Frequency Effect on Peak Pressure Coefficients Using the Narrowband Synthesis Random Flow Generator (NSRFG) Method

Zayuris Del Carmen Atencio Mojica

University of Arkansas, Fayetteville

Follow this and additional works at: https://scholarworks.uark.edu/etd

Citation

This Thesis is brought to you for free and open access by ScholarWorks@UARK. It has been accepted for inclusion in Graduate Theses and Dissertations by an authorized administrator of ScholarWorks@UARK. For more information, please contact scholar@uark.edu.
Frequency Effect on Peak Pressure Coefficients Using the Narrowband Synthesis Random Flow Generator (NSRFG) Method

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Civil Engineering

by

Zayuris Atencio
University of Arkansas
Bachelor of Science in Civil Engineering, 2019

July 2021
University of Arkansas

This thesis is approved for recommendation to the Graduate Council.

_________________________
R. Paneer Selvam, Ph.D.
Thesis Director

_________________________
Eric Fernstrom, Ph.D.
Committee Member

_________________________
Ernie Heymsfield, Ph.D.
Committee Member
Abstract

The collapse of the Tacoma Narrows Bridge in Washington state in 1940 due to high wind forces gave structural engineers notice of the importance of understanding the behavior of wind forces on infrastructure. Wind forces play an important role in the design of buildings and other structures. ASCE 7-16 is the manual used for the design of these buildings and other structures throughout the country. These design criteria have been approved based on research projects involving experimental procedures. For the design of wind loads, ASCE 7-16 uses criteria based on wind tunnel (WT) tests. However, since the development of computing software, numerical methods based on computational fluid dynamics (CFD) have been adopted to study peak pressures produced by wind forces on infrastructure. Once CFD measurements show a good agreement with field measurements and experimental measurements, CFD has the potential to become a cost-effective tool saving time and money.

For this study, peak pressures are computed using CFD procedures and compared with 1:6 scale WT measurements for the Texas Tech University (TTU) building. The Narrowband Synthesis Random Flow Generator (NSRFG) method is used to calculate the inflow turbulence. The behavior of the pressure and the velocity at the building location in the computational domain without building using the NSRFG inflow turbulence method is also investigated. The number of segments of the wind spectrum (N) is varied where a total of 4 different cases are considered. Percentages of error between CFD measurements and WT measurements are investigated. The causes of these errors are discussed as well as limitations of the inflow turbulence generator being used.
Acknowledgments

First, I thank God for giving me the strength and motivation to be here.

I acknowledge the financial support from the University of Arkansas Graduate School. Also, special thanks go to the University of Arkansas Libraries for letting me work as a graduate assistant while taking my classes and working on my research within the civil engineering department.

I also would like to thank my advisor, Dr. R. Paneer Selvam, for his support and encouragement to develop this thesis. Special thanks to Mr. Sumit Verma and Ms. Zahra Mansouri for their help throughout my master’s program.

Finally, but not least, I would like to thank my mom, Rosalia, my dad, Mario, my sister, Yasuris, and all my family back in Panama. Without their support and love, I would not have been able to achieve my goals.
# Table of Contents

1  Introduction .................................................................................................................. 1

1.1 Overview ..................................................................................................................... 1

1.2 Research Objectives .................................................................................................. 4

2  Background .................................................................................................................... 6

2.1 The Texas Tech University (TTU) Building .............................................................. 6

2.2 Atmospheric Boundary Layer (ABL) ......................................................................... 7

2.3 Turbulence .................................................................................................................... 8

2.4 Wind Tunnel (WT) Testing Methods ......................................................................... 10

2.5 Computational Fluid Dynamics (CFD) ..................................................................... 12

2.6 Inflow Turbulence Methods ......................................................................................... 12

2.6.1 The Precursor Database Method ......................................................................... 13

2.6.2 The Recycling Method ......................................................................................... 13

2.6.3 The Synthetic Turbulence Method ....................................................................... 14

2.7 Synthetic Inflow Turbulence Methods Status ........................................................... 15

3  Numerical Models .......................................................................................................... 18

3.1 Navier-Stokes (NS) Equation ..................................................................................... 18

3.2 Large Eddy Simulations (LES) .................................................................................. 19

3.3 Finite Difference Method (FDM) Application on the Grid Spacing $h$ ......................... 20

3.4 Maximum and Minimum Frequencies used in the Inflow Turbulence Models .......... 21

3.4.1 Wind Tunnel (WT) Frequency ............................................................................... 22

3.4.2 Non-Dimensional Grid Frequency ($f_{grid}$) ......................................................... 24

3.4.3 Wind Tunnel (WT) Frequency and Non-Dimensional Grid Frequency ($f_{grid}$) on Large Eddy Simulation (LES) Studies ......................................................................................................................... 25

3.5 Computational Model and Inflow Turbulence Method Details ................................ 26
3.5.1 Computational Region and Boundary Conditions ........................................... 26
3.5.2 Inflow Turbulence Details .................................................................................. 28
  3.5.2.1 “Yif1.out” Description .............................................................................. 29
  3.5.2.2 Data Preparation ......................................................................................... 30
    3.5.2.2.1 Dimensional Frequencies Calculations .............................................. 34
    3.5.2.2.2 The Number of Spectral Segment (N) ............................................... 35
    3.5.2.2.3 Number of Spectral Segment (N) Variations .................................. 37
4 Results and Discussion ............................................................................................. 38
  4.1 Computation without Building: Inflow Turbulence Effect on Interior Velocity and Pressure ........................................................................................................... 38
  4.2 Computation with Building: Comparison of CFD Mean Peak Pressure Coefficients with WT Peak Pressure Coefficients ......................................................... 44
  4.3 Computation with Building: Comparison of CFD Maximum Peak Pressure Coefficients with WT Peak Pressure Coefficients .................................................. 48
  4.4 Computation with Building: Comparison of CFD Minimum Peak Pressure Coefficients with WT Peak Pressure Coefficients .................................................. 52
5 Conclusions and Future Work .................................................................................. 57
6 References .................................................................................................................. 60
7 Appendix .................................................................................................................... 63
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1. Frequency region resolved and modeled by LES (Mansouri et al., 2020)</td>
<td>3</td>
</tr>
<tr>
<td>Figure 1.2. Time history of pressure and velocity of (a) $f_{max} = f_{grid} = 4$ and (b) $f_{max} = 10 &gt; f_{grid} = 4$</td>
<td>4</td>
</tr>
<tr>
<td>Figure 2.1. TTU building structure (Moravej, 2018)</td>
<td>7</td>
</tr>
<tr>
<td>Figure 2.2. Wind velocity profile in the ABL (Ghanadi et al., 2017)</td>
<td>8</td>
</tr>
<tr>
<td>Figure 2.3. Laminar and turbulent flow (Cengel &amp; Cimbala, 2014)</td>
<td>9</td>
</tr>
<tr>
<td>Figure 2.4. Typical wind tunnel setup for boundary layer modeling (Immer, 2016)</td>
<td>11</td>
</tr>
<tr>
<td>Figure 2.5. Recycling technique (Aboshosha et al., 2015)</td>
<td>14</td>
</tr>
<tr>
<td>Figure 2.6. (a) Mean and (b) rms wind pressure coefficients (Yu et al., 2018)</td>
<td>16</td>
</tr>
<tr>
<td>Figure 3.1. Comparison of exact sine waves with the FDM method after 2 cycles of transport (a) wavelength $L=2h$ and (b) wavelength $L=4h$ (Mansouri et al., 2020)</td>
<td>21</td>
</tr>
<tr>
<td>Figure 3.2. Turbulence power spectra ($n=$ frequency, $z=$ model building height, $U=$ mean wind speed at building eave height, $S=$ turbulence power spectra) (Moravej, 2018)</td>
<td>23</td>
</tr>
<tr>
<td>Figure 3.3. Frequency region resolved and modeled by LES (Mansouri et al., 2020)</td>
<td>25</td>
</tr>
<tr>
<td>Figure 3.4. TTU building computational region (a) plan view and (b) elevation view</td>
<td>27</td>
</tr>
<tr>
<td>Figure 3.5. Boundary conditions for the numerical modeling (Mansouri et al., 2020)</td>
<td>28</td>
</tr>
<tr>
<td>Figure 4.1. The time history of (a) pressure and (b) velocity without using the inlet generator at a monitorpoint at the building location (Lebovitz, 2017)</td>
<td>39</td>
</tr>
<tr>
<td>Figure 4.2. The time history of (a) pressure and (b) velocity using the inlet generator at a monitorpoint at the building location (Lebovitz, 2017)</td>
<td>40</td>
</tr>
<tr>
<td>Figure 4.3. Effect of inflow turbulence on interior velocity and pressure for $N=50$ at the building location without building (a) $f_{max}=4$ and (b) $f_{max}=10$</td>
<td>42</td>
</tr>
<tr>
<td>Figure 4.4. Effect of inflow turbulence on interior velocity and pressure for $N=100$ at the building location without building (a) $f_{max}=4$ and (b) $f_{max}=10$</td>
<td>43</td>
</tr>
</tbody>
</table>
Figure 4.5. Effect of inflow turbulence on interior velocity and pressure for N=500 at the building location without building (a) $f_{max}=4$ and (b) $f_{max}=10$ ................................. 43

Figure 4.6. Effect of inflow turbulence on interior velocity and pressure for N=1000 at the building location without building (a) $f_{max}=4$ and (b) $f_{max}=10$ ................................. 44

Figure 4.7. Centerline of the TTU building with the roof as the origin................................................. 45

Figure 4.8. Mean pressure coefficients (Cp-mean) for N=50 (a) $f_{max}=4$ and (b) $f_{max}=10$ .... 46

Figure 4.9. Mean pressure coefficients (Cp-mean) for N=100 (a) $f_{max}=4$ and (b) $f_{max}=10$ ..... 47

Figure 4.10. Mean pressure coefficients (Cp-mean) for N=500 (a) $f_{max}=4$ and (b) $f_{max}=10$ .... 47

Figure 4.11. Mean pressure coefficients (Cp-mean) for N=1000 (a) $f_{max}=4$ and (b) $f_{max}=10$. 48

Figure 4.12. Maximum pressure coefficients (Cp-max) for N=50 (a) $f_{max}=4$ and (b) $f_{max}=10$ 50

Figure 4.13. Maximum pressure coefficients (Cp-max) for N=100 (a) $f_{max}=4$ and (b) $f_{max}=10$ 50

Figure 4.14. Maximum pressure coefficients (Cp-max) for N=500 (a) $f_{max}=4$ and (b) $f_{max}=10$ 51

Figure 4.15. Maximum pressure coefficients (Cp-max) for N=1000 (a) $f_{max}=4$ and (b) $f_{max}=10$ 51

Figure 4.16. Minimum pressure coefficients (Cp-min) for N=50 (a) $f_{max}=4$ and (b) $f_{max}=10$. 54

Figure 4.17. Minimum pressure coefficients (Cp-min) for N=100 (a) $f_{max}=4$ and (b) $f_{max}=10$ 55

Figure 4.18. Minimum pressure coefficients (Cp-min) for N=500 (a) $f_{max}=4$ and (b) $f_{max}=10$ 55

Figure 4.19. Minimum pressure coefficients (Cp-min) for N=1000 (a) $f_{max}=4$ and (b) $f_{max}=10$ 55

Figure 7.1 IFS, IFE, and N calculation and DN variation ................................................................. 64
List of Tables

Table 3.1. DN, IFS, IFE, and N values for non-dimensional and dimensional frequencies $f_{min} = 0.1$, $f_{max} = 4$, and $f_{min} = 0.1$, $f_{max} = 10$ ................................................................. 37

