Structural Analyses of Wind Turbine Tower for 3 *kW* Horizontal-Axis Wind Turbine

A Thesis

presented to

the Faculty of California Polytechnic State University,

San Luis Obispo

In Partial Fulfillment

of the Requirements for the Degree

Master of Science in Mechanical Engineering

by

Tae-gyun (Tom) Gwon

August 9, 2011

©2011

Tae gyun (Tom) Gwon

ALL RIGHTS RESERVED

COMMITTEE MEMBERSHIP

TITLE: Structural Analyses of Wind Turbine Tower for 3 *kW*
Horizontal Axis Wind Turbine

AUTHOR: Tae gyun (Tom) Gwon

DATE SUBMITTED: August 9, 2011

COMMITTEE CHAIR: Dr. Joseph Mello

COMMITTEE MEMBER: Dr. Tom Mase

COMMITTEE MEMBER: Dr. Andrew Davol

ABSTRACT

Structural Analyses of Wind Turbine Tower for 3 *kW* Horizontal Axis Wind Turbine

Tae gyun (Tom) Gwon

Structure analyses of a steel tower for Cal Poly's 3 *kW* small wind turbine is presented. First, some general design aspects of the wind turbine tower are discussed: types, heights, and some other factors that can be considered for the design of wind turbine tower. Cal Poly's wind turbine tower design is presented, highlighting its main design features. Secondly, structure analysis for Cal Poly's wind turbine tower is discussed and presented. The loads that are specific to the wind turbine system and the tower are explained. The loads for the static analysis of the tower were calculated as well. The majority of the structure analysis of the tower was performed using the finite element method (FEM). Using *Abaqus*, a commercial FEM software, both static and dynamic structural analyses were performed. A simplified finite element model that represents the wind turbine tower was created using beam, shell, and inertia elements. An ultimate load condition was applied to check the stress level of the tower in the static analysis. For the dynamic analysis, the frequency extraction was performed in order to obtain the natural frequencies and the mode shapes of the tower. Using the results, the response spectrum analysis and the transient dynamic analysis, which are based on the modal superposition method, were performed in order to see the structure's response for earthquakes that are likely to happen at the wind turbine installation site.

Keywords: Wind Turbine Tower Design, Wind Turbine Tower Load, Finite Element Analysis, Seismic Analysis, Response Spectrum Analysis, Transient Modal Dynamic (Time-History) Analysis

ACKNOWLEDGMENTS

To my advisors, Dr. Andrew Davol, Dr. Joseph Mello, Dr. Tom Mase, and Dr. Patrick Lemiux, thank you so much for all the guidance and encouragement. I have learned so much from this project and you. The advise you gave was crucial for this work. Working with you has been very inspiring and an honor.

To my coworkers at Simms Machinery and Stellar, thank you for the words of advice and encouragement to get my thesis done. “I know it’s hard, grab a bull by the horn, just do it!” - Mr. Simms.

To my friends, Alexandra Andersson, Katie Grady, and Wojtek ‘Vortex’ Wilczynski, you have been always supportive and cheerful. Thank you. I could not have done it without you. Alexandra, thank you for your inspirational messages and many hours of studying together at Kreuzberg.

To Jae and Maria Kim, Stephane Roussel and Raghav Pancholy, thank you so much for your help and support.

Above all, thank you, Mother, for everything. No words can describe my appreciation to you.

Contents

List of Figures	xii
List of Tables	xix
1 Introduction	2
1.1 With Wind, Life Gets Better	2
1.2 Just a Tower?	4
1.3 Small Wind Turbine (SWT) System	5
1.4 Cal Poly's Small Wind Turbine Project	7
1.4.1 Cal Poly's Small Wind Turbine (SWT) System Overview . . .	7
1.4.2 Project Status	11
1.5 Scope of This Thesis Project	12
2 Design of Wind Turbine Tower	13
2.1 Type of Wind Turbine Towers	14

2.1.1	Free-standing or Guy-wired	14
2.1.2	Lattice Tower	15
2.1.3	Tubular Tower	15
2.1.4	Hybrid Tower	16
2.1.5	Tower Design Trend	16
2.2	Materials for Wind Turbine Tower	17
2.3	Height of Wind Turbine Tower	17
3	Cal Poly's Small Wind Turbine (SWT) Tower	19
3.1	Wind Turbine Site	19
3.1.1	Location	19
3.1.2	Soil and Wind Characteristics of Site	20
3.2	Design Features of Cal Poly's Wind Turbine Tower	21
3.3	Installation Plan	26
4	Wind Turbine Design Load	28
4.1	Introduction	28
4.2	DNV/RISO and IEC Guidelines for Load Determination	30
4.2.1	Load Cases and Load Types	30
4.2.2	Methods for Determining Design Loads	32

4.2.3	Using Aeroelastic Method for Load Calculation	33
4.2.4	Using Simplified Load Equations for Load Calculation	35
4.2.5	Determining Loads From Measurements	41
4.3	Wind Loads Given by Cal Poly	41
4.3.1	Cal Poly's Load Case for Tower Design	41
4.3.2	Calculation of Maximum Thrust	42
4.4	Comparison and Conclusion	43
5	Wind Turbine Tower Modeling for Finite Element Analysis	48
5.1	Finite Element Analyses Overview	48
5.2	Finite Element Model of Wind Turbine Tower	49
5.3	Modeling Assumptions	51
5.4	Modeling Space and Coordinate System	51
5.5	Elements	52
5.5.1	Beam Elements	52
5.5.2	Using Beam Elements	55
5.5.3	Shell Elements	56
5.5.4	Using Shell Elements	58
5.5.5	Using Mass and Rotary Inertia Elements	58

5.6	Joining the Elements	59
5.6.1	Multi Point Constraint (MPC) - Tie	60
5.6.2	Multi Point Constraint (MPC) - Beam	61
5.6.3	Multi Point Constraint (MPC) - Hinge	63
5.7	Boundary Conditions	64
5.8	Material Properties	66
6	Static Analyses of Wind Turbine Tower	68
6.1	Load Case: Installation (Tilt-up)	68
6.1.1	Introduction	68
6.1.2	Assumptions	69
6.1.3	Results	70
6.2	Load Case: Maximum Thrust and Gravity	74
6.2.1	Introduction	74
6.2.2	Results for Thrust Load Applied to x-Direction	75
6.2.3	Results for Thrust Load Applied to y-Direction	79
6.3	Conclusion and Design Loads for Tower Components	83
7	Dynamic Analyses of Wind Turbine Tower	86
7.1	Natural Frequency Extraction	86

7.1.1	Introduction	86
7.1.2	Theory	87
7.1.3	Results	88
8	Dynamic Analysis: Seismic Analysis	95
8.1	Evaulating Seismic Risks	95
8.2	Transient Response (Time-history) Analysis	98
8.2.1	Direct Integration Vs. Modal Superposition Method	98
8.2.2	Input	100
8.2.3	About Damping	102
8.3	Response Spectrum Analysis	108
8.3.1	Introduction	108
8.3.2	How Does It Work ?	109
8.3.3	Response Spectrum as Input Force	111
8.4	Results of Analyses	114
8.4.1	Result of Response Spectrum Analysis	114
8.4.2	Result of Transient Modal Dynamic Analysis	119
8.5	Conclusion	121
9	Conclusion	122

9.1	Analyses Results	122
9.2	Future Work	123
9.2.1	Load Verification	123
9.2.2	Improvement on Finite Element Model	125
	Bibliography	127
	A Mechanical Component Analysis	130
	B Bolt Sizing	140
	C Strength Analysis of Flange Welds	151
	D Finite Element Analysis of Mid-Flange	156
D.1	Model	156
D.2	Results	157
	E Cable Load for Installation	159
	F Anchor Bolt Analysis	163
	G Foundation Plans and Drawings	168

List of Figures

1.1	Traditional Use of Wind Power (From Google Image)	3
1.2	Wind Turbine System with Support Structures (Tower and Foundations)	9
1.3	Rotor Assembly and Blade Profiles for Cal Poly SWT	10
1.4	Cal Poly SWT Nacelle and Rotor Assembly	10
2.1	Common Types of Wind Turbine Towers	14
3.1	Cal Poly Wind Turbine Site	20
3.2	Cal Poly Wind Turbine Tower	22
3.3	Wind Turbine Tower Component: Flange	24
3.4	Wind Turbine Tower Component: Ginpole	24
3.5	Wind Turbine Tower Component: Strut	25
3.6	Wind Turbine Tower Component: Gusset	25
3.7	Wind Turbine Tower Component: Bearing	26

3.8	Tower Tilting Process	27
4.1	Determining Design Load Cases and Load Types	30
4.2	Design Load Cases for Simplified Load Calculation Method [4]	38
4.3	1D Actuator Disc Model for Thrust Calculation (from [11])	42
4.4	Thrust Coefficient, C_T Vs. Induction Factor, a (from [11])	43
4.5	Comparison of Calculated Loads for Load Case H (left), and Load Given by Cal Poly (right)	44
4.6	Azimuth Angle of Blades (ψ) (from [11])	45
4.7	Yawing Moment from Each of Three Blades, $M_{X'}$, on Tower as a Blade Rotate through 360° ($ft - lbf$)	46
4.8	Pitching Moment from Each of Three Blades, $M_{Y'}$, on Tower as a Blade Rotate through 360° ($ft - lbf$)	46
5.1	FEA Model Views of Wind Turbine Tower	50
5.2	Global Coordinate System as Specified in IEC61400-2 (from [4])	52
5.3	Timoshenko Beam (B32) (left), and Euler-Bernoulli Beam (B33) (right) Elements (from [19])	54
5.4	Some of Parts Modeled Using Beam Elements with Cross Section Ren- dered	55
5.5	Modeling Tower Using Beam Model for Finite Element Analysis in Abaqus	56

5.6	S4 Shell Element (from [19])	57
5.7	Mass and Rotary Inertia Elements for Wind Turbine (Green Dot) . .	59
5.8	MPC-Tie between Tower and Flange	60
5.9	Multi-point Constraint (MPC), type Beam, picture from [19]	61
5.10	MPC-Beam between beam (tower) and shell (gusset)	62
5.11	MPC-Beam between Ginpole and Ginpole Plate	62
5.12	MPC-Beam between Strut Ring and Tower Mast	63
5.13	Multi-point Constraint (MPC), Type Hinge, from [19]	63
5.14	MPC-Hinge for Bolted Joints: Strut to Tower Mast (Left); Strut to Ginpole Anchor Plate (Center); Ginpole to Tower Mast (Right) . . .	64
5.15	Support Conditions of Tower	64
5.16	Boundary Conditions	65
5.17	ASTM A572 Stress-Strain Plot (from Google Image)	67
6.1	Illustration of Wind Turbine Installation	69
6.2	Axial Stress, $S11$ in psi (Top) and Deflection in x-direction, $U1$ in $inches$ (Bottom) for Load Case: Installation	72
6.3	Section Axial Force, $SF1$ (top) in lbf , and Section Moment, $SM2$ (bottom) in $in - lbf$ for Load Case: Installation (red line). Note that $SM2$ is bending moment with respect to beam's local y-axis (2-axis, pointing out of the paper)	72

6.4	Reaction Forces (RF) for Load Case: Installation	73
6.5	Load Case: $2k\ lbf$ Maximum Thrust	74
6.6	Axial Stress, S_{11} in psi and Deflection, U_1 in $inches$ for Thrust Applied to x-Direction	77
6.7	Section Axial Force, SF_1 (left) in lbf , and Section Moment, SM_2 (right) in $in - lbf$ for Thrust Applied to x-Direction (red line). Note SM_2 is bending moment with respect to beam's local 2-axis (global y-axis, pointing in/out of the paper)	77
6.8	Reaction Forces, RF for Load Applied to x-Direction	78
6.9	Axial Stress, S_{11} in psi (Left), and Deflection in y-direction, U_2 in $inches$ (Right), for Load Applied to y-Direction	81
6.10	Section Bending Moment, SM_1 in $in - lbf$ (Left) - Note SM_1 is bending moment with respect to beam's local 1-axis (global x-axis); Reaction Forces, RF (Right) for Load Applied to y-Direction	81
6.11	Torsional Moment with Respect to Local Beam's Longitudinal Axis, SM_3 , in $in - lbf$ (Left); Induced Shear Stress, S_{12} , from SM_3 in psi (Right), at Lower Section of Tower	82
6.12	Comparison of Axial Stress, S_{11} , in psi (Left), and Mises Stress, $S, Mises$ in psi (Right), at Lower Section of Tower	82
6.13	Key Locations for Mechanical Design Loads	83

6.14	Moments on Strut and Ginpole, $SM1$, $SM2$, $SM3$, in $in-lbf$ Resulting from Thrust in y-direction. $SM1$: bending moment w.r.t beam's local 1 axis. Blue-to-red stick is the beam's local 1-axis; $SM2$: bending moment due to local 2-axis, in and out of paper or same as global y-axis; $SM3$: torsional moment around the beam's longitudinal axis .	84
7.1	1st Mode Shape of Wind Turbine Tower	90
7.2	2nd Mode Shape of Wind Turbine Tower	90
7.3	3rd Mode Shape of Wind Turbine Tower	91
7.4	4th Mode Shape of Wind Turbine Tower	91
7.5	5th Mode Shape of Wind Turbine Tower	92
7.6	6th Mode Shape of Wind Turbine Tower	92
7.7	7th Mode Shape of Wind Turbine Tower	93
7.8	8th Mode Shape of Wind Turbine Tower	93
7.9	9th Mode Shape of Wind Turbine Tower	94
7.10	10th Mode Shape of Wind Turbine Tower	94
8.1	Zone 4 Earthquake Input from Bellcore Environmental Test Requirement: Acceleration (in/s^2) Vs. Time ($seconds$)	101
8.2	Effect of Damping	107

