecules are suspended in a polymer binder to create the paint. The molecules are excited by light at an
appropriate wavelength, and the excited molecules emit light at a longer wavelength. The emission light from the paint surface can be related to the oxygen pressure.

As shown in Figure 4.2, the luminescent molecules are suspended in the polymer binder. The binder is permeable, allowing oxygen molecules to penetrate into the paint, and interact with the luminescent molecules. The excited molecules rise to an upper singlet energy state, and a photon of a longer wavelength is emitted as the molecule returns to its ground state. In the presence of oxygen, the transition back to ground state is radiationless; this process is known as oxygen quenching. With a higher partial pressure of oxygen, more quenching of the luminescent molecules occurs, and thus the intensity of the emission light decreases.

![Figure 4.2. Model of pressure sensitive paint](image)

The intensity signals emitted by the paint are recorded at wind-off (reference) and wind-on (test) conditions using a CCD camera and are subsequently converted into pressure values.
using a pre-determined calibration relation for that specific paint. Such calibration relation is commonly known as Stern-Volmer equation and relates intensity ratio and pressure ratio of the known (reference) and unknown (test) conditions. For the present application, atmospheric pressure is used as the reference condition. The simplified form of the Stern-Volmer equation [58] can be expressed as:

$$\frac{I_{ref}}{I} = A_0 + A_1 \frac{P}{P_{ref}}$$  \hspace{1cm} (4.4)

$A_0$ and $A_1$ are known as the Stern-Volmer coefficients. Equation (4.4) is the most common form of Stern-Volmer equation that is used for PSP measurements due to linearity of the relationship between relative intensity $I_{ref}/I$ and relative pressure $P/P_{ref}$. However, in case of some special coatings and under certain pressure and temperature conditions, this relation may no longer remain linear. As explained by McLachlan and Bell [59], a more general form of Henry’s law required to explain PSP behavior can be expressed as:

$$\frac{I_{ref}}{I} = A_0 + A_1 \frac{P}{P_{ref}} + A_2 \left(\frac{P}{P_{ref}}\right)^2 + \ldots$$  \hspace{1cm} (4.5)

In case of a PSP measurement system, the pressure resolution of the system essentially depends on the intensity resolving capability of the camera used. Using the paint’s calibration relation, intensity information recorded for each camera pixel is converted to pressure values. Depending upon the bit depth of the camera (say N), reflected luminescence signal from the paint is recorded in a number of divisions ($2^N$). Each division corresponds to a particular intensity value and hence a particular pressure value. Difference between any two consecutive pressure levels then becomes the least decipherable pressure difference and hence the pressure resolution of the PSP system. However, in the case of a non-linear calibration relation, a constant intensity resolution of the camera will lead to a non-linear
pressure resolution. This characteristic should be an important consideration while making measurements using the PSP system since a varying pressure resolution restricts the sensitivity and applicability range of the system.

Due to the inherent noise associated with optical components, it is necessary to eliminate the background intensity using black images (no excitation light) [55]. With the removal of the background intensity, the intensity ratio becomes

\[
\frac{I(P)_{\text{ref}} - I_b}{I(P) - I_b} = f\left(\frac{P}{P_{\text{ref}}}\right)
\]

(4.6)

4.2.2. PSP Calibration

Before PSP can be applied to a test surface in a wind tunnel or other application, it must first be calibrated to determine the relationship between the intensity ratios and the pressure ratios. Black images are also recorded to remove the background intensity. To perform the calibration, a pressure control valve is used to control the pressure in the chamber as shown in Fig. 4.3, and the PSP was calibrated from 23.0 – 101.3 kPa. A test plate was sprayed with Binary UniCoat pressure sensitive paint supplied by Innovative Scientific Solutions, Inc. (ISSI). The design of the calibration cell for PSP can be found in Appendix A. At each measurement point, the PSP sample was excited using a constant UV light with a wavelength of 400nm. A CCD camera (PCO. 1600), with a 610nm filter, records the intensity of the light emitted from the paint.
Figure 4.3. The picture and schematic of the PSP calibration setup

Figure 4.4 shows the intensity contours from calibration images. Fig. 4.4(a) represents the intensity contour from the reference image. Fig. 4.4(b) represents the intensity contour from
the vacuum image. Fig. 4.4(c) is the intensity ratio calculated using the intensity values obtained from Fig. 4.4(a) and Fig. 4.4(b). Fig. 4.4(d) shows the error level which is the residuals of intensity ratio value after subtracting the averaged intensity ratio value in each grid.

![Intensity contours](image)

**Figure 4.4.** Intensity contours obtained from calibration: a. Intensity from reference image; b. Intensity from vacuum images $P/P_{ref}=0.23$; c. Intensity ratio $I_{ref}/I$; d. Error Level%

The calibration curve was obtained for the specified PSP with different pressures at the room temperature of $22^\circ$C as shown in Fig. 4.5 based on the averaged intensity ratio data from the Fig. 4.4(c) and the averaged pressure data measured by a DSA module in a same time period. The maximum error level obtained from Fig. 4.4(d) at different pressures was plotted in Figure 4.6. It has been observed that the maximum error is within 1%.
Care must be taken during the calibration to ensure the PSP is calibrated at the same temperature as that will be used during actual tests. The emission intensity of luminescent molecules is affected by the temperature in two ways [60]. First, just as the molecules return to their ground state in the presence of oxygen, they are more likely to return to their ground state at elevated temperatures. Also, most polymer binders are temperature sensitive; the
permeability of the binder changes with temperature. PSP experiments must be performed in isothermal environments, otherwise large errors in pressure measurements can result from variations in temperature which are not taken into account.

4.3. The Cooling Effect Measurement by Using PSP

4.3.1. Experimental Apparatus

After the PSP has been properly calibrated, it can be applied to the test surface. The trailing edge cooling model inside the wind tunnel is coated with Binary Unicoat PSP, and the excitation light source and camera are positioned appropriately so the emission light from the entire surface is recorded in one image. Fig. 4.7 shows the basic setup for PSP measurements, including the test surface and optical components.

Figure 4.7. PSP Experimental Setup
4.3.1.1. PSP measurement theory

Because the focus of this study is to determine the film cooling effectiveness, rather than the surface pressure distribution, an additional element is added to the experiment. As shown in Equation (4.3), the film cooling effectiveness is calculated based on the concentration differences created by air coolant injection and nitrogen coolant injection. Therefore, to accurately determine the film cooling effectiveness, four images are required. The first image is the black image; as with the calibration, this image is required to eliminate any noise in the images. The second image is the reference image; with this image the PSP is excited with the light source, but there is no mainstream or coolant flow. The third image is an air image. This image is taken with mainstream and coolant flow, where the coolant blowing through the breakouts in trailing edge is air. The fourth and final image is the nitrogen image. Similar to the air image, this image is recorded with mainstream and coolant flow, but now the coolant flow is pure nitrogen (N₂).