Table 7.1 Computing time for each performed computer simulation for Cp calculations .......... 63
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TTU</td>
<td>Texas Tech University</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>RFG</td>
<td>Random Flow Generation</td>
</tr>
<tr>
<td>DRFG</td>
<td>Discrete Random Flow Generation</td>
</tr>
<tr>
<td>CDRFG</td>
<td>Consistent Discrete Random Flow Generation</td>
</tr>
<tr>
<td>NSRFG</td>
<td>Narrowband Synthesis Random Flow Generator</td>
</tr>
<tr>
<td>WT</td>
<td>Wind Tunnel</td>
</tr>
<tr>
<td>NOAA</td>
<td>National Oceanic and Atmospheric Administration</td>
</tr>
<tr>
<td>ASCE</td>
<td>American Society of Civil Engineers</td>
</tr>
<tr>
<td>WAWS</td>
<td>Wave Superposition</td>
</tr>
<tr>
<td>ABL</td>
<td>Atmospheric Boundary Layer</td>
</tr>
<tr>
<td>NS</td>
<td>Navier-Stokes</td>
</tr>
<tr>
<td>CAARC</td>
<td>Commonwealth Advisory Aeronautical Council</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulations</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>FDM</td>
<td>Finite Difference Method</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs-Lewy</td>
</tr>
</tbody>
</table>
1 Introduction

1.1 Overview

Strong winds such as hurricanes, tornadoes, and windstorms can cause extensive damage to infrastructure resulting in economic and human losses. In 2019, more than sixteen thousand wind damages were registered in the United States by the National Oceanic and Atmospheric Administration (NOAA); with Arkansas having approximately 300 registered damages (NOAA, 2019). These statistics give an idea about the importance of understanding the behavior of wind forces on buildings.

Peak pressures on buildings have been significantly studied for many years. Peak pressures are calculated and analyzed by using field measurement methods, experimental methods, and numerical methods. Field measurement methods refer to studies performed in-field where a specific structure is located, experimental methods refer to the use of wind tunnels (WT), and numerical methods refer to the use of computational fluid dynamics (CFD) procedures.

ASCE 7-16 is the manual used by engineers to determine the wind load requirements for buildings and other structures. ASCE 7-16 uses data from experimental methods to calculate peak pressures on buildings. However, field measurements are the most accurate method for determining wind loads behavior (Dyrbye & Hansen, 1997). Field measurements or full-scale measurements like the ones performed on the Texas Tech University (TTU) building have been used to validate WT measurements (Dyrbye & Hansen, 1997). However, since the advancement of computing technologies with higher processing speeds and storage, CFD procedures have played an important role in the investigation of these wind forces. CFD measurements consist of computer simulations that generate turbulence in the inflow. Keating et al. (2004) classified the inflow turbulence techniques into three categories: (a) precursor database method, (b) recycling
method, and (c) synthetic turbulence method. These categories are described in Section 2.6 of this thesis. For this work, the synthetic turbulence method is used due to its capability of not requiring any prior inflow domain making it less expensive and computationally faster.

Some limitations can be found in both WT measurements and CFD procedures when results are compared to field measurements. For components and cladding, ASCE 7-16 uses peak pressure coefficients in the range of -3.2 for low-rise buildings. However, Mehta et al. (1992) and Moravej (2018) report peak pressures in the range of -8 to -15.6 for the field measurement of the Texas Tech University building. Richards et al. (2007) in their work, based on field measurements, report peak pressure coefficients also in the range of -8.0 for the Silsoe Cube Building. Both buildings mentioned above are considered to be low-rise structures.

The high error in peak pressure measurements from WT is due to limitations in producing the field wind spectrum as discussed in Mooneghi et al. (2016). Mooneghi et al. (2016) propose modifications to the WT measurement procedures when calculating peak pressure coefficients. In recent years due to extensive developments in computer hardware, computational fluid dynamics (CFD) has emerged as an alternative tool for evaluating wind loads on structures. If the CFD methods are well-validated with field and WT measurements, CFD will become a cost-effective tool.

Mansouri et al. (2020) investigated the Consistent Discrete Random Flow Generation (CDRFG) inflow turbulence generator, a numerical method introduced by Aboshosha et al. (2015) based on synthetic turbulence methods, to compute peak pressure coefficients and found that they predict more than 100% higher peak pressure coefficients than WT measurements. They also proposed modifications in grid spacing all around the computational domain and maximum frequency based on grid spacing to be considered in the inflow turbulence calculation. In the synthetic
turbulence generator methods, the field or WT measured turbulence spectrum will be provided as input as shown in Figure 1.1.

![Turbulence Spectrum](image)

**Figure 1.1. Frequency region resolved and modeled by LES (Mansouri et al., 2020)**

The wind spectrum provides the energy in the wind for different frequencies. In the measured wind spectrum, there is no computational grid restriction. In CFD models, however, the grid spacing restricts the maximum frequency to be considered. If the $f_{\text{max}} > f_{\text{grid}} = f_{\text{LES}}$ range is considered in the inflow, spurious pressure errors are produced. As shown in Figure 1.2 (a), the pressure in time when $f_{\text{max}} = f_{\text{grid}}$ is smoother and more continuous than the pressure when $f_{\text{max}} > f_{\text{grid}}$, shown in Figure 1.2 (b), where the pressure presents more variation in time becoming quite discontinuous. Here, $f_{\text{grid}}$ is calculated as $\frac{U_{\text{ref}}}{4h}$, where $h$ is the grid spacing.

Mansouri et al. (2020) considered 100 segments of the wind spectrum (N) in their inflow turbulence computation. In this work, the number of segments (N) in the energy spectrum will be varied from 50 to 1000 and its effect on the pressure and the velocity variation in time as well as in the peak pressure coefficients are observed. Thus, the recently published Narrowband Synthesis Random Flow Generator (NSRFG) inflow turbulence generator introduced by Yu et al.
(2018) will be investigated for their performance in computing the pressure and the velocity variation in time and the peak pressure coefficients.

![Graphs showing time history of pressure and velocity](image)

Figure 1.2. Time history of pressure and velocity of (a) $f_{\text{max}} = f_{\text{grid}} = 4$ and (b) $f_{\text{max}} = 10 > f_{\text{grid}} = 4$

1.2 Research Objectives

- Investigate the behavior of pressures and velocities when using the Narrowband Synthesis Random Flow Generator (NSRFG) as inflow turbulence generator. To perform this, the pressure and velocity time history is calculated at the building location but without building in the computational domain. Also, the number of spectral segments ($N$) is varied to evaluate their effect.

- Investigate the effect of maximum non-dimensional frequencies on the TTU building pressure by using the Narrowband Synthesis Random Flow Generator (NSRFG) inflow turbulence method and adopting a computational domain with a grid spacing of $H/16$. The number of spectral segment ($N$) is also varied. A total of 4 different cases are considered.
• Compare peak pressure coefficients resulting from CFD measurements of the TTU building with results of 1:6 scale WT measurements reported by Moravej (2018).
2 Background

Pressure on buildings due to wind forces are determined by using in-field testing methods, wind tunnels, and computational fluid dynamics (CFD) techniques. For this work, peak pressure coefficients for the Texas Tech University (TTU) building are calculated using numerical methods and compared with existing data from wind tunnel (WT) measurements.

2.1 The Texas Tech University (TTU) Building

The Texas Tech University (TTU) building is located in Lubbock, Texas on the campus of the Texas Tech University. The TTU building has been used for the study of wind forces on low-rise buildings by collecting field data. This data is used to validate CFD and WT measurements.

The structure consists of a 9.1 m x 13.7 m x 4.0 m (30 ft x 45 ft x 13 ft) flat roof building with prefabricated metal mounted on a rigid frame steel undercarriage (Levitan et al., 1991). The building also has a smooth surface, no architectural features, and a 49 m (160 ft) meteorological tower on the west side (Levitan et al., 1991). Figure 2.1 shows a picture of the TTU building.

For years, the TTU building has been used as an experimental structure to study the behavior of wind forces in low-rise buildings. Levitan et al. (1991) reported field pressure measurements on the TTU building calculating mean, root mean square (rms), and peak pressure coefficients on the walls and roof along the centerline of the building. Later, Selvam (1997) reported peak pressure coefficients for the TTU building using large eddy simulations through CFD techniques. Following this, Moravej (2018) performed WT testing procedures for the TTU building using five model scales. Results from these studies were compared with available field data allowing researchers to validate and analyze results with the existing field data.
For this study, the TTU building is selected as the structure to analyze and compare peak pressures generated by the Narrowband Synthesis Random Flow Generator (NSRFG) inflow turbulence method with existing wind tunnel (WT) data.

Figure 2.1. TTU building structure (Moravej, 2018)

2.2 Atmospheric Boundary Layer (ABL)

Wind varies its speed with height. Liu (1991) explains that the local mean velocity in a terrain, referred to as wind speed, is zero at the surface, and it increases with height above ground in a layer within approximately one kilometer from the ground. This layer is the atmospheric boundary layer. The wind profile is one of the many important characteristics of the ABL (Lebovitz, 2017), which along with the roughness of the terrain influences the behavior of the ABL. WT methods and CFD techniques take into consideration the ABL to simulate turbulence forces present on the surface when winds flow over buildings. Figure 2.2 shows a wind velocity profile in the ABL.
2.3 Turbulence

A flow can be laminar or turbulent. A laminar flow refers to a smooth and steady flow with a very low velocity (Mott & Untener, 2015). Laminar flows can be found in common daily life activities. The flow of oils at low velocities is an example of laminar flow (Cengel & Cimbala, 2014). On the other hand, a turbulent flow is defined as an unsteady and chaotic flow (Mott & Untener, 2015). The flow of low-viscosity fluids such as air at high velocities is typically turbulent (Cengel & Cimbala, 2014).

Laminar and turbulent flow can be seen in the smoke produced when one burns incense or a candle. As seen in Figure 2.3, the smoke formed first has a laminar flow, and after a while, the flow becomes turbulent.
Figure 2.3. Laminar and turbulent flow (Cengel & Cimbala, 2014)

The Reynolds number is a dimensionless number that is used to determine if a flow is laminar or turbulent. Eq. 2.1 shows the definition of the Reynolds number.

\[ R_e = \frac{UL}{\nu} \quad \text{Eq. 2.1} \]

\( U \) and \( L \) are velocity and length scales, respectively, of the flow and \( \nu \) is the kinematic viscosity of the fluid (Wyngaard, 2010). The Reynolds number is the ratio of the inertia force on an element of fluid to the viscous force (Mott & Untener, 2015). Therefore, a flow is turbulent when this ratio is high. However, the value of a Reynolds number will depend on geometry and flow conditions (Cengel & Cimbala, 2014). For example, flows in circular pipes with \( R_e \ll 2300 \) are considered as laminar flows while flows with \( R_e \gg 4000 \) are considered turbulent flows (Cengel & Cimbala, 2014). The flows present in the ABL are almost exclusively turbulent because of the apparent high Reynolds numbers (Lebovitz, 2017).
2.4 Wind Tunnel (WT) Testing Methods

Wind tunnels were first made to study the aerodynamic behavior of aerospace ships. The first wind tunnel in the world was created by British FH Wenham in 1871. It had a length of about 3 meters and wooden bellows open at both ends (Yu, 2016). Wenham’s principles and techniques helped the Wright brothers succeed at creating the world’s first airplane in 1903.

