MQP LDA-1905

Wind Load Design of a Triangular Shaped Building Using Finite Element Analysis

Presented by:

Khant Win Htet

Submitted to:

Professor Leonard Albano and the Worcester Polytechnic Institute Civil & Environmental Engineering Department in partial fulfillment of the requirements for the Degree of Bachelor of Science

2018-2019

Abstract

This major qualifying project investigates wind loading and structural design of a triangular-shaped 5-story building in San Francisco. Finite Element Method (FEM) software, ANSYS 19.1 is adopted to create a virtual wind tunnel test in accordance with *ASCE 7-16* and *ASCE 49-12*. This project focuses on simulating two types of wind-tunnel test, Rigid Pressure Model Technique and High-Frequency Base-Balance (Time-Domain Analysis) Technique. Of those two techniques, the final structural design in based on the Rigid Pressure Model Technique.

Acknowledgements

I would like to extend my gratitude first and foremost to my project advisor, Professor Leonard D. Albano. His extensive knowledge on the project topic and guidance was invaluable throughout the course of our project work.

I would also like to thank Jonathan Palczynski for his contribution to the proposal and early stage of this project. Jon has contributed to the introduction and background knowledge of the proposal. In addition, I would like to acknowledge that some of the materials that are referred to and/or adopted in this project are the contribution of Jonathan Palczynski: (1) determining the site location, (2) proposing building layout, (3) investigating the soil properties of the location, (4) writing introduction and background of the proposal and (5) editing the proposal as a whole.

Capstone Design Statement

To comply with the accreditation requirements established by the Accreditation Board for Engineering and Technology (ABET), the Department of Civil and Environmental Engineering at WPI requires all Major Qualifying Projects (MQP) to include a Capstone Design Experience. The Capstone Design Experience requires students to design a system, component, or process to meet desired needs by applying knowledge and skills acquired in earlier coursework and incorporating engineering standards and realistic design constraints. To fulfill this requirement, this MQP is the design and evaluation of a triangular-shaped 5-story mixed-use residential and office building located in San Francisco, CA. Alternate structure systems were proposed and investigated.

During the structural design process of the building, various effects of lateral loads due to wind were investigated. The design followed standard engineering building codes: *AISC Steel Specifications*, *ACI Concrete Specifications* and *ASCE 7-16 Provisions and Commentary*. To evaluate the constructability and efficiency of the building, Finite Element Analysis was applied with different design scenarios. The main study of the project explored how wind loadings can affect the triangular shaped building. The final focus of the project reported a cost analysis of the structural skeleton of the building. The MQP incorporated economic, constructability, ethical, health and safety, and social design constraints.

Economic

A cost estimate of structural design was completed and broken down into individual components, such as beams, columns and connections. Different beam and columns sizes, column arrangements and flooring systems were considered to determine the most cost-effective design. The recommendation for the final structure considered the estimated cost.

Constructability

The aspect of constructability was highly prioritized during the development of design scenarios. Efficient constructability includes using standard and readily available beam/column sizes and repetitive column spacing.

Ethical

During the design process, the code of ethics for engineers provided by the National Society of Professional Engineers (NSPE) was followed. The building is in an area subject to wind loads and was required to adhere to wind design provisions in *ASCE 7-16*. While this may require additional design work and increased material costs, compromising the overall quality and safety of the structure for cost efficiency was strictly avoided. Moreover, creating a design with structural integrity was the top priority of this capstone project.

Health and Safety

Health and safety during the construction as well as post construction are one of the fundamental concerns within the code of ethics by NSPE. All structural designs followed standard building code provisions, such as *ASCE 7-16* and the *AISC* and *ACI* specifications. Because the building supports office and residential spaces and is in a location subject to seismic loads the appropriate risk category were being assigned. Assigning the appropriate risk category, exposure category for wind loads and following the design procedures associated with it ensured the safety of occupants.

Social

Although this project is mainly aimed to design the structural skeleton of the building, the proposed site is planned to be office/residential building. The proposed building comprised of small and medium-sized office spaces, small shop spaces, and residential apartments for different types of social background. Column grids must account for the spatial layout to ensure that adequate and functional offices and apartments can be implemented into the building.

Professional Licensure Statement

To design a product, either an electronic device or living space, one of the utmost important factors is the safety and health of the public. The engineers are required to train rigorously to a certain level where they are considered as competent to design or review a product. As civil/structural engineers, designing or constructing a building comes with a significant amount of risk, and the necessary training requires years of academic knowledge and work experience.

To create a standard limit of becoming a licensed engineer, the National Society of Professional Engineers (NSPE) specifies to archive a PE (professional Engineer), one must (1) finish a 4-year engineering program from accredited university/college, (2) pass the FE (Fundamental of Engineering) exam. (3) complete 4 years of work experience under a PE, and (4) pass the PE exam. Once an engineer becomes a PE, he/she has a lot of authority as well as responsibility. Since PE can design, review and approve a project, following the ethical guidelines is an essential responsibility.

This capstone design project is strictly related to professional practice for structural engineers. It includes structural member sizing, structural analysis and preliminary design drawings. The outcome of the project will be a product that needs a professional engineers' approval if the project ever come to reality.

Table of Contents

List of Figures

List of Tables

1. Introduction

Modern geometric architecture is becoming more popular; however, it presents new structural design challenges. New structural framing methods, materials, and computer tools must be investigated and used to address the complex geometric structures and associated loading scenarios. Triangular shaped buildings are a type of modern geometric structure but due to structural design challenges they are not very common. Nonetheless, a few examples exist: the Potsdam Platz 11 in Berlin, Germany and the Flatiron Building in New York, New York. Both buildings are considered iconic landmarks of the area and draw tourist attention due to their unique shapes. However, these buildings are not in areas subject to dangerous seismic and wind loads.

To set a standard for the design of a modern triangular shaped building subject to wind loads, this project explored the design and evaluation of a theoretical triangular-shaped 5-story, mixed-use residential and office building located in San Francisco, California. The work was primarily focused on the steel framing system. *ASCE 7-16* specifies that irregularly shaped structures including triangular building must undergo wind-tunnel tests. Historically, wind-tunnel tests were performed within velocity-controlled chambers and scaled building models with pressure taps attached to them to measure the peak pressures. With the increasing use of computational models and the emergence of user-friendly FEM software such as ANSYS and ABAQUS, wind-tunnel tests are performed with computers. However, it becomes critical for the user to create a simulation that can accurately mimic the real-life scenario. The major task of this project were structural frame design, CAD Model, Finite Element Model, Finite Element Analysis and cost estimate. The final outcomes of the project were the completed evaluation and design of the triangular shaped structure along with guidelines to follow when designing a uniquely shaped structure subject to wind loads.

2. Background

This section provides the information needed to understand the major aspects of the project. The major aspects of the project are the building location and its related loads, analysis of wind load, and the utilization of computer software tools.

2.1 Building Site Location

The building site for the proposed triangular-shaped, mixed-use residential and office building is located at 1811 Jerrold Ave San Francisco, CA 94124. It is a 60000 Sq. Ft. triangular shaped lot currently for sale for $8 million. Land in San Francisco is hard to find, and this site was chosen due to its already level surface and large size. The general site location was also chosen due to it being in San Francisco which is subjected to high seismic and wind loads.

Figure 1: 1811 Jerrold Ave San Francisco, CA 94124 Building Site

2.2 Structural Analysis

The selected building site is in a critical location because the building design must address the significant lateral loads. Lateral loads are live loads that act as horizontal forces on the vertical building structure. They consist of seismic loads and wind loads. It is important to follow *ASCE 7-16* as it contains new specifications regarding wind and seismic loading criteria. Many locations have transferred to or have deadlines set to transfer from *ASCE 7-10* to *ASCE 7-16*. *ASCE 7-16* contains safer, updated methods but also stricter methods for Site Class D structures and tall structures.

Wind Loads are caused by air pressure acting on a building's surface. Unlike a rectangular building, a triangular building will not experience symmetrical wind distributions. The maximum and minimum pressures occur on a triangular building when the wind blows parallel to one of the faces. Due to the triangular shape the building will experience much higher twisting moments about its base when compared to a rectangular building (Abdusemed & Ahuja, 2015). *ASCE 7-16* was referred to find the specifications regarding with the wind loads. Chapter 27 to Chapter 31 of *ASCE 7-16* concerns with the specifications of wind loadings and wind-tunnel test specifications (Chapter 31). ASCE has provided with addition provisions specifically on Wind-Tunnel Test Specifications in *ASCE 49-12- Wind Tunnel Testing for Buildings and Other Structures*. Both Specifications are used during investigation of wind loadings

Dead loads are the frame's weight and any object that is permanently fixed to the building. Live loads are loads which can move and are not permanently fixed to the building. Live load design values are generally based on building occupancy classification and can be found in *ASCE 7-16*.

The structural design of a building can be divided into two parts: the superstructure and foundation. The superstructure design consists of the column layout and framing system. The column layout must account for usability and functionality of the spatial layout. The spatial layout is the floor plan and the arrangement of furniture, cubicles, counters, equipment and other items located within the floor plan (Nha & Leblanc, 2002). If columns are placed only to create the best structure with no consideration for the spatial layout it may yield a completely unusable building. The superstructure design also consists of sizing the beams, columns, and framing connections. The foundation design consists of sizing footings, the foundation wall dimensions, and other components of the foundation. Foundation design follows the *American Concrete Institute Manual of Concrete Practice* (ACI Concrete Manual).

Beam and column sizes depend largely on the applied load combinations but also the frame layout. Modern structures typically contain large open spaces which increase the unbraced and effective lengths of beams and columns respectively: adversely affecting their strength. Additionally, bracing (or not bracing) of the frame directly affects column sizing. If a frame is unbraced, substantial Second-Order effects can occur. A properly braced frame can minimize Second-Order effects: lessening the column sizes. Connection variations include welded design, bolted design and coped (bolted) vs. not coped (welded) connections. Connections can be subject to pure shear, pure axial, or a combination of both. Design of beams, columns, and connections all follow *American Institute of Steel Construction: Steel Construction Manual* (AISC Steel Manual).

2.3 Finite Element Analysis

Finite Element Analysis was used to analyze overall structure response and framing connection responses to wind loads present in San Francisco, CA. Finite Element Analysis works by dividing a complex structure into simple shaped elements connected by nodes. It can perform static analysis, modal analysis, transient dynamic analysis, buckling analysis and more. By simplifying a structure into simple shaped elements a computer is able to solve the structure through large sets of simultaneous equations. Properties are calculated at the nodes and then interpolated over the element. Commonly used elements are 1D Beam Elements, 2D Plate Elements, and 3D Solid Elements; a mix of elements can be used in a Finite Element Model. After the Finite Element Model is solved it can display items such as displacement, stress, strain, mode shapes and temperature. A common way to represent results is with a contour plot (Weck & Yong, 2004).

Modal Response Spectrum Analysis is a linear dynamic analysis method which measures the contribution from each natural mode of vibration to determine the maximum wind response of an elastic structure. Because of the unique shape of the building and its location, a linear static analysis cannot be used. Once the analysis is complete, shear forces are computed to be distributed along the height of the building. (Emrah, 2016). The building was designed from a seismic loading point of view and then was checked for wind loading. Similar to the seismic design, because the structure is a unique shape, a normal wind load procedure cannot be used. A Computational Fluid Dynamics model was made to analyze the wind loads acting on the triangular structure. Computational Fluid Dynamics generates fluid flow through a model of the structure and uses a numerical analysis method, such as the Finite Element Method, to find a solution (Lohner, 2009).

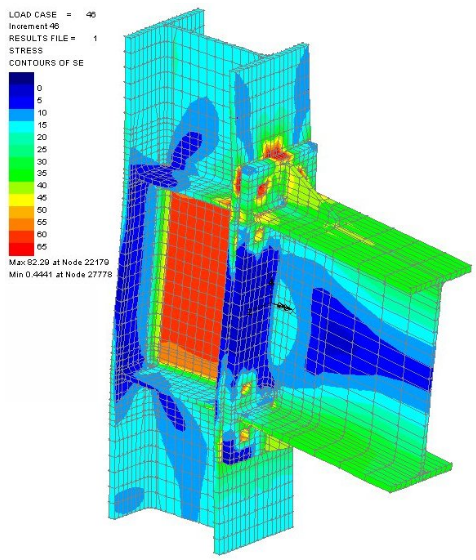


Figure 2: Finite Element Model with Stress Contour Plot (Lindsey, 2017)

Finite Element Analysis can be divided into three main parts: pre-processing, solving, and post-processing. Pre-processing is the development of a Finite Element Model. This is where a structure is meshed into elements, material properties are assigned, loads and boundary conditions are applied, etc. After a Finite Element Model is created it needs to be solved. A text file is created with all components of the pre-processing phase: node locations, material properties, loads, etc. This text file is then sent to the solver. Solving is when the set of equations associated with the structure are put into matrix form and the variables are then solved for. The matrices can be extremely large and require a lot of computing power to be solved within a reasonable amount of time. After the solving is complete the user can view and analyze the results, known as post-processing. Post-processing is where results of stress, strain, displacement, etc. can be viewed as contour plots, graphs, etc. For example, solvers can calculate various types of stresses throughout a structure, including: Mises, Tresca, Principal, Axial and Shear (Roensch, 2008).

There are a lot of commercially available Finite Element Analysis software packages; these consist of pre-processors, solvers, post-processors or "all in one" packages. A few examples include ABAQUS, Hyper Mesh, MSC NASTRAN, MSC PATRAN, ANSYS and STAAD. For this project the software was limited to an educational version that does not have substantial model size limitations. When choosing a pre and post-processor it is important to consider the Finite Element Model creation capabilities, ability to import CAD geometry, solver support, user interface, result presentation, automation and scripting capabilities, and overall value (Wertel, 2018).

From initial investigation it was determined that the ANSYS Workbench software package can create the overall structure and provide member forces through Modal Analysis. ANSYS Workbench contains Design Modeler, a built-in CAD tool to create a framing system that can then be meshed with the built in pre-processor. ANSYS Workbench also includes a built-in solver and post-processor. For connection design, it is generally easier to use CAD software for creating the model geometry rather than creating the geometry within a pre-processor. CATIA is a CAD tool that can create framing connections and then have the geometry directly input into to MSC PATRAN for pre-processing. ANSYS, MSC NASTRAN, MSC PATRAN and CATIA V5 all have free educational versions available to students which make them the most viable options.

2.4 Nature of Wind Load

In order to determine the methodology to accurately mimic the wind loads acting on the building, it is important to first understand the nature of the wind. One of the first terminologies that is important in wind engineering is the boundary layer. A boundary layer is defined as a layer of a fluid that is in contact or in proximity to the bounding fluid or solid [Epifanov, 2011]. Within this layer, the viscosity effects are significant: the velocity profile is almost zero at the contact level, and increases with respect to height. Atmospheric boundary layer (ABL) is the layer of atmosphere that is in contact or proximity to the earth's surface. Atmospheric boundary layer can be influenced by the meteorological parameters such as fluid temperature, pressure, and moisture content of the atmosphere. Therefore, the atmospheric boundary layer can range from 1km to 100m depending on the time of the day. Since the proposed building height does not exceed the atmospheric boundary layer limit, the effect of ABL was considered. One of the effects of ABL is that the velocity of the fluid (air) that is in contact with the earth's surface has the same velocity as the surface due to friction between the air and the surface. As shown in Figure 3, when the layer of air gets further away from the surface, its velocity will increase until the limiting maximum velocity is achieved. From the perspective of fluid mechanics, the fluid responds differently within ABL than outside of ABL due to the development of shear stresses. For this reason, when determining the wind loads of most low-rise buildings and high-rise buildings within the atmospheric boundary layer, the velocity profiles are not constant throughout the height, but rather increasing with respect to the height.

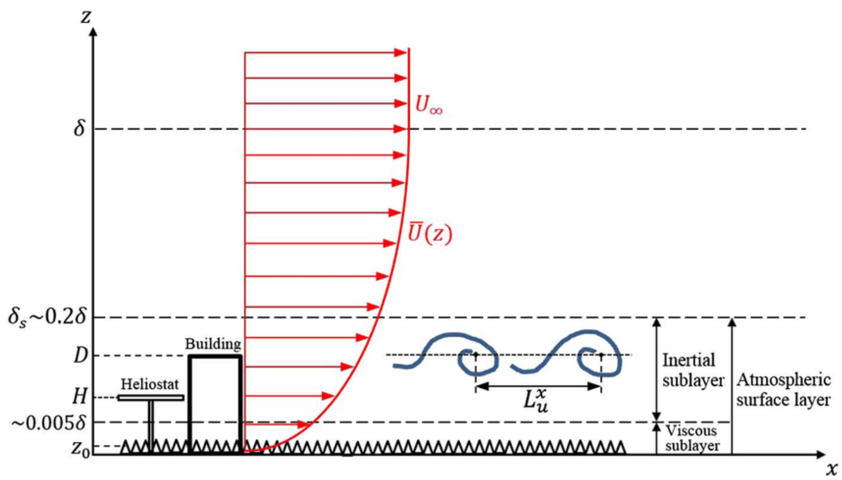


Figure 3: Increasing velocity profile with respect to height, showing the effect of ABL [Emes, 2017]

Apart from the atmospheric boundary layer effect, when wind with a certain velocity profile interacts with a building structure, there is a significant change in velocity around the building, creating positive and negative pressures. Positive pressures are mostly exerted on the windward surface of the building while negative pressures (suctions) are exerted on all surfaces except the windward surface [Epifanov, 2011]. The effects of wind-induced pressures can cause significant stress on the building. Structural engineers have to design a building so that it can withstand the peak wind pressure as well as the dynamic effects of the wind. When ignored, the dynamic effects of the wind can be detrimental to buildings and bridges. One of the examples would be collapsing of Tacoma Narrow Bridge in 1940 which collapsed due to aerodynamic magnifications [Harish, 2019]. Therefore, understanding the nature of wind, and its interaction with structures can be a great asset to present structural engineers.

3. Building Design and Layout

This section includes the basic geometry and dimensions of the building, column locations, and beam-girder configurations. The following sub-sections also discuss the type of bracing used, how the loads are transferred from beams to foundations and other necessary information of the building.

As proposed in the introduction section, the building design is a 5-story triangular shaped (right equilateral triangle). The sides of the triangular building are 200ft long while the hypotenuses side is 283ft long. The first floor story height is 20 feet, and the rest of the floors have a story height of 15 ft. The total height of the building would be approximately 80 ft. The overall building would have 20,000 square feet of floor area per story which would sum up to around 10,000 square foot of total floor area.

The columns are located as square grids with 20-feet spacing except for the 45 degree corners of the buildings which have a column spacing of 10 feet. The reason of choosing 20 feet was that it equally divides 200 feet which are the lengths of the equilateral triangle. The floor areas located at two corners of the triangular would behave as cantilevers supported by the nearest beam or girder. Table 1 presents the numbers of beams, girders and columns that made up each floor of the building. This information is later used as an aid in the sizing of the structural members.

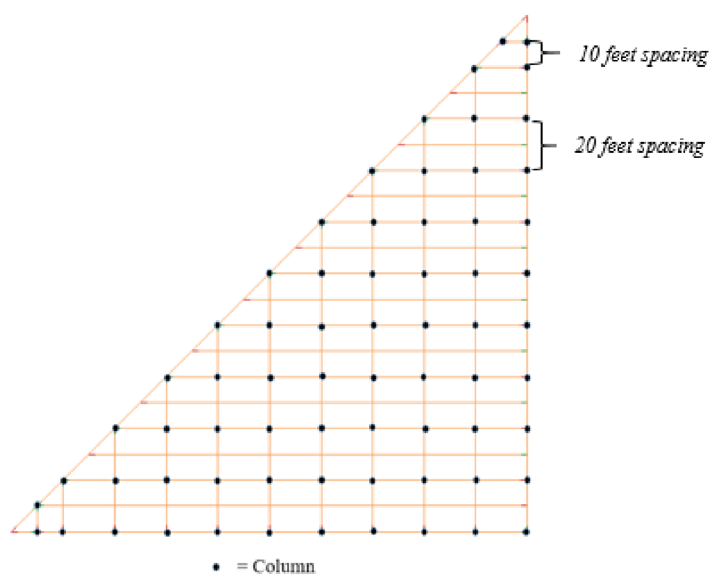


Figure 4. Building Layout showing the column locations

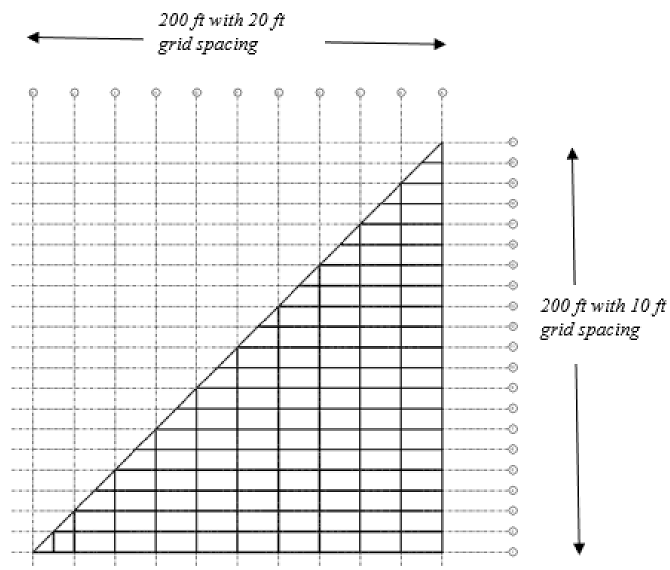


Figure 5. Building Structural Layout showing beams and girders locations

From the building layout view, members parallel to the horizontal grid represent the beam members and members parallel to the vertical grid represent the girder members. The members along the hypotenuse side of the triangular also serve as girders. The bracing system was also set-up to resist the lateral wind loads. The configuration of the bracing system is according to Figure 6. The summary of the total structural members of the building including the bracing system can be found in Table 1.