Expanding Equation (4.6) to include both images with air and nitrogen injection, the intensity ratios are re-written in Equation (4.7) and (4.8).

\[
\frac{I(P)_{\text{ref}} - I_b}{I(P)_{\text{air}} - I_b} = f \left( (P_{O_2})_{\text{air}}; P_{\text{ref}} \right) \quad \text{or} \quad f (P; P_{\text{ref}}) \tag{4.7}
\]

\[
\frac{I(P)_{\text{ref}} - I_b}{I(P)_{\text{mix}} - I_b} = f \left( (P_{O_2})_{\text{mix}}; P_{\text{ref}} \right) \tag{4.8}
\]

\(I(P)_{\text{air}}\) represents the intensity recorded during the test with air injection, and \(I(P)_{\text{mix}}\) represents the intensity recorded from the test for the mixing flow with nitrogen as the coolant. From the calibration of the PSP, the partial pressure of oxygen on the test surface with both air and nitrogen injection can be calculated. Equation (4.1) can be re-written, so
that the film cooling effectiveness is related to the partial pressure of oxygen measured with both air and nitrogen as coolant [55].

\[
\eta = \frac{C_{\infty} - C_{\text{mix}}}{C_{\infty}} = \frac{(P_{O_2})_{\text{air}} - (P_{O_2})_{\text{mix}}}{(P_{O_2})_{\text{air}}}
\] (4.7)

The partial pressure of oxygen with air and nitrogen injection is determined by the calibration of emission intensity and pressure. With Equation (4.7) combined with the PSP calibration, the film cooling effectiveness can be determined at every pixel giving detailed film cooling effectiveness distributions on the surface of trailing edge.

### 4.3.1.2. Studied model

Figure 4.8 shows the trailing edge cooling model used for the present study. The trailing edge model is designed to simulate only a portion of the blade trailing edge as shown in Fig. 4.1. More detail about the design of model can be found in Appendix B. The trailing edge model used in the present study is simplified as a combination of two plates which are made of Plexiglas. The dimensions of the trailing edge model have been shown in Fig. 4.8. There is a long engraved through slot in the lower part that connects the flow channel engraved in the upper part with a cooling air plenum (See Appendix A for detail). Six removable land pieces can be taped on the lower plate between the slots in the spanwise direction. The coolant flow will be adjusted in the plenum then blow out through the long slot in the lower plate into the five channels between the two pieces of plates. The cooling jet streams, which are tangential to main stream flow, eject onto the cutback region through five breakout slots. It should be noted that the wedge shape for the left end as shown in Fig. 4.9 is designed to remove the turbulence boundary from upstream. In the test section, a gap will be remained between the
bottom surface of wind tunnel upstream and the wedge. This design will guarantee a low turbulence intensity level of the main stream flow approaching the trailing edge.

![Figure 4.8. Dimensions of studied trailing edge cooling model](image)

### 4.3.1.3. Experimental facility

In the present study, a low speed blow-down wind tunnel built at Iowa State University is used. The wind tunnel has an inlet cross section of 0.203m (width) × 0.127m (height). The tunnel has a contraction section upstream of the test section with honeycombs and screen structures installed ahead of the contraction section to provide uniform low-turbulence incoming flow into the test section. The main flow velocity at the inlet of the test section was set as $U_\infty = 16.2$ m/s. Based on the length of the upper plate ($L = 177.8$mm) which is right before the breakout, the corresponding Reynolds number is $1.91\times10^5$. A central air-
conditioning system maintained the mainstream temperature at 22°C. The turbulence intensity, measured by using a hotwire anemometer, was about 2% near the wall at the inlet of the test section. The turbulence intensity of the coolant flow at the cutback region is approximately 7%, which is usually between 5%-8% in the cooling flow in a turbine blade.

Figure 4.9. Schematic of the experimental setup

The coolant flow, supplied from a regular gas cylinder or a high pressure air tank depending on the requirement of measurement, passed through flow control valves and entered the cooling gas plenum, which is directly underneath the lower plate. The test surface is sprayed with Binary UniCoat pressure sensitive paint. PSP surfaces were excited using a constant UV light with a wavelength of 400nm. A CCD camera (PCO. 1600) with a 610nm
filter records the intensity of the light emitted by the PSP. The setup layout is shown in Figure 4.9 and Figure 4.10. Although the PSP measurement is not a temperature based technique, the temperature of the cutback surface is measured by using a thermometer. Because the Binary UniCoat paint used in present study will be affected by temperature to some extent. The temperature can be read directly and recorded during the experiment in order to reduce the temperature effect in the post data process. In the present study, the experiment was conducted with five different blow ratios. Since the flow velocity is relatively low, the blow ratio can be defined as the velocity ratio between the cooling stream and main stream \((M = U_{\text{cooling stream}}/U_{\text{mainstream}})\), between 0 and 1.6. The corresponding Reynolds number of the slot jets, \(Re_c\) is in the range of 2800 to 10500 based on the height \((H)\) of the slot, which is 6.35mm as shown in Fig. 4.8.
4.3.2. Results and Discussion

The effect of blow ratios and the existence of lands on the film cooling effectiveness in the cutback region of trailing edge will be discussed in this section.

4.3.2.1. The effect of blow ratios on the cooling effectiveness for the case without lands

The intensity of the excitation on the trailing edge cutback was recorded by a CCD camera and data processing method as described earlier in Section 4.2 was applied in the calculation of film cooling effectiveness. Figure 4.11 shows the contour plots of the effectiveness for the cutback region without lands at the blow ratios of M=0.43, M=0.64, M=0.76, M=1.1 and M=1.6. As shown in Fig. 4.11, as the blow ratio increases, the effectiveness of the film coolant tends to cover more area downstream of the breakout. The higher blow ratios cover more distance in the downstream direction. However, in spanwise direction, the film coolant tends to cover less area as the blow ratio increases. For the lower blow ratios, it is easier for coolant flow to spread out and create more uniform coverage in the lateral direction. But for higher blow ratios, the high momentum of coolant flow restricts the lateral motion, so the coolant flow cannot easily cover the region between the jets. Therefore, optimizing the blow ratio involves finding a balance between uniform lateral coverage and the distance covered downstream of the breakout.
Figure 4.11. Adiabatic film cooling effectiveness on the trailing edge for different blow ratios $M$

From only a comparison of the contour plots it might be difficult to notice any appreciable variation due to the increased blow ratios. Figure 4.12(a) shows the cooling effectiveness distributions along the centerline (i.e. $Z/H=0$) at varied blow ratios. The present data have been compared with a correlation for tangential slot film cooling on a flat plate with no pressure gradient based on a theoretical model given by Mukherjee [61]. Taking into account the influence of specific heat of both the media and assuming a fully developed turbulent boundary layer, the following equation has been derived.

$$
\eta_{adf} = \frac{1.9 Pr^{2/3}}{1 + 0.329 \beta^{0.6} \frac{C_{PM}}{C_{P \infty}}} 
$$

(4.8)
where $P_r$ is Prandtl number, $C_{P_M}$ and $C_{P_c}$ are pressure coefficient of mainstream flow and coolant flow respectively, $\beta$ takes into account the influence of blowing angle and is 1 for tangential blowing.

$$B = \left(\frac{\mu_c}{\mu_m} Re_c\right)^{-0.25} \frac{X}{mH}$$ \hspace{1cm} (4.9)

where $\mu_c$ and $\mu_m$ are viscosity of coolant flow and mainstream flow respectively, $Re_c$ is Reynolds number of coolant flow, $X$ is the distance measured from slot, $m$ is blow ratio and $H$ is the height of the slot as shown in Figure 4.13.