Wind tunnel methods in civil engineering were increasingly used after the Tacoma Narrows Bridge collapse occurred in Washington state in 1940. The reason for this bridge collapse was a flutter on the bridge caused by winds with velocities of about 42 mph. This event alerted structural engineers about the importance of a better understanding of wind forces on structures. After this incident happened, a trend started to test the structural designs in WT in a similar manner as the airplane designs are tested in WTs (Petroski, 1992).

Since wind tunnels were mainly used for aerospace ship tests, civil engineering infrastructure would not have the same accuracy when testing wind forces behavior due to the lack of turbulence generation present in the atmospheric boundary layer (ABL). Thus, in the 1950s, Martin Jensen formulated Jensen’s model law which states that the flow in the wind tunnel should be turbulent in the same way as the flow in the natural wind (Dyrbye & Hansen., 1997). Wind tunnels based on boundary layers started being made after Jensen’s model law, which improved the study of wind forces in structures. Figure 2.4 shows a typical WT based on boundary layers.
The main purpose of wind tunnel tests is to provide designers information on local wind patterns, wind loads, and wind-induced structural vibration (Liu, 1991). However, it does not need to be conducted for every structure. Liu (1991) states that this will depend on factors such as the cost of structures, the likelihood of wind problems, the complexity of the structure, and its importance.

The cost of the structure will be fundamental since wind tunnel test prices in the United States are usually high. Thus, if it is a multi-million-dollar structure, a wind tunnel test can be conducted. Structures with special geographical locations (e.g., a skyscraper in a hurricane area) or wind-sensitive structures (e.g., tall buildings and long-span bridges) will require wind tunnel tests due to their likelihood of wind problems (Liu, 1991). Structures with complex shapes and designs are mostly in need of wind tunnel tests since building codes and standards might not be available for these types of designs. Also, structures designed to hold a considerable number of lives (e.g., risk category IV buildings) will put more attention to wind forces behavior. Liu (1991) explains that even though building codes and standards already require higher wind loads for important structures, some designers and owners will prefer the use of WT tests to reduce the risk of large potential life losses.
Wind tunnels (WT) have been considered as trustworthy resources to analyze wind behaviors on structures. However, as it was already explained, most wind tunnel techniques hold expensive prices that are not always convenient or affordable for the design of some structures.

2.5 Computational Fluid Dynamics (CFD)

Wind tunnel modeling tests are commonly expensive and used mostly on multi-million-dollar structures or special projects. Since computer modeling has been improving in the last decades, computational fluid dynamics (CFD) has become the ideal tool for the study of wind forces on structures. Ding et al. (2019) state that computational simulations are evolving with the promise of becoming versatile, convenient, and reliable means of assessing wind load effects.

CFD consists of computer models that provide details such as pressures and velocities at every point of a computational domain. Selvam (2020) explains the steps to follow when working with CFD methods. The first step consists of generating the grid. Generating a grid usually takes time since it will depend on the simplicity of the domain that is being studied (e.g., shape of the structure). The second step is to solve the Navier-Stokes (NS) equations. For this step, inflow turbulence techniques are implemented. NS equations principles are explained in the Numerical Models part, Chapter 3, of this thesis. The third step is analyzing the results. Visualization software is usually used to perform this step.

2.6 Inflow Turbulence Methods

Keating et al. (2004) classified the inflow turbulence techniques into three categories (a) the precursor database method, (b) the recycling method, and (c) the synthetic turbulence method.
2.6.1 The Precursor Database Method

The precursor database method was introduced by Schlüter et al. (2004) and consists of a generated large eddy simulations (LES) inflow database. Aboshosha et al. (2015) explain that the precursor database method consists of two stages. The first stage involves a simulation of the wind to produce a temporal and spatial distribution of the incoming turbulent velocities that are saved in a database and used in the second stage, where the flow simulation is focused on the zone of interest (Aboshosha et al., 2015). The creation of the precursor database represents a long computational time causing this method to be computationally expensive. Due to this weakness, the precursor database method can be convenient only if the precursor database already exists.

2.6.2 The Recycling Method

The recycling method was used to generate inflow turbulence for smooth surfaces by Lund et al. (1998). Later, Nozawa et al. (2002) used Lund’s method to generate turbulent inflow data for both smooth and rough surfaces. The recycling method is pretty similar to the precursor database method and consists of two computational domains: the driver domain and the computational domain (Aboshosha et al., 2015). Aboshosha et al. (2015) explain that in the driver domain, the flow is recycled over a short domain until the flow becomes statistically stable. Flow characteristics on a mapping plane are stored and used as the inflow condition for the calculation domain. Figure 2.5 shows this procedure. The weakness of this method lays in the extensive computing time, which is also seen in the precursor database method, leading to a computationally expensive method. Also, the recycling method’s susceptibility to the roughness of the surfaces represents another weakness.
2.6.3 The Synthetic Turbulence Method

In contrast to the precursor and recycling methods, the synthetic turbulence method does not require any prior inflow domain, which makes it less expensive. Huang et al. (2010) classified synthetic turbulence method techniques into two groups.

The first group involves the use of a weighted amplitude wave superposition (WAWS) method, which results in a turbulent velocity field that satisfies both the targeted power and the cross-spectra (Aboshosha et al., 2015). However, there is a limitation with the WAWS method, and it lays in the fact that the resulting turbulence does not satisfy the continuity equation requiring a longer computing time at each time step (Selvam et al., 2019).

The second group involves the use of random flow generation (RFG) methods where Gaussian spectra are implemented to generate a divergent-free velocity field (Aboshosha et al., 2015). Unfortunately, RFG methods results do not approach the spectra generally encountered in the atmospheric boundary layer (ABL) (Selvam et al., 2019) making this method not good enough to be used in the analysis of wind forces. However, Huang et al. (2010) suggested the discrete random flow generation (DRFG) method to compute inflow turbulence that can generate
turbulent spectra comparable to those present in the ABL. Later, Aboshosha et al. (2015) improved Huang et al. (2010) method by proposing a new method called the Consistent Discrete Random Inflow Generation (CDRFG). This new method consisted of maintaining proper coherence among the resulting turbulence velocities (Aboshosha et al., 2015). Posterior to Aboshosha et al. (2015) new method, Yu et al. (2018) proposed a new improved inflow turbulence method based on the synthetic turbulence method. This method was named the Narrowband Synthesis Random Flow Generator (NSRFG) method, and it consisted of strictly maintaining a divergence-free condition, coherency function, and turbulent spectra of the ABL (Yu et al., 2018). This method tends to be much simpler and computationally more efficient than the CDRFG method.

2.7 Synthetic Inflow Turbulence Methods Status

Researchers have been adopting both the CDRFG method and the NSRFG method to generate inflow turbulence and analyze peak pressure coefficient behavior on buildings. Aboshosha et al. (2015) used the CDRFG method to compare the spectra of base moments of a tall building to results from WT measurements. Comparison between CDRFG results and WT measurements resulted in good agreement showing the efficacy of the CDRFG method. As mentioned before, Yu et al. (2018) introduced the NSRFG method where a typical ABL flow for the Commonwealth Advisory Aeronautical Council (CAARC) standard building model was generated. The spectra of the base moment, mean, and root mean square (rms) wind pressure coefficients were compared to CDRFG and WT measurements. Yu et al. (2018) reported that the base moment coefficients obtained from the numerical simulations were in good agreement with WT measurements. Mean pressure coefficients from NSRFG and CDRFG were also in good comparison with WT measurements as seen in Figure 2.6 (a). However, Figure 2.6 (b) shows
some discrepancies on the leeward walls and sidewalls of the building in the rms wind pressure coefficients when comparing numerical results from CDRFG and NSRFG measurements to WT measurements. Yu et al. (2018) explained that these differences were due to the fact that the mechanism of fluctuating pressure in numerical processes for rms was more complex than the one used for mean pressure. Thus, both the methods (i.e. the CDRGF and the NSRFG) have some limitations when generating inflow turbulence. However, despite the discrepancies explained above, the NSRFG method still seemed to be more accurate when comparing results to WT measurements.

![Graphs showing mean and rms wind pressure coefficients](image)

Figure 2.6. (a) Mean and (b) rms wind pressure coefficients (Yu et al., 2018)

Mansouri et al. (2020) studied the effect of different grid spacings on the TTU building peak pressures using the CDRFG method. Peak pressures were computed and compared to WT measurements. Mansouri et al. (2020) reported peak pressure coefficients on the windward and leeward sides as well as on the roof of the building. Peak pressure coefficients on the roof of the building were similar to WT measurements. However, peak pressure coefficients on the windward and leeward sides of the building had some errors when comparing them with WT measurements. Mansouri et al. (2020) stated that these discrepancies could be due to the inflow turbulence method used, which was the CDRFG method. Thus, further research is needed to
better understand the reason behind this discrepancy. As a result, for this study, the NSRFG method will be implemented to get CFD measurements for peak pressures on the same places for the TTU building and compare them with WT measurements.
3  Numerical Models

In this chapter, numerical processes used to generate inflow turbulence for the Texas Tech University (TTU) building using the Narrowband Synthesis Random Flow Generator (NSRFG) method are discussed.

3.1 Navier-Stokes (NS) Equation

Lebovitz (2017) explains that fluid motions can be described with the conservation of mass and the conservation of momentum forming a group of equations commonly known as the Navier-Stokes (NS) equations. The Navier-Stokes equations are unsteady, nonlinear, second-order, partial differential equations (Cengel & Cimbala, 2014), and they are considered the foundation of fluid mechanics. Cengel & Cimbala (2014) state that analytical processes to solve the NS equations are unobtainable except for very simple flow fields. Therefore, numerical processes are used to solve the NS equations.

A flow is classified as compressible or incompressible depending on the level of variation of density during flow (Cengel & Cimbala, 2014). Equations for the conservation of mass are usually referred to as the continuity equation. Eq. 3.1 describes the continuity equation in tensorial notation for compressible flow where the density ($\rho$) varies. Here, $i = 1$ to 3 for the 3D flow.

$$\frac{d \rho}{dt} + \frac{du_i}{dx_i} = 0 \quad \text{Eq. 3.1}$$

For this thesis, incompressible flow is considered meaning that the density ($\rho$) is a constant. Eq. 3.2 describes the continuity equation for an incompressible flow.

$$\frac{du_i}{dx_i} = 0 \quad \text{Eq. 3.2}$$
Lebovitz (2017) explains the equation for the conservation of momentum of an incompressible flow as:

\[
\frac{du_i}{dt} + u_j \frac{du_i}{dx_j} = -\frac{1}{\rho} \frac{dp}{dx_i} + \nu \frac{d^2u_i}{dx_j^2} + f_i
\]

Eq. 3.3

where \( p \) is the pressure, \( \nu \) is the kinetic viscosity of the fluid, and \( f_i \) includes body force fields such as gravity. Eq. 3.4 describes the Navier Stokes (NS) for incompressible flows (Cengel & Cimbala, 2014).

\[
\rho \frac{\bar{\nabla}V}{dt} = -\bar{\Delta}P + \rho \bar{g} + \mu \bar{\Delta}^2 \bar{\Delta}
\]

Eq. 3.4

3.2 Large Eddy Simulations (LES)

The Navier Stokes equations are solved by using numerical models. These procedures predict the turbulent flows that happen in the atmospheric boundary layer (ABL). There are two common numerical models used to predict turbulent flows, the direct numerical simulation (DNS) and the large eddy simulation (LES). The direct numerical simulation (DNS) is the most straightforward approach to the solution of turbulent flows (Piomelli, 1999). Piomelli (1999) explains that DNS methods involve the discretization of the NS equations where all scales in motion are solved. On the other hand, the large eddy simulation (LES) method is a technique intermediate between direct numerical simulation of turbulent flows and the solution of the Reynolds-averaged equations where only the smallest scales of turbulence are modeled (Piomelli, 1999). For this thesis, LES is chosen to generate the inflow turbulence flow using the NSRFG method.