8.3	Response Spectra per ASCE-7 using USGS software: Blue, dash-dotted line = MCE Spectrum for Site-class B; Red, solid line = Site-Modified (D) MCE Spectrum; Green, dashed line = Site-modified Design Spectrum	113
8.4	Response Spectrum Analysis Results for Site-modified Design Spectrum Applied in x-direction: Axial stress, S_{11} , in <i>psi</i> (Left); x-direction Displacement U_1 , in <i>inches</i> (Right)	116
8.5	Response Spectrum Analysis Results for Site-modified Design Spectrum Applied in y-direction: Axial stress, S_{11} , in <i>psi</i> (Left); y-direction Displacement U_2 , in <i>inches</i> (Right)	116
8.6	Response Spectrum Analysis Results for Site-modified MCE Spectrum Applied in x-direction: Axial stress, S_{11} , in <i>psi</i> (Left), x-direction Displacement, U_1 , in <i>inches</i> (Right)	118
8.7	Response Spectrum Analysis Results for Site-modified MCE Spectrum Applied in y-direction: Axial stress, S_{11} , in <i>psi</i> (Left), y-direction Displacement, U_2 , in <i>inches</i> (Right)	118
8.8	Transient Dynamic Analysis Results Earthquake load Applied in x-direction: Axial stress, S_{11} , in <i>psi</i> (Left); x-direction Displacement, U_1 , in <i>inches</i> (Right)	120
8.9	Transient Dynamic Analysis Results Earthquake load Applied in y-direction: Mises stress, $S, Mises$, in <i>psi</i> (Left); x-direction Displacement, U_2 , in <i>inches</i> (Right)	120
9.1	Recommended Strain-gage Location for Load Measurements	124

D.1	Result of Finite Element Analysis of Mid-flange: Equivalent Stress, $S, Mises$, in <i>psi</i> (Left); Deflection in y-direction, $U2$, in <i>inches</i> (Right)	158
-----	--	-----

List of Tables

1.1	Brief Summary of Cal Poly Wind Turbine Specification	11
3.1	Geographic Coordinates of Wind Turbine Site	20
4.1	<i>Class III</i> Small Wind Turbine Specified by IEC [4]	38
4.2	Parameters for Load Case H Calculations	40
4.3	Calculated Loads for Load Case H	40
4.4	Design Load Given by Dr. Lemieux [8]	43
5.1	Tower Model Parts Beam Element	55
5.2	Material Properties of ASTM A572 Grade-50 High Strength Structural Steel	66
6.1	Design Loads for Foundation Design Obatined from Static Analyses .	85
7.1	Tower Category Depending on Its Fundamental Natural Frequency, f_{n1}	87
7.2	Natural Frequencoes of Wind Turbine Tower	89

8.1	Typical Published Values of Damping for Different Type of Structures	103
8.2	ASCE 7 Site Classification	113

Chapter 1

Introduction

1.1 With Wind, Life Gets Better

Can you imagine a day without wind? Wind does so much for us and we have fun using it: Kite flying, sailing, wind-surfing, hang-gliding or even little breeze that cools one down on hot summer days. Our ancestors used wind for a little more than leisure. For example, there is the windmill, which converts lateral wind movement to rotary motion. It was used to grind the harvested grains, to saw wood or to pump water. Sailing is another example. People have long used wind to travel across the oceans and around the world until the steam engine was invented. It promoted global trading and led to the discovery of “new India”. Nowadays, wind is used for generating electrical power which is an essential element to modern daily lives. Indeed, it is increasingly getting more popular than conventional methods of generating electricity and there are good reasons for it.

Traditional methods of generating electric power are based largely on the use of fossil fuels such as coal, petroleum, or natural gas. One of the main contributors to

Figure 1.1: Traditional Use of Wind Power (From Google Image)

greenhouse gases, carbon dioxide (CO_2), is emitted in large quantity in the use of such energy sources. Many are concerned with steeply inclining CO_2 levels in the atmosphere as it is directly linked to global warming. In an effort to reduce the emission levels, many countries have signed an international agreement called the *Kyoto Protocol*. The Kyoto Protocol is linked to the United Nations framework convention on climate change. The major feature of the Kyoto Protocol sets binding targets for 37 industrialized countries and the European community for reducing greenhouse gas emissions. One of the ways to achieve this goal is to find and develop ways to harness clean alternative energy sources such as wind.

Finding and securing fossil fuels can be a challenge as many other countries want them and many of them are sourced from regions that are not quite politically stable. Foreign dependency on fossil fuels has caused economic chaos before and there is no guarantee that it will not happen again despite many efforts and measures to prevent it. A good example is the 1973 oil crisis when the members of the Organization of Arab Petroleum Exporting Countries or the OAPEC (consisting of the Arab members of OPEC, plus Egypt, Syria and Tunisia) placed an oil embargo in response to the decision of the U.S. to resupply the Israeli military during the *Yom Kippur War*. Wind is abundant in nature and there is no need to compete for it. Therefore, renewable energy sources such as wind power and solar power have emerged as prominent sources

of alternative energy to gradually replace fossil fuels.

We can witness the shift globally. Still, some are skeptical about using wind as an energy source. Let's look at some facts about wind power as an energy source, especially electricity. The wind power usage and capacity are rapidly increasing in the production of electricity. The capacity in electricity production from wind power increased from around 2.5 GW to 94 GW at the end of 2007, which corresponds to an annual growth of 25% [1]. One of the reasons that contributes to the rapid growth of wind power usage is its competitive cost. Wind energy has the lowest overall cost of all renewable energy sources and now it is almost competitive with conventional energy sources, even without environmental credits. It is due to many investments and research for developing the technologies to harness the wind energy. The dramatic decrease in the cost of energy (COE) from wind over the past two decades is due to improvements in aerodynamics, materials, controls systems, electronics, and reliability that reduce maintenance costs [2].

1.2 Just a Tower?

This project is about the wind turbine tower. Why is the research of the wind turbine tower so important? The wind turbine tower plays an important role in further reducing the cost of wind energy. It can be achieved by the optimal design of the tower, and placing the turbine at higher elevations where more wind can be captured.

Minimizing Cost: Among the costs of the wind turbine system, the wind turbine tower cost may constitute as much as 20% of an entire megawatt-scale horizontal axis wind turbine and 10% of the total cost of energy [2]. It costs an additional \$15,000

on average to increase the height by 10 *m*. Optimally designed towers and foundation systems are key to minimize the cost of the wind turbine system. Tower optimization and advanced structural analyses are active research areas.

Challenge of Putting It Higher: The wind turbine tower holds the wind turbine at the necessary elevation and supports all the loads that the wind turbine experiences. There are increasing demands for a higher tower because the wind speed is higher and more uniform at higher elevation (less boundary layer, wind shear). The wind turbine would be able to capture more wind power at higher height. But a higher height results in higher loads imposed on the tower. The challenge is to make a safe structure with a reasonable cost with the additional constraints imposed by transportation and installation.

1.3 Small Wind Turbine (SWT) System

The International Electro-technical Commission (IEC) is the world's leading organization that prepares and publishes international standards for all electrical, electronic and related technologies, collectively known as electrotechnology [3]. Small-scale wind turbines have different characteristics and design needs compared with larger turbines. A need for the standardization and design guidelines for the new class of wind turbine emerged. The IEC established a standard IEC61400-2: Design Requirements for Small Wind Turbine, which deals with safety philosophy, quality assurance, and engineering integrity and specifies requirements for the safety of Small Wind Turbines (SWT) including design, installation, maintenance and operation under specified external conditions. Its purpose is to provide the appropriate level of protection against damage from hazards from these systems during their planned lifetime [4]. Although this standard establishes useful design guidelines, it is slowly adopted by industry

because it is difficult and costly to apply [5]. So, when situations allowed, the IEC standard was employed for a design or an analysis in this project.

According to [4], the Small Wind Turbine (SWT) is classified as having a rotor swept-area smaller than 200 m^2 (16 m in diameter) generating at a voltage below 1000 VAC or 1500 VDC . Additionally, the standard states that if the rotor swept area is greater than 2 m^2 , then all the support structures shall be included as a part of the SWT system. The support structures refer to the wind turbine tower and foundations and they are important structure elements as their dynamic characteristics and load carrying capacities directly impact the performance of the wind turbine.

It is beneficial to look at the current design practices and trends of the small wind turbine system as a whole before discussing the design of specific component. *Wind Energy - The Facts* is a collection of technical articles about wind turbines that were prepared by the leading wind turbine industry experts. The current trend in the design of a typical horizontal-axis SWT is reviewed and summarized [5]:

1. Rotor, blades:

- In general, three blades are standard for the SWT.
- Design towards lower peak operating speed which gives lower noise emission. Typical design rotor-tip speed to wind speed ratio is 5:1.
- Rotor diameter is less than 10 m . Trend is towards larger rotors.

2. Tower:

- Height is between 12 to 24 m , The trend is towards a taller tower, using a steel tubular structure.

3. Generator:

- Synchronous permanent magnet generator
 - Use rare earth permanent magnet rather than ferrite magnet for superior magnetic properties.
4. Regulation control for gust events:
- Use of yawing or furling - The rotor is turned out of the wind passively, by aerodynamic forces
 - Alternative method - mechanical brake, dynamic brake, stall control, pitch control.

1.4 Cal Poly's Small Wind Turbine Project

Cal Poly's engineering departments are taking part in the global trend to research and develop technologies for the wind turbine system. Led by a mechanical engineering professor, Dr. Patrick Lemieux of Cal Poly Wind Power Research Center (CPWPRC), a multidisciplinary group of students and faculty members from mechanical, civil and electrical engineering departments have collaborated to develop the first wind turbine system at Cal Poly.

1.4.1 Cal Poly's Small Wind Turbine (SWT) System Overview

As shown in Figure 1.2, the Cal Poly's SWT system consists of a horizontal-axis wind turbine (HAWT) and its support structures. A HAWT is a wind turbine whose rotor axis is substantially horizontal. The wind turbine is stall-regulated. For a stall regulated system, the blades of the wind turbine rotors are fixed. But the blades are shaped in a way that they are increasingly stalled with the increasing wind speed in

order to protect the turbine from the excessive wind speed and to maximize power output at the same time. It has a passive yawing system; it aligns itself to the wind direction using a tail fin. The wind turbine consists of a three-bladed rotor and a nacelle. The rotor has fiber-reinforced blades and a hub as sub-components. The nacelle has following components: base-structure, yaw-system, fairing, drivetrain, over-speed protection. The support structure consists of a tapered, tubular steel tower which can tilt on bearings and three main pile-foundations which support the wind turbine tower.

Rotor

The rotor is about 12 *ft* in diameter and it consists of an aluminum hub and three blades. The blades are attached to the hub by a set of bolts (Figure 1.3a). The blades are made from carbon-fiber and E-glass material. They are manufactured from a vacuum-infusion method using low-viscosity epoxy-based resin. The suction side and the opposite side of the blades were manufactured separately, then bonded together including an internal steel root flange. The blade is about 6 *ft* in length (span). It has a combination of RISO-A-27 (root) and RISO-A-18 airfoil profile as shown in Figure 1.3b.

Nacelle

The wind turbine is housed by a fairing which is made from the fiberglass. The fairing is strong enough to protect the components of the nacelle from environmental elements. The shape was designed to minimize the boundary layer separation to minimize the drag and possible oscillatory wind induced vibration. Inside the fairing is a base structure which holds the generator, drive system and over-speed protection

Figure 1.2: Wind Turbine System with Support Structures (Tower and Foundations)

mechanism. The base structure is made from steel square tubes which are welded together. The drive train includes bearings, a driveshaft that connects the generator and the rotor hub by a coupling mechanism. A hydraulic disk brake system is used as an over-speed protection system. The generator is the Ginlong Technologies' GL-PMG-3500 wind turbine specific permanent magnet generator. The generator's rated output power is 3500 W at rotational speed of 250 rpm . It will require $110\text{ ft} - \text{lb}$

Figure 1.3: Rotor Assembly and Blade Profiles for Cal Poly SWT

of torque at the rated power [7]. The nacelle and rotor assembly (the wind turbine) is shown in Figure 1.4. A brief summary of the wind turbine specification is given in Table 1.1.