Correlated with experimental data, three separate regions can be identified when a secondary stream is blown out of a slot on the wall. With the expression of $B$ defined in equation (4.9), the three different regions and the expressions for the effectiveness are found as follows:

For the first region defined by $B < 1$

$$\eta_{adf} = 1.0$$ \hspace{1cm} (4.10)

For the second region which is given by $1 < B < 4$ the best correlation found is

$$\eta_{adf} = \frac{1.9 Pr_e^{2/3}}{1 + 0.525 B^{0.47} \frac{C_{P_M}}{C_{P_c}}}$$ \hspace{1cm} (4.11)

For the turbulent boundary layer region; i.e., $B > 4$ the equation was found to be

$$\eta_{adf} = \frac{1.9 Pr_e^{2/3}}{1 + 0.329 B^{0.47} \frac{C_{P_M}}{C_{P_c}}}$$ \hspace{1cm} (4.12)

Relatively good comparison is obtained for a high blow ratio $M=1.6$. But the comparison at low blow ratio $M=0.43$ is not that good in the region between $X/H=3$ and $X/H=8$. The difference was also observed by Choi et.al [45]. This is because in Mukherjee’s model, the lip thickness was assumed to be zero. This assumption would over-estimate the film cooling.
effectiveness as a conclusion in Mukherjee’s study. Therefore the film cooling effectiveness for a trailing model with a certain lip thickness should be lower than the prediction from Mukherjee’s model. The reduction in effectiveness with increasing lip thickness becomes less significant with increasing blow ratios, which is confirmed by present study. In our study the cooling effectiveness in the center line decreases drastically after $X/H=5$ for blow ratio $M=0.43$. All other curves with a gently decreasing trend do not vary too much for varied blow ratios.

Figure 4.12. Cooling effectiveness profiles for the case without lands

a. Centerline adiabatic effectiveness
b. Spanwise cooling effectiveness ($X/H=2$)

c. Spanwise cooling effectiveness ($X/H=4$)
d. Spanwise cooling effectiveness ($X/H=8$)
However, as seen in Figure 4.12(b) the comparisons in spanwise at \(X/H=2\) reveals an apparent difference at varied blow ratios. In general, the cooling effect with \(\eta \approx 1\) in the jet region performs the best at the center and decreases along spanwise direction. The worst effect was at the center line of the in-between region (the region between two cooling jets) as expected. The cooling effectiveness in the region between the jets decreases drastically with the increased blow ratios, and the worst cooling effect in that region was shown at blow ratio \(M=1.6\). The phenomena could be explained in correlation with the flow features in the in-between region. The flow characteristics will be described and discussed in section 4.4. Figure 4.12(c) shows the spanwise cooling effectiveness at \(X/H=4\) with different blow ratios. It shows very similar trends as those shown in Fig. 4.12(b). It should be noted that the variances between different blow ratios become less. Figure 4.12(d) shows the spanwise cooling effectiveness distribution at \(X/H=8\) with different blow ratios. Since the cooling flow with \(M=0.43\) cannot spread as far as other blow ratios, the cooling effectiveness at \(X/H=8\) is much lower than others. But for all other blow ratios, the cooling effectiveness distributions almost overlap each other without many variations. The maximum cooling effectiveness value at \(X/H=8\) was found approximately to be 0.95.

![Figure 4.13. Slot geometry for the film cooling of a surface in Mukherjee’s model](image)

Figure 4.14 shows the spanwise averaged cooling effectiveness (between \(Z/H = -2\) and \(Z/H = 2\)) at varied blow ratios. As the coolant flow with high blow ratios cannot perform very well in the in-between region, it becomes the worst one before \(X/H \approx 5\). Even after \(X/H = 5\)
the coolant at M = 1.6 cannot work as well as other blow ratios except M = 0.43. The cooling effectiveness curves for M = 0.64 and M = 0.76 are almost overlapped. The similarities are also revealed in the distribution contours in Figure 4.11.

![Spanwise averaged cooling effectiveness profiles for the case without lands](image)

Figure 4.14. Spanwise averaged cooling effectiveness profiles for the case without lands

### 4.3.2.2. The effect of blow ratios on the cooling effectiveness for the case with land

For the case with lands, the PSP measurements were conducted over seven blow ratios in order to show more variances in the comparison. The effectiveness at the cutback region is very high as the coolant exits the slot. Effectiveness magnitudes close to the slot exit are equal to 1. As shown in Figure 4.15, the trends shown at the cutback region between lands in the contour figures are similar to the case without lands shown in Figure 4.11. As the blow ratio increases, the film coolant tends to cover more area in the cutback region downstream. The coolant nitrogen covers the most area at M=1.6 as expected. However, in spanwise direction, the trend of the film cooling effectiveness especially on the land area is not that apparent to reveal the difference. But it could be noticed that it is easier for coolant flow to mix with the mainstream air flow and create more coverage in the lateral direction at the relatively low blow ratios. But at high blow ratios (M=1.1 and M=1.6), it seems that the high
momentum of coolant flow restricts the lateral mixing flow motion hence the coolant flow is not able to cover the top surfaces of lands easily.

From only a comparison of the contour plots it might be difficult to notice any appreciable variation due to the increased blow ratios. On one hand, Figure 4.16(a) shows the cooling
effectiveness distributions along the centerline (i.e. Z/H=0) at varied blow ratios. For lower blow ratios (M=0.25, M=0.43, M=0.52), the film cooling effectiveness drop suddenly at some point as shown in the figure. All other curves for high blow ratios do not vary much with a gradually decreasing trend.

![Graphs showing cooling effectiveness profiles for various blow ratios](image)

a. Centerline adiabatic effectiveness  
b. Spanwise cooling effectiveness at X/H=2  
c. Spanwise cooling effectiveness (X/H=4)  
d. Spanwise cooling effectiveness (X/H=8)

Figure 4.16. Cooling effectiveness profiles for the case with lands

On the other hand, as seen in Figure 4.16(b) the comparisons in spanwise at X/H=2 reveals an apparent difference at varied blow ratios. In general, the cooling effect in the center jet region performs the best, and decreases along spanwise direction until the centerline of lands. For the two lowest blow ratios, the maximum film cooling effectiveness was found to be 0.8 and 0.95 respectively. For other blow ratios, the maximum film cooling
effectiveness is approximately 1.0. It is interesting that the cooling effectiveness on the top surface of lands decreases with the increased blow ratios except for the lowest blow ratio of M=0.25, and the worst cooling effect in that region is observed at the blow ratio of M=1.6. The phenomena could be explained in correlation with the mixing features around the edge of lands. The flow characteristics will be described and discussed in section 4.4. As for the lowest blow ratio, since the flow rate of the coolant is too low, the coverage area of nitrogen is limited too. Figure 4.16(c) shows the spanwise cooling effectiveness at X/H=4 with different blow ratios. It shows very similar trends as those shown in Fig. 4.16(b). The variances between different blow ratios decrease. The film effectiveness curves overlapped in the cutback region for high blow ratios. Figure 4.16(d) shows the spanwise cooling effectiveness distribution at X/H=8 with different blow ratios. It seems the curves are detached along streamwise direction. The cooling flow with low blow ratios cannot spread as far as high blow ratios; therefore, the film effectiveness at X/H=8 is relatively low. The film effectiveness at M=1.6 shows the best performance at the end of the lands.

Figure 4.17. Spanwise averaged cooling effectiveness profiles for the case with lands

Figure 4.17 shows the spanwise averaged cooling effectiveness in the cutback slot region and in the land region at different blow ratios. The trends of the cooling effect curves in the
cutback slot are very similar to those curves found at the center line. In the land region, at low blow ratios, such as M=0.25, M=0.43, the cooling effectiveness increased rapidly along streamwise due to the entrainment of coolant onto the top surface of lands and then started decreasing gently after reaching a maximum due to more addition of mainstream air in the mixing. The trend lines kept increasing with distance at other blow ratios. As the blow ratio increases, more coolant was entrained onto the land surface. However, if the blow ratio keeps increasing over one, the coolant tends to have less chance to reach the top surface of lands.