Table 1. Table showing the quantities of different structural members

Structural Member Type	Quantity per floor	Total Quantity (Ground Level to Roof Level)
Beam	111	666
Girder	67	402
Column	68	340
V braces	27	135

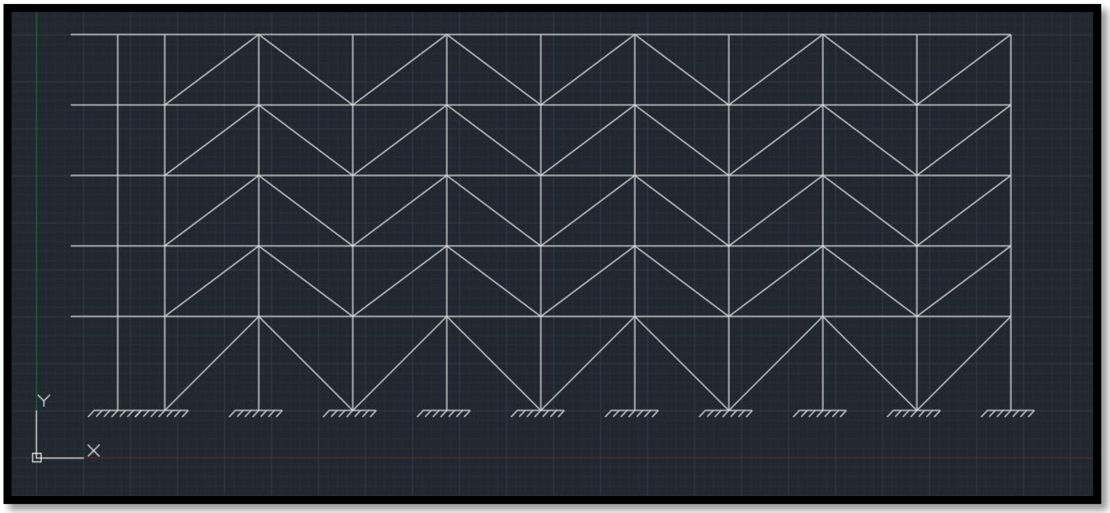


Figure 6: Bracing System on the Equal-Leg Side of the Triangle

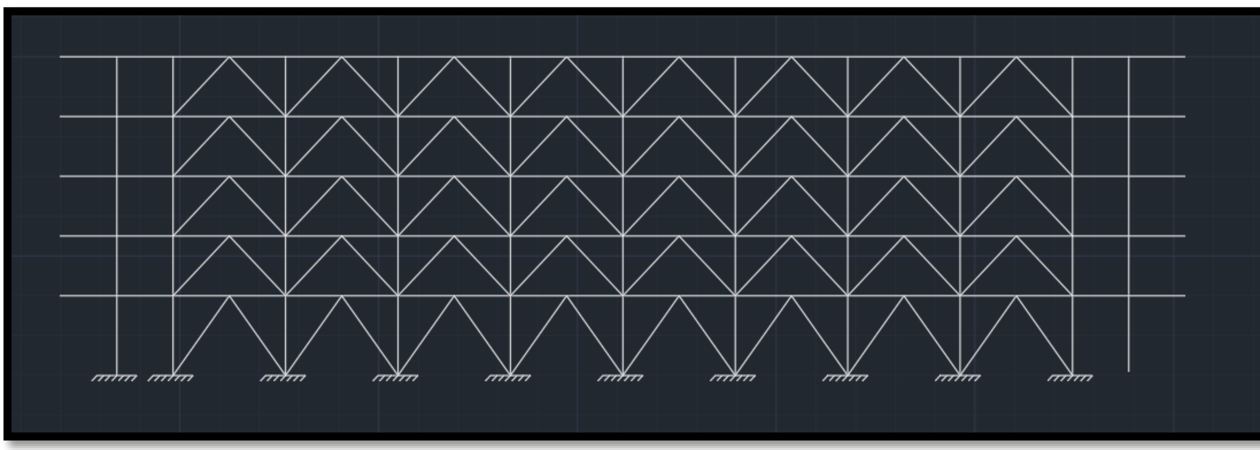


Figure 7: Bracing System on the Hypotenuse Side of the Triangle

4. Structural Loads

This section explains the basic gravity load cases that were considered during the design process and how each load case is calculated using the *ASCE 7-16* specifications.

4.1. Load Combinations

The governing load combination was calculated according to section 2.3.1 and 2.3.6 from *ASCE 7-16*. ASCE also states that wind and seismic loads need not be considered to act simultaneously. *ASCE 7-16* mentions both LRFD and ASD load combinations. For this project, LRFD load combinations were chosen.

4.2 Gravity Load

The gravity load system resists floor live load, roof live load, and superimposed dead load and self-weight of the structural members. The superimposed dead loads in this building include (1) exterior wall loads, (2) Ceiling/ Electrical/Mechanical loads and (3) Flooring loads. Self-weight has not been taken account until the analysis process of the gravity loads.

The structural system is designed in a way that loads are uniformed distributed to the beams first and then transferred to girders as point loads which are then transferred to columns and footings.

4.3 ASCE 7-16 Specifications on Gravity Load

Chapters 3 and 4 of ASCE 7-16 were referenced when considering the gravity load system of the structure. Dead load was divided into two categories: (1) superimposed dead load and (2) self-weight. Superimposed dead load includes ceiling, flooring, estimated exterior wall and HVAC/Electrical system. Since the project is mainly focused in the structural design of the building, the superimposed dead loads were assumed as a certain combined value rather than calculating them individually. Some of the superimposed loads such as Ceiling/ Electric/Mechanical and Flooring Loads are specified in Table 2 which are referred from *ASCE 7-16.* However, when performing member sizing in RISA 3D, the superimposed loads was assumed as 40psf.

According to the building design, it was decided that the first and second floor serve as office space and the rest of the floors are for condominium apartments. This was important in considering the live loads of the building because office space and condominium apartments have different uniform live load values (ASCE 7-16, S\Table 4.3-1). Roof live load values were also obtained from Table 4.3-1. The summary of live load, roof live load and dead load are presented in the following table.

Table 2. Gravity Load Types and their values

Gravity Load Types		**Remark**
Office Uniform Live Load	50 psf	*ASCE 7-16 Provisions* Table 4.3-1
Residential Uniform Live Load	40 psf	*ASCE 7-16 Provisions* Table 4.3-1
Roof Live Load	20 psf	*ASCE 7-16 Provisions* Table 4.3-1
Exterior Wall Load	48 psf (wall area)	*ASCE 7-16 Commentary* Table C3.1-1a
Ceiling/Electrical/Mechanical	4 psf	*ASCE 7-16 Commentary* Table C3.1-1a
Flooring(metal deck concrete)	27 psf	*ASCE 7-16 Commentary* Table C3.1-1a (3” thick Lightweight Concrete = 8psf x 3= 24psf + Metal Deck ,20 guage = 3psf)

5. Wind Load

Although "Wind Load" should be included in *Chapter 4: Structural Loads*, since the primary objective of the proposed project is focused on the wind loadings and structural design, this section serves as a methodology for obtaining the wind load that is acting on the triangular-shaped building. The following sub-sections will demonstrate the procedures and justify iterations that were used to complete the design.

5.1. ASCE Specifications on Wind Loads

ASCE 7-16 specifies that all building configurations that are not mentioned in Chapter 26, 27, 28 and 30 are subjected to the wind tunnel procedure specified in Chapter 31. However, the wind velocity to perform the wind-tunnel test was acquired from Figure 26.5.1C. The risk category of the building is III. Therefore, the basic wind speed for building in the San Francisco area is about 45m/s or 100mph.

Chapter 31 of ASCE 7-16 describes three different types of wind-tunnel tests: (1) Rigid Pressure Model Test (PM), (2) Rigid high-frequency base balance model test (H-FBBM), and (3) Aeroelastic Model (AM). It also specifies that at least one or more test should be perform for all irregular shaped buildings that are not mentioned in Chapter 26,27,28, and 30.More detailed provisions of each wind-tunnel test are referred in ASCE 49-12.

5.2 Wind Tunnel Test

All the wind tunnel test specifications and provisions provided in ASCE 7-16 and ASCE 49-12 are for the actual boundary-layer wind tunnels (BLWTs) of wind engineering labs. Those kinds of test are performed using (1) open-circuit or closed-circuit chamber, and (2) scaled building models equipped with pressure taps. Both ASCE 7-16 and ASCE 49-12 lack provisions regarding with the computational model or virtual wind-tunnel test.

5.3 Rigid Pressure Model Wind Tunnel Test

Rigid Pressure Model are mostly used to measure the peak or local pressures of the building. During Rigid Pressure Model, any dynamic magnification effects were neglected. Moreover, Rigid Pressure Model was only dependent on the shape and geometry of the building rather than the framing systems and connection types. By using the peak pressure from the test, overall wind load exerting on the building was calculated. For this project, only local and overall wind pressures were considered.

Since the building is triangular-shaped, it was important to measure the peak pressures due to different angles of wind direction. The angle of wind direction that caused the maximum peak pressure would be the one that governed. There were several solvers in ANSYS, such as CFX and Fluent, which are compatible for performing fluid dynamic tests. Both solvers have a lot in common with slight differences in techniques of obtaining a solution [Singh, 2016]. For this project, CFX was used because there were more abundant online resources concerning the use of CFX than compared to the use of Fluent. Figure 8 summarizes the procedures for performing Rigid Pressure Model Test. More detailed explanations of the procedures can be also found in the following sub-sections.

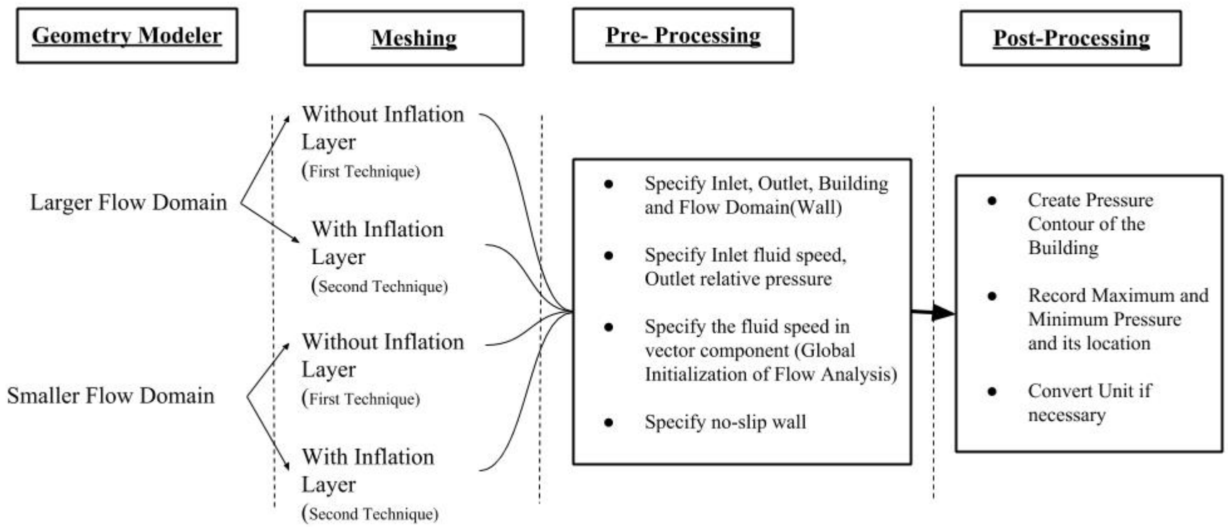


Figure 8: Summary of Rigid Pressure Model Test Procedure Using ANSYS- CFX

5.3.1 Geometry Modeler

The geometry of the rigid pressure model test includes two parts: building and the flow domain. Solidworks was used to build both building and the flow domain. It is mentioned in Section 5.3 that Rigid Pressure Model Test rely on the geometry and configurations of the building rather than the structural details and connections types, simple equilateral triangular block was constructed and confined within the center of the flow domain. The building model dimensions were the same as the actual building.

The objective of the test was to use to ANSYS computational fluid dynamic solver, “CFX” to find the pressure values of the different surfaces of the building. Different angles of wind directions were also considered in order to investigate how the wind directions can influence the peak pressures exerted on the building. Therefore, arbitrary configuration of the building was assumed as α = 0˚ as shown in Figure 9. The α value is increased every 20˚ until it reached 180˚. The reason why α value didn’t reach 360˚ is due to singly symmetric geometry of an equilateral triangle. Some α values such as 45˚, 90˚, and 135˚ are also considered. Building models with different α values are constructed and solved in ANSYS separately. Therefore, a total of 13 models were constructed. Although constructing 13 models in Solidworks was not time-consuming, this method evolved into a more significant inconvenience in ANSYS Set-Up procedures which is discussed in Section 5.3.3.

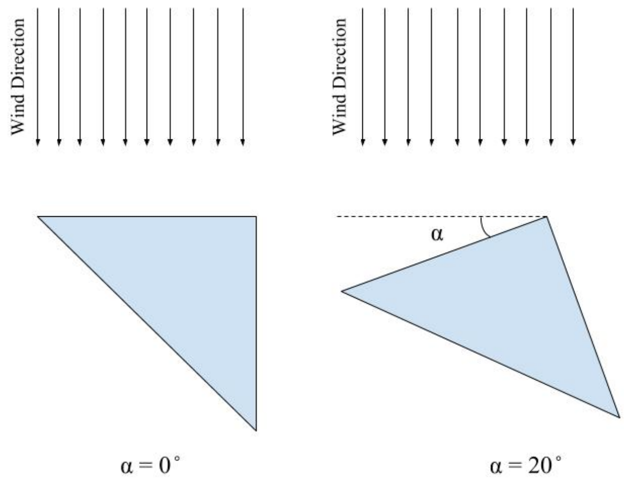


Figure 9: Top View of Buildings with two different wind directions showing the configuration of the arbitrary value α.

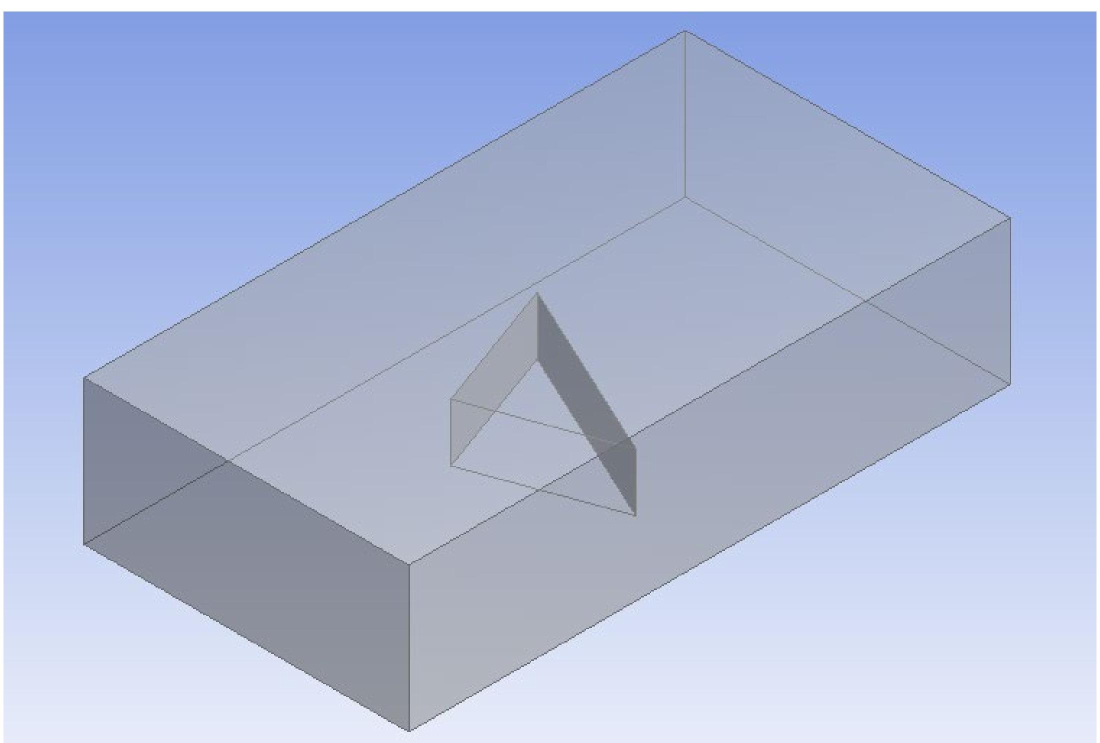


Figure 10: Building Model and Flow Domain

5.3.2 Flow Domain

During the first attempt to conduct Rigid Pressure Model Test, the flow domain had the dimensions of 450 feet wide, 200 feet tall and 830 feet long. The length of the flow domain was almost 3 times longer than the longest side of the triangular. However, in order to validate the results as well as to propose an additional experiment, another test was undergone with a different flow domain size. During the second attempt, the flow domain was updated into a larger confine space with 600 feet wide (33% increase), 250 feet tall (25% increase) and 1500 feet long (80% increase). The final results, and the convergence of results obtained from both flow domains were recorded and compared (Appendix C).

5.3.3 ANSYS 19.1 Set-Up

Unlike the Time-Domain Analysis which used multiple solvers, Rigid Pressure Model Test only used one solver (ANSYS CFX) from geometry modeling stage to post-processing stage. This made the Rigid Pressure Model Test less sophisticated than the Time Domain Analysis. On the other hand, since same procedures had to undergone for 26 different models (13 models for each flow domain), Rigid Pressure Model Test was more time-consuming than Time-Domain Analysis. Although this did not influence the results, more efficient method would have been more desired which will be discussed more in Recommendations (Chapter 10).

5.3.4 Meshing Techniques

Meshing plays an important role in determining the accuracy and the convergence of the solution. Although the smaller and finer the meshing sizes represent more accurate solutions, the processing time to solve the solution will also increase. In this project, a certain level of studies were invested into how to create and improve the meshing process to increase the accuracy and decrease the solving time. Two types meshing techniques were used during the ANSYS CFX process to observe how the solution changes. The first meshing technique was a simple auto-generating mesh with the custom element size of 25.0 feet.

The problem with the first meshing technique was that it created a constant velocity profile throughout the height of the flow domain. As mentioned in Section 2.4, the velocity profile of a fluid within ABL is not uniform, but rather gradient.

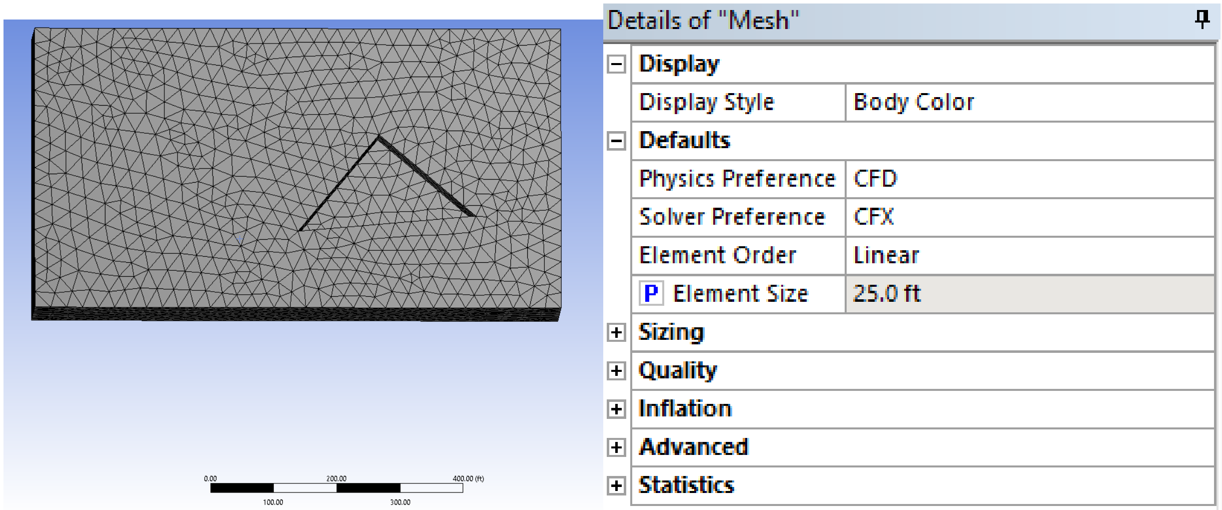


Figure 11: First Meshing Technique with uniform meshing size throughout the flow domain and building (left). Meshing Details (Right).

In order to accommodate the concept of ABL in the flow domain, new method of meshing was adapted. The new meshing technique was to use different meshing element densities in different regions and to mimic the effect of ABL within the flow domain. Inflation layer was used around the building parameter, where the velocity gradient needed to be significant. ANSYS CFX allowed to choose (1) surfaces or edges to add inflation layer, (2) different types of inflation layer, (3) rate of transition of meshing element sizes, (4) maximum meshing element size and (4) growth rate of the elements. Moreover, advanced inflation layer setting with more control on the geometry of the meshing elements is also available in ANSYS CFX. However, in this project, only the parameters shown in Figure 12 were specified. Those values were used throughout different building models (both smaller flow domain and larger flow domain).

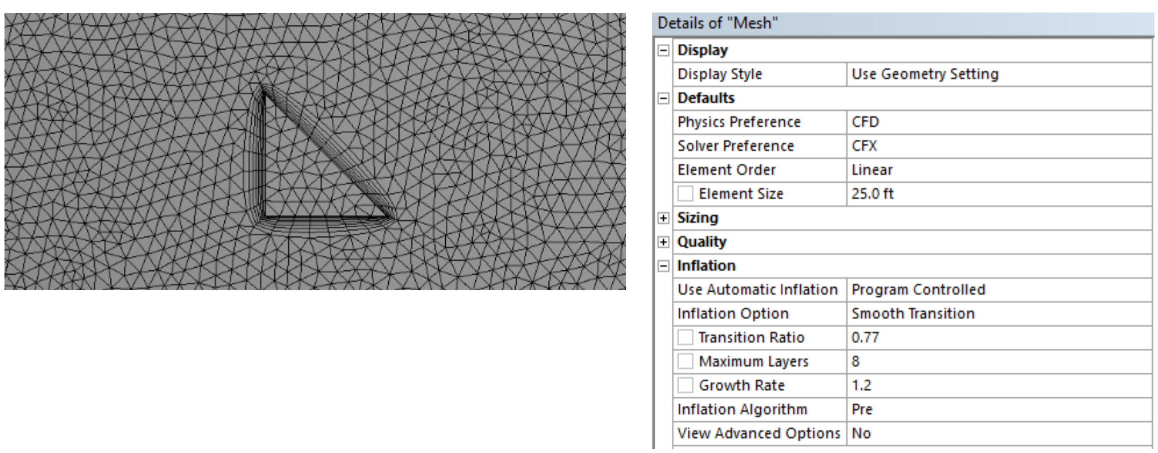


Figure 12: Second Meshing Technique with inflation layers in the vicinity of the building (left). Meshing Details (Right).