Figure 1.4: Cal Poly SWT Nacelle and Rotor Assembly

Table 1.1: Brief Summary of Cal Poly Wind Turbine Specification

Wind Turbine Specification	
Wind Turbine Category	Small Wind Turbine (per IEC61400-2 [4])
Wind Turbine Type	Horizontal-Axis Wind-Turbine (HAWT)
Wind Turbine Configuration	Stall-regulated, Passive-yawing
Wind Turbine Weight	460 <i>lbf</i>
Number of Blades	3
Rotor Diameter	12 <i>ft</i>
Rated Power	3 <i>kW</i>
Rated Rotor Speed	230 <i>rpm</i>
Rated Wind Speed	22.4 <i>mph</i> (10 <i>m/s</i>)
Rotor Tip to Wind Speed Ratio, λ	4
Cut-off Wind Turbine Speed	300 <i>rpm</i> (Wind Speed of 30 <i>mph</i>)
Stopping Mechanism	Disk Brake

Support Structures

The support structures consist of a 70 *ft* (21.34 *m*) tiltable tapered tubular tower and two main pile-foundations which support the main mast and two support pile-foundations which can be seen in Figure 1.2. The tower is made of ASTM A572 structural steel and is galvanized to resist corrosion. The foundation consists of cylindrical piles which are made from reinforced concrete. The turbine has a rotor swept area of 10.5 *m*². Thus all the support structures need to be included in the system design according to the IEC standard [4].

1.4.2 Project Status

The wind turbine nacelle was designed by a group of undergraduate mechanical engineering students for their senior projects [8]. The rotor, which consists of carbon-fiber blades and an aluminum hub, was developed and manufactured by two mechanical engineering graduate students [9]. The foundation design and site soil analysis were

done by a group of civil engineering students and faculty members and private companies who specialize in geotechnical engineering [10]. A mechanical engineering student is currently working on the computational fluid modeling of the wind turbine site area to determine the best spot to install future wind turbines.

1.5 Scope of This Thesis Project

Among other components of Cal Poly's small wind turbine system, this project focuses on the structural analyses of the wind turbine tower. The initial design of the tower was completed by the CPWPRC already. Much of the work presented in this project verified the structural integrity of the tower under various load cases. Various structural analyses of the tower using the finite element analysis and its mechanical components were performed and reported.

Chapter 2

Design of Wind Turbine Tower

What are the functions of a wind turbine tower? The most obvious one is that it places the wind turbine at a certain elevation where desirable wind characteristics are found. It houses many electrical components, connections and the control protection systems and provides access area to the wind turbine [2]. Most importantly, the wind turbine tower supports the wind turbine (a nacelle and a rotor) and carries the loads generated from the turbine. The structural properties of the wind turbine tower are very important as the property such as tower stiffness has a big influence on the performance and structural response of the wind turbine.

As the cost of the tower is a significant portion of the overall wind turbine system cost, they are often optimized in order to minimize the cost of the tower. In certain cases, the structural optimization is performed to meet certain stiffness requirements (frequency-driven) while minimizing the cost. In other cases, static strength of the tower is the main driver (strength-driven) for the optimization. In all cases, the design optimization is a function of many design variables. In this chapter, we look at some of the important design factors that are crucial for the wind turbine tower

optimization: type, material, and height of the wind turbine tower.

2.1 Type of Wind Turbine Towers

Figure 2.1 shows three types of wind turbine towers that are widely seen for the horizontal-axis wind turbines: *lattice tower*, *tubular tower* and *hybrid tower* [11].

Figure 2.1: Common Types of Wind Turbine Towers

2.1.1 Free-standing or Guy-wired

Each of the aforementioned types of towers can be made free-standing or guy-wired. Use of guy wires may bring down the initial cost of the tower as it would require less tower material. Many small wind turbines use guy wires for this reason, however the maintenance costs for the guy wires add more costs to the operation, thus should be avoided if possible [12]. Additionally, the guy wires require a larger footprint and additional foundations. Therefore, it may present a problem with land accessibility and usage which may not be suitable in farm areas.

2.1.2 Lattice Tower

The lattice tower is made from trusses or frames that are bolted or welded together as shown in Figure 2.1a. In general, the initial cost to build a lattice tower is less than the tubular tower because it requires less material for similar stiffness. Although the initial material cost may be lower for the lattice tower, the assembly and maintenance costs may be higher as each bolt needs to be torqued to a specification and checked periodically. Unlike the tubular tower, the base dimension is not restrained by the section size limit (see Tubular Tower). Additionally, the foundation cost may be less than the tubular tower because inexpensive pile foundations at each frame foot can be used. Aesthetically, it is less appealing which is an important factor.

2.1.3 Tubular Tower

The tubular tower is shown in Figure 2.1b. It has a pipe cross-section. Many tubular towers are either tapered (conical) or stepped which has increasing diameter towards the bottom. In addition, for stepped tubular towers, the thickness of the wall can be varied along the height of the tower in order to save the material while satisfying the structural requirements. Bolted flanges are commonly used to join the sections of the tower and to secure the tower to the foundation. Many tubular towers use a gravity-based “matt” foundation which is more expansive than a pile foundation.

The tubular tower has many advantages over the lattice tower. The enclosed area of the tubular tower cavity is useful. First, it provides a covered, protected area for climbing to access the wind turbine in bad weather conditions. Also, it provides a covered area that can house many electrical components. In a cold climate, windy or wet area, this is an important feature. It provides a certain level of security by limiting the access to the turbine unlike the lattice tower. Additionally, it is more

maintenance friendly. Although the initial material cost may be higher than the lattice tower, it does not rely on many bolted-connections which need to be torqued and checked periodically. Aesthetically, it is more appealing than the lattice tower [11]. European countries have always favored tubular towers for aesthetic reasons [13]. However, for very large wind turbines, transportation may be a challenge. The sections of the tubular towers are manufactured and then assembled on the wind turbine site. The current limitation of the tubular section size is 4.3 *m* in diameter [2].

2.1.4 Hybrid Tower

The hybrid tower combines different configurations of the wind turbine tower. The hybrid tower shown in Figure 2.1c is an example. It is called *stayed design* which combines steel tube, steel truss and guy-wire. The name came from the analogy of the support method of a sailing ship mast. For this design, the guy wires can be used without requiring additional foundations. As mentioned the tubular type has a major section size limitation of about 4.3 *m* for larger turbines. This limit may result in site welding and fabrication which may compromise the quality of the tower. Transporting the larger diameter sections may be very costly as well. The *stayed* design may overcome the section size limitation while making use of the tubular tower advantages [2].

2.1.5 Tower Design Trend

Presently, the most common type is the free-standing tubular tower for large wind turbines and it is expected to be more popular in the future. For small wind turbine towers, the use of guy-wires on the tubular tower is widely seen [14].

2.2 Materials for Wind Turbine Tower

The two most common materials used for wind turbine towers are structural steel and reinforced concrete. Use of steel is more popular between the two materials. In developed countries which have higher labor costs, it is more standard to use steel [2]. Steel is also galvanized or painted to protect it from environmental damage such as corrosion [11].

2.3 Height of Wind Turbine Tower

The height of the tower is a site-dependent parameter because it is tailored to the wind characteristics of the site. The design optimization for the least cost could favor tall towers in low wind areas and shorter towers in high wind areas. However, if there are obstacles such as trees or tall objects that may make the wind flow more turbulent, taller tower will be required. In addition, tall towers may prevent the turbine from the effect of wind shear if the site has frequent wind shear occurrence.

In general the taller the tower, the more power the wind system can produce. According to [11], the tower height is to be at minimum 20 *m* (66 *ft*). Favorably, the tower height should be higher than 24 *m* (78 *ft*) because the wind speed is lower and more turbulent close to ground.

With relation to the rotor size, it is typically 1 to 1.5 times the rotor diameter for a normal horizontal axis wind turbine. For a small wind turbine, the tower height to rotor diameter ratio is a lot higher in order to benefit from the higher wind speed at higher height [11].

Indeed, the demand for the taller tower is currently increasing. The challenges that

come with the design of the taller tower are to make it structurally safe at a reasonable cost while meeting requirements imposed by the transportation and installation. The tall tower needs large cranes for the installation and it can be difficult to transport. The use of cranes may not even be possible depending on the installation location. The idea of a self-erecting tower was evaluated and showed some potential to resolve this issue for tall towers [15]. Self-erecting towers use mechanisms such as telescoping mechanisms, tower-climbing devices, A-frame or jacking-up systems.

Chapter 3

Cal Poly's Small Wind Turbine (SWT) Tower

Cal Poly's wind turbine tower was designed by the CPWPRC for a 3 *kW* horizontal-axis wind turbine. It will be installed on a ranch a few miles from the Cal Poly campus. The soil at the site was characterized by a local geotechnical engineering company. Currently, wind is measured at the site for an extended period of time. The main design features of Cal Poly's wind turbine tower and its components are summarized and described in this chapter.

3.1 Wind Turbine Site

3.1.1 Location

Cal Poly's small wind turbine is to be installed on the Escuela Ranch which is located a few miles north of the Cal Poly campus. The site is an unpopulated area, which

is currently utilized as a part of a cattle ranch by the College of Agriculture. The prevailing wind comes from an approximate north-west direction (figure 3.1b). The exact location is shown in Figure 3.1 and the geographic coordinates are specified in Table 3.1.

Figure 3.1: Cal Poly Wind Turbine Site

Table 3.1: Geographic Coordinates of Wind Turbine Site

Longitude	$N35^{\circ}21'7.2''$
Latitude	$W120^{\circ}43'36.8''$

3.1.2 Soil and Wind Characteristics of Site

A local geotechnical engineering company, *Earth Systems Pacific*, performed a soil analysis for the wind turbine site. *ASCE-7: Minimum Design Loads for Buildings and Other Structures* contains a seismic provision in which it specifies the methods to perform the seismic analysis for a structure. The soil analysis results showed that the site can be categorized as *Site Class D - Stiff Soil* per ASCE-7's Site Classification. The site class information is used for obtaining a site-specific (earthquake) design spectrum for seismic analysis. It will be discussed in a later section in detail. The

results of the soil analysis were used in the design of the foundation as well. The type of foundation is strongly influenced by the soil condition.

Just as for soil, the wind characteristics are site-specific. Wind is currently measured using anemometers installed on the site. They are installed on a measuring tower and spaced at equal vertical distances along the tower. As a preliminary analysis, the annual average wind speed of San Luis Obispo and several surrounding locations were reviewed. They are readily available from weather station records. The average wind speed was about 3.14 m/s (7 mph). It was averaged over 10 years from 1996. In the late spring months, however, the average wind speed increased to 5 to 6 m/s (about 12 mph).

3.2 Design Features of Cal Poly's Wind Turbine Tower

The wind turbine tower is shown in Figure 3.2 highlighting some of the main components. The main design features of the tower are summarized and explained:

- Material: ASTM-A570 grade-50, high strength steel; galvanized
- Main structure: tapered, tubular tower
- Tilttable on sleeve bearings
- Permanently installed ginpole and strut
- Gussets (to lower stresses and increase stiffness)
- Two-piece construction

Figure 3.2: Cal Poly Wind Turbine Tower

The tower and all of its components are made from ASTM-572 grade-50 high-strength steel. The material has a higher yield point compared to the more common structural steel, ASTM-A36. In addition, all steel components are galvanized to resist corrosion. The tower is a tubular-type tower as it is simpler to construct and aesthetically more pleasing than a frame-type tower. The tower mast is tapered, having a larger cross-sectional diameter at the bottom and smaller diameter at the top in order to save material and reduce the tower weight.

One of the main design objectives for the tower design is easy access to the wind turbine (nacelle and rotor). This was achieved by a tilting feature. The tower tilts on the two bearings located at the bottom of the tower (Figure 3.7). The ginpole and strut (Figure 3.4, Figure 3.5) are permanently attached to the lower section of the tower by lugs and bolts. The main purpose of these members is to raise and lower the tower but they also provide structural support as well after installation when the turbine is operating. Four gussets (Figure 3.6) are welded at the bottom of the tower in order to lower stresses and increase stiffness in the direction where the ginpole and strut cannot provide the structural support. They are perpendicular to the longitudinal axis of the ginpole and the prevailing wind direction. The need for structural support in the direction is small since the wind is not likely to blow in that direction.

In order to make transportation easy, the tower is built in two sections. The sections of the tower are transported to the installation site, and joined by a bolted flange (Figure 3.3). The wind turbine is placed on top of the tower and secured using a bolted flange as well.

Figure 3.3: Wind Turbine Tower Component: Flange

Figure 3.4: Wind Turbine Tower Component: Ginpole

Figure 3.5: Wind Turbine Tower Component: Strut

Figure 3.6: Wind Turbine Tower Component: Gusset

Figure 3.7: Wind Turbine Tower Component: Bearing

3.3 Installation Plan

One of the main features of the tower is the tilt-on-bearing feature which allows easy access to the nacelle and rotor. The idea is to use a truck which has a high capacity winch to raise/lower the tower without any special equipment (such as a crane). Two anchors are installed on the ground 75 *ft* away from the tower. The front and rear end of the truck are held by a set of anchors. A high strength cable is attached at the end of ginpole-strut assembly and pulled using the truck's winch. Figure 3.8 illustrates the tilting concept.

Figure 3.8: Tower Tilting Process

Chapter 4

Wind Turbine Design Load

4.1 Introduction

A structural analysis begins with determination of loads. Let's look at some of the characteristics of the loads for the wind turbine design.

Comprehensive Load Cases: Design load cases for the wind turbine are comprehensive in order to ensure safe operation. They cover all combinations of operating situations and external conditions that the wind turbine system may experience with reasonable probability for the life of the turbine.