4.3.2.3 The effect of the existence of land on film cooling effectiveness

In order to study the effect of the existence of land on film cooling effectiveness, Figure 4.18 shows a direct comparison of the land effect at both a low blow ratio and a high blow ratio. In Figure 4.18(a), the effectiveness was averaged in the cutback slot for the case with lands, and the effectiveness was averaged between X/H=-2 and X/H=2 for the case without lands. In the comparison of spanwise averaged cooling effectiveness, the coolant for the case with lands performs a little bit better upstream (before X/H ≈ 2) but worse downstream than the case without lands at low blow ratio. The lands make the coolant performing much better at the high blow ratio, M=1.6.

Figure 4.18(b) shows the comparison of film effectiveness with two blow ratios at X/H=2. For a low blow ratio M=0.43, It is apparent that more film effect was found for the case without land, especially in the region between jet flows. The relative low cooling effect on the lands is because the coolant cannot approach the top surface of the lands easily. For high blow ratio M=1.6, the coolant performs better in the slot region for the case with lands than the case without lands. This phenomenon can be attributed to the restriction effect of the lands on spanwise spreading. But on the top surface of lands, the cooling effectiveness is
much worse than that in the same region without lands. As shown in Figure 4.18(c), very similar trends were found at X/H=4. The case with lands shows less cooling effectiveness in the slot valley. This phenomenon might be caused by the entrainment of mainstream flow into the slot valley, which is enhanced by the existence of lands. The difference of the film effectiveness became less for both blow ratios at X/H=8 as shown in Figure 4.18(d).

Figure 4.18. Comparison of cooling effectiveness profiles

4.3.3. Error Analysis on the Cooling Effectiveness

Due to the perspective angle of camera, the complicated three-dimensional structure of the trailing edge model will induce some error in the two dimensional view. As shown in Figure 4.19(a), a 3-D structure of the trailing edge model was drawn in real dimension. In the PSP
experiment, a CCD camera was mounted above the test model. The 2-D view captured by the camera was shown schematically in Figure 4.19(b). The vertical surfaces of the lip and lands were revealed as light green regions around the cutback region between lands. But in present measurements it is assumed that all the data only represents the information from the top surfaces of lands and cutback slot region between lands. Therefore, this assumption will induce some errors in the green region, which is called fake region at the surrounding area of the cutback region in the final PSP results. The width of the fake region is estimated to be 2 mm (about 1/3 of H) at most.

In the present study, even though the temperature difference between mainstream flow and coolant flow is only 2~3°C, the difference still generates uncertainty in the PSP measurements, because the paint Binary Unicoat used in present study is sensitive to temperature. The uncertainty analysis performed on the film effectiveness measurements of
the PSP was based on that described by Kline and McClintock [62]. The uncertainty of the pressure distribution is estimated to be 5%, and film cooling effectiveness was estimated to be 9%.

4.4. PIV Measurements on the Trailing Edge Cooling Flow

4.4.1. Experimental Apparatus

The schematic of the experimental setup used in the present study was shown in Figure 4.20. The wind tunnel facility has been mentioned in section 4.3. The walls of the test section are optically transparent, which is convenient for the PIV measurements. In the present study, a high-resolution 2-D PIV system and a high-resolution stereoscopic PIV system were used to conduct detailed flow field measurements in the cutback region downstream of the breakout of the trailing edge model.

For the PIV measurements, the main flow was seeded with ~ 1 µm oil droplets by using a droplet generator. The film cooling flow was supplied by a large high-pressure air tank which can provide constant airflow for at least 20 minutes. As shown in Figure 4.20, the room-temperature air from the tank serving as coolant passes through a pipe and a bypass valve and then bifurcates through two flow control valves before it enters the plenum under the trailing edge model. One stream enters an aerosol generator to produce aerosol particles for PIV measurements; while the other stream enters the plenum to dilute the condensed aerosol particles. After the mixing of the two cooling streams in the plenum, the coolant air enters the inside channels of the trailing edge model, and ejects from the exits onto the cutback region.

During the PIV measurements, illumination was provided by a double-pulsed Nd:YAG laser (NewWave Gemini 200) adjusted on the second harmonic and emitting two pulses of
200 mJ at the wavelength of 532 nm with a repetition rate of 10 Hz. The laser beam was shaped to a sheet by a set of mirrors, spherical and cylindrical lenses. The thickness of the laser sheet in the measurement region is about 1.5mm. The illuminating laser sheet was first aligned along the main stream flow direction to conduct 2-D PIV measurements in the X-Y planes. Then, the laser sheet was rotated 90 degrees to illuminate flow structures in the cross planes normal to the main stream flow direction in order to conduct stereoscopic PIV measurements in the Y-Z planes at different locations downstream the exit of the slot jets. For the 2-D PIV measurements in X-Y planes, a high resolution 12-bit CCD camera (SensiCam, CookeCorp) was used for PIV image acquisition with the axis of the camera perpendicular to the laser sheet.

For the stereoscopic PIV measurement in the Y-Z planes, two high resolution 14-bit CCD cameras (PCO 1600) were used for Stereo-PIV image acquisitions. The two CCD cameras were arranged in an angular displacement configuration to get a large overlapped view. With the installation of tilt-axis mounts, the lenses and camera bodies were adjusted to satisfy the Scheimpflug condition. The distance between the illuminating laser sheet and image recording planes of the CCD cameras was about 565mm, and the angle $\alpha$ between the view axes of the two cameras was about 60 degrees for the cases without lands. But, for the Stereo-PIV measurements in the slot flow valley between two lands, the distance between the laser sheet and CCD camera was about 730mm, and the angle $\alpha$ between the view axes of the two cameras was 12 degrees which was determined by the limit expansion angle of lands. The CCD cameras and the double-pulsed Nd:YAG lasers were connected to a workstation (host computer) via a Digital Delay Generator (Berkeley Nucleonics, Model 565), which controlled the timing of the laser illumination and the image acquisition. A general in situ
calibration procedure was conducted to obtain the mapping functions between the image planes and object planes for the stereoscopic PIV measurements. A target plate (~ 150mm \times 150 \text{ mm}) with 500-\mu\text{m}-diameter dots spaced at intervals of 2 mm was used for the in situ calibration. The front surface of the target plate was aligned with the center of the laser sheet, and then calibration images were captured at three locations across the depth of the laser sheets. The space interval between these locations was 0.5 mm for the cases with \( \alpha = 60 \text{ deg} \) and 1.25mm for the cases with \( \alpha = 12 \text{ deg} \). The mapping function for the data process was taken from the calibration to be a multi-dimensional polynomial, which is fourth order for the directions (Y and Z directions) parallel to the laser sheet plane and second order for the direction (X direction) normal to the laser sheet plane. The coefficients of the multidimensional polynomial were determined from the calibration images by using a least-square method. More detail about stereoscopic PIV techniques can be found in Appendix D.

For the PIV image processing, instantaneous PIV velocity vectors were obtained by a frame to frame cross-correlation technique involving successive frames of patterns of particle images in an interrogation window 32\times32 \text{ pixels}. An effective overlap of 50\% of the interrogation windows was employed in PIV image processing. For the Stereo-PIV measurements, with the mapping functions obtained by the in situ calibration procedure, the two-dimensional displacements in the two image planes were used to reconstruct all three components of the velocity vectors in the illuminating laser sheet planes (i.e Y-Z planes). After the instantaneous velocity vectors \( (u_i, v_i, w_i) \) were determined, instantaneous vorticity \( (\omega_x, \omega_y) \) could be derived. The time-averaged quantities such as mean velocity \( (U, V, W) \), ensemble-averaged vorticity, turbulent velocity fluctuations \( (\overline{u'}, \overline{v'}, \overline{w'}) \) and normalized...
turbulent kinetic energy \( T.K.E = 0.5 \cdot (u'u' + v'v' + w'w')/U_{\infty}^2 \) distributions were obtained from a cinema sequence of 330 frames of instantaneous velocity fields.