5.3.5 Pre-Processing

Pre-processing stage was where the boundary conditions and initial states of the flow domain were specified. First of all, the fluid speed (wind velocity) of the inlet, in this case 45 m/s (100mph) was assigned. The same wind speed was also specified in “Global Initialization” setup so that the wind flow would be directed parallel to the x-y plane rather than flowing in random directions

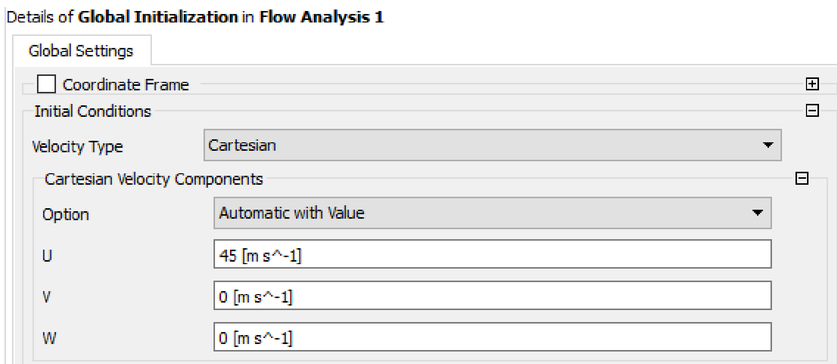


Figure 13: Wind Velocity setup in Global Initialization

The relative pressure of the outlet was also assigned as 1 atm which is also the Atmospheric Pressure. The value of the relative pressure can influence the outcome of the results. During the previous attempts, the outlet relative pressure was specified as 0 Pa. This value can be applicable for testing the peak pressures of the moving vehicles or rotating turbines and other mechanical applications. However, in the case of wind-building interactions where the stationary building is exerted by external fluid pressure, assigning relative pressure to 1 atm would be a better alternative for post-processing stage [CFD Online, 2007].

Both wall and building were assigned and assumed as no-slip wall with smooth surface. Assigning named selections before preprocessing stage was a faster and efficient way to specify boundary conditions since ANSYS application would automatically assigned to respective selections.

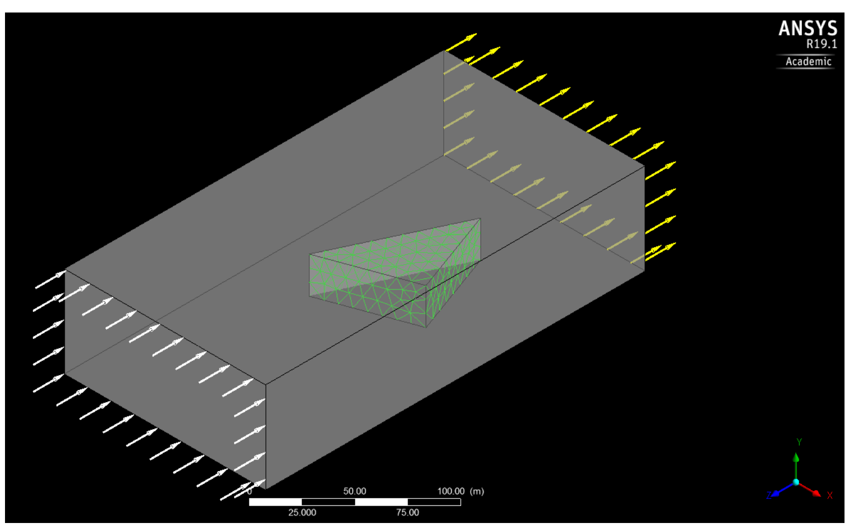


Figure 14: Pre-Processing Stage of Rigid Pressure Model

5.3.6 Post- Processing

ANSYS CFX is capable of measuring the different parameters and has different mode of conveying the results. Since the peak pressure exerting on the building walls was the required parameter, the pressure contour was created on the building as in Figure 15. The peak pressure was measured either from the color map or from the data value at the left of the screen.

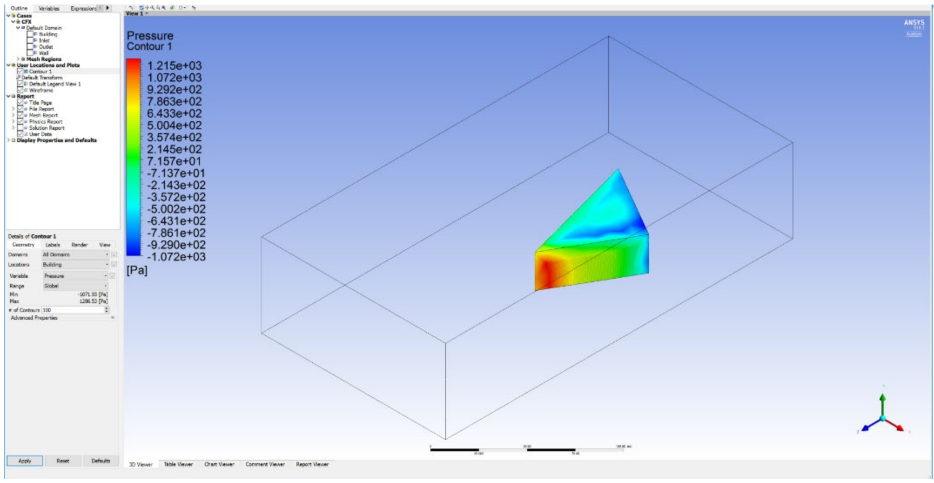
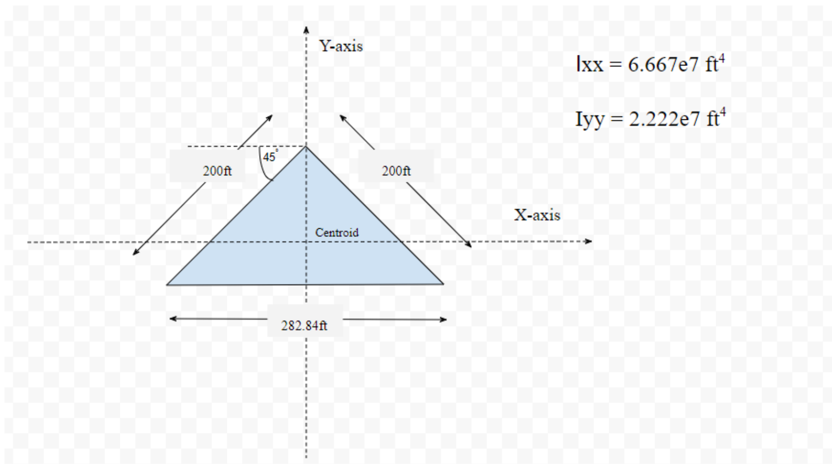
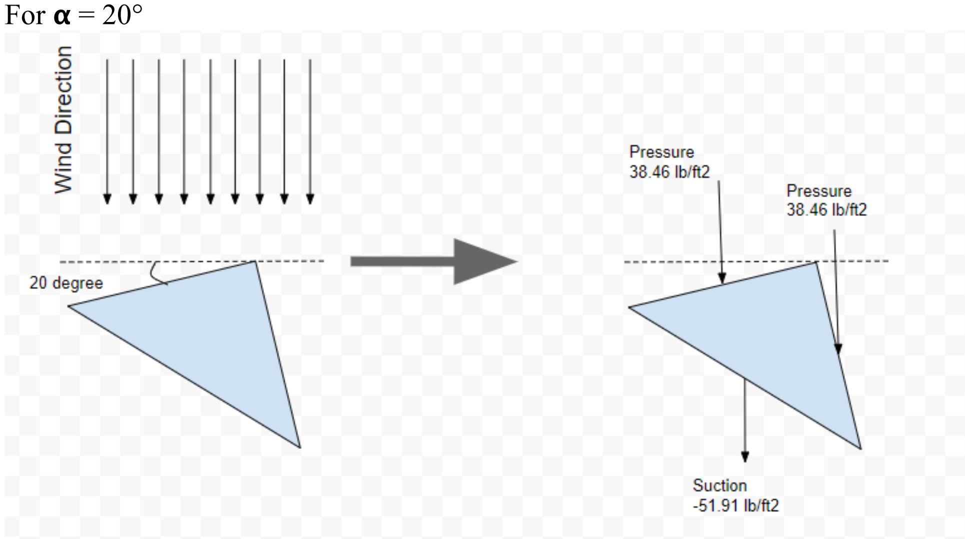


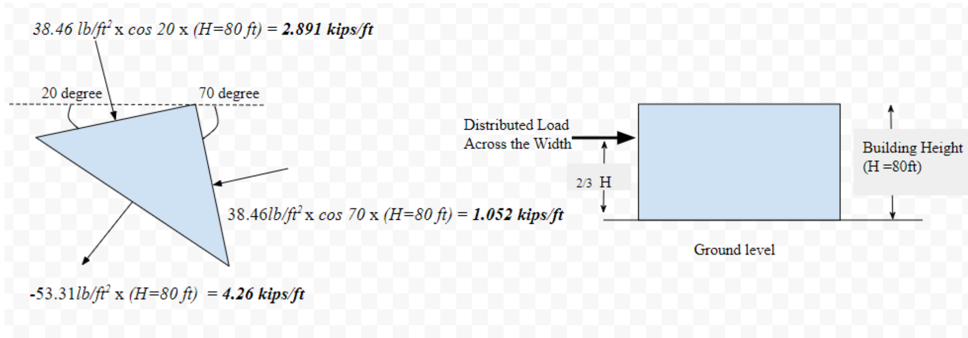
Figure 15: Typical Post-Processing Result

5.3.7 Peak Pressures Using Principal Moment of Inertia

This section describes the detailed procedures for determining the governing wind direction by calculating the peak stresses of the building using the Principal Moment of Inertia. Table 3 illustrates the process along with a descriptive example calculation.

Table 3: Summary of Procedure for Calculating Peak Stresses of the Building

Procedures	Example
1.Determine Principal Moment of Inertia and its axes (Ixx and Iyy)	
2.Apply Peak Pressures from Rigid Pressure Model Test (Appendix C)	For **α** = 20° 

3.Convert Peak Pressures(lb/ft^2) into Distributed linear load (kip/ft)	
4.Calculate the moment in X and Y axes (consider building as a cantilever beam)	Moment in X axis = (*2.891 kips/ft -1.052kips/ft*) x *cos 45* x ⅔(*H*=80ft) x *exposed width (table C1)* = 13,870.6 kips-ft Moment in Y axis = *4.26 kips/ft* x *cos 45* x *⅔ (H=80 ft)* x *exposed width (table C1)* = 24,245 kips-ft
5. Find the coordinates of the triangular building (centroid being the origin)	Coordinates of the triangular building is created in Excel Sheet (x,y) Using the equation $(x,y) = -(M_x\ y / I_{xx}) + (M_y\ x / I_{yy})$, stresses on the outline of the triangular building was determined
6.Calculate the peak stresses of the building	The maximum stress recorded for each **α** configurations (Appendix C,Table C3)

The peak maximum and minimum stresses were measured and recorded for every α values. Figure 16 shows the graph of varying peak stresses with respect to the wind direction (α).The α value with the largest peak stress (α = 45 degree) was chosen to be the governing wind direction for the structure (referred to Figure 16). This wind direction was used in the member sizing stage using RISA 3D which will be further discussed in Chapter 6.

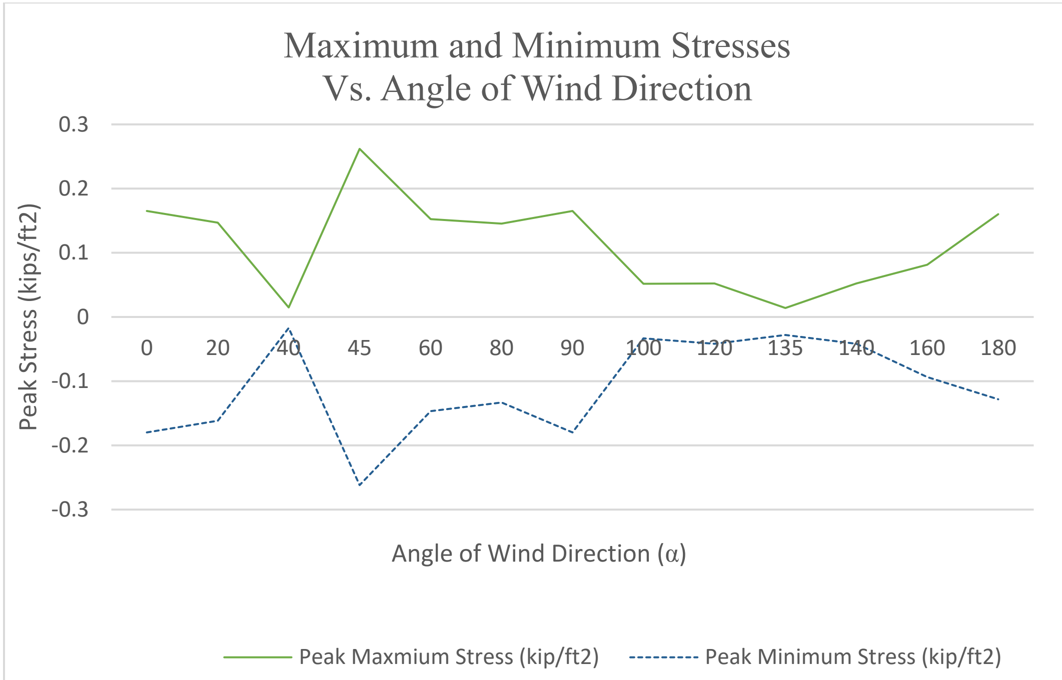


Figure 16: Maximum and Minimum Peak Stresses of the building for different α configurations

5.4 Time-Domain Analysis

Time Domain Analysis is used to measure the dynamic effects of the structure due to time-varying loads. This technique of measuring wind loads used different types of ANSYS solvers, such as (1) static structural, (2) modal analysis, and (3) time-transient structural. This section will discuss methods and some of the significant procedures that were utilized for Time-Domain Analysis.

5.4.1 ANSYS 19.1 Set-up

Creating a building model in ANSYS was a completely different case compared to Rigid Pressure Model. Unlike Rigid Pressure Model of which its solution depends only on the geometry of the building, the solution of Time-Domain Analysis depends on the sizes of each of the structural members and the modal shapes of the building.

One of the first steps of creating ANSYS model is to upload a coordinate file in "txt." format. The x, y, and z coordinates are written in a format shown in Figure 17. Line members were drawn according to the nodes coordinates, and respective I-sections of the members from Rigid Pressure Model test were extruded on the line members. During this stage, it was found that some of the I-beams had incorrect alignments. Those section members were individually chosen, rotated and aligned. Different types of members (columns, exterior beams, girders and beams) were grouped using 'named selection' function mentioned in Section 5.3.5. This saved considerable amount of time in applying gravity and wind loadings to the members.

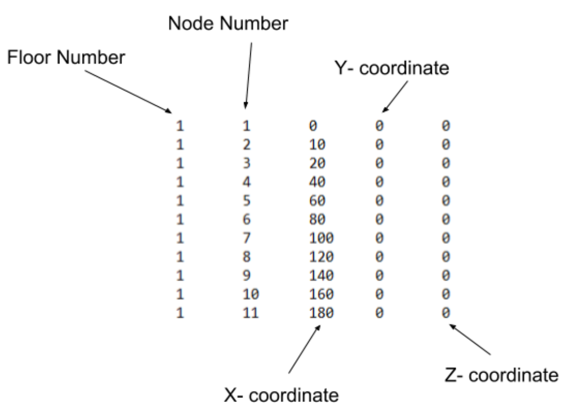


Figure 17: Example of txt. File format which ANSYS accepts as coordinate file

Since multiple solvers were used within a single workflow of the ANSYS, certain relations or connections between the solvers were established in a way that a solution of a particular solver can be transferred and used in subsequent solver. The summary of the ANSYS workflow for Time Domain Analysis can be seen in Figure 18. This workflow can be divided into two parts. The first part (upper flow, Figure 18) served as combined solver to measure the dynamic effects (deformations and principal maximum stress) while the second part solved the static effects of the building due to gravity and steady wind load. The results of two parts were then compared to determine if which solution governed (Appendix F).

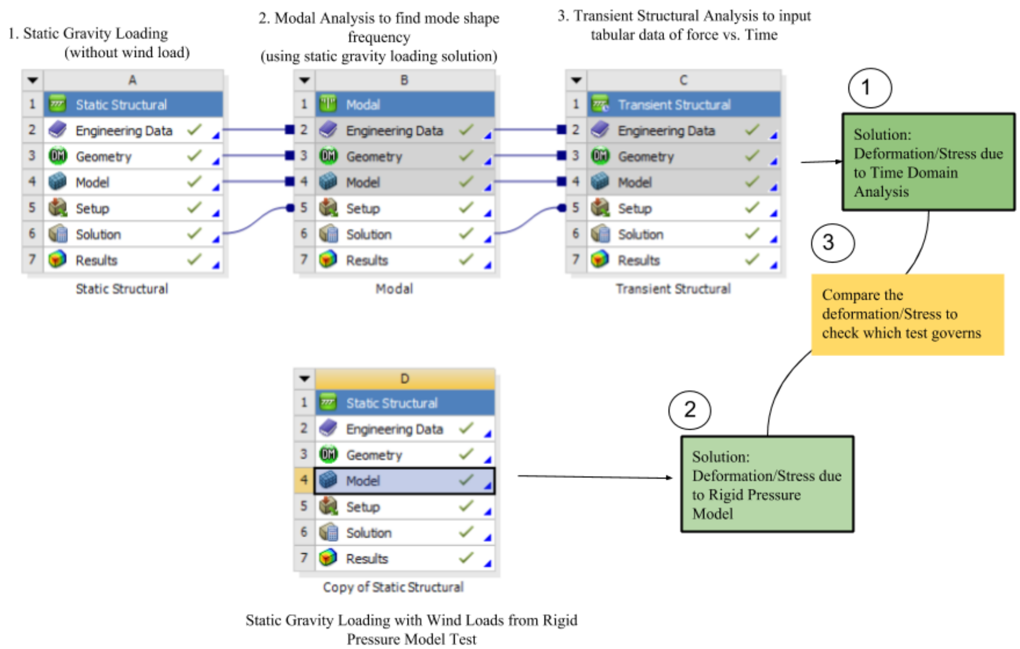


Figure 18: ANSYS Workflow Diagram for Time Domain Analysis

5.4.2 Modal Analysis

Mode shapes of the building and the frequency at which the respective mode shapes occur can be defined by running the modal analysis. Since the set-up of modal analysis was influenced by the solution of the static structural analysis, modal analysis incorporated the gravity loadings of the static structural analysis into the solution of the modal analysis. Therefore the modal shapes of the building were in a condition where the members were pre-stressed by the gravity loads. For this project, six mode shapes and their frequencies were specified and they are presented in the table below. Mode 1, 2 and 4 were associated with the sliding and bending while the rest of the mode shape were associated with twisting. *ASCE 7-16* specified that approximate natural frequency (n_a) for structural steel with lateral resisting system can be calculated using equation "$n_a = 75/h$" from *ASCE 7-16: Section 26.11.3* , where h is equal to mean height of the building (in this case, 40ft). The calculated natural frequency was around 1.875 Hz which is roughly equal to mode 1 from Table 4.

Table 4: Modal Analysis Result

Mode	Frequency (Hz)
1	1.7666
2	2.9035
3	3.9698
4	4.1875
5	4.3931
6	4.7597

5.4.3 Time History Function of Wind Speed

In order to know the structure has a dynamic effect or not, a time history function of wind speed had to be applied on a structure. By considering the mode shapes of the structure, time transient structural analysis was performed to gather the stress and deformation values due to the varying wind loads.

For this project, online wind simulator developed by Nature Hazard (NatHaz) Department of University of Notre Dame was used [Kareem, 2001]. After entering a several input (Table 5), the simulator delivered either Matlab.MAT format or Excel.CSV format of the dataset. The data set includes (1) mean wind speed of the simulation and (2) fluctuating wind speed with respect to time. The NatHaz website also recommends the following Equation 1-1 to convert to fluctuating wind speed into wind forces which can be applied for analysis.

Cut-off frequency was one of the inputs for the wind simulator and the input value range from 1Hz to 5 Hz with discrete values. It was not known that how different cut-off frequencies would influence the results of Time-Domain Analysis. Therefore, each cut-off frequency was simulated and the results were compared in section 5.5, Table 6. After observing the simulated wind graphs (Appendix E), the time range produced by the wind simulator was dependent on the cut-off frequency value in a way that "Time Range simulated (s) = Number of Time steps / cut-off frequency (Hz)". For instance, wind graph simulated with cut-off frequency of 2 Hz has the time range of only 2.5s while cut-off frequency of 1 Hz simulates the time range of 5s.

Table 5: Required Input and their values for NatHaz Wind Simulator

Required Input	Input Value
Unit	English (ft, mph)
Z-coordinate of the point of interest (height)	2/3 of the building height = 2/3 (80ft)
Number of Time Steps	5
Cut-off Frequency	1Hz – 5Hz
Exposure Category of Building (ASCE 7-16)	B
3s gust Wind Speed (ASCE 7-16)	100mph

$$F(t) = \rho \cdot C_D \cdot A \cdot U \cdot u(t) \quad \text{.................................} \quad \text{(Eq 1-1)}$$

Where, $F(t)$ = fluctuating wind force (lbf);
ρ = air density (lbm/ft^3);
C_D = drag coefficient;
A = tributary area (ft^2):
U = mean wind speed (ft/s);
$u(t)$ = fluctuating wind speed (ft/s) (*note: lbf = lbm* x *ft /s^2)*

5.4.4 Post- Processing

Post-Processing of Time-Domain Analysis was to obtain (1) total deformation and (2) principal maximum stress of the structure due to the fluctuating wind force with respect the time. ANSYS automatically provides the total deformation in the post-processing stage. However, to derive the principal maximum stress in the post-processing, one has to specify it in the set-up stage.