LES consist of a filtering operation which is described in Eq. 3.5.

\[
\bar{f}(x) = \int_D f(x') G(x, x'; \bar{\Delta})dx'
\]

Eq. 3.5
D is the entire domain, \( G \) is the filter function based on the grid size and timestep, and \( \Delta \) is the filter width (Piomelli, 1999). After applying Eq. 3.5 and the filtering procedure, one obtains the filtered Navier-Stokes equations.

\[
\frac{\partial \bar{u}_i}{\partial x_i} = 0, \quad \text{Eq. 3.6}
\]

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} \left( \bar{u}_i \bar{u}_j \right) = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j}, \quad \text{Eq. 3.7}
\]

Piomelli (1999) explains that the effect of the small scales appears through a sub grid-scale (SGS) stress term which is described in Eq. 3.8.

\[
\tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \quad \text{Eq. 3.8}
\]

As stated before, in LES only a certain range of turbulence or frequencies are considered. Selvam et al. (2019) explain that the unresolved scales in LES are assumed as isotropic turbulence, and eddy viscosity is introduced in the case of Smogorinsky one equation model. This model accounts for the effect of remaining scales that were not resolved by the LES grid.

### 3.3 Finite Difference Method (FDM) Application on the Grid Spacing \( h \)

Grid spacing \( h \) in inflow turbulence generators represents an important role to obtain accurate turbulence models. Turbulent wind at a point can be represented as a sum of sine and cosine functions as in synthetic RFG methods where each function is a wave (Mansouri et al., 2020). Orszag (1979) states that for a grid spacing \( h \), the theoretical wavelength \( L \) in the form of sine and cosine function transported by a finite difference grid will be \( 2h \). However, Feriziger and Peric (2002) suggest refining the grid spacing to \( L=4h \) since it has more accurate results when computing the turbulence flow. Figure 3.1 shows a comparison of sine waves for \( L=2h \) and \( L=4h \).
where \( L=4h \) shows more accurate results when comparing them to the exact solution. Therefore, taking into consideration Feriziger and Peric’s (2002) suggestions, a wavelength of \( L=4h \) will be used for this study.

![Comparison of exact sine waves with the FDM method after 2 cycles of transport](image)

Figure 3.1. Comparison of exact sine waves with the FDM method after 2 cycles of transport (a) wavelength \( L=2h \) and (b) wavelength \( L=4h \) (Mansouri et al., 2020)

Mansouri et al. (2020) studied the effect of grid spacing when synthetic inflow turbulence generators are used. Grid spacings equal to \( H/8 \), \( H/16 \), and \( H/24 \) were analyzed. From the three grid spacings, \( H/24 \) and \( H/16 \) performed with good agreement with WT measurements. According to Mansouri et al. (2020), the \( H/24 \) grid took close to 8 days to run whereas the \( H/16 \) grid took only a day. Therefore, it is more convenient to use a grid spacing equal to \( H/16 \) to save computing time. Thus, for this work, a grid spacing of \( H/16 \) is used to run the computational program. Appendix A shows the computing time for the simulations performed for this work.

### 3.4 Maximum and Minimum Frequencies used in the Inflow Turbulence Models

As established before, for this study, the Narrowband Synthesis Random Flow Generator (NSRFG) method is used to generate the turbulent flow in the ABL to obtain pressure
coefficients for the TTU building. Different frequencies are required by the computational domain to run the random flow generator method. These frequencies are explained in the following sections.

3.4.1 Wind Tunnel (WT) Frequency

Mooneghi et al. (2016) reported data from WT measurements for the TTU building and Silsoe Cube building while Moravej (2018) also reported data from WT measurements for the TTU building. For this work, WT data from Moravej’s (2018) thesis work, shown in Figure 3.2, is used to obtain non-dimensional frequencies from the TTU building to use in the inflow turbulence generator. These non-dimensional frequencies are referred to as $f_{\text{max}}$, for maximum WT frequencies, and as $f_{\text{min}}$ for minimum WT frequencies.
Figure 3.2. Turbulence power spectra (n= frequency, z= model building height, U= mean wind speed at building eave height, S= turbulence power spectra) (Moravej, 2018)
A non-dimensional frequency \( f \) can be calculated using the following relationship shown in Eq. 3.9.

\[
f = \frac{nH}{U_H}
\]  
\text{Eq. 3.9}

Here, \( n \) is the dimensional frequency in Hz, \( H \) is the reference height of the building in m, and \( U_H \) is the reference velocity at the building height in m/s. A non-dimensional frequency \( f \) can be converted into dimensional frequency \( n \) by rewriting Eq. 3.9 as shown in Eq. 3.10.

\[
n = \frac{fU_H}{H}
\]  
\text{Eq. 3.10}

### 3.4.2 Non-Dimensional Grid Frequency (\( f_{grid} \))

In Eq. 3.9, the non-dimensional frequency \( f \) is defined. In this section, the non-dimensional grid frequency \( f_{grid} \) will be defined. Therefore, let us relate the non-dimensional wavelength \( \lambda \) to the non-dimensional frequency \( f \) using the actual wavelength \( L \) in meters as shown in Eq. 3.11.

\[
\lambda = \frac{L}{H} = \frac{U_H}{nH} = \frac{1}{f}
\]  
\text{Eq. 3.11}

Here, \( \lambda \) is inversely proportional to \( f \). In the case of finite difference method (FDM), to represent a wave minimum, 5 grid points or 4 grid spacings are needed. Thus, a minimum wavelength \( L=4h \) or \( \lambda=4h/H \) is needed to transport a wave. The same length is suggested by Feriziger and Peric (2002), as mentioned in Section 3.3 above. Here, \( h \) is the maximum grid spacing used in the computer model. This wavelength \( \lambda \) is called grid wavelength and \( f \) is called \( f_{grid} \).

Therefore, \( f_{grid} \) can be calculated as:

\[
f_{grid} = \frac{1}{\lambda} = \frac{H}{4h}
\]  
\text{Eq. 3.12}
This frequency \( f_{\text{grid}} \) is also called \( f_{\text{LES}} \). This \( f_{\text{LES}} \) is explained in the following section below. Thus, for a grid spacing of \( h=H/16 \), \( f_{\text{grid}} \) comes to:

\[
f_{\text{grid}} = f_{\text{LES}} = \frac{16h}{4h} = 4
\]

Eq. 3.13

3.4.3 Wind Tunnel (WT) Frequency and Non-Dimensional Grid Frequency \( f_{\text{grid}} \) on Large Eddy Simulation (LES) Studies

In the LES method, the highest frequency transported by the grid is \( f_{\text{grid}} \) which is equal to \( f_{\text{LES}} \) as shown in Figure 3.3 meaning that frequencies greater than \( f_{\text{LES}} \) are modeled by the sub-grid-scale model explained in Section 3.2.

![Figure 3.3. Frequency region resolved and modeled by LES (Mansouri et al., 2020)](image)

For LES studies, non-dimensional frequencies from WT measurements are used to run the computer simulations. Generally, \( f_{\text{max}} \) is equal to \( f_{\text{max}}^{\text{e}} \) and \( f_{\text{min}} \) is equal to \( f_{\text{min}}^{\text{e}} \). However, Lamberti et al. (2018) reported that the turbulence generated at the building location without building using \( f_{\text{max}} = f_{\text{max}}^{\text{e}} \) is quite different from what is in the inlet. Thus, results from inflow turbulence generators are different from what is already reported from WT measurements. Mansouri et al. (2020) used the CDRFG method to investigate the effect of \( f_{\text{max}} \) on peak pressure coefficients by considering a \( H/16 \) grid for the TTU building. Mansouri et al. (2020)
calculated peak pressures for various $f_{\text{max}}$ and compared them with WT measurement results reported by Moravej (2018). Different $f_{\text{max}}$ were used to generate peak pressure coefficients. First, Mansouri et al. (2020) used data from WT measurements stating that $f_{\text{max}} = f_{\text{max}e}$, and then LES method principles were applied where $f_{\text{max}}$ was taken as $f_{\text{LES}}$. For $f_{\text{max}}, f_{\text{max}e}$ from WT measurements was taken as 10 while $f_{\text{LES}}$ was calculated by using Eq. 3.11 for different grid spacings. It was noticed that by reducing $f_{\text{max}}$ values, the error was reduced from 100% to 33% on the roof and reduced from 600% to 200% on the sidewalls, which showed that using $f_{\text{max}} = f_{\text{max}e}$ produces more error than using $f_{\text{max}} = f_{\text{LES}}$. However, Mansouri et al. (2020) argued that the error continued to be high on the leeward and windward sides of the TTU building but these errors are reduced somewhat for lower $f_{\text{max}}$. Therefore, similar procedures applied by Mansouri et al. (2020) are used in this work to calculate peak pressure coefficients in the TTU building. However, the Narrowband Synthesis Random Flow Generator (NSRFG) inflow turbulence method is used. The numerical processes used to apply this method are described in the following sections.

3.5 Computational Model and Inflow Turbulence Method Details

3.5.1 Computational Region and Boundary Conditions

As mentioned before, a grid spacing of H/16, where H is the height of the TTU building, is considered in all directions of the building. The dimensions of the building are 2.25H x 3.375H x H, where H is the reference height of the building, which is equal to 3.96 m as reported by Mooneghi et al. (2016). The windward and leeward dimensions of the building are 4H and 7H, respectively, in the x-direction, 3H for the y directions, and 4H for the z-direction. Figure 3.4 shows the computational region for the TTU building with its dimensions.
Figure 3.4. TTU building computational region (a) plan view and (b) elevation view

The computational region for the x, y, and z-axis is calculated below. Figure 3.5 shows the TTU building boundary conditions for better visualization and understanding of these measurements. These boundary conditions are indicated for all surfaces of the building. Eq. 3.14, Eq. 3.15, and
Eq. 3.16 show the calculations of the computational region for the TTU building, which results in 13.3H x 9.375H x 5H.

\[ x = 4H + 2.25H + 7H = 13.3H \]  \hspace{1cm} \text{Eq. 3.14}

\[ y = 3H + 3.375H + 3H = 9.375H \]  \hspace{1cm} \text{Eq. 3.15}

\[ z = 4H + H = 5H \]  \hspace{1cm} \text{Eq. 3.16}

Figure 3.5. Boundary conditions for the numerical modeling (Mansouri et al., 2020)

3.5.2 Inflow Turbulence Details

As stated in previous sections, the inflow turbulence generator method chosen to use in this work is the NSRFG method. This method was proposed by Yu et al. (2018), and it overcomes some problems from random flow generator (RFG) techniques by strictly maintaining a divergence-free condition, coherency function, and turbulent spectra of the atmospheric boundary layer (ABL) flow theoretically (Yu et al., 2018).
3.5.2.1 “Yif1.out” Description

“Yif1.out” is a computer program developed by Dr. Selvam to obtain peak pressure coefficients by performing 3D-CFD calculations using the NSRFG method. It consists of two input files that simultaneously generate inflow turbulence results. These files are “yif-i.txt” and “char.txt”.

The “yif-i.txt” file carries data involving building and wind force characteristics such as dimensions, roughness, velocities, frequencies, and viscosity. The “chart.txt” file consists of 999 names of the movie files stored to run data (Selvam, 2020).