Figure 4.20. The schematic of the PIV setup for flow measurement

a. 2-D PIV setup

b. Stereo-PIV setup
4.4.2. Experimental Uncertainties

The 2-D PIV measurement-uncertainty level for the velocity vectors is estimated to be within 2% and that of T.K.E is about 10%. The 3-D Stereo-PIV measurement-uncertainty level for velocity vectors with camera angle $\alpha/2=30$ degrees was calculated to be about 2% by using the method of Lawson and Wu [63]. All cases without lands and the case with lands at $X/H=8$ were conducted with camera angle $\alpha/2=30$ degrees. The measurements-uncertainty level with camera angle $\alpha/2=6$ degrees was calculated at about 10%. The measurements of cases with lands at $X/H=0.1$, $X/H=2$, $X/H=4$, $X/H=6$ were conducted with $\alpha/2=6$ degrees.

4.4.3. Results and Discussion

For the prescribed main air velocity of 16.2m/s, the slot flow velocity was set by six blow ratios ranging from 0 to 1.6. The slot jet velocity was measured in the center of the exit by hotwire manometer. The turbulence intensity of the slot jet flow is ranging from 5% - 7% for blow ratio varied from 0 to 1.6. 2D PIV measurements were conducted on a vertical mid-span plane at six blow ratios without lands. Stereo-PIV measurements were conducted in five lateral cross-planes along X-direction for six blow ratios. The effects of blow ratios and the existence of lands were investigated by using Stereo-PIV technique.

4.4.3.1. Results in mid-span plane

The detailed flow measurements on the vertical mid-span plane for the case without lands are revealed in the following figures from Figure 4.21 to Figure 4.25. Figure 4.21(a) shows the instantaneous velocity vector field on the vertical mid-span plane at the blow ratio of $M=0.43$. The suddenly expanding laminar jet flow keeps the flow rates low by forming a buffer layer between the bottom surface and mainstream flow until $X/H=4$ (around 4 times of the slot height $H$, $H=6.35$mm) after the exit of slots. A similar phenomenon was also found at blow
ratios of 0.37 and 0.47 by Caken and Taslim [41]. This could also be seen from the ensemble-averaged streamwise velocity field in Fig. 4.21(c). However as both flows proceed downstream, they mix and form a mixed turbulent boundary layer flow downstream after about 40mm (around 6H). The shearing effect generated by the differences of momentum between slot and mainstream flow triggered mixing turbulent flow between the flows. Fig. 4.21(b) shows the instantaneous vorticity distribution for the same condition as above. Apparently, paired-vortex was generated in the region between the slot and main flow with adverse sign of the vorticity value. While it can be predicted that a two-dimensional geometry will produce a Von Karman vortex street, it is far from obvious in the instantaneous vorticity field. It has been interpreted by Medic and Durbin [38] that the three-dimensionality can suppress coherent shedding. The negative value of vorticity found near the bottom wall is generated by boundary layer flow. Fig. 4.21(d) shows the ensemble-averaged turbulent kinetic energy distribution. The low T.K.E region after the breakout until $X/H \approx 3$ supports the finding about the length of the buffer layer between slot flow and bottom wall. The high T.K.E region observed after $X/H=4.5$ in Fig. 4.21(d) may indicate that the fully developed turbulent flow enhances the mixing process.

Figure 4.22(a) shows the instantaneous velocity vector field on the vertical mid-span plane at the blow ratio of $M=0.64$. The instantaneous vorticity contour in Figure 4.22(b) reveals a similar pattern as the one at the blow ratio of $M=0.43$. The jet shape of the coolant flow has been extended longer as shown in streamwise velocity field in Figure 4.22(c). It is interesting that the turbulence kinetic energy at $M=0.64$ tends to be less than that at $M=0.43$, which means the coolant flow with blow ratio of 0.64 generate less turbulence though it mixes with the mainstream flow downstream. At blow ratio $M=0.76$, Figure 4.23(a-d) shows very
similar flow feature as those found in the flow at M=0.64. But a relatively high TKE region was found between the coolant flow and mainstream flow, which does not appear at M=0.64.

Figure 4.24(a-d) shows the PIV measurement results for blow ratio M=1.1. Figure 4.25(a-d) shows the PIV measurement results for blow ratio M=1.6. At higher blow ratios, on one hand the slot is filled more and more with laminar or transitional slot flow and this prevents effective entrainment of mainstream flow onto the slot jet flow. It induces the buffer boundary layer of the slot flow to extend to X/H=6 for M=1.1 and X/H=8 for M=1.6, which can be observed from the instantaneous vorticity field and averaged T.K.E field in Fig. 4.24 and Fig. 4.25. On the other hand, the mix layer at high blow ratio triggers the mainstream flow entrainment into the slot valley more easily. This flow pattern is different from the observation at low blow ratios. At low blow ratios, the streamlines before X/H=4 are observed to tilt upward for M=0.43 because of the less flow momentum of the slot flow. Due to the effect of momentum difference, the downward slope of the streamlines of main flow starting from the lip end is much larger at M=1.6 than the one at M=0.43.

Figure 4.21. PIV measurement results at blow ratio of M =0.43
Figure 4.22. PIV measurement results at blow ratio of $M = 0.64$

a. Instantaneous velocity vector field
b. Instantaneous vorticity contour
c. Averaged velocity with streamlines
d. Averaged turbulent kinetic energy

Figure 4.23. PIV measurement results at blow ratio of $M = 0.76$

a. Instantaneous velocity vector field
b. Instantaneous vorticity contour
c. Averaged velocity with streamlines
d. Averaged turbulent kinetic energy
The effect of blow ratios on the X-component velocity at different distances downstream of the slot exit is shown in the velocity profiles in Figure 4.26. There was negative velocity for M=0 because the mainstream flow will generate a circulation region when passing the lip end. The velocity profiles do not vary much in the near wall region between X/H=0.5 to X/H=2, which indicates that the slot flow remains laminar in this region. In the region between X/H=2 to X/H=4, the slot flow still keeps a laminar jet shape, but the difference
between the velocity in the laminar region and mixing region is below 1.5m/s for M=0.43, which means the mixing process is almost fully developed. The slot flow further decelerates and the flow accelerates in the mixing region at X/H=8. The mainstream flow decelerates a little as well due to the mixing. At X/H=8, the profiles tend to be monotonically increased curves for blow ratios less than 1, which indicates the combined flow has been developed to a turbulent boundary layer flow. Even though the relatively simple flow pattern in the mid-span plane, to some extent, could be related to explain the cooling effectiveness on the bottom surface, it must be emphasized that it is definitely not enough to reveal the flow characteristics because of the three-dimensional unsteady flow feature [38].

Figure 4.26. Streamwise velocity profiles on the center vertical plane with various blow ratios at different distances after the exits.
4.4.3.2. Results in lateral cross-planes

In order to show the more complicated three-dimensional vortex, a high-resolution Stereo-PIV system was used to investigate the complex flow near the walls and lands. The Stereo-PIV measurements were conducted in five lateral cross planes at X/H=0.1, X/H=2, X/H=4, X/H=6 and X/H respectively.