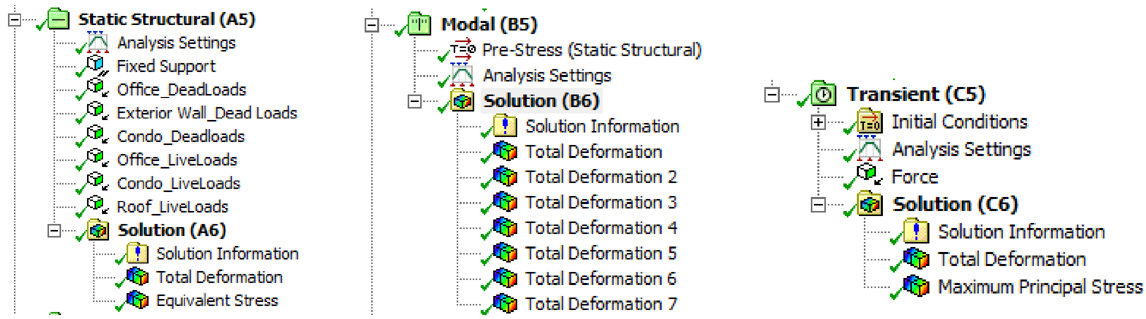


Figure 19: Post Processing Stage of Time-Domain Test

5.5. Summary of Wind-Tunnel Test

This section discuss the summary results of the two wind-tunnel tests that were performed during this project. Rigid Pressure Model provided with the governing wind direction (α) and its wind forces. On the other hand, Time-Domain Analysis incorporated the mode shapes of the statically pre-stressed building and offered the dynamic response of the building under time-varying wind forces from NatHaz Wind Simulator.

Before advancing into member sizing stage, it is necessary to determine which wind-tunnel test governs the design stage of the project. The work flow mentioned in Figure 18 was used to determine which test governed. By comparing the total deformation and principal maximum stress resulted from each test, static wind loads governed design process. Table 6 shows that comparing the results of different cut-off frequencies, the value of 1 Hz and 2Hz had the more significant deformation and stress compared to the rest. However, Time-Domain analysis result did not exceed the results of Static Analysis.

Table 6: Tabular Result of Static Analysis vs. Time Domain Analysis Showing Maximum Deformation (in) and Principal Maximum Stress (ksi)

Cut-Off Frequency (Hz)	***Principal Maximum Stress (ksi)***		***Total Deformation (in)***	
	Static Analysis	**Time-Domain Analysis**	**Static Analysis**	**Time-Domain Analysis**
1Hz	11.179 ksi	1.5577 ksi	0.654 in	0.1694 in
2Hz	11.179 ksi	8.7784 ksi	0.654 in	0.6379 in
3Hz	11.179 ksi	7.9556 ksi	0.654 in	0.1505 in
4Hz	11.179 ksi	3.1872 ksi	0.654 in	0.2001 in
5Hz	11.179 ksi	6.6581ksi	0.654 in	0.4282 in

6. Structural Analysis and Member Sizing

This section explains the structural analysis procedures using the RISA 3-D and how member sizes were calculated. Different iterations of creating a RISA model are mentioned. The justification of choosing load cases and load combinations are also included.

6.1 Structural Model

When building a five-story, 3-D model using RISA 3D, there were several trials and errors. In order to be less time-consuming and more efficient with the RISA model building, the naming of structural members and nodes became the most essential according to project experience. However, those techniques can vary according to the building geometry and layout. The naming system explained in the following table can only be applied to the proposed building. This naming system relied on the types of structural member and the floor number in which the member is located. Exterior beams is mentioned in the Table 7 since it significantly saved time when applying exterior wall loads and wind loads onto the exterior beams.

Table 7: Naming System of Nodes and Structural Members in RISA 3D

Structural Members	Name Label
Nodes	"Floor number- 1, 2, 3…etc."
Beam Members	"B-floor number- 1, 2, 3…etc."
Girder Members	"G-floor number-1, 2, 3…etc."
Exterior Beams	"E-B-floor number-1, 2, 3…etc."
Columns	"C-floor number-1, 2, 3…etc."

Specifying the connection constraints was challenging and time-consuming when creating a building with over 3000 structural members. After several tries, it was found that it saved a lot of time if the reaction constraints were specified at the same time as creating structural members. This also relied significantly on the naming system of the nodes and members.

6.2 Gravity Loading

Gravity load specifications were already mentioned in section 4.2 and 4.3. Appendix B lists the values of the different types of distributed load for individual beams of different floors.

6.3 Wind Loading

The values of the lateral wind loads were obtained from Rigid Pressure Model Test. After considering the concept of principal moment of inertia, it was found that α =45 degree had the governing peak stress. Therefore, wind direction of α =45 degree was used to apply as distributed wind load across each floor. Since the direction of the wind did not align with either global or local coordinate system created in RISA 3D model, wind loads were divided into x and y components. One of the crucial aspects of applying wind loads in RISA 3D was that diaphragm planes had to be specified in order for the loads to transfer into lateral resisting members and then eventually to columns.

6.4 Final Design Member Sizes

After inserting gravity and wind loadings on the building, structural analysis was performed and member sizes were calculated according the load combinations referred in *ASCE 7-16.* RISA 3D optimizes the member sizes by assigning the lightest possible section for individual members. However, considering the constructability aspect of the project, members sizes were repeated according to different types of structural components provided in Table 8. Due to the exterior wall loadings and wind loadings, member forces in the exterior beams were larger than the interior members. In the RISA model, 1st floor beams and girders were not included nor calculated by RISA. Since 1st floor concrete slab on grad was supported by the backfill soil, the member sizes of the 1st floor would be lighter and smaller compared to those of the upper floors. Therefore, same sections used in roof level were used in the first floor to increase the constructability of the overall building.

Table 8: Final Member Sizes from RISA 3D

Floor Number	Exterior Beams	Interior Beams	Girder	Columns	Brace Members
1	W8x24	W8x24	W12x30	W12x65	W8x31
2	W14x43	W10x33	W12x30	W12x45	W8x31
3	W14x43	W10x33	W12x30	W10x39	W8x31
4	W14x43	W10x33	W12x30	W10x39	W8x31
5	W14x43	W10x33	W12x30	W8x28	W8x31
Roof	W8x24	W8x24	W12x30	*N/A*	*N/A*

7. Foundation Design

Simple square column footings were used in this building. Foundation plan view and elevation view of the foundation are provided in the following figures (20, 21). RISA 3D provided the column reaction values. The bearing capacity of the soil is assumed as 5kips/ft^2. The final foundation design was reinforced with 12 No.7 rebar with 9.5" spacing. Detailed calculation of foundation design can be found in Appendix I.

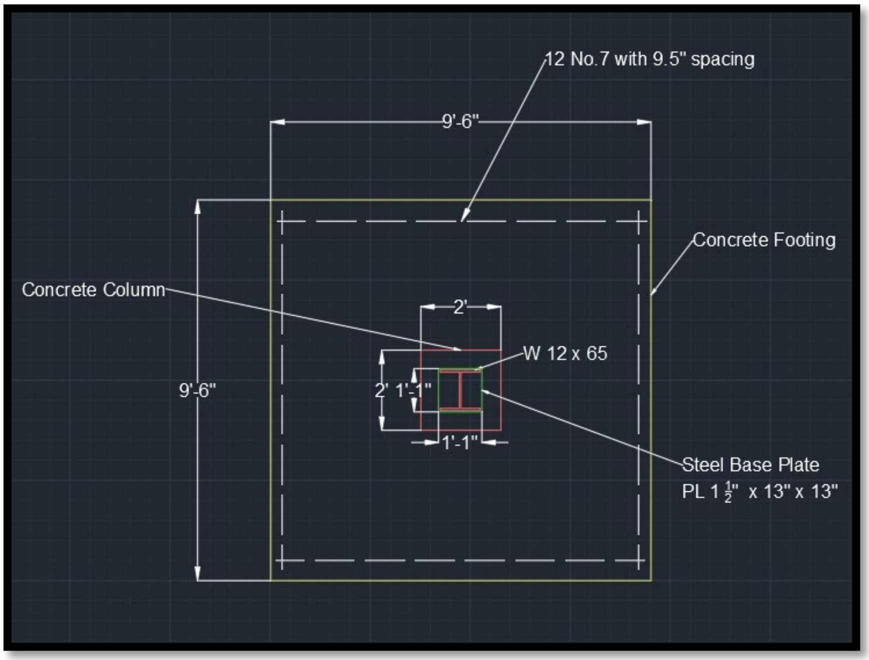


Figure 20: Plan view Detailed Drawing of Foundation and Steel Column Base Plate

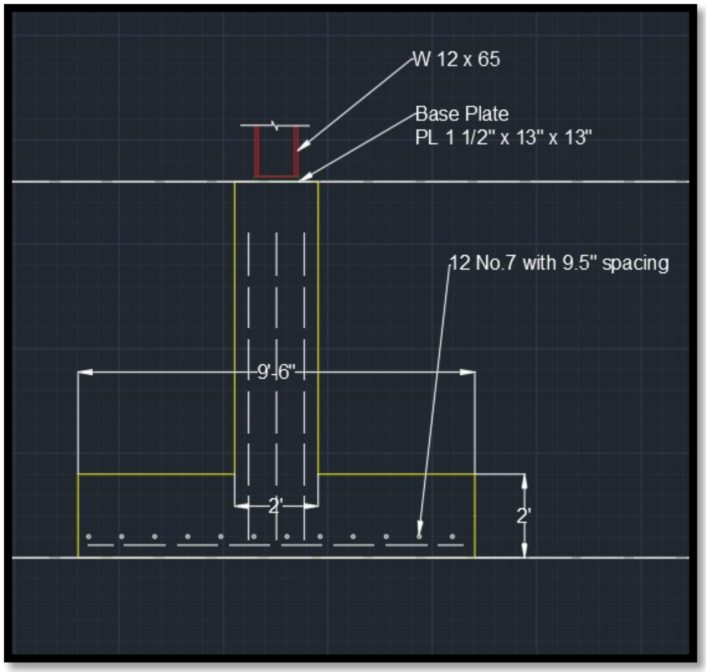


Figure 21: Elevation View of the Foundation

8. Connection Design

There are two connection types used throughout the building: (1) beam-to-girder, (2) girder/beam-to- column. All-bolted double-angle, simple shear connections were used. All the connections were designed using ASTM A325-N and ASTM A36 angles. Detailed drawings of each connection type are provided in the following figure. Necessary member forces such as (1) beam and girder end shear reactions, (2) column base reactions were obtained from RISA 3D analysis and were also mentioned in Appendix H. The maximum reaction forces was measured from RISA 3D analysis. 2L 6" x 3 ½" x ½" with 3 rows of ¾" diameter bolts were used as shear connection. Moreover, step-by-step calculations of the connection designs can be found in Appendix J.

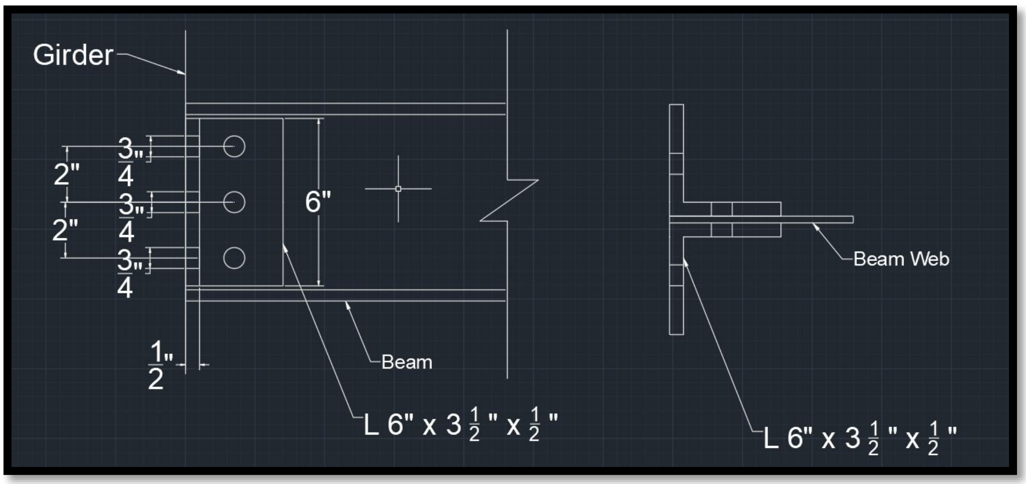


Figure 22: Side View and Overview of the Connection Details

9. Cost Estimate

This section describes the method used to determine the costs of the overall building including both material costs and installation costs using *RS Means Square Foot Costs 2015*. The final cost of the building was multiplied with time-value index to ensure that the cost reflects 2019 estimate. Although the historical cost indexes does not provide with future years estimate increase in cost, the estimate increase in building cost in 2018 was calculated by looking at the trends of historical cost data (Table 9). In addition to the structural costs, the final cost estimate included the electrical, mechanical and other types of non-structural components. *RS Means Square Foot Costs* provide examples of different types and uses of buildings ranging from schools, and condominiums to gas stations. Each example includes detailed costs per square foot of every aspects of the building. The book also provides the total cost (material+ installations) per unit of various types of components.

In order to calculate the cost estimate of the building, excel sheet was created which can be found in Appendix I. The final cost were then adjusted for San Francisco Area by multiplying with Location factor and Time Value Index. Since the proposed project mainly focused on the structural components of the building, the accuracy of the structural member cost would be higher compared to that of non-structural elements such as, interiors and mechanical components. The cost breakdown of the building according to category is shown as in Figure 22.

The final weight of the superstructure components (Beams, Girders, and Columns) of the building were around 440.2 tons. The estimate material and installation fees for the superstructure is calculated using the vertical linear height for columns, and floor square foot area for Girders and Columns. The material and installation of the superstructure is roughly 4.2 million, around 24.15% of the total cost of the building without additional fees (contractors' and architects' fees)

Table 9: Average Percentage Increase in Cost using Historical Data (San Francisco Area)

Year	**% increase in cost(compared to previous year)**
2012	4.26%
2013	1.43%
2014	2.90%
2015	2.42%
Average Increase in %	**2.75%**

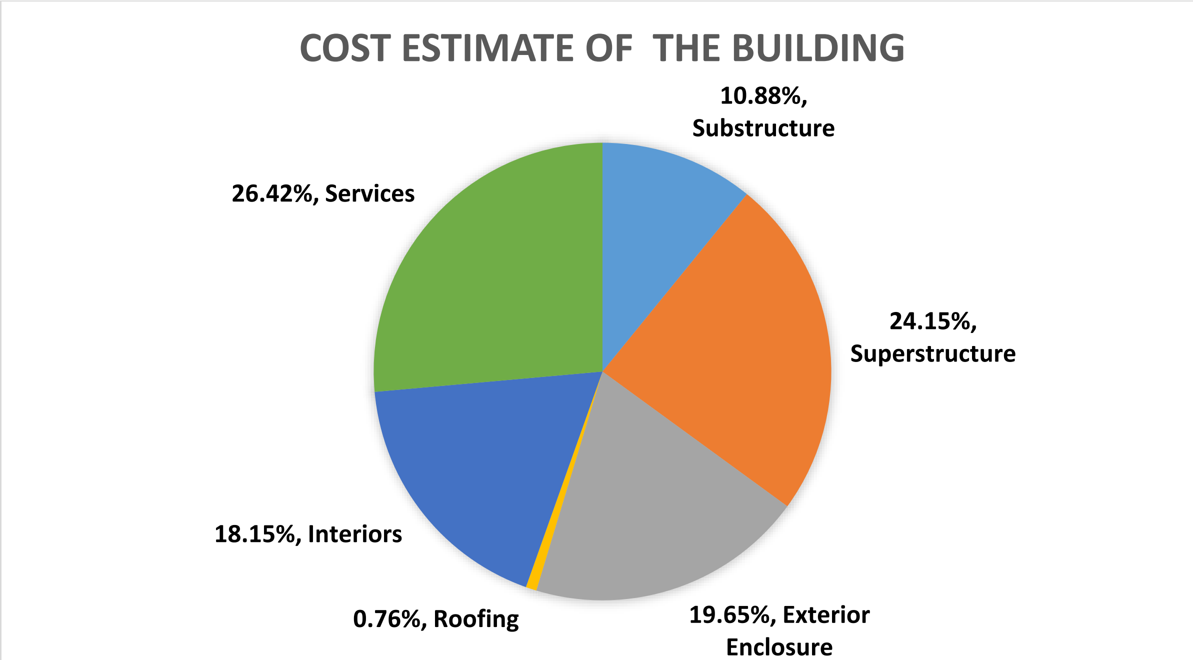


Figure 23: Pie-chart of the cost estimate of the building according to categories

Table 10: Cost Estimate of Building Materials and Installations

Building Component Category	Percentage of total cost	Cost per square foot	Total Estimate Cost
Substructure	10.88%	$ 19.06	$ 1,906,000
Superstructure	24.15%	$ 42.32	$ 4,232,000
Exterior Enclosure	19.65%	$ 34.43	$3,443,000
Roofing	0.76%	$ 1.332	$133,200
Interiors	18.15%	$ 31.804	$3,180,400
Services	26.42%	$ 46.294	$4,629,400
		Estimate Sub-Total	$ 17,824,000

Table 11: Cost Estimate of Anticipated Additional Fees

Addition Fees	Amount
Sub-Total x Location Factor (1.25 for San Francisco)	$22,280,000
2018 Time Value Index[1] (1 .0275^3)	$24,169,111
Contractor's Fees (25% of sub-total)	$6,042,277
Architect's Fee (8% of sub-total)	$ 1,933,529
Total Building Cost	$32,144,917

[1] Time value Index of present year = 1.0275$^{(\text{present year -2015})}$

10. Conclusion and Recommendations

10.1 Conclusion

As a capstone design project, the final aim is to deliver the structural design with sound resistance to gravity loading and wind loading specified by ASCE specifications. During this project, the interactions between wind and structure were intensively studied performing two wind-tunnel tests: (1) Rigid Pressure Model Test and (2) Time-Domain Analysis Test. Unlike conventional wind-tunnel tests, design and simulation softwares such as ANSYS 19.1 and RISA 3D were heavily relied for creating computational models. By incorporating the result of the governing wind-tunnel test, structural members and connections were designed in accordance to the *ASCE*, *ACI* specifications. Constructability was also considered in choosing the member sizes. Cost estimate of both structural components and non-structural components were calculated using RS Means Square Foot Costs.

10.2 Recommendations

The primary purpose of the project was to explore and delve into the subject of wind engineering, which involves multiple disciplinary such as, meteorology, fluid mechanics, structural dynamics and probability and statistics [Holmes, 2001]. Finite element software and structural analysis software were practiced to a certain level of depth to deliver a final product. However, it is learned that there were several aspects of the project that needed improvement and aspects that should be done differently in similar future projects.

One of the first recommendations would be scheduling of the project. Although two wind-tunnel tests were undergone during the project, more significant amount of time was spent on the first wind-tunnel test (Rigid Pressure Model) which left the second wind-tunnel test only a few weeks to accomplish. This was primarily due to software unfamiliarity as well as several unforeseeable problems which took up significant amount of time to solve.

Second recommendation would be to improve the understanding of the subject and software in the early stage. It would be better to give considerable amount of time to understand and familiarize yourself with software and subjects of interest. This will result in faster executions and more reliable results. Understanding the subject in depth will also be helpful in solving unforeseeable problems mentioned in first recommendations.

Last recommendation would be to begin with a simpler model for the design process and focus more on the basic concepts of the topics. This would help ease the learning curves during the early stage. Once one get familiar with the topics and software, more advanced and complex models can be designed.

References

1. Abdusemed, Muftha Ahmed, and Ashok Kumar Ahuja. “Wind Pressure Distribution on Triangular Shape Tall Buildings.” *International Journal of Innovative Research in Science, Engineering and Technology*, vol. 4, no. 8, Aug. 2015.

2. *ACI Manual of Concrete Practice 2017*. American Concrete Institute, 2017.

3. *ANSYS® Academic Research Mechanical, Release 19.1*

4. *ASCE/SEI 7-16 Minimum Design Loads and Associated Criteria for Buildings and Other Structures*. American Society of Civil Engineers, 2017.

5. *ASCE 49-12 Wind Tunnel Testing for Buildings and Other Structures,* American Society of Civil Engineers, 2012.

6. Britannica, The Editors of Encyclopaedia. “Boundary Layer.” *Encyclopædia Britannica*, Encyclopædia Britannica, Inc., 20 July 2018, www.britannica.com/science/boundary-layer.

7. Eduran, Emrah. “Response Spectrum Analysis: Theory, Benefits and Limitations.” njtf.no/wp-content/uploads/2016/10/Technical-review-of-Modal-Analysis.pdf.

8. Gargallo-Peiro, A, et al. “Mesh Generation for Atmospheric Boundary Layer Simulation in Wind Farm Design and Management.” *NeuroImage*, Academic Press, 3 Nov. 2015,
9. www.sciencedirect.com/science/article/pii/S1877705815032373?via=ihub.

10. “Generating Inflation Layers for CFD - ANSYS e-Learning.” *Speaking the Engineering Language | CAE Associates*, CAE Associates, caeai.com/resources/generating-inflation-layers-cfd-ansys-e-learning.

11. Ghanadi, Farzin, et al. “Investigation of the Atmospheric Boundary Layer Characteristics on Gust Factor for the Calculation of Wind Load.” *Stability of Plane Couette Flow of Carreau Fluids Past a Deformable Solid at Arbitrary Reynolds Numbers: Physics of Fluids: Vol 30, No 7*, AIP Publishing LLC, 27 June 2017, aip.scitation.org/doi/10.1063/1.4984496.

12. Harish, Ajay. “Why the Tacoma Narrows Bridge Collapsed: An Engineering Analysis.” *SimScale*, SimScale, 15 Jan. 2019, www.simscale.com/blog/2018/07/tacoma-narrows-bridge-collapse/.

13. Holmes, John D. *Wind Loading of Structures*. Spon, 2003.

14. Kareem, Ahsan. “NatHaz Wind Simulator.” *NatHaz On-Line Wind Simulator*, 2015, windsim.ce.nd.edu/int_winsim.html.

15. Lee, Huei-Huang. *Finite Element Simulations with ANSYS Workbench 19: Theory, Applications, Case Studies*. SDC Publications, 2018.