Obtaining accurate loads is difficult: Major parts of the design loads stem from an external condition, wind. However, determining accurate wind loads can be difficult because of the characteristics of wind. Wind is time-varying and random in nature (turbulence, gust). The extreme wind speed of a 50-year occurrence period, for example, at a specific location is at most a guess from statistical extrapolation. And wind speed and direction vary greatly depending on the location. Luckily, industry

standards and guidelines exist that provide methods to calculate accurate design loads, not just the wind loads but all the loads that can result from all the load cases.

DNV/RISO Guidelines for Design of Wind Turbines [14] and *IEC 61400-2, Design requirements for small wind turbines* [4] contain good guidelines for determining the design loads for wind turbine design. They cover comprehensive lists of design load cases, and provide detailed methods to calculate different types of loads. The DNV/RISO guidelines provide information which is obtained from a combination of specifications and studies from the wind turbine industry and research agencies for all classes of wind turbines. The IEC 61400-2 is unique in that it was prepared specifically for small wind turbines.

In order to explain the wind turbine loads that are relevant to the tower (some load cases have little or no relevance or a small effect on tower loads), both sources are reviewed and briefly discussed in this chapter. The load that would create the maximum bending load is of interest in order to perform the ultimate strength analysis. Using the IEC's simplified load equation method, some design loads that are important to Cal Poly's wind turbine tower were calculated. Design loads provided from the Cal Poly team are also summarized, and then compared with the calculated IEC's simplified method results. It turned out that the design load for Cal Poly's wind turbine turned out to be more conservative than the load calculated from the simplified method. Therefore, the more conservative load, Cal Poly's design load, was used as a basis for the ultimate strength analysis of the wind turbine tower.

4.2 DNV/RISO and IEC Guidelines for Load Determination

Load determination methods presented in the *DNV/RISO Guidelines for Design of Wind Turbines* [14] and *IEC 61400-2: Design requirements for small wind turbines* [4] are reviewed and summarized. It is beneficial to understand them in order to get accurate tower loads.

4.2.1 Load Cases and Load Types

A wind turbine system must be analyzed for various loads that it will experience during its design life including the behavior of the control system and protection system, such as braking or pitch-regulation. The first step in determining the design loads is to define the load cases.

Design load cases are created by combining relevant *design situations* with various *external conditions* as illustrated in Figure 4.1.

Figure 4.1: Determining Design Load Cases and Load Types

Design situations refer to various operating conditions of wind turbines. Usually they are categorized into two: operational and temporary conditions. Operational conditions are normal operations such as: power production conditions, idling, cut-

in, cut-out, or standing-still condition. Temporary conditions include cases such as transportation, installation, fault, repair/testing [14].

External conditions usually refer to wind and environmental conditions/events where the turbines are installed. They can be categorized as normal or extreme conditions [14].

In general, the following lists of combinations are considered as load cases for small wind turbines [4]:

- Turbine operation without fault + normal external conditions (i.e., wind at design wind speed) or extreme external condition.
- Turbine at fault + appropriate external conditions
- Maintenance, transportation, installation + appropriate external conditions

For each design load case, then, different types of loads are considered. Each type of load can be categorized from its source. A combination of different types of loads is applied to the relevant components of the wind turbine system for design.

- Aerodynamic loads that result from wind: drag, lift force
- Inertia loads that result from gravity, rotation, vibration, or gyroscopic effects
- Functional loads from transient operation conditions of turbine such as braking, yawing, blade pitching, or transmitting power to generator: braking torque, yawing moment, blade pitching moment
- Other loads resulting from other environment sources such as wave, current, ice, and earthquake

4.2.2 Methods for Determining Design Loads

Three methods were suggested for finding design loads [4]:

- Aeroelastic Modeling
- Simplified load equations
- Data measurements/extrapolation

In the following sub-sections, some of the methods for determining loads were reviewed and summarized. The aeroelastic modeling seems to be a popular method in the wind turbine industry for determining design loads. With relatively inexpensive computational costs these days, this method calculates the most realistic loads all the way down to the wind turbine structure's component level. It accounts for the complicated flow patterns of the wind turbine blades, and even includes effects of the control, protection system of the wind turbine. For seismic analysis, it can calculate the loads for the wind turbine components resulting from earthquake loading. IEC's simplified equations provide methods to calculate the loads only using the key wind turbine parameters. It is a simple and economical method of obtaining loads but load verification is required. Finally, loads can be directly measured and extrapolated from the measured data.

For the Cal Poly's tower, the IEC's simplified load-equation method was used to determine the ultimate load for static analysis.

4.2.3 Using Aeroelastic Method for Load Calculation

Aeroelastic Modeling

An aeroelastic method specified in the standards employs computer codes specially developed for load calculation on wind turbine systems. A well-recognized program is *HAWC2* and it was developed by RISO laboratory. *HAWC2* is a tool for simulating wind turbine system response in the time domain. It was started as a modified non-linear finite element analysis based program but the most recent version is based on a multibody formulation applied to a six-degree of freedom (three translational, three rotational) Timoshenko beam element. The program is continuously improved and modified. Its goal is to simulate all loading conditions and loads for a wind turbine system all the way down to the wind turbine component level. Other notable aeroelastic modeling software for wind turbines are *GL-BLADED*, *FLEX5*, and *FAST*, which was developed by NREL.

Structure Modeling

For *HAWC2* (and for many other programs as well), the structural modeling of a wind turbine system is accomplished by multibody formulation. The multibody formulation models each body (for example, blades) using a set of Timoshenko beam elements. Then, a component (turbine nacelle or tower, for example), which is composed of one or more bodies, is created with its own reference frame. Each body is constrained using algebraic coupling such as joints, fixed relative positions, or controlled position. A body itself assumes a small deformation and rotation as it uses linear formulation element. However non-linearity can be simulated: Large deformation or rotation can be accounted for in the coupling points as deformation states

of last nodes are passed to next body reference frame; and forces are applied in the deformed state. This multibody modeling gives great flexibility by breaking complex system into components.

Modules

The aeroelastic model attempts to incorporate all possible factors that affect loading including earthquake, aerodynamic effects, rotor dynamics, soil structure interaction, electrical system dynamics, and others. In order to achieve this goal, individual modules that simulate aerodynamics, turbine's control systems, or soil conditions are developed (currently modified and new features are added). The modules are linked to the structural model; the calculated results from each module are transferred to the model. They provide realistic, time-varying effects which are incorporated into the structure characteristics such as mass (M), damping matrices (C), etc. For example, for the aerodynamic module, both deterministic (shear, gust) and stochastic (turbulence and wakes) wind components are modeled using various physical models to describe realistic wind conditions. With specified blade geometry, the module calculates the aerodynamic forces on the blades. Then the forces are applied to the structure accounting for complex flow patterns such as wake. Additionally, gyroscopic effects which result from the rotation of the rotor are applied to the structure's equation of motion modifying the damping characteristic of the structure.

Verification and Conclusion

The calculated loads from the aeroelastic analysis are subject to verification [14], [4]. According to load validation studies, the loads obtained from the aeroelastic analysis agree well with the measured loads. RISO developed the aeroelastic models for Ves-

tas V39 and Nordtank 500/37, which are pitch- and stall-controlled horizontal-axis wind turbines respectively, to investigate the load sensitivity to wind and turbulence parameters for a complex terrain. They performed load validation studies of the aeroelastic models and concluded that the measured load data and the loads obtained from the aeroelastic model have a good agreement [16], [17]. Because of the accurate load prediction capability of the aeroelastic analysis, the IEC specification allows use of lower safety factors on the loads calculated from the aeroelastic analysis [4]. If the load measurement is difficult for a wind turbine, the load verification can be performed by comparing the calculated loads of a wind turbine to the measured loads obtained from similar-size wind turbines that have similar configurations.

Nowadays, computational resources are readily available. Thus, the aeroelastic method for determining load is the most common in the wind turbine industry [14]. However, for small wind turbines and prototypes such as our project in which not much wind data or load verification data are readily available, this method can pose some challenges. For low budget wind turbines, such as small wind turbine systems, the cost of computation may actually be a significant portion of the overall cost and may not be justifiable unless the system is manufactured for large quantities.

4.2.4 Using Simplified Load Equations for Load Calculation

Introduction

IEC 61400-2 provides a simple method for determining loads for small wind turbines if certain conditions are met. The method is simple and thorough, covering most of the load cases that a small wind turbine would experience for its design life. It is simple because it only requires basic wind turbine parameters that can be easily measured from the prototype wind turbine. Therefore, this method provides a good starting

point to get the design loads for an initial component design. This method provides loads at specific turbine locations such as blade root or shaft of the wind turbine (but not the tower itself). The loads determined from this method are supposed to be conservative [18].

The wind measurement at the wind turbine installation site is on-going and some of the turbine's characteristics still need to be measured and verified. Using some of the known parameters of Cal Poly's wind turbine, some of the loads were calculated per IEC's simplified load equation method. The selected loads were determined to be the worst load case for the tower for the ultimate strength analysis.

Wind Turbine Requirements for Simplified Equation Method

In order to use the simplified equations, the wind turbine must meet the following criteria:

- Horizontal axis wind turbine
- 2 or more blades
- Cantilevered Blade
- Have rigid hub (not teetering or hinged hub)

The criteria apply to all downwind or upwind; variable or constant speed; active, passive, or fixed pitch; and furling or no furling turbines. Cal Poly's wind turbine meets the specified requirements (see specification described in Chapter 1), thus the simplified load equation method can be used to calculate the loads.

Small Wind Turbine Classes and Wind Speeds

Small wind turbine classes are categorized according to wind speed and turbulent parameters in the IEC standard [4]. It was assumed that our wind turbine was the *Class III* small wind turbine, and corresponding wind speeds were used for load calculations. Class III turbines can withstand wind conditions of $V_{ref} = 37.5m/s$ (83.9mph) and more. V_{ref} is a reference wind speed and it is a basic parameter for wind speed used for defining wind turbine classes. The average wind speed, V_{ave} , and V_{ref} for Class III SWT are given by the standard and shown in Table 4.1. From V_{ref} , V_{e50} and V_{e1} were scaled using Equations 4.1 and 4.2. They are expected extreme wind speeds (averaged over 3 seconds) with a recurrence time interval of 50 years and 1 year respectively. The design wind speed, V_{design} , was defined using Equation 4.3.

$$V_{e50} = 1.4V_{ref} \text{ (at hub height)} \quad (4.1)$$

$$V_{e1} = 0.75V_{e50} \quad (4.2)$$

$$V_{design} = 1.4V_{ave} \quad (4.3)$$

Design Load Cases

Load cases for the simplified load equation method are provided by IEC as shown in Figure 4.2.

For each design load case, different types of loads are calculated using the given

Table 4.1: *Class III* Small Wind Turbine Specified by IEC [4]

SWT Class	Wind Speed, m/s (mph)
V_{ref}	37.5 (83.9)
V_{ave}	7.5 (16.8)
V_{e50}	52.5 (117.4)
V_{e1}	39.4 (88.1)
V_{design}	10.5 (23.5)

Design situation		Load cases	Wind inflow	Type of analysis	Remarks
Power production	A	Normal operation		F	
	B	Yawing	$V_{hub} = V_{design}$	U	
	C	Yaw error	$V_{hub} = V_{design}$	U	
	D	Maximum thrust	$V_{hub} = 2.5V_{ave}$	U	Rotor spinning but could be furling or fluttering
Power production plus occurrence of fault	E	Maximum rotational speed		U	
	F	Short at load connection	$V_{hub} = V_{design}$	U	Maximum short-circuit generator torque
Shutdown	G	Shutdown (braking)	$V_{hub} = V_{design}$	U	
Parked (idling or standstill)	H	Parked wind loading	$V_{hub} = V_{e50}$	U	
Parked and fault conditions	I	Parked wind loading, maximum exposure	$V_{hub} = V_{ref}$	U	Turbine is loaded with most unfavourable exposure
Transport, assembly, maintenance and repair	J	To be stated by manufacturer		U	

Figure 4.2: Design Load Cases for Simplified Load Calculation Method [4]

equations in the specifications. Simple equations for calculating the moments at the root of the blades, bending moment and torque on the shaft, and axial load along the length of the shafts were commonly found. None of the loads are given for the tower specifically except for the load case J: transportation, assembly.

Loads for Design of Tower

Which load cases produce the largest load for the wind turbine tower? For the tower, the horizontal axial-load that acts along the turbine's shaft produces the largest load as it creates a bending moment at the lower parts of the tower. The braking torque and moments applied to the shaft due to power generation and other conditions are not significant compared to the tower bending moment. Thus, for static ultimate strength analysis, the largest bending moment for the tower needs to be found. According to [14], the following list of load cases usually produces such load:

1. Stand-still + 50-year recurrence wind speed
2. Fault + High wind speed
3. Normal operation + Near cut-off wind speed

The first case in the list corresponds to the load case H, the second case to the load case I, and the third to the load case D of the load case table shown in Figure 4.2.

In our case, with V_{e50} of 117 *mph*, the load case H (first case in the list) produced the most significant bending moment at two locations of the tower amongst all other load cases listed in Figure 4.2. Wind speed has the largest effect on the aerodynamic loads as the speed is squared in the load calculation (see Equation 4.4, 4.5). Case H uses the largest wind speed for the calculation.