4.4.3.2a. The effect of blow ratios

Figure 4.27 shows the three-dimensional velocity vectors and the contour of the streamwise velocity values for both the flows without lands and with lands at blow ratio of 0.43. In general, it can be seen clearly that the coolant flows with a jet shape coming out from the breakouts. In the last two cut planes a4 and a5 near the end shown in Fig. 4.27, the jet shape disappears, which indicates the mixing process is well done. For the case without lands, as shown in Fig. 4.27(a), the velocity vectors tend towards adverse-flow direction at two sides of the measurement plane a1 and a2. Because there will be a low-speed ‘vacuum’ region between the slot flow under the lip step as one might expect. The low pressure in the vacuum region results in the suction effect which induces the adverse vectors. For the case with lands as shown in Fig. 4.27(b), no reversed flow vectors were found due to the existence of lands. But at a first glance, very similar flow distribution is found in the slot flow valley as that without lands. Small velocity values, even negative values, can be found near the top surface of the land due to the flow separation at the lip region. More detailed flow comparison will be revealed in the next section 4.4.3.2b.

Figure 4.28 and Figure 4.29 show the three-dimensional flow feature for the blow ratio of 0.64 and 0.76 respectively. As similar as the results at blow ratio of 0.43, apparent jet shapes
of the coolant flow were found until the cross plane at $X/H=4$. But the velocity at jet center increases with the increasing blow ratios.

Figure 4.30 and Figure 4.31 show the three-dimensional velocity vectors and the contour of the streamwise velocity values at higher blow ratios of 1.1 and 1.6 respectively. The jet shapes of coolant flow are much easier to observe compared with the case at low blow ratios due to the high velocity of jet. The jet shapes of flow are kept until the very end cross plane ($X/H=8$). The longer jet shape indicates more coverage along X-direction, which could be confirmed with the results of cooling effectiveness measurements as shown in Figure 4.11 and 4.15. The high-speed jet regions for the case with lands are wider than those without lands, which indicates that the existence of land suppress the slot flow to move forward instead of expanding in the spanwise direction, and even turning back in the case without lands. This finding could also be related to the measurement results of cooling effectiveness as shown in Figure 4.18 in section 4.3.
Figure 4.27. Three-dimensional velocity vector and axial velocity contour at blow ratio of $M=0.43$ at five different distances after the slot exit: a. without lands b. with lands.
Figure 4.28. Three-dimensional velocity vector and axial velocity contour at blow ratio of $M=0.64$ at five different distances after the slot exit: a. without lands b. with lands
Figure 4.29. Three-dimensional velocity vector and axial velocity contour at blow ratio of $M=0.76$ at five different distances after the slot exit: a. without lands b. with lands
Figure 4.30. Three-dimensional velocity vector and axial velocity contour at blow ratio of M=1.1 at five different distances after the slot exit: a. without lands b. with lands
Figure 4.31. Three-dimensional velocity vector and axial velocity contour at blow ratio of $M=1.6$ at five different distances after the slot exit: a. without lands b. with lands
The vortex structure is very complicated in the instantaneous vector field [38]. Since the present measurements were only conducted in one cut plane each time by using two cameras, only streamwise vorticity can be obtained instantaneously. It is impossible to get instantaneous three-dimensional vortex structures. Hence the ensemble-averaged vortex structure in main flow direction is revealed in Figure 4.32 for varied blow ratios. The streamwise vorticity is known to greatly enhance mixing in shear layers [64]. Basically, there always appear paired vortex structures with adverse signs of the vorticity value which means an adverse rotational direction. The vortices tend to be stronger in the very end cut plane for the case with lands because of the development of mixing. The unsteady vortex flow originates at the upper wall of the breakout which agrees with the results found by Medic and Durbin [38]. This vortex flow pattern is kept in the downstream flow. For lower blow ratios, the entrainment of mainstream flow tends to drive the coolant flow in the valley to escape from the valley to form vortex structure around the edge of lands. This phenomenon does not happen when the velocity of coolant flow is higher than mainstream flow (M=1.1, M=1.6) as shown in Fig. 4.32(d, e), because the momentum of slot flow is higher than the mainstream flow. In conclusion, the interaction between the mainstream flow and slot flow is very strong. The blow ratios have significant effects on the vortex structures in the mixing process.
Figure 4.32. Averaged streamwise vorticity contour with varied blow ratios at three different distances after the exits for the cases with lands

Figure 4.33 shows the flow field at different cross planes for the case without lands with different blow ratios. The flow pattern in the side region is totally different for these two blow ratios. At low blow ratio, the mainstream entrainment produces updraft flow in the side
regions, which will allow the coolant flow to cover this region easier. Therefore higher cooling effectiveness could be expected as shown in Figure 4.11. But at high blow ratios, since the coolant flow has more momentum, the jet flow entrainment produces downdraft flow in the side regions. This behavior will bring more mainstream flow into the side regions hence it results in lower cooling effectiveness as shown in Figure 4.11.

Figure 4.33. Flow field at different cross planes for M=0.43 and M=1.6

4.4.3.2b. The effect of the existence of lands

In order to show the effect of the existence of lands, Figure 4.34 shows a direct comparison of the land effect in five lateral cross planes at different distances downstream of the breakout for both low and high blow ratios. For the low blow ratio of M=0.43 and high blow ratio of M=1.6, the existence of lands suppresses the expanding motion in the lateral direction of the coolant flow. As mentioned before, the restriction of lands makes the high-speed region of coolant flow a little bit wider than that without lands.
As shown in Fig. 4.34, for the case without lands at a low blow ratio, on one hand the coolant flow spreads laterally and turns up mixing with the mainstream flow at upper level in the side regions. On the other hand, the updraft coolant flow in the mid-span region mixes with the mainstream flow above the upper wall of the exit before X/H=4. After X/H=4 the mainstream flow comes down drastically in the whole region and mixes with coolant jet flow. However, for the case with lands, since the lateral flow has been restricted, the coolant flow tends to climb the side wall of lands and reach the top surfaces of lands to mix with the mainstream flow. In the slot region, the updraft coolant flow mixes with the mainstream flow above the upper wall of the exit before X/H=4. After X/H=4 the mainstream flow comes down and mixes with the coolant flow. Compared to the case without lands, the streamwise velocity contour shows less high-speed region in the cross planes downstream. It indicates the existence of lands limits the invasion of mainstream flow to some extent.

As shown in Figure 4.35, for the case without lands at a high blow ratio, on one hand the coolant flow spreads laterally and mixes with the downdraft mainstream flow at lower level close to the bottom in the side regions, which is different from the findings at low blow ratio. On the other hand, the mainstream flow suddenly comes down near the exit and mixes with the coolant flow. This downdraft flow pattern is kept until the end cross plane. For the case with lands, since the lateral flow has been restricted, the velocity vectors are much shorter than those for the case without lands and the coolant flow tends to form vortices at both sides of the exit. The vorticity contour has been shown in Figure 4.32. The slot flow tries to climb the wall of lands and meets the down draft flow around the corner edges of the lands. High vorticity contours were also found in this region.
Figure 4.34. Flow field comparisons at different distance downstream.
Figure 4.35. Flow field comparisons at different distance downstream.
According to the flow features revealed in the PIV measurements, the three-dimensional flow structure was reconstructed schematically. Figure 4.36(a) shows the flow structure in the trailing edge region at low blow ratios. It can be shown that the mainstream flow tends to mix with the coolant flow downstream of the slot. The coolant flow in the slot tends to climbing up around the edges of lands due to the entrainment of high-momentum mainstream
flow. Figure 4.36(b) shows the flow structure in the trailing edge region at high blow ratios. The mainstream flow is entrained by the high-momentum coolant flow, which results in the downward flow pattern of the mainstream flow. But the high-momentum coolant flow keeps a laminar boundary layer flow above the surface of the slot. The mainstream flow near the edges of lands tends to climbing down around the edges due to the entrainment.