16. Lindsey, Jon. *Stresses in Exterior Connection with Full-Depth Column Web Stiffeners.* 2017, www.lusas.com/case/civil/connection_research.html.

17. Lohner, Rainald. *Applied CFD Techniques: An Introduction Based on Finite Element Methods*. 2nd ed., Wiley, 2009.

18. Mahendran, Mahen. "International Conference on Computer Aided Engineering." 11 Mar. 2008.

19. Nguyen, Nha, and Gaston Leblanc. "Contact Personnel, Physical Environment and the Perceived Corporate Image of Intangible Services by New Clients." *International Journal of Service Industry Management*, vol. 13, no. 3, 2002, pp. 242–262.

20. Roensch, Steve. "Finite Element Analysis: Post-Processing." *Finite Element*, 2008, www.finiteelement.com/feawhite4.html.

21. *RISA 3D v17.0.2, 2018*, A Nemetschek Company

22. "Setting Pressure Boundary Conditions in ANSYS CFX Pre." *Www.cfd-Online.com*, www.cfd-online.com/Forums/cfx/81634-setting-pressure-boundary-conditions-ansys-cfx-pre.html.

23. *Solidworks Academic 2018,* Dassault Systems, France

24. Spaulding, Erin. "IBC 2018/ASCE 7-16: DCI ENGINEERS PREPARES FOR BIG CHANGES." *DCI Engineers*, 18 May 2018, www.dci-engineers.com/news/ibc-2018asce-7-16-dci-engineers-prepares-big-changes.

25. *Steel Construction Manual Fifteenth Edition*. American Institute of Steel Construction, 2017.

26. "SUCTION EFFECTS." *THERMOPEDIA™*, 2011, www.thermopedia.com/content/1166/.

27. *The Severity of an Earthquake*. U.S. Geological Survey, pubs.usgs.gov/gip/earthq4/severitygip.html.

28. Weck, Olivier de, and Il Yong Kim. "Finite Element Method." Engineering Design and Rapid Prototyping. web.mit.edu/16.810/www/16.810_L4_CAE.pdf.

29. Wertel, Scott. *FEA Buyer's Guide for Pre- and Post-processing Software*. 27 Sept. 2013, www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/6391/FEA-Buyers-Guide-for-Pre-and-Postprocessing-Software.aspx.

Appendix

Appendix A: Proposal

Design and Finite Element Analysis of a Triangular Shaped Building Structure

A Major Qualifying Project proposal submitted to the faculty of Worcester Polytechnic Institute in partial fulfillment of the requirements for the Degree of Bachelor of Science

Submitted by:

Jon Palczynski

Khant Win Htet

Submitted to:

Professor Leonard Albano

October 11, 2018

Abstract

The primary purpose of this project is to design the structure of a five-story triangular shaped mixed-use office and residential building in San Francisco, CA. The geometry of the building will be triangular shaped and during the design process, certain scopes of work will be intensely focused. Since the building will be approximately 80 feet tall and located in earthquake zones of San Francisco, the effects of lateral loads will be considered in the design process. Application of Finite Element Analysis software such as ANSYS and PATRAN is one of the goals of the project. Moreover, design evaluation and cost analysis will be undergone as the final scope of the project. Alterations of different structural elements (bracing systems, shear wall systems and column positions/spacings) will be tested out to find a cost-efficient and structurally sound building. The project also aims as a case study for buildings with irregular configurations.

Table of Contents

List of Figures

List of Tables

1. Introduction

Modern geometric architecture is becoming more popular; however, it presents new structural design challenges. New structural framing methods, materials and computer tools must be investigated and used to address the complex geometric structures and associated loading scenarios. Triangular shaped buildings are a type of modern geometric structure but due to structural design challenges they are not very common. Nonetheless, a few examples exist; the Potsdamer Platz 11 in Berlin, Germany and the Flatiron Building in New York, New York. Both buildings are considered iconic landmarks of the area and draw tourist attention due to their unique shapes. However, these buildings are not in areas subject to dangerous seismic and wind loads.

To set a standard for the design of a modern triangular shaped building subject to seismic and wind loads, this project will explore the design and evaluation of a theoretical five-story triangular shaped mixed-use residential and office building located in San Francisco, CA, primarily focused on the steel framing system. The project will investigate different structural framing methods and computer software to aid in the design of a complex building structure subject to seismic and wind loads. Additionally, a cost estimate will be performed for each framing method to better evaluate and assess the viability of each option. The major tasks of this project will be two structural frame designs, CAD Models, Finite Element Models, Finite Element Analysis and cost estimates. The final outcome of the project will be the completed evaluation and design of the triangular shaped structure along with guidelines to follow when designing a uniquely shaped structure subject to seismic and wind loads.

2. Background

This section provides the information needed to understand the major components of the project. The major components of the project are the building location, structural analysis and the utilization of computer software tools.

2.1 Building Site Location

The building site for the triangular shaped mixed-use residential and office building is located at 1811 Jerrold Ave San Francisco, CA 94124. It is a 60000 Sq. Ft. triangular shaped lot currently for sale for $8 million. This site was chosen due to its already level surface and large size. The general site location was chosen due to it being in an area subject to high seismic and wind loads.

Figure 1: 1811 Jerrold Ave San Francisco, CA 94124 Building Site

2.2 Structural Analysis

Design loads will be calculated from the *American Society of Civil Engineering Specification 7-16* (ASCE 7-16). This building site is in a critical location because the building design must address the significant lateral loads. Lateral loads are live loads that act like a horizontal force acting on the side of a building, consisting of seismic loads and wind loads. It is important to follow ASCE 7-16 as it contains new specification regarding wind and seismic loading criteria. Many locations have transferred to or have deadlines set to transfer from ASCE 7-10 to ASCE 7-16. ASCE 7-16 contains safer, updated methods but also stricter methods for Site Class D structures and tall structures. The seismic design loads for tall Site Class D structures will have an additional increase of up to 50%. ASCE 7-16 also removed the provision that allowed for an 18% reduction in seismic design loads for tall structures. Updated ground motion maps and soil coefficients have also contributed to the increase in seismic design loads (Spaulding, 2018). For our structure seismic loading is the controlling lateral load.

Seismic loads are caused by earthquakes. The building site location is classified as a Seismic Design Category D zone by ASCE 7-16. A Seismic Design Category is given to a structure based on its usage and level of possible ground motion. Sites classified as Seismic

Design Category D risk loss of soil strength. Structures classified as Seismic Design Category D could experience very strong shaking capable of producing a Modified Mercalli Intensity Scale (MMI) of VIII. The Modified Mercalli Intensity Scale has values assigned to a location after an earthquake has occurred; they are a measure of the viewable physical effects that the location experienced. An MMI of VIII is classified as damage slight in specially designed structures and considerable damage and partial collapse in ordinary buildings (U.S. Geological Survey, 1989).

Wind Loads are caused by air pressure acting on a building's surface. Unlike a rectangular building, a triangular building will not experience symmetrical wind distributions. The maximum and minimum pressures occur on a triangular building when the wind blows parallel to one of the faces. Due to the triangular shape the building will experience much higher twisting moments about its base when compared to a rectangular building (Abdusemed & Ahuja, 2015).

Dead loads are the frame's weight and any object that is permanently fixed to the building. Live loads are loads which can move and are not permanently fixed to the building. Live load design values are generally based on building occupancy classification and can be found in ASCE 7-16.

The structural design of a building can be divided into two parts; the superstructure and foundation. The superstructure design consists of the column layout and framing system. The column layout must account for usability and functionality of the spatial layout. The spatial layout is the floor plan and the arrangement of furniture, cubicles, counters, equipment and other items located within the floor plan (Nha & Leblanc, 2002). If columns are placed only to create the best structure with no consideration for the spatial layout, it may yield a completely unusable building. The superstructure design also consists of sizing the beams, columns and framing connections. The foundation design consists of sizing footings, the foundation wall dimensions and other components of the foundation. Foundation design follows the *American Concrete Institute Manual of Concrete Practice* (ACI Concrete Manual).

Beam and column sizes depend largely on the applied load combinations but also the frame layout. Modern structures typically contain large open spaces which increase the unbraced and effective lengths of beams and columns respectively; adversely affecting their strength. Connection variations include welded design, bolted design and coped (bolt) vs. not coped (weld) connections. Connections can be subject to pure shear, pure axial or a combination of both. Design of beams, columns and connections all follow *American Institute of Steel Construction: Steel Construction Manual* (AISC Steel Manual).

2.3 Finite Element Analysis

Finite Element Analysis will be used to analyze overall structure response and framing connection responses to seismic and wind loads present in San Francisco, CA. Finite Element Analysis works by dividing a complex structure into simple shaped elements connected by nodes. It can perform static analysis, modal analysis, transient dynamic analysis, buckling analysis and more. By simplifying a structure into simple shaped elements, a computer is able to solve the structure through large sets of simultaneous equations. Properties are calculated at the nodes and then interpolated over the element. Commonly used elements are 1D Beam Elements, 2D Plate Elements and 3D Solid Elements; a mix of elements can be used in a Finite Element Model. After the Finite Element Model is solved it can display items such as displacement, stress, strain, mode shapes and temperature. A common way to represent results is with a contour plot (Weck & Yong, 2004).

Modal Response Spectrum Analysis must be used to analyze the building structure due to the seismic loads. Modal Response Spectrum Analysis is a linear dynamic analysis method which measures the contribution from each natural mode of vibration to determine the maximum seismic response of an elastic structure. Because of the unique shape of the building and its location, a linear static analysis cannot be used. Once the analysis is complete, shear forces are computed to be distributed along the height of the building. (Emrah, 2016). The building will be designed from a seismic loading point of view and then will be checked for wind loading. Similar to the seismic design, because the structure is a unique shape, a normal wind load procedure cannot be used. A Computational Fluid Dynamics model must be made to analyze the wind loads acting on the triangular structure. Computational Fluid Dynamics generates fluid flow through a model of the structure and uses a numerical analysis method, such as the Finite Element Method, to find a solution (Lohner, 2009).

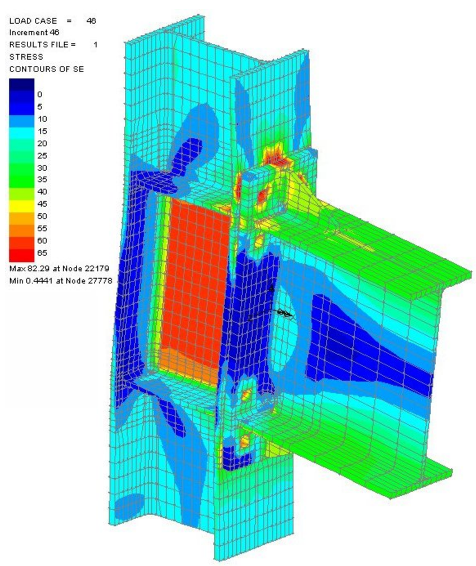


Figure 2: Finite Element Model with Stress Contour Plot (Lindsey, 2017)

Finite Element Analysis can be divided into three main parts; pre-processing, solving and post-processing. Pre-processing is the development of a Finite Element Model. This is where a structure is meshed into elements, material properties are assigned, loads and boundary conditions are applied, etc. After a Finite Element Model is created it needs to be solved. A text file is created with all components of the pre-processing phase; node locations, material properties, loads, etc. This text file is then sent to the solver. Solving is when the set of equations associated with the structure are put into matrix form and the variables are then solved for. The matrices can be extremely large and require a lot of computing power to be solved within a reasonable amount of time. After the solving is complete the user can view and analyze the results, known as post-processing. Post-processing is where results of stress, strain, displacement, etc. can be viewed as contour plots, graphs, etc. For example, solvers can calculate various types of stresses throughout a structure including; Mises, Tresca, Principal, Axial and Shear (Roensch, 2008).

There are a lot of commercially available Finite Element Analysis software packages; these consist of pre-processors, solvers, post-processors or "all in one" packages. A few examples include ABAQUS, HyperMesh, MSC NASTRAN, MSC PATRAN, ANSYS and STAAD. For this project the software will be limited to an educational version that does not have substantial model size limitations. When choosing a pre and post-processor it is important to consider the Finite Element Model creation capabilities, ability to import CAD geometry, solver support, user interface, results presentation, automation and scripting capabilities and overall value (Wertel, 2018).

From our initial investigation we have found that the ANSYS Workbench software package can create the overall structure and provide member forces through Modal Response Spectrum Analysis. ANSYS Workbench contains Design Modeler, a built-in CAD tool to create a framing system that can then be meshed with the built-in pre-processor. ANSYS Workbench also includes a built-in solver and post-processor. For connection design we have found it is generally easier to use CAD software for creating the model geometry rather than creating the geometry within a pre-processor. CATIA is a CAD tool that can create framing connections and then have the geometry directly inputted to MSC PATRAN for pre-processing. ANSYS, MSC NASTRAN, MSC PATRAN and CATIA V5 all have free educational versions available to students which make them the most viable options.

3. Methodology

This section outlines the major tasks of the project, how they will be completed and who will complete them. Each task includes the associated resources to assist in completion. The flowchart below lists the major steps.

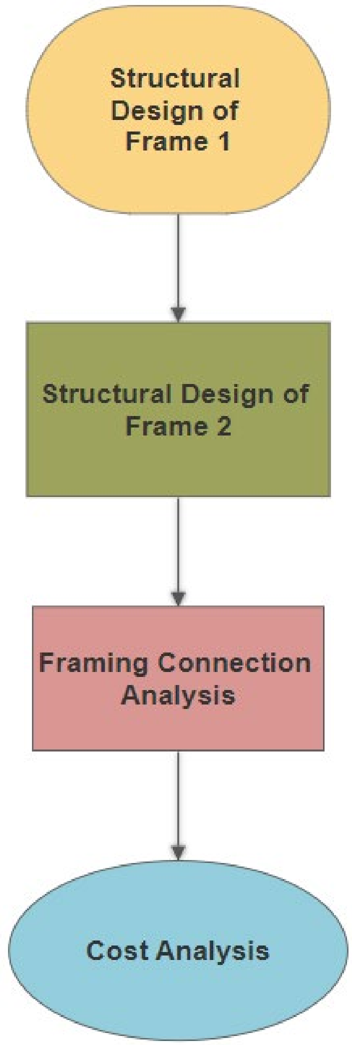


Figure 3: Methodology Flowchart

3.1 Structural Design

The structural design includes the superstructure and foundation design. Two frame designs (each of a different type) will be created and both will follow the same overall design procedure.

Table 1: Building Structure Design Tasks

Task	Resources	Team Member
Choose Framing System	ASCE 7-16	Jon P. & Khant H.
Set Column Grid	ASCE 7-16	Khant H.
Determine Design Loads	ASCE 7-16	Khant H.
Modal Response Spectrum Analysis	ASCE 7-16	Jon P.
Compute Member Sizes	AISC Steel Manual	Khant H.
Design Framing Connections	AISC Steel Manual	Jon P.
Design Foundation Elements	ACI Concrete Manual	Khant H.
Check Wind Loads	ASCE 7-16	Khant H.

3.1.1 Choose Framing System

Seismic resisting frame systems presented in ASCE 7-16 will be investigated and two will be chosen to develop into a full structure. Framing system types include steel eccentrically braced frames, steel special moment frames, steel special truss moment frames, etc. The following steps apply to both frame choices.

3.1.2 Set Column Grid

A column layout will be set to maximize the spatial layout but also provide for a safe structure. The column layout will be designed in AutoCAD.

3.1.3 Determine Design Loads

Design loads will be determined from ASCE 7-16 based on building occupancy type and other classifications. Design loads include gravity and lateral loads. Because the building is located in an earthquake zone, a separate seismic analysis will need to be performed for lateral loading. The tributary areas of each beam will need to be calculated to determine the distributed load to be applied along the beam from gravity load combinations.

3.1.3.1 Load Combinations

Governing load combination is calculated according to section 2.3.1 and 2.3.6 from ASCE 7-16. ASCE also states that wind and seismic loads need not to be considered to act simultaneously.

3.1.3.2 Gravity Load

Gravity loads that is applied for the building include dead load, live load, snow load and rain load. Uniformly distributed and concentrated live load is selected according to Table 4.3-1 from ASCE 7-16. Whether the live load reduction is permitted or not has to be further calculated as refers in Section 4.7.

To calculate the ground snow load, figure 7.2-1 from ASCE 7-16 is used as a reference. Ground snow load is then converted into flat roof snow loads from section 7.3-1 using different factor values such as exposure factor, thermal factor, importance factor.

Chapter 8 of ASCE 7-16 discuss design of rain loads. It says rain loads is based on the static head and hydraulic head values which are associated with the primary and secondary drainage system.

3.1.4 Modal Response Spectrum Analysis

Modal Response Spectrum Analysis will follow the general guidelines set forth by ASCE 7-16. First, a 3D Finite Element Model will be created in accordance with ASCE 7-16 using ANSYS. After the 3D Finite Element Model is created, Modal Response Spectrum Analysis will be performed. An adequate number of mode shapes will be found to have a total of 90% modal mass participation in each orthogonal direction. The base shear forces to be distributed along the height of the building will be combined with the design loads to compute the member forces utilizing the original 3D Finite Element Model.

3.1.5 Compute Member Sizes

After the member forces in the columns, beams and bracing members are found, the steel members will be sized. Excel sheets will be made to automate the procedure. All steel members will be designed in accordance with the AISC Steel Manual.

3.1.6 Design Framing Connections

Once the members are sized, the framing connections will be designed. Excel sheets will be made to automate the procedure. The connections will be designed in accordance with the AISC Steel Manual.

3.1.7 Design Foundation Elements

The pile footings and connections to the superstructure will be designed once the superstructure is complete. The concrete elements will be designed in accordance with the ACI Concrete Manual.

3.1.8 Check Wind Loads

After the building is designed for the seismic loads, it will be checked for the wind loads. A Computational Fluid Dynamics model will need to be created. This will be completed in ANSYS. Wind will be applied at various angles to see the effects on the triangular structure. The wind loads will be determined and the building will be checked to see if it is still compliant.

3.2 Framing Connection Analysis

Finite Element Analysis will be used to see how different types of framing connections respond to seismic loads. The connections will also be checked for fatigue and other conditions.

Table 2: Framing Connection Analysis Tasks

Task	Resource	Team Member
3D CAD Model Creation	CATIA V5 Workbook	Jon P.
Finite Element Analysis of Connections	Patran User's Guide	Jon P.

3.2.1 3D Model Creation

The framing connection geometry for different types of connections including the angles and bolts will be modeled in CATIA V5.

3.2.2 Finite Element Analysis of Connections

The CATIA V5 models will be directly imported to MSC PATRAN for meshing. MSC NASTRAN will be used as a solver. The results will be viewed in the MSC PATRAN post-processor. A comparison of the effects of seismic loading on different types of framing connections will be laid out.

3.3 Cost Estimate

Table 3: Cost Estimate Tasks

Task	Resource	Team Member
Cost Estimate	RS Means Construction Data	Jon P.

3.3.1 Cost Estimate

A cost estimate will be performed for each framing system using RS Means Construction Data. Additionally, full prices (non-student editions) of all software used will be found to give a perspective on how much it would cost for a design firm to utilize them all.

4. Capstone Design Statement

To comply with the Accreditation Board for Engineering and Technology (ABET), the Department of Civil and Environmental Engineering at WPI requires all Major Qualifying Projects (MQP) to include a Capstone Design Experience. The Capstone Design Experience requires students to design a system, component or process to meet desired needs by applying knowledge and skills acquired in earlier coursework and incorporating engineering standards and realistic design constraints. To fulfill this requirement, this MQP is the design and evaluation of a five-story triangular shaped mixed-use residential and office building located in San Francisco, CA. Alternate building designs will be investigated and proposed.

During the design process of the building, various effects of lateral loads due to wind and earthquakes will be investigated. Since the proposed property is situated in a location geologically vulnerable to earthquakes, seismic-resisting structures will be intensely studied and utilized. The design will follow standard engineering building codes; AISC Steel Manual, ACI Concrete Manual and ASCE 7-16. To evaluate the constructability and efficiency of the building, Finite Element Analysis will be undergone with different design scenarios. The main study of the project will explore how slight alteration of different design components, (beam/column sizes, floor-to-ceiling height, column positioning) will influence and improve the overall design of the structure. The final focus of the project will be reporting a cost analysis of the structural skeleton of the building. The MQP will incorporate economic, constructability, ethical, health and safety, and social design constraints.

4.1 Economic

A cost estimate of each structure will be completed and broken down into individual components like beams, columns and connections. Different beam and columns sizes, column arrangements and flooring systems will be considered to determine the most cost-effective design. The recommendation for the final structure will consider cost.

4.2 Constructability

The aspect of constructability will be highly prioritized during development of design scenarios. Efficient constructability includes using beam/column sizes and column spacings.

4.3 Ethical

During the design process, the code of ethics for engineers provided by National Society of Professional Engineers (NSPE) will be followed. The building is in an area subject to seismic loads and will be required to adhere to seismic design provisions in ASCE 7-16. While this may require additional design work and increased material costs, compromising the overall quality of the structure for cost-efficiency will be strictly avoided. Moreover, creating a design with structural integrity will be the top priority of this capstone project.

4.4 Health and Safety

Health and safety during the construction as well as post construction is one of the fundamental code of ethics by NSPE. All structural designs will be followed by standard building codes such as ASCE 7-16. Because the building will support office and residential spaces and is in a location subject to seismic loads the appropriate risk category will be assigned.

Assigning the appropriate risk category and following the design procedures associated with it will ensure the safety of occupants.

4.5 Social

Although this project is mainly aimed to design to structural skeleton of the building, the proposed site is planned to be office/residential building. The building will comprise of small and medium sized office spaces, small shop spaces and residential apartments for different types of social background. Column grids must account for the spatial layout to ensure adequate and functional offices and apartments can be implemented into the building.

5. Professional Licensure Statement

To design a product, either electronic device or living space, one of the utmost important factors is safety and health of the public. The engineers are required to train rigorously to a certain level where they are considered as competent to design or review a product. As civil/structural engineers, designing or constructing a building comes with a significant amount of risk and require years of academic knowledge and work experience.