The “yif-i.txt” is used as an input file to read data related to the TTU building dimensions. This file consists of 2 lines which are described below.

Line 1 consists of 10 variables where each variable represents a specific point in the TTU building.

Line 1: IM, JM, KM, IMK1, IMK2, JMK1, JMK2, KH, DTT, TTIME

IM: # of points in x
JM: # of points in y
KM: # of points in z
IMK1: Building start point in x.
IMK2: Building endpoint in x.
JMK1: Building start point in y.
JMK2: Building endpoint in y.
KH: Building point in z.
DTT: Time step
TTIME: Total time to run the computer program.
Line 2 consists of 13 variables where each variable also represents a specific point of the TTU building. These points are detailed below.

Line 2: C11, C2, IPLOT, VISC, XL3, YL3, ZL, DN, IFS, IFE, HREF, UREF, INFLT

- **C11**: Log law profile coefficient
- **C2**: Non-dimensional roughness length
- **IPLOT**: Frequency of the file called “chart.txt”
- **VICS**: Viscosity
- **XL3**: x domain length
- **YL3**: y domain length
- **ZL**: z domain length
- **DN**: Frequency width (Hz)
- **IFS**: Starting range of whole number of dimensional frequency
- **IFE**: Ending range of whole number of dimensional frequency
- **HREF**: Reference height of the building
- **UREF**: Reference velocity in the building
- **INFLT**: Inflow turbulence (INFLT=1)

### 3.5.2.2 Data Preparation

Now that the variables are defined, numerical operations are performed. As mentioned before, a grid spacing of $\frac{H}{16}$ is used to perform computational processes. As calculated in Section 3.5.1, a computational domain size of $13.3H \times 9.375H \times 5H$ is used. Thus, the building dimensions needed in Line 1 of the computer program are calculated.

The values for IM, JM, and KM are calculated below.
\[ IM = 13.3(16) + 1 = 213 \quad \text{Eq. 3.17} \]

\[ JM = 9.375(16) + 1 = 151 \quad \text{Eq. 3.18} \]

\[ IM = 5(16) + 1 = 81 \quad \text{Eq. 3.19} \]

By multiplying IM x JM x IM, the total number of grid points used by the program can be determined. Thus, \(213 \times 151 \times 81 = 2,605,203\) or 2.6 million grid points are used by the program to run the data.

To calculate the building start and endpoints (IMK1, IMK2, JMK1, JMK2) at the x and y-axis and the building point at the z-axis, TTU building computational domain, shown in Figure 3.4, is applied.

The values for IMK1, IMK2, JMK1, JMK2, and KM are calculated below.

\[ IMK1 = 4(16) + 1 = 65 \quad \text{Eq. 3.20} \]

\[ IMK2 = (4 + 2.25)(16) + 1 = 101 \quad \text{Eq. 3.21} \]

\[ JMK1 = 3(16) + 1 = 49 \quad \text{Eq. 3.22} \]

\[ JMK2 = (3 + 3.375)(16) + 1 = 103 \quad \text{Eq. 3.23} \]

\[ KH = (16) + 1 = 17 \quad \text{Eq. 3.24} \]

Time step (DTT) is taken as non-dimensional time step of 0.02 in order to keep a CFL (Courant-Friedrichs-Lewy) less than 1. The CFL condition is defined by Laney (1998) as the full numerical domain of dependence must contain the physical domain of dependence. Thus, the distance for any data that travels at the specific time step (DTT) needs to be less than the grid spacing. This condition is explained below in Eq. 3.25. Here, \(\Delta t\) is the non-dimensional time step.
(DTT), $\Delta x$ is the grid spacing ($h=H/16$), and $v$ is the maximum velocity ($U_{max}$) around the TTU building, which based on computation procedures is $2U_H$ (Mansouri et al., 2020).

$$\Delta t \leq \frac{\Delta x}{v} \rightarrow \Delta t \leq \frac{h}{U_{max}}$$  \hspace{1cm} \text{Eq. 3.25}

$$\Delta t \leq \frac{H}{2U_H} \rightarrow \Delta t \leq 0.03125 \frac{H}{U_H}$$  \hspace{1cm} \text{Eq. 3.26}

As shown in Eq. 3.26, a DTT less than 0.03125 should be used. Thus, a non-dimensional DTT equals to 0.02 is considered.

By using Eq. 3.27 below, dimensional DTT in seconds can be calculated. In Eq. 3.27, $t^*$ is the dimensionless DTT value, $t$ is the time in s, $U_H$ is the reference velocity at the building height in m/s, which is 7.66 m/s for this case, and $H$ is the reference height of the TTU building in m, which is 3.96 m for this case (Mooneghi et al., 2016).

$$t^* = \frac{t U_H}{H}$$  \hspace{1cm} \text{Eq. 3.27}

Therefore, using Eq. 3.27 and solving for $t$, DTT dimension is calculated below.

$$t = \frac{t^* H}{U_H} = \frac{0.02 \cdot 3.96 \, \text{m}}{7.66 \, \text{m/s}} = 0.01 \, \text{s}$$  \hspace{1cm} \text{Eq. 3.28}

As mentioned above, a CFL less than 1 needs to be also considered when calculating the DTT. A CFL less than 1 capture all the time-variant issues than can be generated during the running of the computer simulations (Mansouri et al., 2020). This CLF is obtained by calculating the Courant number ($C$) using Eq. 3.29 below.

$$C = a \frac{\Delta t}{\Delta x}$$  \hspace{1cm} \text{Eq. 3.29}
Here, $a$ is the maximum velocity around the TTU building in m/s, which is $2U_H$ (Mansouri et al., 2020) where $U_H$ is equal to 7.66 m/s, $\Delta t$ is the dimensional time step (DTT) in s, and $\Delta x$ is the grid spacing used, which is $H/16$ where $H$ is equal to 3.96 m. Thus, Eq. 3.30 below shows the calculation of the Courant number ($C$) for a dimensional DTT of 0.01 s.

$$C = a \frac{\Delta t}{\Delta x} = \frac{2(7.66\frac{m}{s})(0.01 \text{ s})}{\frac{3.96 \text{ m}}{16}} = 0.62 < 1$$  
Eq. 3.30

Therefore, a non-dimensional DTT of 0.02 is chosen to be used since it keeps a CFL less than 1 and a value less than the grid spacing when using Eq. 3.25.

The total time to run the computer program (TTIME) is taken as 100-time units.

Therefore, Line 1 for a grid with spacing $H/16$ is:

213,151,81,65,101,49,103,17,0.02,100

Values for Line 2 involve data related to the wind flow behavior on the building. The log law profile coefficient ($C_{11}$) and the non-dimensional roughness length ($C_2$) are equal to 0.195 and 0.006 respectively and were calculated using Eq. 3.31 and Eq. 3.32. For $C_2$ calculation in Eq. 3.32, $z_0$ represents the roughness length of the ground for the TTU building, which, in this case, is equal to 0.024 m according to data reported by Levitan et al. (1991).

$$C_{11} = \frac{1}{\ln\left(\frac{1+C_2}{C_2}\right)} = \frac{1}{\ln\left(\frac{1+0.006}{0.006}\right)} = 0.195$$  
Eq. 3.31

$$C_2 = \frac{z_0}{4} = \frac{0.024}{4} = 0.006$$  
Eq. 3.32

The frequency of the file “chart.txt” (IPLOT) is equal to 5000 according to data reported by Mansouri et al. (2020). The viscosity (VICS) is equal to $4 \times 10^{-7}$, and it is calculated using Eq. 3.33.
\[ VISC = \frac{1}{R_e} \]  

Eq. 3.33

where \( R_e \) is calculated using Eq. 3.34

\[ R_e = \frac{U_H H}{v} \]  

Eq. 3.34

where \( U_H \) is the reference wind velocity in the TTU building, which is 7.66 m/s, \( H \) is the reference height of the TTU building, which is 3.96 m (Mooneghi et al., 2016), and \( v \) is taken as 1.52 \( \times 10^{-5} \).

By solving Eq. 3.34, a \( R_e \) of 2.3 \( \times 10^6 \) is calculated. XL3, YL3, and ZL are the domain length of the building in the x, y, and z-axis, respectively. These values are calculated in Section 3.5.1. Thus, XL3 = 13.3, YL3 = 9.375, and ZL = 5.

### 3.5.2.2.1 Dimensional Frequencies Calculations

The frequency width (DN), the starting range of the whole number of dimensional frequency (IFS), and the ending range of the whole number of dimensional frequency (IFE) are calculated based on non-dimensional frequencies, \( f_{min} \) and \( f_{max} \), explained in Section 3.4.

Since the NSRFG method is based on random flow generator techniques, non-dimensional frequencies used come from WT test measurements already reported by Moravej (2018). Thus, \( f_{min} \) is taken as 0.1. As mentioned in Section 3.4.3, the use of maximum frequencies in LES studies has resulted in some errors when computing peak pressure coefficients due to the limitations in the simulation process. Thus, for this work, two different non-dimensional maximum frequencies are used. The first one is taken as \( f_{max} = f_{LES} \), following the LES principles, and the second one is taken as \( f_{max} = f_{maxe} \) based on WT test measurements reported by Moravej (2018).
An example of DN, IFS, and IFE calculations for \( f_{\text{min}} = 0.1 \) and \( f_{\text{max}} = 4 \) is shown to explain this procedure. Same procedure is followed when \( f_{\text{min}} = 0.1 \) and \( f_{\text{max}} = 10 \).

First, Eq. 3.10 is used to calculate dimensional frequencies \( n \) from non-dimensional \( f \) frequencies. Here, \( H \) is the reference height of the TTU building (3.96 m), and \( U_H \) is the reference wind velocity in the TTU building (7.66 m/s).

\[
\begin{align*}
    n_{\text{min}} &= \frac{f_{\text{min}} \cdot U_H}{H} = \frac{0.1 \cdot 7.66 \text{ m/s}}{3.96 \text{ m}} = 0.19 \text{ Hz} & \text{Eq. 3.35} \\
    n_{\text{max}} &= \frac{f_{\text{max}} \cdot U_H}{H} = \frac{4 \cdot 7.66 \text{ m/s}}{3.96 \text{ m}} = 7.74 \text{ Hz} & \text{Eq. 3.36}
\end{align*}
\]

Now, let us say that a DN= 0.75 Hz is used. Thus, values for IFS and IFE can be calculated using Eq. 3.37 and Eq. 3.38, respectively.

\[
\begin{align*}
    IFS &= \frac{2(n_{\text{min}})}{DN} + 1 \\
    IFE &= \frac{2(n_{\text{max}})}{DN} + 1 & \text{Eq. 3.37, 3.38}
\end{align*}
\]

Thus, for \( n_{\text{min}} = 0.19 \text{ Hz} \) and \( n_{\text{max}} = 7.74 \text{ Hz} \),

\[
\begin{align*}
    IFS &= \frac{2(0.19 \text{ Hz})}{0.75 \text{ Hz}} + 1 \approx 1 \\
    IFE &= \frac{2(7.74 \text{ Hz})}{0.75 \text{ Hz}} + 1 \approx 11
\end{align*}
\]

Therefore, for a frequency width (DN) equals to 0.75 Hz, starting and ending ranges of the whole number of dimensional frequencies are 1 and 11, respectively.