For the load case H: *Parked Wind Loading*, aerodynamic loads from extreme wind speed (V_{e50}) is calculated. Horizontal drag force of the rotor applied to the shaft, $F_{x-shaft}$, is calculated using Equation 4.4 for the non-spinning rotor as our turbine's cut-off speed is reached before the wind speed reaches V_{e50} . This equation assumes drag coefficient, C_d , of 1.5. Aerodynamic forces on the tower and nacelle are calculated

using Equation 4.5. Force coefficients, C_f , which takes drag and lift into account for different body shapes, were used in the calculation.

$$F_{x-shaft} = 0.75B\rho V_{e50}^2 A_{proj,B} \quad (4.4)$$

$$F_{body} = C_{f,body} \frac{1}{2} \rho V_{e50}^2 A_{proj,body} \quad (4.5)$$

Loads for the load case H were calculated for Cal Poly's turbine. The parameters that were used in the calculation are summarized in Table 4.2. All calculations were carried out in metric units. The calculation results are summarized in Table 4.3.

Table 4.2: Parameters for Load Case H Calculations

Symbol	Description	Value
ρ	Density of Air, Standard	1.225 kg/m^3
$A_{proj,B}$	Projected Area of Blade	0.418 m^2
$A_{proj,tower}$	Projected Area of Tower	5.42 m^2
$A_{proj,nacelle}$	Projected Area of Nacelle	0.393 m^2
B	Number of Blade	3
$C_{f,tow}$	Force Coefficient, Tower	1.3
$C_{f,nac}$	Force Coefficient, Nacelle	1.5
R	Rotor Radius	1.829 m

Table 4.3: Calculated Loads for Load Case H

Symbol	Description	Value, N (lbf)
$F_{x,shaft}$	Drag force on parked rotor applied to shaft	3175.5 (713.9)
$F_{x,tower}$	Combined force (lift, drag) on Tower	11895.8 (2674)
$F_{x,nacelle}$	Combined force on Nacelle	995.2 (223.7)

Note that the load case H, as calculated above, turned out to be very conservative according to the study, *Tower Design Load Verification on a 1-kW Wind Turbine* sponsored by the National Renewable Energy Laboratory (NREL) [18].

4.2.5 Determining Loads From Measurements

Loads can be measured directly and used for all or a particular load case [4]. Measurement processes and guidelines are given in the IEC specification for accurate measurement. Enough data need to be collected to perform statistical analysis. This method is good in that the load verification is not necessary but it is time-consuming and costly.

4.3 Wind Loads Given by Cal Poly

A set of loads for the Cal Poly's wind turbine tower was calculated using the IEC's simplified load equation method. This was the extreme load case which gave the largest load for the tower. Dr. Patrick Lemieux (Cal Poly) calculated the extreme thrust load as well but used a slightly differently approach than the IEC's method.

4.3.1 Cal Poly's Load Case for Tower Design

Cal Poly's load case is similar to the load case D: *Maximum Thrust* of the IEC's simplified method, but with an increased cut-off speed of 60 *mph* (30 *mph* is the actual cut-off speed) and ideal thrust coefficient, C_T , of 2.

4.3.2 Calculation of Maximum Thrust

The maximum thrust was calculated using Equation 4.6.

$$T = C_T \frac{1}{2} \rho \pi R^2 U^2 \quad (4.6)$$

where T is the thrust; C_T is thrust coefficient; ρ is density of air; R is radius of rotor; and U is free stream velocity.

The equation 4.6 is derived by applying one-dimensional momentum theory to an “actuator disk” in a “stream tube” as illustrated in Figure 4.3 [11]. As seen in Figure 4.4, for the Betz turbine model (dashed-line), maximum C_T is 1 occurring at induction factor, a , of 0.5. After $a = 0.5$, this model is invalid to predict C_T . After $a = 0.5$, Glauert Empirical Relations is used (solid-line in Figure 4.4). In reality, the complicated flow (turbulent wake state) may drive C_T as high as 2 as shown in the Figure 4.4 [11]. Note that $a = 1$ means that the downwind wind speed (U_3 in Figure 4.3) is zero after the wind goes through the rotor which is not likely to occur in the operation. Thus the corresponding $C_T = 2$ is a conservative estimate.

Figure 4.3: 1D Actuator Disc Model for Thrust Calculation (from [11])

A maximum possible value of $C_T = 2$ at induction factor, $a = 1$ was used for the thrust calculation. Density of air at the standard condition, $\rho_{air} = 1.229 \text{ kg/m}^3$, was

Figure 4.4: Thrust Coefficient, C_T Vs. Induction Factor, a (from [11])

used, and the increased cut-off wind velocity of 26.82 m/s (60 mph) was assumed. Again, the actual cut-off wind speed is about 30 mph .

The resulting loads are summarized in Table 4.4.

Table 4.4: Design Load Given by Dr. Lemieux [8]

Load Type	Load
Max. Thrust Load on Shaft	2100 lbf
Max. Torque	$150 \text{ ft} - \text{lbf}$

4.4 Comparison and Conclusion

Various methods of determining load cases and different types of loads were reviewed. Some of the loads that are relevant to the tower were reviewed. For Cal Poly's tower, load case H induced the largest loads on the tower among other cases considered in the simplified equations. Note that the case H is very conservative according to NREL's load verification study. Dr. Patrick Lemieux (Cal Poly) also calculated the extreme load using a slightly different method and assumptions than the IEC's method.

Now the two loads are compared. The calculated thrust loads were converted to the

bending moments at two locations of the tower for comparison, just above the strut attachment and the bottom of the tower. As shown in Figure 4.5, the total combined lateral loads for the load case H are higher but Cal Poly's load induced more bending moment on the tower. Therefore, Cal Poly's calculated load was used for the ultimate strength analysis of the tower.

Figure 4.5: Comparison of Calculated Loads for Load Case H (left), and Load Given by Cal Poly (right)

How do moment loads on blades affect the tower?

X' , Y' , and Z' represent the tower coordinate system where the X' -axis is along the tower, the Z' -axis is along the axis of rotor rotation, and the Y' -axis is perpendicular to both X' and Z' . For example, if the tower coordinate system is placed on a nacelle, a motion around the X' -axis is yawing, the Y' -axis is pitching-backwards and the Z' -axis is rolling of the nacelle. ψ is the Azimuth angle of the blade with respect to the tower (Figure 4.6).

M_β is the flapwise bending moment at the root of a blade. It can be calculated using

Figure 4.6: Azimuth Angle of Blades (ψ) (from [11])

Equation 4.7. The thrust, T , is 2100 *lbf* for our case; B is the number of blades (3); and R is the radius of the turbine rotor (6 *ft*). M_β (2800 *ft – lbf*) is, then, transferred to the tower coordinate using Equation 4.8, and Equation 4.9 which are called the yawing moment and the pitching moment respectively.

$$M_\beta = \frac{2}{3} \frac{T}{B} R \quad (4.7)$$

$$M_{X'} = M_\beta \sin(\psi) \quad (4.8)$$

$$M_{Y'} = -M_\beta \cos(\psi) \quad (4.9)$$

The yawing moment, $M_{X'}$, and the pitching moment, $M_{Y'}$ resulted from M_β of each blade are transferred to the tower; they are plotted as shown in Figure 4.7 and 4.8. As seen in Figure 4.7 and Figure 4.8, the moment loads created from a blade is canceled out by the other two moment loads for both $M_{X'}$ and $M_{Y'}$. The blades are located at an equal angular distance from each other, resulting zero net-moment when combined (the sum of moment vector is zero).

Figure 4.7: Yawing Moment from Each of Three Blades, $M_{X'}$, on Tower as a Blade Rotate through 360° ($ft-lbf$)

Figure 4.8: Pitching Moment from Each of Three Blades, $M_{Y'}$, on Tower as a Blade Rotate through 360° ($ft-lbf$)

The torsional load (or moment) along the generator shaft, M_ξ , is calculated using Equation 4.10. P is power produced by the turbine, which was assumed as 3 kW in our case. W is the angular speed of the turbine (230 rpm). The calculated M_ξ is 41.5 N-m per blade. The total M_ξ on the tower is about 125 N-m (92 ft-lbf), which is small compared to the bending moment of the tower. Note the maximum M_ξ given in Table 4.4 which is larger than the calculated M_ξ using Equation 4.10.

But, it is still small compared to the tower bending moment resulting from the thrust load. Therefore, the main static load of our concern is the thrust load, the weight of the turbine and the tower itself.

$$M_{Z'} = M_{\xi} = \frac{(P/W)}{B} \quad (4.10)$$

Chapter 5

Wind Turbine Tower Modeling for Finite Element Analysis

5.1 Finite Element Analyses Overview

Various structural analyses of Cal Poly's wind turbine tower were performed using the finite element method. All the analyses were performed using *Abaqus* which is a suite of engineering simulation programs based on the finite element method. The wind turbine tower including the nacelle assembly was modeled using beam, shell and inertia elements in Abaqus. The model was used to perform static and dynamic analyses of the wind turbine tower. For static analysis, the displacements, reaction forces and stresses of the tower structure under the static loads (not time-varying loads such as maximum thrust load) were calculated. Dynamic analysis, in our case, consists of a modal analysis, response spectrum analysis and transient dynamic (time-history) analysis. In modal analysis, a set of undamped natural frequencies and mode shapes of the tower structures were calculated. The results of modal analysis were

used as a basis for the response spectrum analysis and the transient response analysis. The two analyses were performed in order to study the structural response of the tower from a time-varying transient load: earthquake.

5.2 Finite Element Model of Wind Turbine Tower

As shown in Figure 5.1, the wind turbine tower is a slender structure. Its main components such as the tower mast, the ginpole and the strut have large longitudinal dimensions such as a length, compared to the cross sectional dimensions such as a diameter. The tower is exposed to the large bending moments which induce large longitudinal stresses (also known as axial or normal stress).

Based on the observations as stated above, certain simplifying assumptions can be made for the finite element model which will significantly reduce computational time and efforts. The biggest simplification comes from the use of beam elements in this model. In general, geometrically slender structures where the longitudinal stress is of main concern, such as the wind turbine tower, can be modeled using the beam elements. They are a one-dimensional approximations of the three-dimensional structures. Details of the beam elements and its use in the tower modeling are discussed in later sections.

The main features of the finite element model shown in Figure 5.1 are summarized as follows:

- Beam, shell, inertia elements were used for modeling wind turbine tower structure
- Multi-body (part) construction (body or part: a group of beam/shell elements)

(a) FEA Model of Wind Turbine Tower (b) FEA Model of Wind Turbine Tower with Beam Profile Shown

Figure 5.1: FEA Model Views of Wind Turbine Tower

- Bodies were joined using multi-point constraints

In the subsequent sections, modeling details of the wind turbine tower are discussed.

5.3 Modeling Assumptions

The following list summarizes the general assumptions made for the finite element analyses.

1. Linear Elastic Analysis (Structure response is linear and elastic) - material will remain in elastic region; Contacts between parts are ignored; Applied loads remain constant in direction and magnitude.
2. Beam theory assumptions apply (Refer to the beam element section).
3. The wind turbine is modeled as a lumped mass with inertia terms and its center of mass lies directly above tower, one foot from the top flange.
4. Ground is assumed to be rigid.

5.4 Modeling Space and Coordinate System

The tower is modeled in three-dimensional space, and a Cartesian coordinate system was chosen for the finite element modeling. The orientation of the coordinate system follows the convention as specified as in IEC61400-2 (Figure 5.2).

x is positive in the downwind direction, z is positive upward, and y follows right-hand rule. In Abaqus, x , y , z coordinates are denoted as 1, 2, 3 respectively.

Figure 5.2: Global Coordinate System as Specified in IEC61400-2 (from [4])

5.5 Elements

5.5.1 Beam Elements

Slender structures such as the wind turbine tower can be modeled using beam elements. Beams are a one-dimensional approximation of a three-dimensional continuum. According to Abaqus manual, the beam element can be used to model structures when one dimension such as the length is greater than other two dimensions such as the cross sectional dimensions, and in which the longitudinal stress is most important.

The use of the beam element over solid or shell element in order to perform analyses such as time-history analysis will significantly reduce the computational time and effort. Additionally, the axial stress of the tower from the bending moment is of our primary concern. Thus, the structure was modeled mainly using the beam elements in order to perform the static and the dynamic analysis.

Requirements for Using Beam Elements

In order for this approximation to be realistic, certain slenderness conditions need to be met. Slenderness conditions specify geometry conditions of the part in order to use the beam element. Abaqus recommends the use of beam elements being limited to the structures that have cross-sectional dimension smaller than 1/10 th of axial dimension. The axial dimension refers to distance between two supports, between gross change in cross section, or wave length of the highest vibration mode of interest. It does not refer to the element length.

Beam Theory Assumptions

In addition to the general modeling assumptions, the assumptions associated with beam theory are also applied to the finite element model as follows:

1. Plane sections remain plane during deformation
2. Plane sections remain perpendicular to the axis of the beam for Euler-Bernoulli beam element. Timoshenko beam elements allow transverse shear deformation, thus plane sections do not stay perpendicular to the axis of the beam.
3. Deformation of the structure can be determined entirely from variables that are functions of position along the structural length.