To create a standard limit of becoming a credential engineer, the National Society of Professional Engineers (NSPE) specifies, in order to archive a PE (professional Engineer), one must: (1) finish a 4-year engineering program from accredited university/college, (2) pass the FE (Fundamental of Engineering) exam. (3) complete 4 years of work experience under a PE and (4) pass the PE exam. Once an engineer becomes a certified PE, it comes with a lot of authority as well as responsibility. Since PE engineer can design, review and approve a project, following the ethical guidelines would be the main responsibility.

This capstone design project is almost strictly related structural professional practice. It includes structural members sizing, structural analysis and detail drawings. The outcome of the project will be a product which need a professional engineers’ approval if the project ever come to reality.

6. Deliverables

The following deliverables will be included in addition to the final MQP report.

Table 4: Deliverables

ANSYS Building Model
CATIA Connection Models
CAD Drawings
Excel Calculation Sheets
Hand Calculations

7. Conclusion

As a capstone design project, the final aim is to not only utilize the theoretical knowledge that is gained from 4-years of civil engineering modules but also connect the different civil engineering aspects and consider the most efficient outcome with structural integrity. At the end of the project, it is also expected that the team will gain knowledge of different design processes, structural analysis applications, simulation software and design optimization processes. Moreover, the project is expected to extend the capacity of the team as structural engineers and also improve the problem-solving skills. Properly utilizing personal experience and academic knowledge as well as the guidance of the advisor will be one of the primary intent of the project.

8. Project Schedule

The schedule below outlines the process the project will follow. The two frame designs will be completed one after another to better structure the project outcome. After the two frame designs are complete, Finite Element Analysis will be used to investigate different connection combinations for each structure to see which performs the best under seismic loading. When preparing the final report a final recommendation will be made as to which is the best framing design and what are the best connection designs for an irregular shaped building subject to seismic loads.

Table 5: Project Schedule

Task	**Month**							
	Sep	**Oct**	**Nov**	**Dec**	**Jan**	**Feb**	**Mar**	**Apr**
Site Evaluation								
Frame Design 1								
Column Design 1								
Beam Design 1								
Connection Design 1								
Foundation Design 1								
Frame Design 2								
Beam Design 2								
Column Design 2								
Connection Design 2								
Foundation Design 2								
FEA of Connections								
Cost Analysis								
Proposal								
Final Report								
Project Presentation								

Appendix B: Gravity Loading

Table B1: Gravity loadings on the beams of first and second floor

	First Floor/ Second Floor										
Beam Number	x	z	y	x'	z'	y'	Tributary Area (ft2)	Length (ft)	Total Dead Load	Total Live Load	Load Combination(lb/ft)
A1	10	0	0	20	0	0	87.5	10	73	50	1466.5
A2	20	0	0	40	0	0	100	20	73	50	838
A3	40	0	0	60	0	0	100	20	73	50	838
A4	60	0	0	80	0	0	100	20	73	50	838
A5	80	0	0	100	0	0	100	20	73	50	838
A6	100	0	0	120	0	0	100	20	73	50	838
A7	120	0	0	140	0	0	100	20	73	50	838
A8	140	0	0	160	0	0	100	20	73	50	838
A9	160	0	0	180	0	0	100	20	73	50	838
A10	180	0	0	200	0	0	100	20	73	50	838
AA1	10	10	0	20	10	0	112.5	10	25	50	1237.5
AA2	20	10	0	40	10	0	200	20	25	50	1100
AA3	40	10	0	60	10	0	200	20	25	50	1100
AA4	60	10	0	80	10	0	200	20	25	50	1100
AA5	80	10	0	100	10	0	200	20	25	50	1100
AA6	100	10	0	120	10	0	200	20	25	50	1100
AA7	120	10	0	140	10	0	200	20	25	50	1100
AA8	140	10	0	160	10	0	200	20	25	50	1100
AA9	160	10	0	180	10	0	200	20	25	50	1100
AA10	180	10	0	200	10	0	200	20	25	50	1100
B2	20	20	0	40	20	0	187.5	20	25	50	1031.25
B3	40	20	0	60	20	0	200	20	25	50	1100
B4	60	20	0	80	20	0	200	20	25	50	1100
B5	80	20	0	100	20	0	200	20	25	50	1100

B6	100	20	0	120	20	0	200	20	25	50	1100
B7	120	20	0	140	20	0	200	20	25	50	1100
B8	140	20	0	160	20	0	200	20	25	50	1100
B9	160	20	0	180	20	0	200	20	25	50	1100
B10	180	20	0	200	20	0	200	20	25	50	1100
BB2	30	30	0	40	30	0	112.5	10	25	50	1237.5
BB3	40	30	0	60	30	0	200	20	25	50	1100
BB4	60	30	0	80	30	0	200	20	25	50	1100
BB5	80	30	0	100	30	0	200	20	25	50	1100
BB6	100	30	0	120	30	0	200	20	25	50	1100
BB7	120	30	0	140	30	0	200	20	25	50	1100
BB8	140	30	0	160	30	0	200	20	25	50	1100
BB9	160	30	0	180	30	0	200	20	25	50	1100
BB10	180	30	0	200	30	0	200	20	25	50	1100
C3	40	40	0	60	40	0	187.5	20	25	50	1031.25
C4	60	40	0	80	40	0	200	20	25	50	1100
C5	80	40	0	100	40	0	200	20	25	50	1100
C6	100	40	0	120	40	0	200	20	25	50	1100
C7	120	40	0	140	40	0	200	20	25	50	1100
C8	140	40	0	160	40	0	200	20	25	50	1100
C9	160	40	0	180	40	0	200	20	25	50	1100
C10	180	40	0	200	40	0	200	20	25	50	1100
CC3	50	50	0	60	50	0	112.5	10	25	50	1237.5
CC4	60	50	0	80	50	0	200	20	25	50	1100
CC5	80	50	0	100	50	0	200	20	25	50	1100
CC6	100	50	0	120	50	0	200	20	25	50	1100
CC7	120	50	0	140	50	0	200	20	25	50	1100
CC8	140	50	0	160	50	0	200	20	25	50	1100
CC9	160	50	0	180	50	0	200	20	25	50	1100
CC10	180	50	0	200	50	0	200	20	25	50	1100

D4	60	60	0	80	60	0	187.5	20	25	50	1031.25
D5	80	60	0	100	60	0	200	20	25	50	1100
D6	100	60	0	120	60	0	200	20	25	50	1100
D7	120	60	0	140	60	0	200	20	25	50	1100
D8	140	60	0	160	60	0	200	20	25	50	1100
D9	160	60	0	180	60	0	200	20	25	50	1100
D10	180	60	0	200	60	0	200	20	25	50	1100
DD4	70	90	0	80	90	0	112.5	10	25	50	1237.5
DD5	80	90	0	100	90	0	200	20	25	50	1100
DD6	100	90	0	120	90	0	200	20	25	50	1100
DD7	120	90	0	140	90	0	200	20	25	50	1100
DD8	140	90	0	160	90	0	200	20	25	50	1100
DD9	160	90	0	180	90	0	200	20	25	50	1100
DD10	180	90	0	200	90	0	200	20	25	50	1100
E5	80	80	0	100	80	0	187.5	20	25	50	1031.25
E6	100	80	0	120	80	0	200	20	25	50	1100
E7	120	80	0	140	80	0	200	20	25	50	1100
E8	140	80	0	160	80	0	200	20	25	50	1100
E9	160	80	0	180	80	0	200	20	25	50	1100
E10	180	80	0	200	80	0	200	20	25	50	1100
EE5	90	90	0	100	90	0	112.5	10	25	50	1237.5
EE6	100	90	0	120	90	0	200	20	25	50	1100
EE7	120	90	0	140	90	0	200	20	25	50	1100
EE8	140	90	0	160	90	0	200	20	25	50	1100
EE9	160	90	0	180	90	0	200	20	25	50	1100
EE10	180	90	0	200	90	0	200	20	25	50	1100
F6	100	100	0	120	100	0	187.5	20	25	50	1031.25
F7	120	100	0	140	100	0	200	20	25	50	1100
F8	140	100	0	160	100	0	200	20	25	50	1100
F9	160	100	0	180	100	0	200	20	25	50	1100

F10	180	100	0	200	100	0	200	20	25	50	1100
FF6	110	110	0	120	110	0	112.5	10	25	50	1237.5
FF7	120	110	0	140	110	0	200	20	25	50	1100
FF8	140	110	0	160	110	0	200	20	25	50	1100
FF9	160	110	0	180	110	0	200	20	25	50	1100
FF10	180	110	0	200	110	0	200	20	25	50	1100
G7	120	120	0	140	120	0	187.5	20	25	50	1031.25
G8	140	120	0	160	120	0	200	20	25	50	1100
G9	160	120	0	180	120	0	200	20	25	50	1100
G10	180	120	0	200	120	0	200	20	25	50	1100
GG7	130	130	0	140	130	0	112.5	10	25	50	1237.5
GG8	140	130	0	160	130	0	200	20	25	50	1100
GG9	160	130	0	180	130	0	200	20	25	50	1100
GG10	180	130	0	200	130	0	200	20	25	50	1100
H8	140	140	0	160	140	0	187.5	20	25	50	1031.25
H9	160	140	0	180	140	0	200	20	25	50	1100
H10	180	140	0	200	140	0	200	20	25	50	1100
HH8	150	150	0	160	150	0	112.5	10	25	50	1237.5
HH9	160	150	0	180	150	0	200	20	25	50	1100
HH10	180	150	0	200	150	0	200	20	25	50	1100
I9	160	160	0	180	160	0	187.5	20	25	50	1031.25
I10	180	160	0	200	160	0	200	20	25	50	1100
II9	170	170	0	180	170	0	112.5	10	25	50	1237.5
II10	180	170	0	200	170	0	200	20	25	50	1100
J10	180	180	0	200	180	0	187.5	20	25	50	1031.25
K10	190	190	0	200	190	0	112.5	10	25	50	1237.5

Table B2: Gravity loadings on the beams of third, fourth, and fifth floor

	Third Floor/Fourth Floor/Fifth Floor										
Beam Number	x	z	y	x'	z'	y'	Tributary Area (ft2)	Length (ft)	Total Dead Load	Total Live Load	Load Combination(lb/ft)
A1	10	0	35	20	0	35	87.5	10	77	40	1368.5
A2	20	0	35	40	0	35	100	20	77	40	782
A3	40	0	35	60	0	35	100	20	77	40	782
A4	60	0	35	80	0	35	100	20	77	40	782
A5	80	0	35	100	0	35	100	20	77	40	782
A6	100	0	35	120	0	35	100	20	77	40	782
A7	120	0	35	140	0	35	100	20	77	40	782
A8	140	0	35	160	0	35	100	20	77	40	782
A9	160	0	35	180	0	35	100	20	77	40	782
A10	180	0	35	200	0	35	100	20	77	40	782
AA1	10	10	35	20	10	35	112.5	10	29	40	1111.5
AA2	20	10	35	40	10	35	200	20	29	40	988
AA3	40	10	35	60	10	35	200	20	29	40	988
AA4	60	10	35	80	10	35	200	20	29	40	988
AA5	80	10	35	100	10	35	200	20	29	40	988
AA6	100	10	35	120	10	35	200	20	29	40	988
AA7	120	10	35	140	10	35	200	20	29	40	988
AA8	140	10	35	160	10	35	200	20	29	40	988
AA9	160	10	35	180	10	35	200	20	29	40	988
AA10	180	10	35	200	10	35	200	20	29	40	988
B2	20	20	35	40	20	35	187.5	20	29	40	926.25
B3	40	20	35	60	20	35	200	20	29	40	988
B4	60	20	35	80	20	35	200	20	29	40	988
B5	80	20	35	100	20	35	200	20	29	40	988
B6	100	20	35	120	20	35	200	20	29	40	988
B7	120	20	35	140	20	35	200	20	29	40	988

B8	140	20	35	160	20	35	200	20	29	40	988
B9	160	20	35	180	20	35	200	20	29	40	988
B10	180	20	35	200	20	35	200	20	29	40	988
BB2	30	30	35	40	30	35	112.5	10	29	40	1111.5
BB3	40	30	35	60	30	35	200	20	29	40	988
BB4	60	30	35	80	30	35	200	20	29	40	988
BB5	80	30	35	100	30	35	200	20	29	40	988
BB6	100	30	35	120	30	35	200	20	29	40	988
BB7	120	30	35	140	30	35	200	20	29	40	988
BB8	140	30	35	160	30	35	200	20	29	40	988
BB9	160	30	35	180	30	35	200	20	29	40	988
BB10	180	30	35	200	30	35	200	20	29	40	988
C3	40	40	35	60	40	35	187.5	20	29	40	926.25
C4	60	40	35	80	40	35	200	20	29	40	988
C5	80	40	35	100	40	35	200	20	29	40	988
C6	100	40	35	120	40	35	200	20	29	40	988
C7	120	40	35	140	40	35	200	20	29	40	988
C8	140	40	35	160	40	35	200	20	29	40	988
C9	160	40	35	180	40	35	200	20	29	40	988
C10	180	40	35	200	40	35	200	20	29	40	988
CC3	50	50	35	60	50	35	112.5	10	29	40	1111.5
CC4	60	50	35	80	50	35	200	20	29	40	988
CC5	80	50	35	100	50	35	200	20	29	40	988
CC6	100	50	35	120	50	35	200	20	29	40	988
CC7	120	50	35	140	50	35	200	20	29	40	988
CC8	140	50	35	160	50	35	200	20	29	40	988
CC9	160	50	35	180	50	35	200	20	29	40	988
CC10	180	50	35	200	50	35	200	20	29	40	988
D4	60	60	35	80	60	35	187.5	20	29	40	926.25
D5	80	60	35	100	60	35	200	20	29	40	988

D6	100	60	35	120	60	35	200	20	29	40	988
D7	120	60	35	140	60	35	200	20	29	40	988
D8	140	60	35	160	60	35	200	20	29	40	988
D9	160	60	35	180	60	35	200	20	29	40	988
D10	180	60	35	200	60	35	200	20	29	40	988
DD4	70	90	35	80	90	35	112.5	10	29	40	1111.5
DD5	80	90	35	100	90	35	200	20	29	40	988
DD6	100	90	35	120	90	35	200	20	29	40	988
DD7	120	90	35	140	90	35	200	20	29	40	988
DD8	140	90	35	160	90	35	200	20	29	40	988
DD9	160	90	35	180	90	35	200	20	29	40	988
DD10	180	90	35	200	90	35	200	20	29	40	988
E5	80	80	35	100	80	35	187.5	20	29	40	926.25
E6	100	80	35	120	80	35	200	20	29	40	988
E7	120	80	35	140	80	35	200	20	29	40	988
E8	140	80	35	160	80	35	200	20	29	40	988
E9	160	80	35	180	80	35	200	20	29	40	988
E10	180	80	35	200	80	35	200	20	29	40	988
EE5	90	90	35	100	90	35	112.5	10	29	40	1111.5
EE6	100	90	35	120	90	35	200	20	29	40	988
EE7	120	90	35	140	90	35	200	20	29	40	988
EE8	140	90	35	160	90	35	200	20	29	40	988
EE9	160	90	35	180	90	35	200	20	29	40	988
EE10	180	90	35	200	90	35	200	20	29	40	988
F6	100	100	35	120	100	35	187.5	20	29	40	926.25
F7	120	100	35	140	100	35	200	20	29	40	988
F8	140	100	35	160	100	35	200	20	29	40	988
F9	160	100	35	180	100	35	200	20	29	40	988
F10	180	100	35	200	100	35	200	20	29	40	988
FF6	110	110	35	120	110	35	112.5	10	29	40	1111.5

FF7	120	110	35	140	110	35	200	20	29	40	988
FF8	140	110	35	160	110	35	200	20	29	40	988
FF9	160	110	35	180	110	35	200	20	29	40	988
FF10	180	110	35	200	110	35	200	20	29	40	988
G7	120	120	35	140	120	35	187.5	20	29	40	926.25
G8	140	120	35	160	120	35	200	20	29	40	988
G9	160	120	35	180	120	35	200	20	29	40	988
G10	180	120	35	200	120	35	200	20	29	40	988
GG7	130	130	35	140	130	35	112.5	10	29	40	1111.5
GG8	140	130	35	160	130	35	200	20	29	40	988
GG9	160	130	35	180	130	35	200	20	29	40	988
GG10	180	130	35	200	130	35	200	20	29	40	988
H8	140	140	35	160	140	35	187.5	20	29	40	926.25
H9	160	140	35	180	140	35	200	20	29	40	988
H10	180	140	35	200	140	35	200	20	29	40	988
HH8	150	150	35	160	150	35	112.5	10	29	40	1111.5
HH9	160	150	35	180	150	35	200	20	29	40	988
HH10	180	150	35	200	150	35	200	20	29	40	988
I9	160	160	35	180	160	35	187.5	20	29	40	926.25
I10	180	160	35	200	160	35	200	20	29	40	988
II9	170	170	35	180	170	35	112.5	10	29	40	1111.5
II10	180	170	35	200	170	35	200	20	29	40	988
J10	180	180	35	200	180	35	187.5	20	29	40	926.25
K10	190	190	35	200	190	35	112.5	10	29	40	1111.5

Table B3: Gravity loadings on the beams of the roof

Roof											
Beam Number	z	y	x'	z'	y'	Tributary Area (ft2)	Length (ft)	Total Dead Load	Total Live Load	Roof Live Load	Load Combination(lb/ft)
A1	0	80	20	0	80	87.5	10	29	0	20	90.98
A2	0	80	40	0	80	100	20	77	0	20	54.62
A3	0	80	60	0	80	100	20	77	0	20	54.62
A4	0	80	80	0	80	100	20	77	0	20	54.62
A5	0	80	100	0	80	100	20	77	0	20	54.62
A6	0	80	120	0	80	100	20	77	0	20	54.62
A7	0	80	140	0	80	100	20	77	0	20	54.62
A8	0	80	160	0	80	100	20	77	0	20	54.62
A9	0	80	180	0	80	100	20	77	0	20	54.62
A10	0	80	200	0	80	100	20	77	0	20	54.62
AA1	10	80	20	10	80	112.5	10	25	0	20	115.5
AA2	10	80	40	10	80	200	20	25	0	20	101.5
AA3	10	80	60	10	80	200	20	25	0	20	101.5
AA4	10	80	80	10	80	200	20	25	0	20	101.5
AA5	10	80	100	10	80	200	20	25	0	20	101.5
AA6	10	80	120	10	80	200	20	25	0	20	101.5
AA7	10	80	140	10	80	200	20	25	0	20	101.5
AA8	10	80	160	10	80	200	20	25	0	20	101.5
AA9	10	80	180	10	80	200	20	25	0	20	101.5
AA10	10	80	200	10	80	200	20	25	0	20	101.5
B2	20	80	40	20	80	187.5	20	25	0	20	95.25
B3	20	80	60	20	80	200	20	25	0	20	101.5
B4	20	80	80	20	80	200	20	25	0	20	101.5
B5	20	80	100	20	80	200	20	25	0	20	101.5
B6	20	80	120	20	80	200	20	25	0	20	101.5
B7	20	80	140	20	80	200	20	25	0	20	101.5

B8	20	80	160	20	80	200	20	25	0	20	101.5
B9	20	80	180	20	80	200	20	25	0	20	101.5
B10	20	80	200	20	80	200	20	25	0	20	101.5
BB2	30	80	40	30	80	112.5	10	25	0	20	115.5
BB3	30	80	60	30	80	200	20	25	0	20	101.5
BB4	30	80	80	30	80	200	20	25	0	20	101.5
BB5	30	80	100	30	80	200	20	25	0	20	101.5
BB6	30	80	120	30	80	200	20	25	0	20	101.5
BB7	30	80	140	30	80	200	20	25	0	20	101.5
BB8	30	80	160	30	80	200	20	25	0	20	101.5
BB9	30	80	180	30	80	200	20	25	0	20	101.5
BB10	30	80	200	30	80	200	20	25	0	20	101.5
C3	40	80	60	40	80	187.5	20	25	0	20	95.25
C4	40	80	80	40	80	200	20	25	0	20	101.5
C5	40	80	100	40	80	200	20	25	0	20	101.5
C6	40	80	120	40	80	200	20	25	0	20	101.5
C7	40	80	140	40	80	200	20	25	0	20	101.5
C8	40	80	160	40	80	200	20	25	0	20	101.5
C9	40	80	180	40	80	200	20	25	0	20	101.5
C10	40	80	200	40	80	200	20	25	0	20	101.5
CC3	50	80	60	50	80	112.5	10	25	0	20	115.5
CC4	50	80	80	50	80	200	20	25	0	20	101.5
CC5	50	80	100	50	80	200	20	25	0	20	101.5
CC6	50	80	120	50	80	200	20	25	0	20	101.5
CC7	50	80	140	50	80	200	20	25	0	20	101.5
CC8	50	80	160	50	80	200	20	25	0	20	101.5
CC9	50	80	180	50	80	200	20	25	0	20	101.5
CC10	50	80	200	50	80	200	20	25	0	20	101.5
D4	60	80	80	60	80	187.5	20	25	0	20	95.25
D5	60	80	100	60	80	200	20	25	0	20	101.5

D6	60	80	120	60	80	200	20	25	0	20	101.5
D7	60	80	140	60	80	200	20	25	0	20	101.5
D8	60	80	160	60	80	200	20	25	0	20	101.5
D9	60	80	180	60	80	200	20	25	0	20	101.5
D10	60	80	200	60	80	200	20	25	0	20	101.5
DD4	90	80	80	90	80	112.5	10	25	0	20	115.5
DD5	90	80	100	90	80	200	20	25	0	20	101.5
DD6	90	80	120	90	80	200	20	25	0	20	101.5
DD7	90	80	140	90	80	200	20	25	0	20	101.5
DD8	90	80	160	90	80	200	20	25	0	20	101.5
DD9	90	80	180	90	80	200	20	25	0	20	101.5
DD10	90	80	200	90	80	200	20	25	0	20	101.5
E5	80	80	100	80	80	187.5	20	25	0	20	95.25
E6	80	80	120	80	80	200	20	25	0	20	101.5
E7	80	80	140	80	80	200	20	25	0	20	101.5
E8	80	80	160	80	80	200	20	25	0	20	101.5
E9	80	80	180	80	80	200	20	25	0	20	101.5
E10	80	80	200	80	80	200	20	25	0	20	101.5
EE5	90	80	100	90	80	112.5	10	25	0	20	115.5
EE6	90	80	120	90	80	200	20	25	0	20	101.5
EE7	90	80	140	90	80	200	20	25	0	20	101.5
EE8	90	80	160	90	80	200	20	25	0	20	101.5
EE9	90	80	180	90	80	200	20	25	0	20	101.5
EE10	90	80	200	90	80	200	20	25	0	20	101.5
F6	100	80	120	100	80	187.5	20	25	0	20	95.25
F7	100	80	140	100	80	200	20	25	0	20	101.5
F8	100	80	160	100	80	200	20	25	0	20	101.5
F9	100	80	180	100	80	200	20	25	0	20	101.5
F10	100	80	200	100	80	200	20	25	0	20	101.5
FF6	110	80	120	110	80	112.5	10	25	0	20	115.5