### 3.5.2.2.2 The Number of Spectral Segment (N)

In RFG techniques, the number of spectral segment (N) is defined as the number of frequencies used in one segment of the computational grid when generating inflow turbulence using the NSRFG method. The number of spectral segment will depend on the variation of the frequency step or frequency width (DN). For this research, the dimensional frequencies (\( n \)) were first
calculated as explained in Section 3.5.2.2.1. Once DN is varied, values for IFS and IFE can be calculated as well as the number of spectral segment (N). This principle comes from the analysis of the NSRFG equation, shown in Eq. 3.39, introduced by Yu et al.(2018).

\[ u_i(x, t) = \sum_{n=1}^{N} \sqrt{2S_{u,i}(f_n)} \Delta f \sin (k_{j,n} \bar{x}_{j,n} + 2\pi f_n t + \phi_n) \]  

Eq. 3.39

Here, Yu et al. (2018) explains that \( u_i \) represent the velocities in the three directions (along wind, across wind, and vertically when \( i = 1, 2, \) and \( 3, \) respectively); \( j = 1, 2, \) and \( 3 \) denote the \( x, y, \) and \( z \) directions respectively; \( x = \{x, y, z\} \) is the coordinate vector, \( S_{u,i}(f_n) \) can be calculated by the von Karman spectrum in the \( i \) direction at frequency \( f_n \) and \( \Delta f \) is the frequency bandwidth; \( N \) is the number of spectral segments; \( f_n = \frac{2n-1}{2} \Delta f \), which was used to calculate IFS and IFE above in Section 3.5.2.2.1, and \( \Delta f \) is the frequency step, in this case the non-dimensional value for DN, for discretizing the target spectra, \( \phi_n \sim U(0,2\pi) \) is a random phase obeying a uniform distribution; and \( \bar{x}_{j,n} \) is given by \( \bar{x}_{j,n} = \frac{x_j}{L_{j,n}} \).

The non-dimensional value \( \Delta f \) for DN can be calculated by using Eq. 3.9 (\( f = \frac{nH}{U_H} \)) introduced above in Section 3.4.1. Here, \( f \) is the non-dimensional value (\( \Delta f \)), \( n \) is the dimensional value for DN in Hz, \( H \) is the reference height of the building in m, and \( U_H \) is the reference wind velocity in the building in m/s. \( \Delta f \) calculation is shown below in Eq. 3.40.

\[ \Delta f = f = \frac{nH}{U_H} = \frac{0.75 \left( \frac{1}{3.96 \text{ m}} \right)}{7.66 \text{ m/s}} = 0.39 \]  

Eq. 3.40

Now, the number of spectral segment (N) calculation for \( f_{\text{min}} = 0.1, f_{\text{max}} = 4, \) and DN=0.75 Hz is shown in Eq. 3.41 below.

\[ N = IFE - IFS \]  

Eq. 3.41
\[ N = 11 - 1 = 10 \]

Thus, having calculated IFS, IFE, and the number of spectral segment (N), Line 2 for a grid with spacing H/16 is:

0.195, 0.006, 5000, 4e-7, 13.3, 9.375, 5, 0.75, 1, 11, 3.96, 7.66, 1

3.5.2.2.3 Number of Spectral Segment (N) Variations

The number of spectral segment (N) is varied to compare peak pressure coefficients obtained from the NSRFG method with peak pressure obtained from WT measurements. The frequency width (DN) is varied in order to obtain these number of spectral segments (N) variations. A total of four N values are used to run the computer simulations. Table 3.1 shows a summary of the non-dimensional and dimensional frequencies as well as the N variations.

Table 3.1. DN, IFS, IFE, and N values for non-dimensional and dimensional frequencies \( f_{\text{min}} = 0.1 \ f_{\text{max}} = 4 \), and \( f_{\text{min}} = 0.1 \ f_{\text{max}} = 10 \)

<table>
<thead>
<tr>
<th>DN</th>
<th>IFS</th>
<th>IFE</th>
<th>N</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Non-Dimensional (f)</td>
<td>Non-Dimensional (Hz)</td>
<td>Dimensional (f)</td>
</tr>
<tr>
<td>Min</td>
<td>0.1</td>
<td>0.0775</td>
<td>0.15</td>
</tr>
<tr>
<td></td>
<td>4.0</td>
<td>0.0078</td>
<td>0.0151</td>
</tr>
<tr>
<td>Max</td>
<td>0.0390</td>
<td>0.0075</td>
<td>26</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DN</th>
<th>IFS</th>
<th>IFE</th>
<th>N</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Non-Dimensional (f)</td>
<td>Non-Dimensional (Hz)</td>
<td>Dimensional (f)</td>
</tr>
<tr>
<td>Min</td>
<td>0.1</td>
<td>0.1964</td>
<td>0.38</td>
</tr>
<tr>
<td></td>
<td>0.0987</td>
<td>0.191</td>
<td>2</td>
</tr>
<tr>
<td>Max</td>
<td>10.0</td>
<td>0.0198</td>
<td>0.0383</td>
</tr>
<tr>
<td></td>
<td>0.0099</td>
<td>0.0192</td>
<td>11</td>
</tr>
</tbody>
</table>
4 Results and Discussion

The first computation is done without building. Here, the inlet velocities are compared with the interior velocity at the building location. Also, the pressures at the building location are monitored to see the effect of inflow turbulence. Next, the computation is done with the building in the computational domain. Here, the mean and peak pressures on the building are obtained from CFD computation. The inflow turbulence effect on interior velocity and pressure on the Texas Tech University (TTU) building are analyzed using the Narrowband Synthesis Random Flow Generator (NSRFG) method. The computation simulations are performed using $f_{max}$ equals 4 and 10 for a grid spacing of H/16. Minimum pressure ($f_{min}$) is taken to be equal to 0.1 following experimental results from Moravej’s (2018) thesis work. Yu et al. (2018) used a number of spectral segments (N) of 1000 in their work. For this thesis, the number of spectral segments (N) is varied from 50 to 1000. Figures and comparisons are explained in the following sections.

4.1 Computation without Building: Inflow Turbulence Effect on Interior Velocity and Pressure

The velocity at the inlet and at the building location is plotted for comparison. In addition, the pressure at the building location without building is also plotted to evaluate the level of spurious pressure due to the number of spectral segment (N) range considered.

Lebovitz (2017) investigated peak pressure coefficients along a line around a cube from front to back using LES methods. In his study, Lebovitz (2017) found that the pressure in the CFD domain is strongly varying at a frequency equal to the simulation timestep when using the inlet generator. Lebovitz (2017) showed the time history of the pressure and velocity for a chosen monitorpoint, located at the windward edge of the roof, with and without the inlet generator.
being used. Figure 4.1 shows the pressure and the velocity when the inlet generator was not being used. Here, the time history of the pressure and the velocity at the chosen monitorpoint shows that the frequency behavior of the pressure matched the behavior of the velocity. On the other hand, Figure 4.2 shows the pressure and the velocity with the inlet generator being used. Here, the time history of the pressure and velocity showed that the pressure history did not match the velocity history and its values jump from one timestep to another. Lebovitz (2017) assumed that these differences are due to the use of an Autoregressive (AR) model, which causes global pressure vibrations and high-frequency fluctuations in the CFD domain distorting the pressure measurements and their statistics.

Thus, the pressure and velocity time history are calculated and analyzed for the TTU building at the building location without building using the NSRFG inflow turbulence generator. For this study, N values for $f_{\text{max}}$ of 4 and 10 are varied from 50 to 1000. Figure 4.3 through Figure 4.6 show results for this analysis.

![Figure 4.1](image-url)  
**Figure 4.1.** The time history of (a) pressure and (b) velocity without using the inlet generator at a monitorpoint at the building location (Lebovitz, 2017)
Figure 4.2. The time history of (a) pressure and (b) velocity using the inlet generator at a monitorpoint at the building location (Lebovitz, 2017)

From the figures, it can be noticed that pressures and velocities do not match, and pressure fluctuations are observed. This agrees with Lebovitz’s (2017) statement about the effect of pressure and velocities behavior in the inlet when using a turbulence generator. Nevertheless, it can be seen that pressure fluctuations using $f_{\text{max}} = 4$ are more continuous and smoother than the pressures obtained when using $f_{\text{max}} = 10$. This effect can be due to the filtration process that LES studies use where maximum frequencies greater than $f_{\text{LES}}$ are filtered out. Due to this effect, pressures from simulations where $f_{\text{max}} = 10$ is used have more variations in time.

Figure 4.3 shows the time history of the pressure when N=50. The pressure resulted from $f_{\text{max}} = 4$ is continuous and does not present too many variations. The pressure resulted from $f_{\text{max}} = 10$ is pretty discontinuous with higher variations. Figure 4.4 shows results for the pressure when N=100. For this case, the pressure resulting from $f_{\text{max}} = 4$ presents higher variations than the pressure from N=50. A different case happened when using $f_{\text{max}} = 10$ for N=100. Here the pressure presents lesser variations in comparison with the pressure from N=50 when using
\( f_{\text{max}} = 10 \). For \( N=500 \) and \( N=1000 \), as shown in Figure 4.5 and Figure 4.6, respectively, the pressure from \( f_{\text{max}} = 4 \) has variations similar to the ones seen when \( N=100 \). These variations are quite high in comparison to the ones from \( N=50 \) but less discontinuous than the pressures from \( f_{\text{max}} = 10 \). Figure 4.5 (b) shows the pressure from \( f_{\text{max}} = 10 \) using \( N=500 \), and the pressure has higher variations than the pressure from \( N=100 \) but lesser variations than the pressure from \( N=50 \). When \( N=1000 \) for \( f_{\text{max}} = 10 \), shown in Figure 4.6 (b), variations in pressure are higher than the ones seen when \( N=500 \) and \( N=100 \). Pressure variations for \( f_{\text{max}} = 10 \) resulted from \( N=1000 \) are pretty similar to ones seen when \( N=50 \).

Therefore, the number of spectral segment (\( N \)) variations show that the bigger the \( N \) value is, the more fluctuated the pressure becomes when using \( f_{\text{max}} = f_{\text{grid}} = 4.0 \). Figure 4.3 (a), where \( N=50 \), presents the smoothest and most continuous pressure flow over time in comparison with the other \( N \) variations when using \( f_{\text{max}} = f_{\text{grid}} = 4.0 \). For \( f_{\text{max}} = 10 \), \( N \) variations do not show the same results as when \( f_{\text{max}} = f_{\text{grid}} = 4.0 \). The pressure and velocity are more discontinuous and present higher variations when using \( f_{\text{max}} = 10 \). When \( N=50 \) and \( N=1000 \), these variations in time have the highest behaviors which results in more discontinuous pressure flows. When \( N=100 \) and \( N=500 \), however, the pressure flow is still discontinuous but with fewer variations in time than results from when \( N=50 \) and \( N=1000 \). In fact, for \( f_{\text{max}} = 10 \), pressure flow from \( N=100 \) has the least fluctuated behavior, but it is still more discontinuous than results from when \( N=100 \) for \( f_{\text{max}} = f_{\text{grid}} = 4.0 \).