Types of Beam Elements

Many different types of beam elements are available in Abaqus. But they can be broadly categorized into two types: Euler-Bernoulli and Timoshenko. For modeling the wind turbine tower, Abaqus's three-dimensional beam elements, B33 and B32 elements, were used (Figure 5.3). They both have three translational degrees of freedom (U_1, U_2, U_3), and three rotational degrees of freedom (UR_1, UR_2, UR_3).

Figure 5.3: Timoshenko Beam (B32) (left), and Euler-Bernoulli Beam (B33) (right) Elements (from [19])

Element B33

B33 is the Euler-Bernoulli type beam that does not allow transverse shear deformation. B33 has two nodes. And it uses cubic interpolation functions as shown in Figure 5.3. B33 gives reasonably accurate results for cases involving distributed loading along the beam. Therefore, they are well suited for dynamic vibration studies, where the d'Alembert (inertia) forces provide such distributed loading [19]. The cubic beam elements are written for small-strain, large-rotation analysis.

Element B32

B32 is the Timoshenko beam in 3D space that allows transverse shear deformation. It is good for modeling either a stubby or slender beam. It has three nodes and uses quadratic interpolation functions as shown in Figure 5.3. Abaqus assumes that the transverse shear behavior of Timoshenko beams is linear elastic with a fixed modulus and, thus, independent of the response of the beam section to axial stretch and bending [19].

5.5.2 Using Beam Elements

There are a total of 11 tower parts that use beam elements in the model created. Thick and stubby parts were modeled using the Timoshenko beam (B32 element). Slender parts were modeled using the Euler-Bernoulli beam (B33 element). The parts that used beam elements are summarized in Table 5.1 and shown in Figure 5.4.

Table 5.1: Tower Model Parts Beam Element

Part Name	Element type	Cross Section
Tower mast	B33, Euler-Bernoulli	Pipe
Ginpole	B33, Euler-Bernoulli	Pipe
Strut	B33, Euler-Bernoulli	Pipe
Reinforcement Rings (4X)	B32, Timoshenko	Pipe
Flanges (top and middle)	B32, Timoshenko	Pipe
Lugs (ginpole and stut)	B32, Timoshenko	General

Figure 5.4: Some of Parts Modeled Using Beam Elements with Cross Section Rendered

The tower mast is a tapered pole that has a tubular cross section. It has a larger diameter at the bottom that, then, decreases as the height increases. Abaqus does not have a beam element that can model the tapered geometry. In order to model

the tapered geometry of the tower, the mast was discretized into six sections, each of which has constant cross sectional properties (Moment of inertia (I), area (A), etc.) as shown in Figure 5.5.

Figure 5.5: Modeling Tower Using Beam Model for Finite Element Analysis in Abaqus

The pipe cross-section in the beam cross-section library was used for most of the parts except for parts that represents the lugs, gusset and ginpole plate (Figure 3.2). Using the section library is convenient because of the automatic generation of the beam properties such as moment of inertia. Only the radius and thickness are required to specify the beam properties. Abaqus calculates the required section properties. For lugs, general profiles were used instead of built-in cross-section profiles.

5.5.3 Shell Elements

According to the Abaqus manual [19], the shell element is a two-dimensional representation of a three-dimensional continuum. The shell element is used to model

structures where one dimension (thickness) is significantly smaller than other dimensions and the stress in thickness direction is not significant. Typically, if the thickness of the part is less than 1/10 th of the global structural dimensions, the use of the shell element is acceptable. Typical global structural dimensions are: distance between supports, stiffeners or large change in section thickness, radius of curvature, or wave length of highest vibration mode of interest.

Abaqus *S4* shell elements with full integration were used in modeling the gusset and ginpole plate (Figure 5.6). The S4 has three translational degrees of freedom (U_1, U_2, U_3), and three rotational degrees of freedom (UR_1, UR_2, UR_3). According to the Abaqus manual, element type S4 is a fully integrated (4 integration points), general-purpose, finite-membrane-strain shell element. The element's membrane response is treated with an assumed strain formulation that gives accurate solutions to in-plane bending problems, is not sensitive to element distortion, and avoids parasitic locking. S4 can be used for problems prone to membrane- or bending-mode hourglassing, in areas where greater solution accuracy is required, or for problems where in-plane bending is expected [19].

Figure 5.6: S4 Shell Element (from [19])

5.5.4 Using Shell Elements

Gussets are used in a structure to increase joint stiffness. When modeling the gussets, two different approaches were considered. Assuming that the membrane stiffness of the gusset plate is greater than the beam bending stiffness, the gusset can be modeled using a beam element having a very large stiffness. Rigid beam or beam with a greater (orders of magnitude) Young's modulus may be used to model the gusset. The other approach was using shell elements. The meshed nodes of the shell elements are tied to the beam nodes at the corresponding points using rigid beams (explained in Joining Elements). For the tower modeling, the shell elements were used in order to model the gussets.

5.5.5 Using Mass and Rotary Inertia Elements

The mass element is a one-node element. It can be used to introduce inertia at a point. It has three translational degrees of freedom (U_1, U_2, U_3), and gravity loading can be applied to a specified direction. The element can output the element kinetic energy.

The rotary inertia element is a one-node element as well, and it is used to apply the rotary inertia (moment of inertia) at a point. It is associated with three rotational degrees of freedom at the point (UR_1, UR_2, UR_3), and can be used in conjunction with the mass element.

The wind turbine tower (a nacelle and a rotor assembly) was modeled as a lumped point mass, using the mass element and the rotary inertia element as shown in Figure 5.7. The center of mass was assumed to be 1 *ft* above the top of the tower. A node was created at that location for the mass and rotary inertia element. Then both

elements were connected to the tower using a multi-point constraint (beam), equating the kinematics of the node of the top of the tower to the mass/inertia element. Mass inertia of $1.1905 \text{ lbf} - \text{sec}^2/\text{in}$ (converted from 460 *lbf* of turbine weight) was used. The moment of inertia for the assembly was calculated using Solidworks.

Figure 5.7: Mass and Rotary Inertia Elements for Wind Turbine (Green Dot)

5.6 Joining the Elements

All the wind turbine parts were modeled using different types of elements. Each set of elements was joined, with a kinematic relationship between them. They are assembled using a node-based constraint method, multi-point constraint (MPC). The MPC specifies linear or non-linear constraints between the nodes that are connected [19]. Many different MPCs are available in Abaqus, and each type imposes different kinematic constraints between the nodes. Three types of MPCs were used in the modeling: MPC-tie, MPC-beam and MPC-hinge. In the following section, each type of MPCs is briefly explained.

It is important to note a general rule for connecting different elements. When joining

the two different types of elements, both elements should have the same number and degrees of freedom (translational and rotational) at the nodes that are connected. In our case, all the elements used have the same types and degrees of freedom at their nodes: three-translational, and three-rotational degrees of freedom.

5.6.1 Multi Point Constraint (MPC) - Tie

MPC-tie is usually used to join two parts of a mesh when corresponding nodes on the two parts are to be fully connected (“zipping up” a mesh) [19]. It makes the global displacements and rotations as well as all other active degrees of freedom equal at two nodes [19]. The requirement of the MPC-tie is that the two nodes involved in the MPC must overlay each other.

The parts that were welded to the tower masts were constrained using MPC-tie. The nodes of the welded parts were tied to each other making all the degrees of freedom equal. For the tower, the ring stiffeners and flanges were welded to the tower, thus the “tie” was used to connect these parts as shown in Figure 5.8.

Figure 5.8: MPC-Tie between Tower and Flange

5.6.2 Multi Point Constraint (MPC) - Beam

MPC-beam provides a rigid link between two nodes constraining all the translational and rotational degrees of freedom at the first node (a) to those of the second node (b): $(U_1, U_2, U_3, UR_1, UR_2, UR_3)_a = (U_1, U_2, U_3, UR_1, UR_2, UR_3)_b$ [19]. It is like placing a rigid beam between the two nodes. For our case, it was the same kinematic constraints as the MPC-tie, but the nodes did not have to overlay each other. The main use of the MPC-beam was to constrain the shell elements to the beam elements where there is a gap between the location of the shell and the center of the beam (Figure 5.9).

Figure 5.9: Multi-point Constraint (MPC), type Beam, picture from [19]

MPC-beam was used to constrain the welded edges of the gussets to the beam as shown in Figure 5.10. MPC-beam's kinematic constraints simulated the welded joint very well, just like the MPC-tie. But a gap existed between the welded edges of the gusset and the beam because the beam in the finite element model represented the center of the beam. The gussets were welded to the beam few inches off the center axis.

It was also used to constrain the welded joint between the ginpole hold down plate and ginpole as well, as shown in Figure 5.11.

Figure 5.10: MPC-Beam between beam (tower) and shell (gusset)

Figure 5.11: MPC-Beam between Ginpole and Ginpole Plate

The MPC-beam is great for tying the nodes that are not aligned as well. Since the MPC-tie and MPC-beam give the same kinematic constraints for our case, it was used to constrain the nodes that were not overlaying each other. Initially, the strut ring and the tower mast were to be joined using the MPC-tie as there was no offset between the nodes to be joined. However, a very slight misalignment existed between the nodes of the two parts (the element of the ring was slightly shorter). Rather than re-meshing and fixing the geometry of the part, the MPC-beam was used to apply

the kinematic constraints (Figure 5.12).

Figure 5.12: MPC-Beam between Strut Ring and Tower Mast

5.6.3 Multi Point Constraint (MPC) - Hinge

The MPC-hinge is a constraint combined from the MPC-join and the MPC-revolute. It makes translational degrees of freedom of two nodes (a, b) equal, and provides a revolute constraint between their rotational degrees of freedom. That is, only two rotational degrees of freedom are constrained. The rotational degree of freedom with respect to the local 1-axis (x -axis) will not be constrained: $(U_1, U_2, U_3, UR_2, UR_3)_a = (U_1, U_2, U_3, UR_2, UR_3)_b$; $(UR_{1,a} \neq UR_{1,b})$. Local axes need to be defined at each node, and the local 1-axis of each node need to be parallel to each other.

Figure 5.13: Multi-point Constraint (MPC), Type Hinge, from [19]

The MPC-hinge was used to simulate the bolted connection as shown in Figure 5.14.

Figure 5.14: MPC-Hinge for Bolted Joints: Strut to Tower Mast (Left); Strut to Ginpole Anchor Plate (Center); Ginpole to Tower Mast (Right)

5.7 Boundary Conditions

The support conditions of the tower are shown in Figure 5.15. Two bearings are the main support members for the tower assembly which allows rotation with respect to the y-axis. The rear end of the tower is supported at the anchor plate which holds the tower in the z-direction.

Figure 5.15: Support Conditions of Tower

Initially, for the bearings, all degrees of the freedom were set to zero except for the that of the bearing rotation axis, UR_2 . But the reaction forces with the boundary conditions induced unrealistically large loads at the bearing supports. It was found that the rotations, UR_1 , and UR_3 at the bearings are small and no moments could be induced at the supports. Therefore, it behaves as a pinned joint. The pinned boundary conditions, $U_1, U_2, U_3 = 0$ and $U_1, U_3 = 0$ were determined to be reasonable and, therefore, used as the boundary conditions for the bearings (Figure 5.16a). Indeed, the induced reaction forces at the support matched well with the simple statics calculation.

At the ginpole anchor plate, a boundary condition of the simply-supported type, $U_3 = 0$, was found to be reasonable (Figure 5.16b). For dynamic analyses, $U_2 = 0$ was applied additionally.

Figure 5.16: Boundary Conditions

5.8 Material Properties

Two materials were considered for the wind turbine tower fabrication: ASTM A36 Carbon Structure Steel, and ASTM A572 High Strength Low-Alloy Columbium-Vanadium Structural Steel. ASTM A572 Grade-50 high strength steel was chosen over A36 because of its high strength and reasonable cost. The price difference of fabrication using A572 over A36 was marginal according to the vendor quote. It was used to fabricate the main mast and all of its mechanical components. The material properties are summarized in Table 5.2. The linear elastic region was assumed for the material (Figure 5.17).

Table 5.2: Material Properties of ASTM A572 Grade-50 High Strength Structural Steel

ASTM A572 Grade-50 Steel	
Ultimate Tensile Strength (<i>ksi</i>)	65.3
Yield Tensile Stength (<i>ksi</i>)	50.0
Poisson's Ratio	0.26
Young's Modulus (<i>Msi</i>)	29.2
Shear Modulus (<i>Msi</i>)	11.6
Bulk Modulus (<i>Msi</i>)	20.3
Elongatin at Break	0.21
Density (<i>lbm/in³</i>)	0.284

Using the material density specified combined with the weight of the wind turbine nacelle, rotor assembly, the overall structure's weight for the finite element's model is 4474 *lbf*, which agreed well with the Solidwork's weight calculations.

Figure 5.17: ASTM A572 Stress-Strain Plot (from Google Image)

Chapter 6

Static Analyses of Wind Turbine Tower

Using the model described earlier, the static analyses of the wind turbine tower for two load cases were performed: Installation (tilt-up) case, $2k - lbf$ Maximum Thrust case. These two load cases created the largest loads for the tower. The stress, the deflection of the tower, the section axial-loads and the section bending-moments along the tower were calculated and reviewed.