FF7	110	80	140	110	80	200	20	25	0	20	101.5
FF8	110	80	160	110	80	200	20	25	0	20	101.5
FF9	110	80	180	110	80	200	20	25	0	20	101.5
FF10	110	80	200	110	80	200	20	25	0	20	101.5
G7	120	80	140	120	80	187.5	20	25	0	20	95.25
G8	120	80	160	120	80	200	20	25	0	20	101.5
G9	120	80	180	120	80	200	20	25	0	20	101.5
G10	120	80	200	120	80	200	20	25	0	20	101.5
GG7	130	80	140	130	80	112.5	10	25	0	20	115.5
GG8	130	80	160	130	80	200	20	25	0	20	101.5
GG9	130	80	180	130	80	200	20	25	0	20	101.5
GG10	130	80	200	130	80	200	20	25	0	20	101.5
H8	140	80	160	140	80	187.5	20	25	0	20	95.25
H9	140	80	180	140	80	200	20	25	0	20	101.5
H10	140	80	200	140	80	200	20	25	0	20	101.5
HH8	150	80	160	150	80	112.5	10	25	0	20	115.5
HH9	150	80	180	150	80	200	20	25	0	20	101.5
HH10	150	80	200	150	80	200	20	25	0	20	101.5
I9	160	80	180	160	80	187.5	20	25	0	20	95.25
I10	160	80	200	160	80	200	20	25	0	20	101.5
II9	170	80	180	170	80	112.5	10	25	0	20	115.5
II10	170	80	200	170	80	200	20	25	0	20	101.5
J10	180	80	200	180	80	187.5	20	25	0	20	95.25
K10	190	80	200	190	80	112.5	10	25	0	20	115.5

Appendix C: Rigid Pressure Model Test

Larger Flow Domain (600’ x 250’ x 1500’)

Table C1: Rigid Pressure Model Test Results of Larger Flow Domain

Angle of Wind Direction (α)	*Maximum Pressure (Pa)*	*Minimum Pressure (Pa)*	*Exposed Surface Area (ft^2) (Pressure)*	*Exposed Surface Area (ft^2) (Suction)*	*Wind Load (Pressure)*	*Wind Load (Suction)*	*Maximum Pressure (lbs/ ft^2)*	*Minimum Pressure (lbs/ ft^2)*
0	1555.44	-2485.35	16000.00	22627.42	519.78	-1174.54	32.49	-51.91
20	1841.4	-2552.41	20507.40	22627.42	788.68	-1206.23	38.46	-53.31
40	1541.66	-2313.43	22541.35	22627.42	725.79	-1093.29	32.20	-48.32
45	1506.15	-2206.58	22627.42	22627.42	711.78	-1042.79	31.46	-46.09
60	1643.48	-2472.69	21856.41	22627.42	750.22	-1168.55	34.32	-51.64
80	1581.28	-2306.92	18535.29	22627.42	612.14	-1090.21	33.03	-48.18
90	1545.42	-2440.15	16000.00	38627.42	516.43	-1968.60	32.28	-50.96
100	1653.09	-1897.91	15756.92	38627.42	544.02	-1531.14	34.53	-39.64
120	1813.02	-2245.22	13856.40	38627.42	524.68	-1811.34	37.87	-46.89
135	1716.69	-1720.05	11313.71	16000.00	405.64	-574.78	35.85	-35.92
140	1590.46	-1281.8	10284.60	38627.42	341.63	-1034.09	33.22	-26.77
160	1021.67	-1522.17	12234.21	16000.00	261.05	-508.66	21.34	-31.79
180	1691.46	-2177.55	16000.00	16000.00	565.23	-727.67	35.33	-45.48

Smaller Flow Domain (450' x 200' x 833')

Table C2: Rigid Pressure Model Test Results of Smaller Flow Domain

Angle of Wind Direction (α)	***Maximum Pressure (Pa)***	***Minimum Pressure (Pa)***	***Exposed Surface Area (ft^2) (Pressure)***	***Exposed Surface Area (ft^2) (Suction)***	***Wind Load (Pressure)***	***Wind Load (Suction)***	***Maximum Pressure (lbs/ft^2)***	***Minimum Pressure (lbs/ ft^2)***
0	2013.68	-2984.66	16000.00	22627.42	672.91	-1410.50	42.06	-62.34
20	2431.78	-2870.86	20507.40	22627.42	1041.55	-1356.72	50.79	-59.96
40	1987.02	-2849.92	22541.35	22627.42	935.46	-1346.83	41.50	-59.52
45	1975.51	-2720.15	22627.42	22627.42	933.59	-1285.50	41.26	-56.81
60	2124.03	-2891.83	21856.41	22627.42	969.58	-1366.63	44.36	-60.40
80	2090.9	-2471.41	18535.29	22627.42	809.43	-1167.95	43.67	-51.62
90	2068.27	-3298.89	16000.00	38627.42	691.15	-2661.39	43.20	-68.90
100	1995.42	-2546.24	15756.92	38627.42	656.67	-2054.18	41.68	-53.18
120	2084.9	-2589.27	13856.40	38627.42	603.36	-2088.90	43.54	-54.08
135	1913.4	-1863.88	11313.71	16000.00	452.12	-622.85	39.96	-38.93
140	1741.21	-1413.62	10284.60	38627.42	374.01	-1140.44	36.37	-29.52
160	1165.29	-1827.03	12234.21	16000.00	297.75	-610.53	24.34	-38.16
180	1973.54	-2460.76	16000.00	16000.00	659.49	-822.31	41.22	-51.39

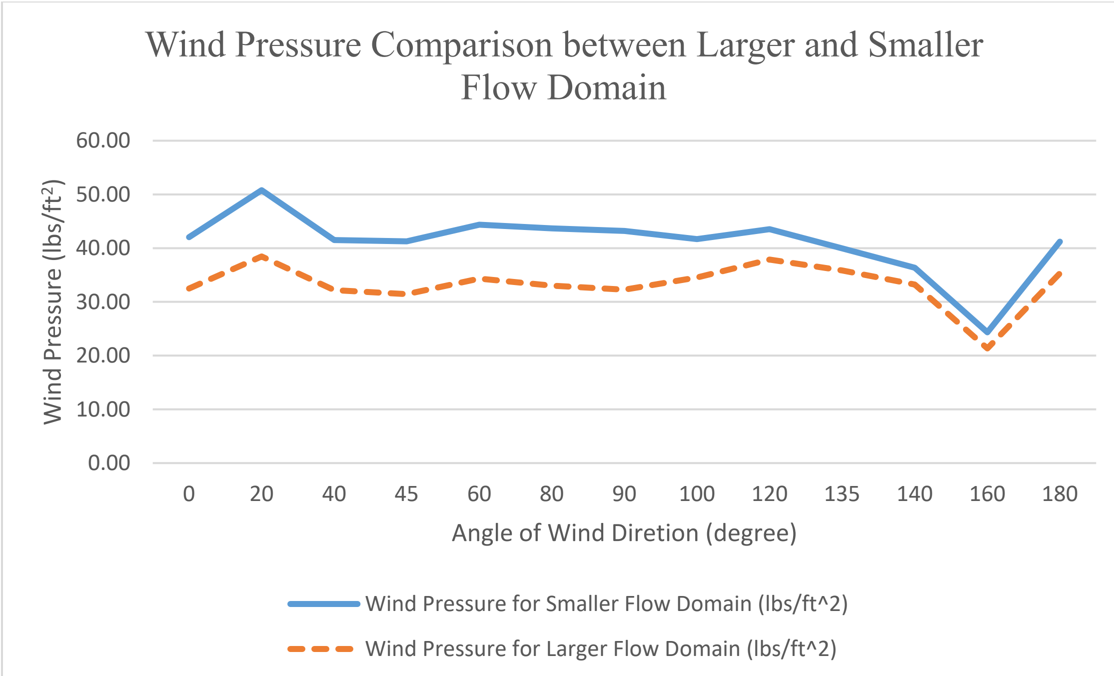


Figure C1: Wind Pressure Comparison between Larger and Smaller Flow Domain

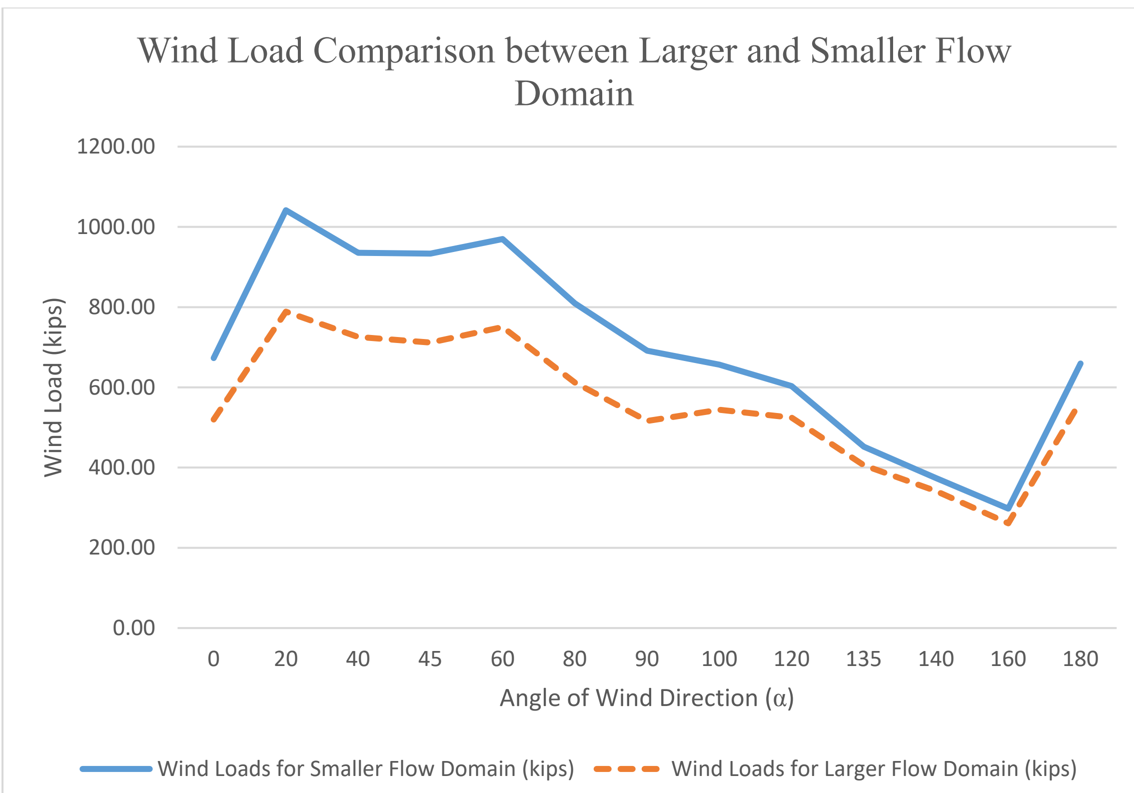


Figure C2: Wind Load Comparison between Larger and Smaller Domain

Table C3: Maximum and Minimum Peak Pressures of the building

Angle of Wind Direction (α)	Peak Maxmium Stress (kip/ft2)	Peak Minimum Stress (kip/ft2)
0	0.16519472	-0.179914489
20	0.146822283	-0.161635954
40	0.014813671	-0.017181786
45	0.261887215	-0.261887215
60	0.152311236	-0.146639333
80	0.145536	-0.133317373
90	0.16519472	-0.179914489
100	0.051620339	-0.033186865
120	0.052193378	-0.041754753
135	0.01398354	-0.027967079
140	0.05222355	-0.041724581
160	0.08129204	-0.093571006
180	0.160200497	-0.128160542

Appendix D: Graphical data of Fluctuating Wind Speed (NatHaz)

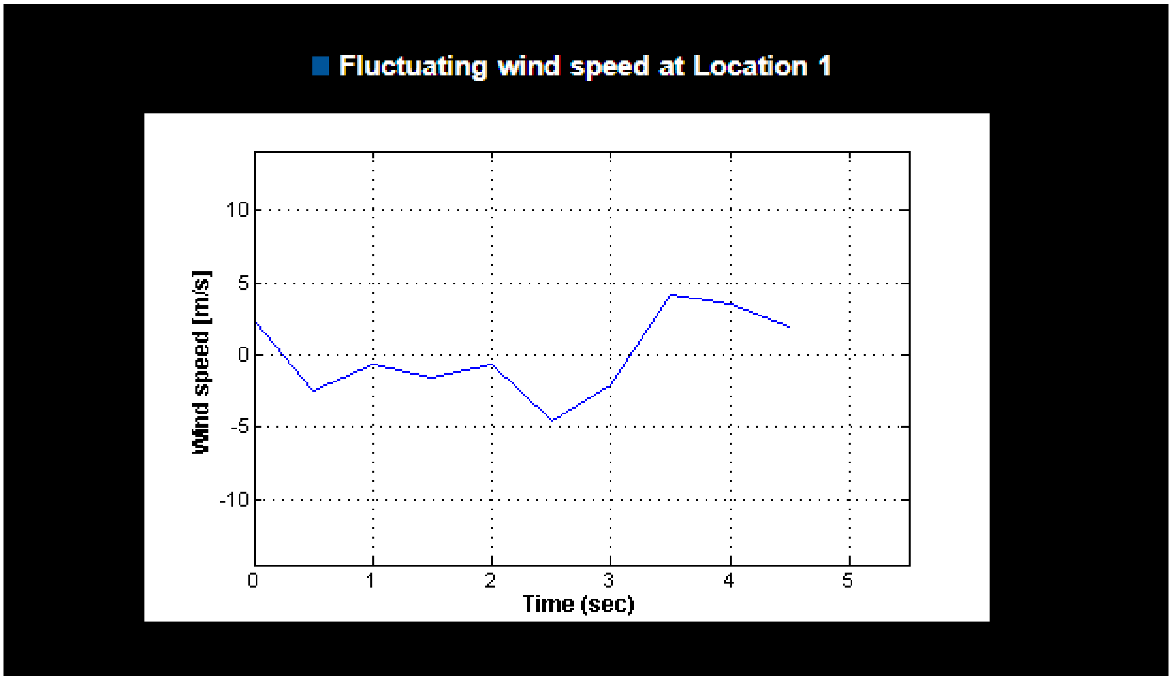


Figure D1: Fluctuating Wind Speed [with cut-off frequency = 1Hz]

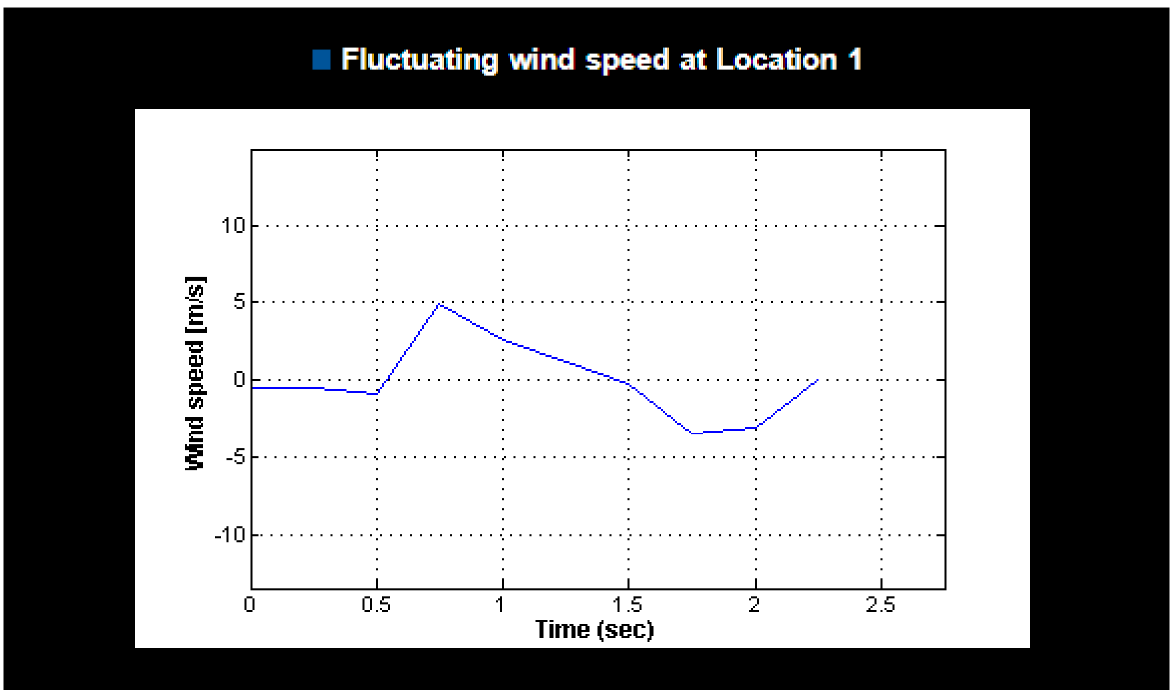


Figure D2: Fluctuating Wind Speed [with cut-off frequency = 2Hz]

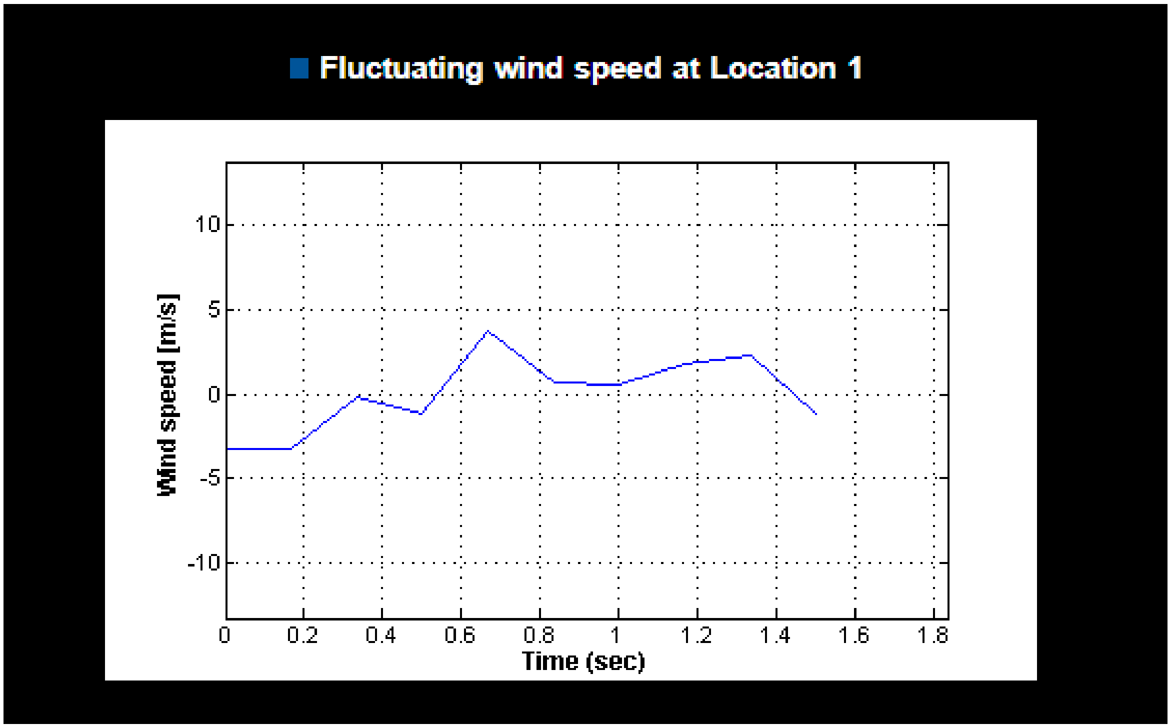


Figure D3: Fluctuating Wind Speed [with cut-off frequency = 3Hz]

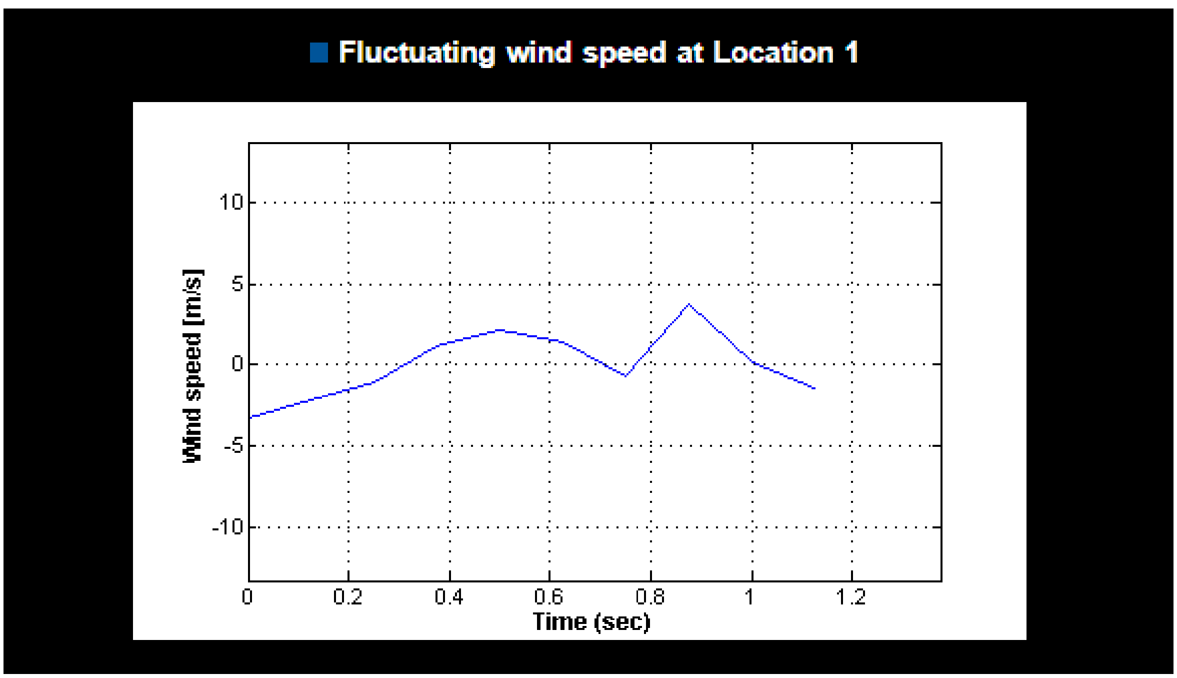


Figure D4: Fluctuating Wind Speed [with cut-off frequency = 4Hz]