Thus, from running inflow turbulence at the building location without building in the computational domain, it was observed that by using \( f_{\text{max}} = f_{\text{grid}} \), spurious pressures are reduced comparing to using \( f_{\text{max}} = 10 \). It was also observed that keeping a number of spectral
segment (N) equals 50 results in a more continuous pressure. Therefore, when using inflow turbulence generators such as the NSRFG one, it is more convenient to use maximum frequencies equal to $f_{grid}$ and keep the number of spectral segment (N) low to avoid spurious results in the pressure flow and keep a smoother and continuous flow. Even though the common understanding is that when greater numbers of spectral segments (N) are used, more accurate results for pressures and velocities should be obtained. In the CFD computation, it does not look that way. Therefore, further research is needed to understand the reason behind this issue.

\[ f_{min} = 0.1, f_{max} = 4, D_N = 0.15 \text{ Hz}, N = 50 \quad \text{and} \quad f_{min} = 0.1, f_{max} = 10, D_N = 0.38 \text{ Hz}, N = 50 \]

Figure 4.3. Effect of inflow turbulence on interior velocity and pressure for N=50 at the building location without building (a) fmax=4 and (b) fmax=10
fmin = 0.1, fmax = 4, DN = 0.0755 Hz, N = 100
fmin = 0.1, fmax = 10, DN = 0.191 Hz, N = 100

Figure 4.4. Effect of inflow turbulence on interior velocity and pressure for N=100 at the building location without building (a) fmax=4 and (b) fmax=10

fmin = 0.1, fmax = 4, DN = 0.0151 Hz, N = 500
fmin = 0.1, fmax = 10, DN = 0.0383 Hz, N = 500

Figure 4.5. Effect of inflow turbulence on interior velocity and pressure for N=500 at the building location without building (a) fmax=4 and (b) fmax=10
f_{min} = 0.1, f_{max} = 4, \text{DN} = 0.0075 \text{ Hz}, N = 1000 \quad f_{min} = 0.1, f_{max} = 10, \text{DN} = 0.0192 \text{ Hz}, N = 1000

(a)

(b)

Figure 4.6. Effect of inflow turbulence on interior velocity and pressure for N=1000 at the building location without building (a) f_{max}=4 and (b) f_{max}=10

4.2 Computation with Building: Comparison of CFD Mean Peak Pressure Coefficients with WT Peak Pressure Coefficients

The mean peak pressure coefficients for \(f_{min} = 0.1\) and \(f_{max} = 4\), and \(f_{min} = 0.1\) and \(f_{max} = 10\) are calculated and compared with WT mean peak pressure coefficients. As mentioned before, the number of spectral segment (N) is varied. The inflow generator was run for a total time (TTIME) of 100 units with a time step (DTT) of 0.02 at each point along the centerline of the TTU building. Figure 4.7 shows a diagram of the representation of the centerline of the TTU building where the roof is the origin. The peak pressure coefficient plots are reported following this diagram.

In order to calculate the percentage of error at the windward side, roof, and leeward side of the TTU building, critical points from the resulting plots were taken into consideration. For the comparison of CFD mean pressure coefficients with WT measurements, points equal to -0.13H from the roof top for the windward side, 1.30H from the roof top for the roof, and 2.5H from the
roof top for the leeward side were considered. These are the most critical points along the centerline of the TTU building in the resulted plots. Therefore, these points were considered to calculate percentage of errors between CFD measurements and WT measurements.

Figure 4.7. Centerline of the TTU building with the roof as the origin

Figure 4.8 through Figure 4.11 show results of CFD simulations for mean peak pressure coefficients for the TTU building. CFD results for mean peak pressure coefficients are reasonably in agreement with WT measurements. However, small discrepancies between CFD and WT measurements are noticed at the windward roof edge, the middle roof, and the leeward side of the TTU building. The windward roof edge has the maximum errors while the middle roof and the leeward side of the building present smaller errors. For \( f_{max} = 4 \), the percentage of error at the windward roof edge is 105%, 113.2%, 115.8%, and 118.4% for N=50, 100, 500, and 1000, respectively while that for \( f_{max} = 10 \), the percentage of error at the same location is 126.3%, 121.1%, 113.2%, and 121.1% for N=50, 100, 500, and 1000, respectively. It is noticed that once the N value increases, the error at the windward roof edge of the building increases when using \( f_{max} = 4 \). For \( f_{max} = 10 \), the percentage of error at the windward roof edge of the building has a small decrease when N increases. However, the percentage of error for \( f_{max} = 10 \) at the windward roof edge of the building is still much higher than results from \( f_{max} = 4 \).

Mansouri et al. (2020) reported the same discrepancies between WT measurements and CFD
calculations at the windward roof edge of the TTU building but using the CDRFG method. Mansouri et al. (2020) state that these discrepancies could be due to the inflow turbulence method used.

Therefore, using the NSRFG method, CFD measurements obtained from using $f_{max} = 4$ show a better agreement with WT measurements than when $f_{max} = 10$ is used. There are still some discrepancies, principally at the windward roof edge of the building, between the experimental data from WT measurements and CFD results when using $f_{max} = 4$, and this issue can be due to the inflow turbulence generator used as stated by Mansouri et al. (2020). Further research needs to be performed to investigate this problem more deeply.

$$f_{min}= 0.1, f_{max}=4, D_{N}= 0.15 \text{ Hz}, N= 50$$

$$f_{min}= 0.1, f_{max}=10, D_{N}= 0.38 \text{ Hz}, N=50$$

Figure 4.8. Mean pressure coefficients ($C_p\text{-mean}$) for $N=50$ (a) $f_{max}=4$ and (b) $f_{max}=10$
\[ \text{fmin} = 0.1, \text{fmax} = 4, \text{DN} = 0.0755 \text{ Hz}, N = 100 \]

\[ \text{fmin} = 0.1, \text{fmax} = 10, \text{DN} = 0.191 \text{ Hz}, N = 100 \]

Figure 4.9. Mean pressure coefficients (Cp-mean) for N=100 (a) fmax=4 and (b) fmax=10

\[ \text{fmin} = 0.1, \text{fmax} = 4, \text{DN} = 0.0151 \text{ Hz}, N = 500 \]

\[ \text{fmin} = 0.1, \text{fmax} = 10, \text{DN} = 0.0383 \text{ Hz}, N = 500 \]

Figure 4.10. Mean pressure coefficients (Cp-mean) for N=500 (a) fmax=4 and (b) fmax=10
\[ f_{\text{min}} = 0.1, f_{\text{max}} = 4, DN = 0.0075 \, \text{Hz}, N = 1000 \]

\[ f_{\text{min}} = 0.1, f_{\text{max}} = 10, DN = 0.0192 \, \text{Hz}, N = 1000 \]

Figure 4.11. Mean pressure coefficients (Cp-mean) for N=1000 (a) \( f_{\text{max}} = 4 \) and (b) \( f_{\text{max}} = 10 \)

4.3 Computation with Building: Comparison of CFD Maximum Peak Pressure Coefficients with WT Peak Pressure Coefficients

The maximum peak pressure coefficients for \( f_{\text{min}} = 0.1 \) and \( f_{\text{max}} = 4 \), and \( f_{\text{min}} = 0.1 \) and \( f_{\text{max}} = 10 \) are calculated and compared with maximum peak pressure coefficients from WT measurements in Figure 4.12 through Figure 4.15. The inflow generator was run for a total time (TTIME) of 100 units with a time step (DTT) of 0.02 at each point along the centerline of the TTU building. The maximum peak pressure coefficient plots are reported following the TTU building diagram showed in Section 4.2 in Figure 4.7.

In order to calculate the percentage of error at the windward side, roof, and leeward side of the TTU building, critical points from the resulting plots were taken into consideration. For the comparison of CFD maximum pressure coefficients with WT measurements, points equal to -0.18H from the roof top for the windward side, 1.18H from the roof top for the roof, and 2.75H from the roof for the leeward side were considered. These are the most critical points along the
centerline of the TTU building in the resulted plots. Therefore, these points were considered to calculate percentage of errors between CFD measurements and WT measurements.

Maximum peak pressure coefficients resulted from using $f_{\text{min}} = 0.1$ and $f_{\text{max}} = 4$ showed a better agreement with WT measurements than maximum peak pressure coefficients obtained from $f_{\text{min}} = 0.1$ and $f_{\text{max}} = 10$. However, errors are still observed at the windward side, roof edge, and leeward side of the TTU building for $f_{\text{max}} = 4$ results when changing the number of spectral segment (N). The maximum errors on the windward side of the building are around 51.2%, 83.2%, 126.4%, and 108.8% for $f_{\text{max}} = 4$ for N values of 50, 100, 500, and 1000, respectively. For the roof, CFD results for $f_{\text{max}} = 4$ also have a better agreement with WT measurements than from $f_{\text{max}} = 10$ computations. For the leeward side of the building, $f_{\text{max}} = 4$ CFD results also agree better with WT measurements than $f_{\text{max}} = 10$ results. As seen in Figure 4.12 (a), $f_{\text{max}} = 4$ using N=50 has the best agreement with WT measurements with a percentage of error of 828.6%. This percentage of error increased to 1523.6% when using $f_{\text{max}} = 4$ and N=1000. Therefore, it can be seen how the increment of the N value affects the accuracy of CFD computations when comparing results with WT measurements.

Thus, increasing the number of spectral segment (N) results in an increment of error between CFD and WT measurements for maximum peak pressure coefficients at the windward, roof, and leeward sides of the TTU building. This error is again more noticeable in maximum peak pressure coefficients resulted from using $f_{\text{max}} = 10$, and they are due to the use of the cutoff filtration method used by LES studies. Since $f_{\text{max}} = 10 > f_{\text{LES}} = 4$, the inflow turbulence method filters out these frequencies causing discrepancies between CFD calculations and WT measurements. Thus, considering these inconsistencies, it could be recommended to use a
frequency equal to or lesser than $f_{LES}$ to get a more accurate agreement between CFD calculations and WT measurements.

$f_{min} = 0.1$, $f_{max} = 4$, $D_N = 0.15$ Hz, $N = 50$

$f_{min} = 0.1$, $f_{max} = 10$, $D_N = 0.38$ Hz, $N = 50$

Figure 4.12. Maximum pressure coefficients ($C_p$-max) for $N=50$ (a) $f_{max}=4$ and (b) $f_{max}=10$

$f_{min} = 0.1$, $f_{max} = 4$, $D_N = 0.0755$ Hz, $N = 100$

$f_{min} = 0.1$, $f_{max} = 10$, $D_N = 0.191$ Hz, $N = 100$

Figure 4.13. Maximum pressure coefficients ($C_p$-max) for $N=100$ (a) $f_{max}=4$ and (b) $f_{max}=10$
Figure 4.14. Maximum pressure coefficients (Cp-max) for N=500 (a) fmax=4 and (b) fmax=10

Figure 4.15. Maximum pressure coefficients (Cp-max) for N=1000 (a) fmax=4 and (b) fmax=10
4.4 Computation with Building: Comparison of CFD Minimum Peak Pressure Coefficients with WT Peak Pressure Coefficients

The minimum peak pressure coefficients for $f_{\text{min}} = 0.1$ and $f_{\text{max}} = 4$, and $f_{\text{min}} = 0.1$ and $f_{\text{max}} = 10$ are calculated and compared with minimum peak pressure coefficients from WT measurements. These results are shown in Figure 4.16 through Figure 4.19. The flow calculations were done for a non-dimensional time (TTIME) of 100 units with a time step (DTT) of 0.02. The maximum peak pressure coefficient plots are reported following the TTU building diagram showed in Section 4.2 in Figure 4.7.

Critical points are again taken into consideration to calculate the percentage of error at the windward side, roof, and leeward side of the TTU building. For the comparison of CFD minimum pressure coefficients with WT measurements, points equal to -0.15H from the roof top for the windward side, 0.94H from the roof top for the roof, and 2.7H from the roof top for the leeward side were considered. These are the most critical points along the centerline of the TTU building in the resulted plots. Therefore, these points were considered to calculate percentage of errors between CFD measurements and WT measurements.

As seen in Section 4.3, maximum peak pressure coefficients resulted from CFD measurements using $f_{\text{max}} = 4$ are more accurate with WT measurements than maximum peak pressure coefficients resulted from CFD measurements using $f_{\text{max}} = 10$. Different results are seen for minimum peak pressure coefficients.