6.1 Load Case: Installation (Tilt-up)

6.1.1 Introduction

The installation process of the tower with the wind turbine is illustrated in Figure 6.1. The tower with the wind turbine is tilted on the tower's two bearings. A cable from the truck's winch was attached to the tower's anchor plate and used to pull the

tower up.

Figure 6.1: Illustration of Wind Turbine Installation

The installation process can induce large loads on the tower. The wind turbine tower when combined with the wind turbine weighs about 4500 lbf . The tower is at the horizontal position, so the weight is applied perpendicularly to the tower's long axis, inducing a large bending moment on the tower. When the tower is tilted up, the weight could “effectively” increase due to accelerations.

6.1.2 Assumptions

For this analysis, the following assumptions were made (in addition to the general assumption in Chapter 5):

- The worst case arises when the tower is at the horizontal position
- A dynamic factor of 2 is applied to amplify load as specified in IEC61400-2 [4] (g is multiplied by factor of 2)
- The combined weight of the wind turbine nacelle and rotor is 460 lbf [8]. It is approximated as a point mass and inertia at the top of the tower.
- Gravity acts in the negative x-direction (coordinate as shown in the results)

6.1.3 Results

Stress and Deflection

In the absence of torsional load (moment around the beam axis), the *B33* element used in the tower does not output the shear stress, *S12*. Therefore, Mises stress, *S, Mises*, is the same as *S11* in magnitude. We looked at the axial stress, *S11*, for this case. As shown in Figure 6.2, the maximum axial stress, *S11*, of the tower due to the bending moment was about 36 *ksi*, giving factor of safety of 1.4 using the material's yield strength of 50 *ksi*. As shown in Figure 6.2, the maximum deflection in the negative x-direction, *U1*, was about 34 *inches* at the tower tip.

Section Force and Section Bending Moment

The section force, *SF1*, is the axial force at the beam element nodes (1 denotes beam's axial direction in this case), and the section moment, *SM2*, is the bending moment with respect to beam's local y-axis (2-axis, pointing out of the paper in this case). They were reviewed at each section of the tower (Figure 6.3). The results were used as the design loads for the specific components such as bolts, lugs, welds and flanges.

The largest bending moment was $1.2 \times 10^6 \text{ in} - \text{lb}$ which was found just above the strut attachment. This location also corresponds to the largest axial stress as seen in Figure 6.2. The axial forces for the strut was about 17 *kips* in tension and 12.4 *kips* for the ginpole in compression.

Reaction Forces

The reaction forces were reviewed at the tower's support locations: bearings and the anchor plate. The results give design loads for the foundation and the anchor bolts. For the tilt-up load case, the reaction forces, RF , were calculated as shown in Figure 6.4.

Result Plots

Figure 6.2: Axial Stress, S_{11} in psi (Top) and Deflection in x-direction, U_1 in inches (Bottom) for Load Case: Installation

Figure 6.3: Section Axial Force, SF_1 (top) in lbf , and Section Moment, SM_2 (bottom) in $in-lbf$ for Load Case: Installation (red line). Note that SM_2 is bending moment with respect to beam's local y-axis (2-axis, pointing out of the paper)

Figure 6.4: Reaction Forces (RF) for Load Case: Installation

6.2 Load Case: Maximum Thrust and Gravity

6.2.1 Introduction

The 2100 *lbf* of maximum thrust calculated in Chapter 4 was determined to be the worst load case for the tower among others considered. The thrust load was applied in two directions, x-, and y-direction, as shown in Figure 6.5. The prevailing wind direction is the x-direction. Although the probability of the maximum thrust applied in the y-direction was low at this site, it was considered for the analysis. The gravitational load was applied in the negative z-direction for the entire body as well. The same general assumptions were applied as listed in Chapter 5 for this analysis.

Figure 6.5: Load Case: 2k *lbf* Maximum Thrust

6.2.2 Results for Thrust Load Applied to x-Direction

Stress and Deflection

For this load case, the results produced similar levels of stress and deflection as the previous load case, the installation case. As shown in Figure 6.6, the maximum axial stress of the tower was found to be little over 36 *ksi* which gives the factor of safety of 1.4 on yield. The deflection at the tip was about 40 *inches*. The overall length of the tower is 840 *inches*, so the deflection is small, accounting for only 4 % of the overall tower length.

Section Axial Force and Section Bending Moment

The axial force and the bending moment of the tower were reviewed just as the previous load case. The largest bending moment of was $1.12 \times 10^6 \text{ in} - \text{lb}f$ which was found just above the strut attachment. The moment in this load case was a little bit lower than the installation case. The axial force for the strut was about 10 *kips* in compression, and 6.72 *kips* in tension for the strut. These results matched well with the static hand calculations as shown in Appendix A.

Reaction Forces

At support locations, bearings (2X) and the anchor plate, the reaction forces were calculated. The two bearing reaction forces were identical. For a bearing, the base shear force (horizontal or x-direction) was 1050 *lb**f*, and pull-out force (vertical or z-direction) was 1500 *lb**f* as shown in Figure 6.4. For the anchor plate, the total reactions were calculated to be about 7600 *lb**f* in z-direction, which is compressive force

applied to the foundation. The direction of the pull-out force and the compressive force may be reversed for the wind blowing from the opposite direction.

Result Plots

Figure 6.6: Axial Stress, S_{11} in psi and Deflection, U_1 in $inches$ for Thrust Applied to x-Direction

Figure 6.7: Section Axial Force, SF_1 (left) in lbf , and Section Moment, SM_2 (right) in $in-lbf$ for Thrust Applied to x-Direction (red line). Note SM_2 is bending moment with respect to beam's local 2-axis (global y-axis, pointing in/out of the paper)

Figure 6.8: Reaction Forces, RF for Load Applied to x-Direction

6.2.3 Results for Thrust Load Applied to y-Direction

Stress and Deflection

Figure 6.9 shows the axial stress, S_{11} , and the corresponding deflection in the y-direction, U_2 , of the tower. The maximum stress on the tower was about 38 *ksi* at the lower part of the tower, and the largest deflection was about 69 *inches* at the top of the tower. The stress distribution along the tower mast seems fairly uniform due to the tapered cross-section. The resulted stress has the factor of safety of 1.3 to yield, and the deflection is about 8% of the overall tower length.

As seen in Figure 6.11, a small amount of torsional loads, SM_3 , were found on the strut, ginpole and lower part of the tower for this load case (presumably due to the hinge constraints imposed on the strut, ginpole and their attachment points of the tower which are subjected to a slight rotation with respect to one of the “tied” rotational axes). The resulting shear stress, S_{12} , was small compared to the axial stress, S_{11} (1.58 *ksi* Vs. 38 *ksi*). Therefore, it did not have a significant effect on the overall combined stress, $S, Mises$ as shown in Figure 6.12. Therefore, looking at S_{11} instead of $S, Mises$ is reasonable as well.

Section Axial Force and Section Bending Moment

There was no significant axial force contribution in the structure for this load case. However, the bending moment on the tower was large. As shown in Figure 6.10, the largest bending moment with respect to beam’s 1-axis (global x-axis) was 1.72×10^6 *in – lbf* at the base of tower mast. Note that the rainbow stick-measure in this plot represents each beam’s local 1-axis.

Reaction Forces

This load case produced the largest load at the main bearing support locations (Figure 6.10). The bearing located upwind of the tower induced the pull-out force of 27.5 *kips* while the other bearing induced 31.7 *kips* of compressive force and 2.1 *kips* of the base shear force. The anchor plate had marginal reaction forces compared to the one at the bearings.

Result Plots

Figure 6.9: Axial Stress, $S11$ in psi (Left), and Deflection in y -direction, $U2$ in $inches$ (Right), for Load Applied to y -Direction

Figure 6.10: Section Bending Moment, $SM1$ in $in-lbf$ (Left) - Note $SM1$ is bending moment with respect to beam's local 1-axis (global x -axis); Reaction Forces, RF (Right) for Load Applied to y -Direction

Figure 6.11: Torsional Moment with Respect to Local Beam's Longitudinal Axis, $SM3$, in $in-lbf$ (Left); Induced Shear Stress, $S12$, from $SM3$ in psi (Right), at Lower Section of Tower

Figure 6.12: Comparison of Axial Stress, $S11$, in psi (Left), and Mises Stress, $S, Mises$ in psi (Right), at Lower Section of Tower

6.3 Conclusion and Design Loads for Tower Components

The largest stress on the tower was approximately 38 *ksi*, which was induced from the thrust applied to y-direction. For 50 *ksi* of material yield-strength, the stress level produced the factor of safety of 1.3. The maximum deflection of 69 *inches* occurred with the same load case. The maximum deflection was about 8 % of the overall tower length which is 840 *inches*.

Figure 6.13: Key Locations for Mechanical Design Loads

The section forces and section moments at certain locations were reviewed for the design of the specific mechanical components such as welded-joints, bolts, lugs, and flanges. The locations of interest are shown in Figure 6.13, and the largest loads at each location were compiled in order to use them as the design loads of the mechanical components.

The bending moment was the driver for the flange design (location 1 in Figure 6.13). At the mid-flange (1), the maximum bending moment was found to be 9.1×10^5 *in-lbf* which was induced from the thrust load case.

Maximum tensile/compressive loads for the strut (3) were 17 *kips*; and 12.4 *kips* for the ginpole (4). Both loads were found in the installation case. For the case where thrust applied in y-direction, moment loads were predominant for the strut and the ginpole as seen in Figure 6.14. These moment loads were used for the design loads for the lugs (2, 5).

Figure 6.14: Moments on Strut and Ginpole, *SM1*, *SM2*, *SM3*, in *in-lbf* Resulting from Thrust in y-direction. *SM1*: bending moment w.r.t beam's local 1 axis. Blue-to-red stick is the beam's local 1-axis; *SM2*: bending moment due to local 2-axis, in and out of paper or same as global y-axis; *SM3*: torsional moment around the beam's longitudinal axis

The foundation loads were obtained by looking at the reaction forces for each load case. The largest reaction forces were compiled, and summarized in Table 6.1. The foundation and the anchor bolt should be able to withstand the listed loads.

Table 6.1: Design Loads for Foundation Design Obatined from Static Analyses

Location	Design Base Shear Load (<i>kips</i>)	Design Compressive or pull-out Load (<i>kips</i>)
Main Support (Bearings)	4.5 from installation	31.7 from thrust in y-dir.
Rear Support (Anchor Plate)	-	7.6 from thrust in x-dir.

Chapter 7

Dynamic Analyses of Wind Turbine Tower

7.1 Natural Frequency Extraction

7.1.1 Introduction

A dynamic effect such as vibration can be a significant source of loads for a wind turbine tower. The tower can be categorized by its dynamic characteristics: fundamental natural frequency (f_{n1}), and its relationship with the excitation frequencies such as the rotor frequency (f_{rotor}) or the blade passing frequency (f_{bp}). These are denoted as $1p$ or np respectively in the literature and n is number of blades of the rotor. f_{rotor} is the rotor's rotation speed, and f_{bp} is the number of blade times f_{rotor} . This relationship is summarized in Table 7.1.

A tower should be designed so that its natural frequency (f_n) does not coincide with the excitation frequencies such as f_{rotor} ($1p$) or f_{bp} (np). Soft towers and soft-soft

Table 7.1: Tower Category Depending on Its Fundamental Natural Frequency, f_{n1}

Category	Relationship between f_{n1} and f_{rotor} , f_{bp}
Soft-Soft Tower	$f_{n1} < f_{rotor} \text{ (1p)}$
Soft Tower	$f_{rotor} \text{ (1p)} < f_{n1} < f_{bp} \text{ (np)}$
Stiff Tower	$f_{n1} > f_{bp} \text{ (np)}$

towers may be excited during the start-up and shut-down. Stiff towers are desirable but may not be cost-effective to build. Although not proven, some believe that soft-soft towers attenuate the fatigue load better throughout the system [2]. Most large turbines have soft towers.

The results of natural frequency extraction of wind turbine towers can be used to determine the category of our wind turbine tower as specified in Table 7.1 and to determine if the tower will be stable during operation. The result of this analysis is also used in other dynamic analyses such as time-history (transient analysis) or response spectrum analysis.

7.1.2 Theory

Abaqus performs eigenvalue extraction to calculate the natural frequency and corresponding mode shapes of a structure. The eigenvalue problem for the natural frequencies of an undamped finite element model is to solve:

$$(-w^2 M + K)\phi = 0 \quad (7.1)$$

where M is the mass matrix (symmetric, positive definite) of the structure, K is the stiffness matrix, and ϕ is the eigenvector which is the mode of vibration (the effect of

damping can be included by performing complex eigenvalue analysis).

Abaqus uses three numerical methods to solve the eigenvalue problem: Lanczos, Automatic multilevel substructuring (AMS), and subspace iteration. According to the Abaqus Analysis User's Manual, the choice of solving mechanism has minimal impact on the frequency extraction procedure except for the computational performance [19]. The Lanczo method, which has the most general capabilities, was chosen for the analysis.

7.1.3 Results

For this analyses, all the applied loads (thrust, gravity) were ignored and the specified boundary conditions were applied as mentioned in Chapter 5.

The plotted solution, specifically the displacement of the mode shape, was normalized to unity as shown in the subsequent figures. Table 7.2 summarizes the natural frequencies (f_n) of the wind turbine and tower assembly. Only the first ten modes were reported.