Another point worth considering is the turbulent kinetic energy in the cross plane at the end. It is observed from Figure 4.37 that the flow is more chaotic with higher turbulent kinetic energy for the case without lands than that with lands for the case of $M=1.6$. The larger T.K.E value indicates more eddy mixing between the mainstream and coolant flow which will result in lower cooling effectiveness. The T.K.E could be also related to the total pressure loss which is also significant for the design of the trailing edge cooling. Larger turbulent kinetic energy generates more total pressure loss. The trailing edge design with lands in the present study might induce less aerodynamic loss after the mixing.

Figure 4.37. Averaged 3-D turbulent kinetic energy at $X/H=8$ at blow ratio of $M=1.6$
CHAPTER 5. CONCLUSIONS AND FUTURE WORK

In this chapter, the major objectives met in this dissertation are summarized and discussed. Future work with respect to flow-structure interaction in a tornado-like wind and trailing edge film cooling is also suggested.

5.1. The Characteristics of a Tornado-like Vortex

The PIV measurement results reveal clearly that a tornado-like vortex is a very complex, highly turbulent, three-dimensional vortex flow. In addition to a strong upward flow in the outer region and downdraft jet flow in the vortex core along vertical direction, axisymmetric flow pattern in the form of a well-defined vortex structure can be seen clearly in the horizontal planes. The flow field of a tornado-like vortex in each horizontal plane could be divided into two regions: an inner core region and an outer region. In the inner core region, air streams were found to flow concentrically with the wind speed increasing linearly with the increasing radial distance away from the center of the tornado-like vortex. The airflows inside the tornado-like vortex core rotate like a rigid column with the flow velocity vectors being tangential and the radial components being negligible (i.e., $v_r \approx 0$). Wind speed was found to reach its maximum value at the outer boundary of the tornado-like vortex core, and then, begins to decrease with increasing distance from the center of the vortex core in the outer region. The flow velocity vectors in the outer region of the tornado-like vortex were found to have significant radial components, which results in a strong spiral motion that sucks surrounding air flowing towards the vortex core.

To the author’s best knowledge, this work represents the first comprehensive PIV measurements on the flow structure of such a complex and large-size tornado like vortex.
5.2. Flow-structure Interaction for a Building Model in a Tornado-like Vortex

By using the world-largest tornado simulator of Iowa State University, an experimental study was conducted to quantify the characteristics of wake vortex and flow structures around a low-rise gable-roofed building model as well as the wind loads and pressure acting on the test model induced by tornado-like winds. The detailed flow field measurements were correlated with the wind loads and pressure data to elucidate underlying physics, and to quantify the dynamics of flow-structure interactions in a tornado-like vortex.

The wind loads (both force and moments) acting on the test model induced by tornado-like wind were found to vary significantly with the position of the test model relative to the center of the tornado-like vortex. The maximum aerodynamic force in tangential direction and radial direction and corresponding moments were found to reach their maximum values when the test model was mounted on the outer boundary of the tornado-like vortex core. The maximum lift force was observed at about $R/R_o=0.5$, which was confirmed with the pressure distribution data. Unlike those in straight-line winds, the wake vortex and flow structures around the test model in tornado-like winds were found to become quite unsymmetrical related to the approaching flow. The wake vortices in vertical plane were found to be closer to the leeward gable roof. The orientation angle of the test model related to the tornado-like vortex was found to have considerable effects on the wake vortex and flow structures around the test model as well as the resultant wind loads. Interestingly, wind loads acting on the square-shaped high-rise model were found to be maximum at orientation angle $OA \approx 30-45$ degrees, instead of $OA=45$ degrees as that observed for the same test models placed in straight-line winds. The lift force acting on the gable-roofed model, which is the most
important force according to the damage pattern induced by tornado, was found to be maximum at orientation angle \( OA \approx 15 \sim 30 \) degrees. This maximum value is almost 3.5 times higher than the maximum lift coefficient in straight-line wind. The force in tangential direction reached the maximum value at orientation angle \( OA \approx 45 \) degrees, which is less than \( OA=52.5 \) degrees in straight-line wind.

This work represents the first comprehensive study on the flow characteristics around a building model combined with the measurements of wind loads and pressure acting on the building models.

5.3. **Film Cooling Effect on a Trailing Edge**

The influence of blow ratio and existence of lands on the film cooling effectiveness was investigated experimentally in a low speed wind tunnel. The film cooling effectiveness on the cutback region of trailing edge was measured by using the pressure sensitive paint (PSP) technique. Before the PSP applied to the study in trailing edge cooling, a careful calibration of PSP were conducted by using a self-designed calibration cell.

It has been found that increasing blow ratio increases the film effectiveness along streamwise direction but decrease the film effectiveness in spanwise direction for the case without lands. Therefore, optimizing the blow ratio involves finding a balance between uniform lateral coverage and the distance covered downstream of the breakout. For the case with lands, the coolant with high blow ratios prevents effective entrainment of mainstream flow onto the slot valleys; as a result, it covers more area downstream of the cutback region. But in spanwise, the coolant flow with high blow ratio restricts the lateral spreading and mixing process hence it is not easy to cover the top surfaces of lands.
The existence of lands enhance the entrainment of mainstream flow on to the cutback slots at low blow ratio which results in relatively bad cooling effectiveness in the cutback region. However, at high blow ratio, it prevents the spreading of coolant flow which leads to ideal cooling effectiveness in the cutback slots region although the coolant cannot perform very well on the top surface of lands at high blow ratio.

To the author’s best knowledge, this work represents the first investigation of the trailing edge film cooling effectiveness by using a relatively new technique, pressure sensitive paint (PSP), considering different blow ratios and land effect.

5.4. The Characteristics of the Trailing Edge Cooling Flow

In this experimental study, a high-resolution Stereo-PIV system was used to conduct the comprehensive flow measurements in the slot region of a test setup simulating typical airfoil trailing edge slots. The effect of blow ratio and the existence of lands on the mixing flow were studied by conducting a comprehensive comparison of the vortical flow structures.

The 2-D PIV measurement results for the case without lands indicate that the region at 4H~6H could be an critical mixing region between the mainstream flow and coolant flow at low blow ratios. The region at 6H~8H could be the critical mixing region under large blow ratios, i.e. M=1.1, M=1.6.

The 3-D Stereo-PIV measurement results indicate that the streamwise vortices would play an important role in the mixing process. The effect of blow ratios and the existence of lands on the flow feature were analyzed combining with the cooling effectiveness results. For the case without lands, the coolant flow with high blow ratio tends to cover more area in streamwise direction and results in high cooling effectiveness downstream. But in the lateral direction, the coolant flow entrainment brings more mainstream flow onto the bottom surface
and leads to relatively low film effectiveness on the regions between slots for a high blow ratio. The existence of lands could constrain the lateral spreading flow and induce the coherent vortical flow in the slots. This will result in effect on the final cooling effect which has been confirmed by the film cooling effectiveness measurements. From the view of flow characteristics at high blow ratio, the insertion of lands tends to benefit the cooling effect in the slot valley, which agrees with the found in film cooling effectiveness measurements.

To the author’s best knowledge, this work represents the first experimental investigation of the flow characteristics of the trailing edge cooling flow considering different blow ratios and land effect.

5.5. **Future Work**

As for the investigation of the flow around a building model and resultant wind force in tornado-like vortex, the present study only considers a single building model in a static tornado-like vortex. For a more practical application, a group of buildings (like a community) should be considered in a moving tornado-like vortex (like a moving natural tornado). A more realistic boundary condition may be applied in the future study.