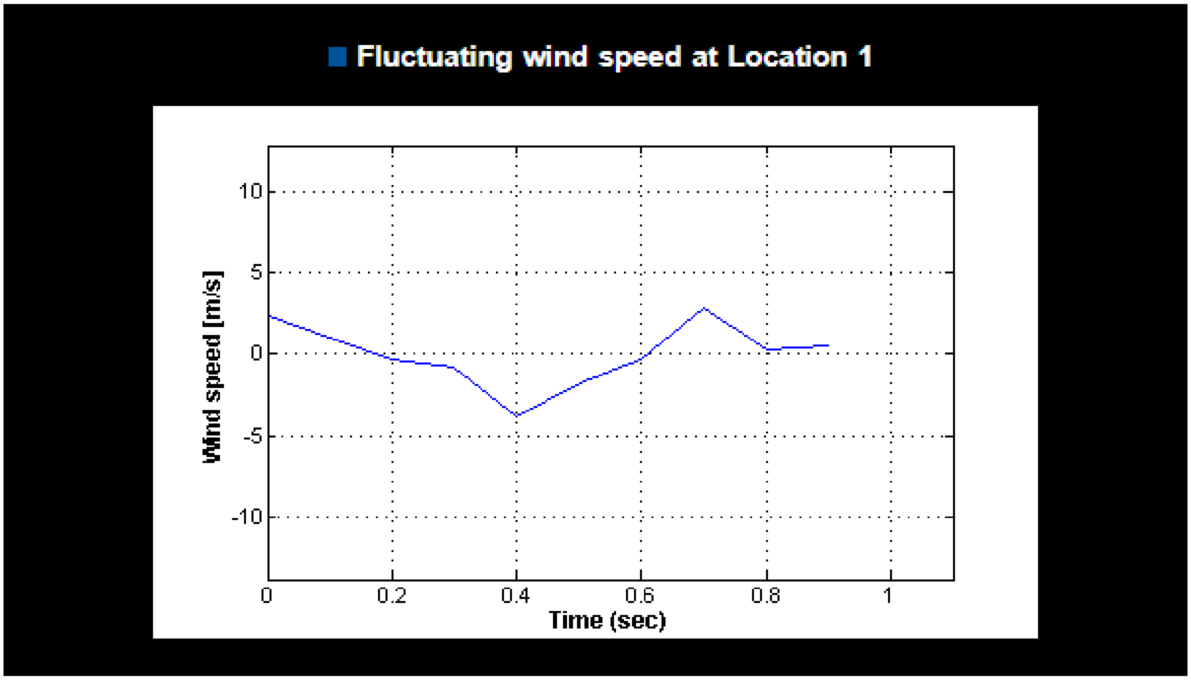


Figure D5: Fluctuating Wind Speed [with cut-off frequency = 5Hz]

Appendix E: Tabular data of Fluctuating Wind Speed (NatHaz)

Table 1: Fluctuating Wind Speed and Wind Force (Cut-Off Frequency = 1 Hz)

Cut Off Frequency	1 Hz	
Mean Wind Speed (ft/s)	94.03027	
Time Step (s)	**Fluctuating Wind Speed (ft/s)**	**Fluctuating wind force (lbf)**
0	14.85458	2568373
0.5	5.022318	868364.1
1	-3.95077	-683092
1.5	5.716493	988387.7
2	-0.41733	-72156.2
2.5	-5.38767	-931533
3	-20.623	-3565737
3.5	-2.90151	-501675
4	1.401301	242286.3
4.5	6.285561	1086780

Table 2: Fluctuating Wind Speed and Wind Force (Cut-Off Frequency = 2 Hz)

Cut Off Frequency	2 Hz	
Mean Wind Speed (ft/s)=	94.03027	
Time Step (s)	**Fluctuating Wind Speed (ft/s)**	**Fluctuating wind force (lbf)**
0	9.744995	1684920
0.25	0.539251	93236.99
0.5	3.130167	541209.2
0.75	2.969567	513441.2
1	12.9768	2243703
1.25	4.581584	792160.7
1.5	-5.65225	-977280
1.75	-7.86751	-1360301
2	-10.7222	-1853886
2.25	-9.70041	-1677211

Table 3: Fluctuating Wind Speed and Wind Force (Cut-Off Frequency = 3 Hz)

Cut Off Frequency	3 Hz	
Mean Wind Speed (ft/s)=	94.03027	
Time Step (s)	**Fluctuating Wind Speed (ft/s)**	**Fluctuating wind force (lbf)**
0	-4.05608	-701300
0.16667	4.599184	795203.7
0.33333	14.50669	2508221
0.5	6.861376	1186339
0.66667	-1.81955	-314602
0.83333	1.928378	333418.5
1	-4.27872	-739795
1.1667	-5.92197	-1023915
1.3333	-1.43135	-247482
1.5	-10.388	-1796093

Table 4: Fluctuating Wind Speed and Wind Force (Cut-Off Frequency = 4 Hz)

Cut Off Frequency	4 Hz	
Mean Wind Speed (ft/s)=	94.03027	
Time Step (s)	**Fluctuating Wind Speed (ft/s)**	**Fluctuating wind force (lbf)**
0	4.989465	862683.7
0.125	-3.02501	-523027
0.25	1.215547	210169.3
0.375	3.409421	589492.5
0.5	-11.4259	-1975558
0.625	-8.24562	-1425676
0.75	-4.48244	-775018
0.875	1.68007	290486
1	7.928378	1370825
1.125	7.955951	1375592

Table 5: Fluctuating Wind Speed and Wind Force (Cut-Off Frequency = 5 Hz)

Cut Off Frequency	5 Hz	
Mean Wind Speed (ft/s)=	94.03027	
Time Step (s)	**Fluctuating Wind Speed (ft/s)**	**Fluctuating wind force (lbf)**
0	-4.4861	-775652
0.1	2.026058	350307.6
0.2	6.671442	1153499
0.3	12.26635	2120864
0.4	3.953702	683599.3
0.5	-7.2609	-1255417
0.6	-1.8945	-327561
0.7	-2.65453	-458970
0.8	-6.8624	-1186517
0.9	-1.75912	-304154

Appendix F: Time Domain Analysis Test

Table F1: Tabular Result of Time Domain Analysis Showing Maximum Deformation (ft) and Principal Maximum Stress (psf)

Cut-Off Frequency (Hz)	Principal Maximum Stress (**Static Analysis**)	Principal Maximum Stress (**Time Domain Analysis**)	Total Deformation (**Static Analysis**)	Total Deformation (**Time Domain Analysis**)
1 Hz	1.6098e6 psf	2.2421e5 psf	0.054509 ft	0.014121 ft
2 Hz	1.6098e6 psf	1.2641e6 psf	0.054509 ft	0.053159 ft
3 Hz	1.6098e6 psf	1.1456e6 psf	0.054509 ft	0.012544 ft
4 Hz	1.6098e6 psf	4.5896e5 psf	0.054509 ft	0.016678 ft
5 Hz	1.6098e6 psf	9.5876e5 psf	0.054509 ft	0.03568 ft

Table F2: Tabular Result of Time Domain Analysis Showing Maximum Deformation (in) and Principal Maximum Stress (ksi)

Cut-Off Frequency (Hz)	Principal Maximum Stress (**Static Analysis**)	Principal Maximum Stress (**Time Domain Analysis**)	Total Deformation (**Static Analysis**)	Total Deformation (**Time Domain Analysis**)
1 Hz	11.179ksi	1.5577 ksi	0.654in	0.1694 in
2 Hz	11.179ksi	8.7784 ksi	0.654in	0.6379 in
3 Hz	11.179ksi	7.9556 ksi	0.654in	0.1505 in
4 Hz	11.179ksi	3.1872 ksi	0.654in	0.2001 in
5 Hz	11.179ksi	6.6581 ksi	0.654in	0.4282 in

Appendix G: Member Sizes (Gravity +Wind)

Table G1: Suggested Member Sizes from RISA 3D

Floor Number	Exterior Beams	Interior Beams	Girder	Columns	Brace Members
1	*Slab on Grade*	*Slab on Grad*	*Slab on Grade*	W12x65	W8x31
2	W14x43	W10x33	W12x30	W12x45	W8x31
3	W14x43	W10x33	W12x30	W10x39	W8x31
4	W14x43	W10x33	W12x30	W10x39	W8x31
5	W14x43	W10x33	W12x30	W8x28	W8x31
Roof	W8x24	W8x24	W 12x30	*N/A*	*N/A*

Appendix H: Column Reaction Forces for Foundation Design

Table H1: Column Reaction Forces in kips from RISA 3D (Maximum Value Highlighted)

Column Joint Label	Column1	Column Reaction Forces (kips)
1_131	max	160.71
	min	36.105
1_130	max	154.86
	min	34.24
1_129	max	186.944
	min	59.68
1_128	max	251.102
	min	76.858
1_124	max	218.876
	min	66.563
1_123	max	328.868
	min	87.276
1_117	max	270.864
	min	81.951
1_122	max	301.666
	min	87.206
1_116	max	338.572
	min	87.82
1_108	max	227.242
	min	68.711
1_115	max	332.999
	min	88.974
1_107	max	335.933
	min	87.491
1_97	max	273.009
	min	82.261
1_114	max	304.693
	min	87.591
1_96	max	335.676
	min	87.374
1_84	max	227.442
	min	68.726
1_105	max	334.432
	min	89.296
1_95	max	335.844
	min	87.398
1_83	max	335.681

	min	87.384
1_69	max	272.77
	min	82.175
1_52	max	226.479
	min	68.428
1_68	max	335.479
	min	87.304
1_82	max	335.638
	min	87.397
1_33	max	261.33
	min	78.445
1_51	max	337.305
	min	87.324
1_67	max	335.466
	min	87.307
1_81	max	335.71
	min	87.381
1_93	max	334.47
	min	89.32
1_80	max	336.549
	min	86.734
1_66	max	335.431
	min	87.322
1_50	max	337.695
	min	87.44
1_32	max	349.603
	min	92.362
1_12	max	164.798
	min	46.114
1_11	max	229.678
	min	70.848
1_31	max	348.937
	min	91.91
1_49	max	337.147
	min	87.26
1_65	max	335.487
	min	87.301
1_64	max	336.338
	min	86.657
1_48	max	337.635
	min	87.45

1_30	max	350.263
	min	92.345
1_10	max	197.006
	min	61.408
1_9	max	237.937
	min	74.355
1_29	max	349.02
	min	91.93
1_47	max	337.164
	min	87.253
1_63	max	334.133
	min	89.221
1_62	max	304.716
	min	87.518
1_46	max	338.545
	min	86.788
1_28	max	350.256
	min	92.37
1_8	max	197.61
	min	61.685
1_7	max	238.098
	min	74.429
1_27	max	349.052
	min	91.928
1_45	max	335.846
	min	89.165
1_44	max	301.59
	min	87.201
1_26	max	351.176 (MAX)
	min	91.687
1_6	max	197.459
	min	61.639
1_25	max	347.459
	min	93.67
1_5	max	237.428
	min	74.303
1_4	max	194.961
	min	61.34
1_24	max	252.94
	min	76.322
1_13	max	175.439

	min	44.173
1_3	max	202.355
	min	63.582
1_79	max	334.385
	min	89.311
1_78	max	305.194
	min	87.599
1_104	max	305.177
	min	87.617
1_94	max	336.651
	min	86.754
1_106	max	336.595
	min	86.75
1_92	max	305.265
	min	87.627
1_2	max	129.713
	min	32.692

Appendix I: Foundation Design Calculations

Square Column Footing Design.

base column reaction (maximum) from RISA 3D = 352 kips.

average weight of concrete and fill = 125 pcf

The foundation height below grade = 8 ft.

height of the foundation = 12"

self weight + weight of fill = 125 pcf × 8 ft

= 1,000 psf.

$q_a = 5000\ lb/ft^2$.

$q_e = 5000 - 1000 = 4000\ lb/ft^2$.

$A_{req} = (352\ kips.) / 4\ kips/ft^2 = 88\ ft^2$.

$b = h = \sqrt{88\ ft^2} \approx 9.5'$

$q_u = 352/(9.5)^2 = 3.9\ kips/ft^2$

Trial calculation , $d = 19 in.$

$b_o = 4(18 + d)$

$= 148 \text{ in.}$

$V_{u1} = 3.9 \left[9.5^2 - \left(\frac{43}{12}\right)^2 \right] = 472.2 \text{ kips.}$

$V_c = 4\lambda\sqrt{f'_c}\, b_o d.$

$= 4(1)\sqrt{4000}\,(148)(19)$

$= 711 \text{ kips.} > V_{u1}$ [OK]

$\phi V_c = 0.75\,(711) \text{ kips.} = 533.5 \text{ kips} > V_{u1}$ [OK]

$V_{u2} = 3.9 \times 2.42' \times 9.5$

$V_{u2} = 89,661 \text{ kips.}$

$V_c = 2\lambda\sqrt{f'_c}\, bd.$

$= 2(1)(\sqrt{4000}\,(9.5)(12)$

$V_c = 144.2 \text{ kips.}$

$\phi V_c = (0.75)(144.2) \text{ kips.}$

$\phi V_c = 108.15 \text{ kips.} > 144.2 \text{ kips.}$

$M_u = 3.9 \times 9.5 \left(\frac{4.0^2}{2}\right) 12 = 3556.8 \text{ kip-in.}$

$= 296.4 \text{ kip-ft.}$

$A_s = \dfrac{296.4 \times 12}{(0.90)(60)\left(19 - \frac{2}{2}\right)}$ $(a = 2)$

$A_s = 3.47 \text{ in}^2.$

$$A_{s,min} = \frac{3\sqrt{f'_c}}{f_y} bd.$$

$$= \frac{3(\sqrt{4000})}{60,000} (9.5 \times 12)(19).$$

$$= 6.85 \text{ in}^2.$$

$$A_{s,min} = \frac{200}{f_y} bd.$$

$$= \frac{200}{60,000} (9.5 \times 12)(19)$$

$$= 7.22 \text{ in}^2.$$

$A_{s,min} > A_s$

$\therefore$ 12 #7 (No. 22) bars are used.

Appendix J: Connection Design Calculations

Base Plote Column Design.

$P_u = 352$ kips (from RISA 3D)

$A_2 = 9.5\ ft^2 \times 144\ in^2/ft^2 = 12,996\ in^2$.

assume $\sqrt{\frac{A_2}{A_1}} = 2$.

$A_1 = BN$

$A_1 = \frac{P_u}{\phi(0.85 f'_c)\sqrt{\frac{A_2}{A_1}}}$

$A_1 = \frac{352\ kips}{(0.65)(0.85)(4)(2)} = 79.63\ ft^2$ ($f'_c = 4$ kips).

$b_f d = (12.12)(12) = 145.44^2$ [W 12x65]

$A_1 = 150\ in^2 > 145.44\ in^2$ [OK]

$B = \frac{A_1}{N} = \frac{150}{13} = 11.5$ in.

$\Delta = 0.95d - 0.8 b_f$.

$= \frac{0.95(12.1) - (0.8)(12.0)}{2}$

$\Delta = 0.947$ in.

$N = \sqrt{A_1} + \Delta = \sqrt{150} + 0.467$

$= 12.71$ in.

sqy = 13 in.

$\phi_c P_n = \phi_c\ 0.85\ f'_c\ A_1 \sqrt{\frac{A_2}{A_1}}$

$= 528.19$ kips > 352 kips [OK]

Plate Thickness

$$m = \frac{N - 0.95d}{2} = \frac{13 - 0.95(12.1)}{2} = 0.75 \text{ in.}$$

$$n = \frac{B - 0.8b_f}{2} = \frac{13 - (0.8)(12)}{2} = 1.7 \text{ in.}$$

$$n' = \frac{\sqrt{db_f}}{4} = \frac{\sqrt{(12.1)(12)}}{4} = 3.012 \text{ in.}$$

$$l = 3.012 \text{ in.}$$

$$t_{req} = l\sqrt{\frac{2P_u}{0.9F_y BN}}$$

$$= 3.012\sqrt{\frac{2(352)}{0.9(36)(13\times13)}}$$

$$t_{req} = 1.08 \text{ in.}$$

Base Plate Final Design = $1\frac{1}{2}" \times 13" \times 13"$

Connection Design.

R_u = 30.5 kips (maximum).

Beam to Girder Connections.

Beam = W 8 × 24 , F_y = 50 ksi , F_u = 65 ksi
Girder = W 12 × 30 , F_y = 50 ksi , F_u = 65 ksi

Angles : F_y = 36 ksi
F_u = 58 ksi] - ASTM 36.

Beam W 8 × 24 , t_w = 0.245 in.
Girder W 12 × 30 , t_w = 0.260 in.

AISC Manual , Table 10 - 1.

3 rows of bolts and 2L 6 × $3\frac{1}{2}$" × $\frac{1}{2}$" [$\phi R_n > P_u$.]

Appendix K: Detailed Cost Estimate

Table K1: RS Means Cost Estimate by Square Foot of the Building

	Building Category	Description	Unit	Unit Cost	Cost per S.F	% of Sub Total
A	**Substructure**					
1010	Standard Foundations	[002/004]12" thick, 4000 psi concrete 9.5 feet wide footing	each	40.95	7.32	4.18%
1030	Slab on Grade	[003/1020] 5" thick, reinforced, non-industrial	S.F floor	5.5	5.5	3.14%
2010	Foundation Excavation	[001/008] Excavate, Back fill, 8 feet deep 20' x20'	each	960	6.25	3.57%
B	**B10 Superstructure**					
1010	Floor Constructions	[208/6400] 500kips load, 20ft unsupported	V.L.F	194.1	88.10	7.20%
		[254/0720] 20'x20' bay size 5"slab thickness-126psf-W shape. Composite Deck	S.F floor	22.25	22.25	12.70%
		[720/3450] Column Fireproof Gypsum Board	V.L.F	26.11	1.77	1.01%
1020	Roof Constructions	[112/1500] 15'x20' bay size, 40psf steel joists-beams-deck on columns	S.F floor	5.68	5.68	3.24%
	B20 Exterior Enclosure					
2010	Exterior Wall	[103/5950]6" thickness, 20x10 panel size, precast concrete with rigid insulation	S.F wall	46	31.41	17.92%
2020	Exterior Windows	[106/6450]Aluminum, Double Hung, Insulated 4'5" x 5'3"	each	620	1.16	0.66%
2030	Exterior Doors	[110/6300]Glazed Door with Aluminum and Glass 3' x 7'	each	2975	1.86	1.06%
	B30 Roofing					
3010	Roof Coverings	[120/1000]Single Ply Membrane	S.F floor	3.56	0.712	0.41%

3010	Roof Deck Rigid Insulations	[320/1650]Roof Deck Rigid Insulations 2.5" thick	S.F floor	1.93	0.386	0.22%
3010	Roof Edges	[420/1000] Aluminium Roof Edges 0.050" thick	L.F floor	25.3	0.173	0.10%
3010	Gutters	[610/0050] Gutter Box, Aluminium 0.027" thick	L.F floor	8.97	0.061	0.03%
C	**Interiors**					
1010	Partitions	[126/7850]Dry Wall Paritions/ Metal Stud Framing 5/8" FR wall	S.F wall	7.38	5.039	2.88%
1020	Interior Doors	[102/2500]Single Leaf wood 3' x 7' x 1 3/8"	each	626	1.96	1.12%
2010	Stairs	[110/0700] 24 risers, with landing	flight	16,625	0.665	0.38%
3010	Wall Finishes	[230/0080] Painting Interior /primer & 2 coats	S.F wall	1.2	3.82	2.18%
3020	Floor Finishes	[410/0660] Tile& coverings 3/4" thick	S.F Floor	7.78	7.78	4.44%
3030	Ceiling Finishes	[105/4500] Plaster Ceiling 3.4# metal	S.F Floor	12.54	12.54	7.16%
D	**Services**					
	D10 Conveying					
1010	Elevators and Lift	[140/1300] Passenger 2000lbs, 5 floors	each	171,100	6.844	3.91%
	D20 Plumbing					
2010	Plumbing Fixtures	[932/1360] Bathtub, water closet, stall shower and lavatories	each	9775	4.89	2.79%
2040	Rain Water Drainage	Roof drains	S.F floor	1.68	1.68	0.96%
	D30 HVAC					
3010	Energy Supply	Oil fired hot water, basedboard radiations	S.F floor	7.87	7.87	4.49%
3020	Heating Generating Systems	Air heating system	S.F floor	9.54	9.54	5.44%
	D40 Fire Protection					
4010	Sprinklers	Wet pipes spinkler system, light hazard	S.F floor	2.78	2.78	1.59%
4020	Standpipes	Standpipes and hose systems, with pumps	S.F floor	0.95	0.95	0.54%

	D50 Electrical					
5010	Electrical Service/Distribution	1600 Ampere service, panel board and feeders	S.F floor	2.45	2.45	1.40%
5020	Lighitng & Branch Wiring	Incandecent Fixtures, receptacles, swithced, A.C and misc. power	S.F floor	7.59	7.59	4.33%
5030	Communications & Sercurity	Addressable Alarm Systems, Emergency lighting, Internet and Phone Wiring	S.F floor	1.7	1.7	0.97%
E	**Equipment and Furnishings**					
	N/A					
F	**Special Construction**					
	N/A					
G	**Building Sitework**					
	N/A					
				Sub-Total	175.25	100.00%
	Contractor Fees			25%	43.8125	
	Architect Fees			8%	14.02	
				Total Building Cost	**233.0825**	