The maximum errors on the roof for $f_{\text{max}} = 4$ are around 22.4%, 8.6%, 4.1%, and 17.1% for N=50, 100, 500, and 1000, respectively whereas that the maximum errors for $f_{\text{max}} = 10$ at the same location are around 45.7%, 4.5%, 1.0%, and 8.6% also for N=50, 100, 500, and 1000,
respectively. Here for a number of spectral segment (N) equal to 50, results from $f_{\text{max}} = 4$ calculations have better agreement with WT measurements than $f_{\text{max}} = 10$ results. Nevertheless, for N=100, 500, and 1000, results from $f_{\text{max}} = 10$ measurements show better agreements with WT measurements than $f_{\text{max}} = 4$ results. For the windward side of the building, errors are around 32.3%, 22.6%, 62.9%, and 91.9% for N values equal to 50, 100, 500, and 1000, respectively whereas the errors on the windward side of the building for $f_{\text{max}} = 10$ are 124.2%, 1.6%, 98.4%, and 121% for N values equal to 50, 100, 500, 1000, respectively. Overall, results from $f_{\text{max}} = 4$ measurements show better agreements with WT calculations than $f_{\text{max}} = 10$ results, and these percentage of errors usually increase once the N value increases for both $f_{\text{max}} = 4$ and $f_{\text{max}} = 10$ measurements. However, when N=100, the percentage of error from $f_{\text{max}} = 10$ calculations is much lesser than the percentage of error from $f_{\text{max}} = 4$ calculations. For the leeward side of the building, it is clear that overall $f_{\text{max}} = 4$ results keep a much better agreement with WT measurements than $f_{\text{max}} = 10$ results. It is also observed that once the number of spectral segment (N) increases, the percentage of error also increases for both $f_{\text{max}} = 4$ and $f_{\text{max}} = 10$. However, as stated before, measurements from $f_{\text{max}} = 4$ are still much closer to WT results.

Overall, for the windward and leeward sides of the building, results from $f_{\text{max}} = 4$ calculations show a better agreement with WT measurements than results from $f_{\text{max}} = 10$ calculations. Thus, one can see the $f_{\text{LES}}$ cutoff issues on the minimum peak pressure coefficients. However, these percentages of error significantly increase when the number of spectral segments (N) increases. On the roof side of the building, both $f_{\text{max}} = 4$ and $f_{\text{max}} = 10$ results have in general good agreements with WT measurements. In fact, once the N value increases, the percentage of error decreases, which is the opposite of what happens at the windward and leeward sides of the building.
building. One would think that results from $f_{\text{max}} = 4$ calculations at the roof side of the building would show a better agreement with WT measurements. However, only N=50 for $f_{\text{max}} = 4$ has a better percentage of error at the roof than when using N=50 for $f_{\text{max}} = 10$. For N=100, 500, and 1000, results from $f_{\text{max}} = 10$ have a better agreement with WT measurements than results from using $f_{\text{max}} = 4$. Overall, results from $f_{\text{max}} = 4$ calculations still show much better agreement with WT measurements due to the filtration process used in LES methods.

From this study, it can be stated that using a maximum frequency ($f_{\text{max}}$) lesser or equal to $f_{\text{LES}}$ would result in less percentage of error when comparing CFD measurements to WT measurements. However, further research needs to be performed to investigate more thoroughly the effect of the variations of the number of spectral segment (N) when using inflow turbulence methods based on LES methods.

$f_{\text{min}} = 0.1$, $f_{\text{max}} = 4$, $D_{\text{N}} = 0.15$ Hz, $N = 50$

$f_{\text{min}} = 0.1$, $f_{\text{max}} = 10$, $D_{\text{N}} = 0.38$ Hz, $N = 50$

Figure 4.16. Minimum pressure coefficients (Cp-min) for N=50 (a) $f_{\text{max}}=4$ and (b) $f_{\text{max}}=10$
Figure 4.17. Minimum pressure coefficients (Cp-min) for N=100 (a) fmax=4 and (b) fmax=10

Figure 4.18. Minimum pressure coefficients (Cp-min) for N=500 (a) fmax=4 and (b) fmax=10
f_{min} = 0.1, f_{max} = 4, D\text{N} = 0.0075\ Hz, N = 1000

f_{min} = 0.1, f_{max} = 10, D\text{N} = 0.0192\ Hz, N = 1000

Figure 4.19. Minimum pressure coefficients (Cp-min) for N=1000 (a) fmax=4 and (b) fmax=10
5 Conclusions and Future Work

The behavior of the pressure and the velocity in the use of the Narrowband Synthesis Random Flow Generator (NSRFG) inflow turbulence method was investigated. The pressure and velocity time history were calculated without building in the computational domain but at the building location for the Texas Tech University (TTU) building. The number of spectral segment \( N \) was varied from 50 to 1000.

From CFD measurements, it was observed that by using \( f_{max} = f_{grid} = 4 \) spurious pressures are reduced compared to using \( f_{max} = 10 \). Keeping a number of spectral segment \( (N) \) low keeps a smoother and more continuous pressure which avoids spurious results. However, further research needs to be performed related to variations of \( N \) values. Usually, the greater the \( N \) value is, the more continuous and less spurious results for pressure are obtained. In the CFD computations performed in this thesis, the lesser the \( N \) value was, the less spurious pressure was obtained. Thus, further investigation is needed to understand the reason behind this issue. Therefore, according to these results, it is recommended that when working with inflow turbulence generators based on LES studies such as the NSRFG to use maximum frequencies equal to LES frequencies \( (f_{max} = f_{grid}) \) to keep the continuous pressure and less margin of error. The \( N \) value should be kept low to assure less spurious pressure. However, as mentioned before, more research needs to be performed related to the \( N \) value variations.

Mean and peak pressure coefficients on the 1:6 scale of the TTU building were calculated using the NSRFG inflow turbulence method adopting a computational domain with a grid spacing of \( H/16 \). The CFD results were later compared to WT measurements reported by Moravej (2018). Overall, CFD measurements for mean peak pressure coefficients are fairly accurate with WT measurements for both \( f_{max} = 4 \) and \( f_{max} = 10 \). However, mean peak pressure coefficients
from $f_{\text{max}} = 4$ calculations showed lesser discrepancies when comparing them with WT measurements. The fact that $f_{\text{max}} = 4$ calculations had a better agreement with WT measurements is attributed to the LES cutoff principle where maximum frequencies ($f_{\text{max}}$) greater than LES frequencies ($f_{\text{LES}}$) are filtered out. It was also observed that once the N value increases, the percentage of error increases too. Therefore, one can conclude that when using the NSRFG method, it is better to use maximum frequencies ($f_{\text{max}}$) less than or equal to LES frequencies ($f_{\text{LES}}$) keeping a number of spectral segment (N) low to obtain less percentage of error when comparing results with WT measurements.

Maximum peak pressure coefficients resulting from using $f_{\text{max}} = 4$ had a better agreement with WT measurements than using $f_{\text{max}} = 10$. However, errors were still observed at the windward side, roof edge, and leeward side of the building for $f_{\text{max}} = 4$ results. These errors increase by large percentages once the number of spectral segment (N) is also increased. Thus, considering these inconsistencies, it could be recommended to use a frequency equal or lesser than $f_{\text{LES}}$ keeping the N value small to get a more accurate agreement between CFD calculations and WT measurements. For minimum peak pressure coefficients, overall, for the windward and leeward side of the building, results from $f_{\text{max}} = 4$ measurements keep a better agreement with WT measurements than results from $f_{\text{max}} = 10$. Therefore, it is seen again how the $f_{\text{LES}}$ filtration process cutoff frequencies greater than LES frequencies results in discrepancies between CFD calculations and WT measurements. For the roof side of the building, a different scenario is seen. Here, results from using both $f_{\text{max}} = 4$ and $f_{\text{max}} = 10$ have pretty good agreements with WT measurements. Also, it is seen that a decrease in the percentage of error occurs once the N value increases which is the opposite of what happens at the windward and leeward sides of the building. However, this is only seen on the roof side of the building. For the other sides, overall,
calculations from $f_{\text{max}} = 4$ still show much better agreement with WT measurements. Thus, it can be stated that using a maximum frequency ($f_{\text{max}}$) lesser or equal to $f_{\text{LES}}$ would result in less percentage of error when comparing CFD measurements with WT measurements.

Therefore, it is demonstrated that inflow random flow generators based on large eddy simulations (LES) still experience some complications related to the use of maximum frequencies. However, if the principle of $f_{\text{max}} = f_{\text{LES}} = f_{\text{grid}}$ is applied, results become considerably more accurate when comparing them to WT measurements. For future work, more examination of the inflow turbulence generators needs to be performed to decrease the discrepancies between CFD measurements and WT measurements. If this is achieved, CFD techniques can become a more convenient and cost-effective tool in the measurement of peak pressures in low-rise buildings. Also, further research needs to be done to investigate the effect of the variations of the number of spectral segment ($N$) when using inflow turbulence methods based on LES.
6 References


Appendix A. Computing Time

Computer simulations were run using the graduate computer lab located in the Civil Engineering Department. This computer lab runs under the Windows operating system. The computer was being used only by one user at the time. Table 7.1 below shows the computing time in hours took by the computer to run the simulations for the peak pressure coefficients (Cp) calculations for the TTU building.

Table 7.1 Computing time for each performed computer simulation for Cp calculations

<table>
<thead>
<tr>
<th>Data</th>
<th>Time (Hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>With Building</td>
<td></td>
</tr>
<tr>
<td>fmax= 4.0, DN=0.15 Hz, N=50</td>
<td>17.97</td>
</tr>
<tr>
<td>fmax= 4.0, DN= 0.0755 Hz, N=100</td>
<td>32.37</td>
</tr>
<tr>
<td>fmax= 4.0, DN= 0.0151 Hz, N=500</td>
<td>33.85</td>
</tr>
<tr>
<td>fmax= 4.0, DN=0.0075 Hz, N=1000</td>
<td>30.62</td>
</tr>
<tr>
<td>fmax= 10.0, DN= 0.38 Hz, N=50</td>
<td>35.18</td>
</tr>
<tr>
<td>fmax= 10.0, DN=0.191 Hz, N= 100</td>
<td>22.12</td>
</tr>
<tr>
<td>fmax= 10.0, DN= 0.0383 Hz, N= 500</td>
<td>34.87</td>
</tr>
<tr>
<td>fmax= 10.0, DN= 0.0192 Hz N=1000</td>
<td>31.50</td>
</tr>
</tbody>
</table>
Appendix B. Dimensional Frequencies and Number of Spectral Segment (N) Calculations, and the Frequency Width (DN) Variation

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>f1=</td>
<td>0.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>f2=</td>
<td>4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Uh=</td>
<td>7.66</td>
<td>m/s</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>H</td>
<td>3.96</td>
<td>m</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>n1=</td>
<td>=(B1*B3)/(B4) Hz</td>
<td>0.19 Hz</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>n2=</td>
<td>=(B2*B3)/(B4) Hz</td>
<td>7.74 Hz</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>DN=</td>
<td>0.75</td>
<td>Hz</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>IFS=</td>
<td>=SUM((2*(B6)/B9),1))/2</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>IFE=</td>
<td>=SUM((2*(B7)/B9),1))/2</td>
<td>11</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>N=</td>
<td>=B12-B11</td>
<td>10</td>
<td></td>
</tr>
</tbody>
</table>

Figure 7.1 IFS, IFE, and N calculation and DN variation