As for the study on trailing edge film cooling, a dual-plane stereoscopic PIV might be necessary to show the three-dimensional vortical structure in the slot flow instantaneous. Based on that, more accurate analysis of the unsteady effect of the flow on the film cooling effect could be realized, and the numerical simulation could be verified experimentally. The Binary UniCoat is a very bright paint, however, the temperature sensitivity of the paint if relatively high compared to FIB based paints. Therefore, FIB based paints can be applied to reduce the errors in the PSP measurements on film cooling effectiveness.
APPENDIX A. THE DESIGN OF CALIBRATION CELL

A self-designed calibration cell was used to conduct the calibration of pressure sensitive paint. Figure A-1 shows the blueprint of the assembly calibration cell. It can be divided into three main parts. The bottom part A, shown in Figure A-2, is used for water circulation to control the temperature on the sample plate.
Figure A-3. Bottom face

The middle part B, shown in Figure A-4, is designed to connect part A and part C. The combination of part A and part C forms a water channel inside, and is sealed up to enable the water flow without leaks. The threaded hole facing up in the middle of part B is designed for the sample plate to mount.

Figure A-4. Middle part B
The sample plate, shown in Figure A-5, is designed for painting. The pressure sensitive paint was painted on the top surface of the sample plate. In order to get better heat conductivity in the plate, the material copper was used to make this plate.

![Figure A-5. Sample plate](image)

The upper part A, shown in Figure A-6, is designed to hold a silica quartz window and connect the pressure air control. A circular silica window can be mounted in the threaded hole facing up. Air pipes can be connected through the threaded holes at both sides.

![Figure A-6. Upper part C](image)
A ring bolt, shown in Figure A-7, was designed to fix a silica quartz window. Figure A-8 shows the vertical slice view of the final assembly.

Figure A-7. The bolt to fix a silica quartz plate

Figure A-8. The slice view of assembly
APPENDIX B. THE DESIGN OF A TRAILING EDGE MODEL

Figure B-1 shows a schematic of the trailing edge of a turbine blade. In order to study the principle flow characteristics of the trailing edge flow, a simplified trailing model was designed to conduct the experimental study. Figure B-2 and Figure B-3 shows the two parts of the trailing edge model. Figure B-4 shows the land model.

Figure B-1. A schematic of the trailing edge of a turbine blade

Figure B-2. The upper part of the trailing edge model
Figure B-3. The lower plate of the trailing edge model

Figure B-4. The land of the trailing edge model
APPENDIX C. SYNCHRONIZATION SETUP FOR PIV

Figure C-1. The synchronization setup for laser and camera

Note: A, B, C, D, E are output channels of the Delay Generator

Figure C-1. The synchronization setup for laser and camera
The following timing settings are default values used for general PIV measurements:

<table>
<thead>
<tr>
<th>Pulse Output Channel</th>
<th>Time Delay</th>
<th>Pulse Width</th>
</tr>
</thead>
<tbody>
<tr>
<td>Channel A (to flash lamp #1)</td>
<td>0</td>
<td>100 µs</td>
</tr>
<tr>
<td>Channel B (to Q-switch #1)</td>
<td>200 µs</td>
<td>2 ms</td>
</tr>
<tr>
<td>Channel C (to flash lamp #2)</td>
<td>$\Delta t$ for PIV measurement</td>
<td>100 µs</td>
</tr>
<tr>
<td>Channel D (to Q-switch #2)</td>
<td>$\Delta t + 200 \mu s$</td>
<td>2 ms</td>
</tr>
<tr>
<td>Channel E* (to Pixelfly Camera)</td>
<td>$170 \mu s \left( t_{ed} \right)$</td>
<td>$100 \mu s \left( t_{ew} \right)$</td>
</tr>
</tbody>
</table>

Table C-1. The timing setting for PIV measurement

* Note: following requirement should be met for the timing setting of the Channel E (time delay, $t_{ed}$ and pulse width, $t_{ew}$) in order to make sure that the two laser pulses can be imaged at two PIV image frames:

$$200 \mu s + 130 \text{ ns} < (\approx 1 \mu s \text{ delay of camera}) + t_{ed} + t_{ew} < \Delta t + 200 \mu s - 70 \text{ns}$$
APPENDIX D. THE PRINCIPLE OF STEREOSCOPIC PIV

A three component PIV (Stereo-PIV) is based on the same fundamental principle as human eye-sight: Stereo vision. Our two eyes see slightly different images of the world surrounding us, and comparing these images, the brain is able to make a 3-dimensional interpretation. With only one eye, it is perfectly able to recognize motion up, down or sideways, but it is difficult to judge distances and motion towards or away from you. Here, cameras play the role of "eyes". The most accurate determination of the out-of-plane displacement (velocity) is accomplished when there is 90° between the two cameras.

Figure D-1. Scheimpflüg configuration for stereoscopic PIV

Steroscopic PIV uses two cameras to view a flow field from two perspectives so that the out-of-plane velocity component can be measured. The two components of velocity nominally perpendicular to the camera optical axis are measured from each camera viewpoint.
The pair of two-dimensional velocity vectors for a point in the flow is then combined to yield a three-dimensional velocity vector. By combining the vector fields from the two cameras, the three-dimensional velocity field for the measurement plane in the fluid is captured. The Scheimpflüg configuration of PIV setup was used in present study. The Scheimpflüg configuration requires the object plane, image plane, and lens principle plane to intersect at a common point, as shown in Figure D-1. By tilting the image sensor plane and the lens principle plane to the Scheimpflüg condition, the plane of best focus can be adjusted so that it is aligned with the light sheet.

Since the Scheimpflüg arrangement introduces perspective distortion, as shown in Figure D-2, a correction method is required to remove the distortion from the vector fields so that the velocity vectors no longer represent the distorted image, but the authentic area of interest. Therefore, a calibration was performed to measure and correct the perspective distortion due to the camera tilt and any other image distortions in the optical system. A calibration target with marker points at known locations is aligned to the laser light sheet. By recording and analyzing the images of the target, mapping functions were created using third-order calibration polynomial equations for each camera. Mapping function is the matrix which transforms the pixel coordinate system into real coordinate systems. By using these mapping functions, the two cameras do not have to be precisely aligned because they correct the center of the fields of view for the two cameras. The mapping functions also correct distortions caused by lens aberrations.
Figure D-2. Image perspective effect due to camera tilt in Scheimpflüg arrangement
REFERENCES


ACKNOWLEDGEMENTS

I would like to take this opportunity to express thanks from the bottom of my heart to those who helped me with various aspects of conducting research and the writing of this thesis. First and most, I would like to thank my major advisor Dr. Hui Hu for his guidance, inspiration, patience and encouragement throughout my PhD study. His insights and words of encouragement have often inspired me and renewed my hopes for completing my graduate education in a relatively short time. I would also like to thank my committee members for their efforts and contributions to this work: Dr. Tom Shih, Dr. Zhijian Wang, Dr. Terry Meyer and Dr. Chris R. Rehmann. I would additionally like to thank Dr. Partha Sarkar and Dr. Fred L. Haan for their guidance throughout the initial stage of my graduate career.

I would like to thank Mr. Bill Richard, Mr. Kevin Brownfield and Mr. Thomas J. Elliott for their help on building experimental setups during this study. I would also like to thank all staffs in the main office in our department, especially Delora Pfeiffer and Gayle Fay, for their help to prepare a lot of paper works during my PhD study.

I would like to specially thank Dr. Wei Zhang (for her suggestions on the research on tornado-like vortex), Mr. Vasanth Balaramudu(for his help in conducting pressure measurement on the gable-roof building model), Mr. Anand Gopa Kumar (for his help in building the setup for the research of trailing edge cooling), Mr. Hirofumi Igarashi (for his help in conducting the stereoscopic PIV measurements), Mr. Matt Martin and Mr. Travis Grager (for checking the grammar errors in this thesis) , and my girl friend Danlei Wang (for her help in checking many format details in this thesis ).