Our turbine's design rotating speed is about 230 *rpm* (Table 1.1) which corresponds to the rotor frequency, f_{rotor} ($1p$) of 3.83 *Hz*. The 1st mode of the tower is 0.584 *Hz* which is less than the rotor frequency. Thus, the tower is categorized as a *soft-soft* tower. f_{rotor} lies between 3rd and 4th mode of the tower natural frequency, avoiding excitation of the tower's natural mode of vibration during the normal operation. The blade passing frequency, f_{bp} ($3p$), is 11.5 *Hz*; it lies between 7th and 8th mode of vibration. Thus, it avoids the excitation during the normal operation. However, the tower may be temporarily excited as the turbine starts up or shuts down.

Table 7.2: Natural Frequencies of Wind Turbine Tower

Mode	f_n (Hz)	Mode	f_n (Hz)
1	0.584	6	6.378
2	0.834	7	8.125
3	2.805	8	13.898
4	4.673	9	14.083
5	4.959	10	15.609

Figure 7.1: 1st Mode Shape of Wind Turbine Tower

Figure 7.2: 2nd Mode Shape of Wind Turbine Tower

Figure 7.3: 3rd Mode Shape of Wind Turbine Tower

Figure 7.4: 4th Mode Shape of Wind Turbine Tower

Figure 7.5: 5th Mode Shape of Wind Turbine Tower

Figure 7.6: 6th Mode Shape of Wind Turbine Tower

Figure 7.7: 7th Mode Shape of Wind Turbine Tower

Figure 7.8: 8th Mode Shape of Wind Turbine Tower

Figure 7.9: 9th Mode Shape of Wind Turbine Tower

Figure 7.10: 10th Mode Shape of Wind Turbine Tower

Chapter 8

Dynamic Analysis: Seismic Analysis

8.1 Evaluating Seismic Risks

Evaluating seismic risk for a structure can be thought of as two parts. The first part is evaluating the probability that a particular level of earthquake will happen at a site where the wind turbine is built. The level of earthquake can be measured in many ways but, in the context of seismic analysis, peak ground acceleration (PGA) is relevant. The second part is assessing the likelihood that any structural damage could happen from the anticipated earthquake.

For the first part, probabilistic seismic hazard assessment (PSHA) is commonly conducted. It is performed to obtain a statistical probability of a particular level of earthquake at a specific location. According to a study performed by California Geographic Survey in 2002, it was found that the peak ground acceleration that has a 10 % probability of being exceeded in 50 years in California was about 0.8g (while Asia

and Europe has 0.5g). A statistical model, Homogeneous Poisson Process (HPP) is often used to get the probability of earthquake occurrence.

There are a number of Earth's plates that form the Earth's surface. Two of them meet in the western California region: the Pacific Plate and the North American Plate. The boundary between the two plates is the San Andreas Fault. It's a master fault of an intricate fault network that cut through rocks of the California coastal region. Also, it is the source of many earthquakes in California. The site on which the wind turbine is built has a close proximity to the fault. Thus, the seismic analysis shall be performed to check the structural integrity of the wind turbine tower and to prevent potential economic loss.

For the second part, assessing the structural risk, several methods of such assessments are available. These include codified methods provided from building codes and wind turbine design standards; and computer modeling such as one using the finite element method. Both building codes and the wind turbine standards use similar methods, a single degree of freedom (SDOF) frequency-domain analysis, for simplicity. A few building codes and the wind turbine standards for the seismic analysis are listed as follows:

- Building Code
 - International Building Code (IBC)
 - Uniform Building Code (UBC)
 - American Society of Civil Engineers (ASCE)
- Wind Turbine Standard
 - IEC 61400-1, Wind Turbines - Part 1: Design Requirements. (ANNEX C and section 11.6)

- RISO, Guideline for Design of Wind Turbines
- GL (Germanischer Lloyd), Guideline for the Certification of Wind Turbine

All three wind turbine guidelines suggest that there are few regions throughout the world where seismic loads may drive the design. In all cases, seismic analysis is only required in regions of high seismic hazard or as required by local authorities (building codes). Building codes do not have seismic provisions specific to wind turbine towers. Clearly, wind turbine tower structures are very different from traditional buildings. However, they are often governed by the same seismic building codes such as the ones mentioned.

The provisions for seismic requirements were developed from studying earthquake effects from past events, and are mainly concerned with minimizing the loss of life when the structure is subjected to the most intense earthquake possible in the life of the structure. Some structural damage can be expected to occur as most building codes allow inelastic energy dissipation in the structure. For example, local yielding of the structure may be observed in the event of such an earthquake.

Simple static methods specified in the codes are based on the single-mode response with simple corrections for including higher mode effects [20]. This method is acceptable for simple regular-shaped buildings. However, in order to capture the detailed seismic behavior of the structure that is complex in shape, time history or response spectrum analyses are the preferred methods. The finite element model which was developed earlier was used to perform a response spectrum analysis and a transient modal dynamic analysis (time-history). Both analyses were based on the modal superposition method, and performed to assess the tower's behavior under seismic loading.

8.2 Transient Response (Time-history) Analysis

For transient response (time-history) analysis, we calculated how a structure responds to an arbitrary time-dependent loading. Two methods are commonly available for performing this analysis using finite element methods: Direct integration method and modal superposition method.

8.2.1 Direct Integration Vs. Modal Superposition Method

Direct Integration Method

The direct integration method is a step-by-step direct integration method. Equations of motion of a structure are solved for a discretized time-interval using solutions of a previous time step as an initial value. This gives the most accurate results compared to many other modal based methods. It is used for both linear and non-linear analyses.

$$M\ddot{D}_n + C\dot{D}_n + KD_n = R_n \quad (8.1)$$

where, n is integer corresponding to a time step, t ($t = 1\Delta t, 2\Delta t, \dots$), R_n is a forcing function at n_{th} time step instance. Using a numerical scheme D , $\dot{D}$, and $\ddot{D}$ are calculated for each time, t . M is mass, C is damping, K is stiffness for the system.

The two common numerical methods to solve equations are the central difference method, and the Newmark Method. The central difference method is based on the finite difference formulas. It is good for wave propagation problems in which many higher modes are excited. The Newmark Method is good for structural dynamics under earthquake loading in which only the lower few modes are of importance.

The direct integration method can be used for both linear and non-linear analyses. It is used for systems that have non-symmetric stiffness, and complex damping (i.e., damping that is dependent on frequency, etc.) [19]. The downside of the direct integration method, however, is that it requires more computational resources and efforts when compared to the modal superposition method [19]. The analysis produces a large number of output, which may require a large post-processing effort to conduct all possible design checks as a function of time.

Modal Superposition Method

Modal equations of motion (transformed from physical equations of motion of a system (Equation 8.1)) are used for this method. Modal equations are a reduced form of the equations of motion by expressing the displacement of the full physical system in terms of a limited number of its vibration modes, especially lower frequency modes. It results in a n -set of uncoupled equations of motion where n is same as number of modes used in the transformation. It was emphasized that *only the lowest few modes need to be retained in the transformation*. The transformed equation is:

$$w^2 z_i + c' z_i + z_i = p_i \text{ for mode } i \quad (8.2)$$

where, w is the natural frequency of mode i , z_i is modal (principal) coordinates, p_i is transformed modal time-dependent loads, and c' is damping. Commonly, proportional (Rayleigh) damping or modal damping is used; it will be explained in later section.

The transformed p_i was the earthquake loads for our analysis. After each uncoupled equation was solved, the solution was re-transformed to D , the physical degree of freedom.

The transient analysis using the modal superposition method has advantages over the direct integration method because it requires less computational time and resources. Therefore, it is a cost-effective option for performing linear or mildly nonlinear dynamic analyses (the principle of superposition is not valid for non-linear analysis) [19]. Indeed, the response spectrum analysis, as explained in later sections, is also based on the principle of the modal superposition.

Abaqus's Transient Modal Dynamic Analysis

In Abaqus, the transient analysis procedure is based on the theory of modal superposition. First, the structure's natural frequencies and modes are calculated. Then, a time-dependent loading is applied to the structure. The structure's response is calculated based on a subset of extracted modes and as a function of time. As long as the system is linear and is represented correctly by the modes being used, the method is very accurate [19]. Using this analysis procedure, instantaneous stresses and deflections of the wind turbine tower were calculated for the duration of an earthquake input.

8.2.2 Input

The first step of the analysis is to select the appropriate earthquake input and its direction.

Source of Input

Network computing equipment is often subject to seismic assessment if installed in an earthquake sensitive zone. Bell Communication Research (Bellcore) established a

test standard for their equipment to assess its structural integrity and functionality during and after an earthquake. The standard is called Network Equipment-Building System (NEBS) Requirements: Physical Protection (GR-63-Core, section 5) [21]. The acceleration-time history waveform, VERTEQII, was used as the earthquake input (Figure 8.1). The VERTEQII is a synthesized waveform from several typical earthquakes and for different buildings and site-soil conditions. Also, it was for structures inside a building (our structure is the building effectively); it is somewhat conservative as the maximum acceleration was 1.6g (typically less than 1g for CA as mentioned before). This waveform with high accelerations was used for the transient dynamic analysis of the tower.

Figure 8.1: Zone 4 Earthquake Input from Bellcore Environmental Test Requirement: Acceleration (in/s^2) Vs. Time (*seconds*)

Direction

A well designed structure should be capable of equally resisting earthquake motion from all possible directions. One option in existing design codes for buildings and

bridges requires that the structure member be designed for 100 % of the prescribed seismic forces in one direction and 30 or 40 % of the prescribed forces in the perpendicular direction. However, it is reasonable to assume that the motion that takes place during an earthquake has one principal direction [22]. The 30/40 % rule does not have theoretical basis. [22]

Using the finite element model developed, a transient modal dynamic analysis was performed. The Bellcore input was applied in each of the two principal directions, x, and y, assuming that the earthquake has one principle direction. The output of the analysis was reviewed to find the maximum stresses in the structure.

8.2.3 About Damping

Choosing the right value of damping can be complicated and tricky. Some background about damping and implementation methods in finite element analysis is reviewed in this section.

What is Damping?

Damping describes the structure's ability to dissipate energy. It causes the amplitude of a free vibration to decay with time. Depending on the characteristics of damping, a structure can be categorized as under-damped, over-damped and critically damped. Critical damping (C_{cr}) is a key to such characterization. C_{cr} is the amount of damping which will cause a vibrating structure to reach an equilibrium state without any oscillatory behavior. One can find the damping ratio for a given structure by test measurement, which is the most accurate method. Some published data are available as well for analysis, although careful thought has to be given for the choice of damping

value. Certain mathematical models are available to implement damping in the finite element analysis, and are explained below.

Typical Values of Damping

Various reference literature specify typical ranges of damping for a type of structure and are summarized for review in Table 8.1.

Table 8.1: Typical Published Values of Damping for Different Type of Structures

Type of Structure	Range of Damping (% of C_{cr})
Continuous Metal Structure	2 to 4 [23]
Jointed (bolted) Structure	4 to 7 [23]
Reinforced Concrete Structures	4 to 7 [23]
Small Diameter Piping Systems	1 to 2 [23]
Equipments, Large Diameter Pipes	2 to 3 [23]
Typical Building Codes	5 [20]
Wind Turbine Structures	0.5 to 5 (or more) [24]
Wind Turbine Structures	1 [25]
Wind Turbine Blade	3 [14]
Wind Turbine Shaft and Tower	5 [14]

For wind turbine specific structures, RISO reports that 3% for blades, 5% for shaft and tower [14]. For seismic analysis based on SDOF response spectrum analysis, IEC recommends use of 1% damping [25]. The Sandia Laboratory study reported that the use of 0.5% to 5% or more for modeling wind turbines was commonly found in literature. The ASCE Building Code uses 5% damping to generate the site-specific design spectrum.

Types of Damping and Sources

There are many factors and design details that affect damping. Common sources of damping are said to be material and friction (the internal friction in the materials and Coulomb friction in connections of the structure). However, the exact sources of damping are complex, it is not easy to measure or represent mathematically, and it is often not linear. Lower values of damping may occur when the structure undergoes small deflections at low levels of stress. At higher levels of stress and larger vibration amplitudes the damping may be at the upper end of the range given [23]. However, damping in a structural problem is small enough that it can be idealized as viscous damping regardless of the actual damping mechanism [26]. Viscous damping applies a force to a structure proportional to velocity but oppositely directed (often seen as term $C\dot{X}$ in x-direction).

Implementation of Viscous Damping in FEA

Viscous damping can be implemented in many ways in a finite element analysis depending on the software, but two are commonly found: Proportional damping (Rayleigh damping) and modal damping. They are explained briefly.

Rayleigh Damping

For a direct integration method, a physical damping mechanism such as a dashpot is often used to introduce damping. For structural models that do not have such dissipation sources, general mechanisms of damping are introduced. The Rayleigh damping model is one of them; it is also known as proportional damping. Although the model may not be physically correct (infinite damping at $\omega = 0$), it may be acceptable for

general use of damping. This model uses two damping factors α and β to dampen the lower mode (mass proportional damping) and higher mode (stiffness proportional damping) respectively. From two sets of natural frequencies (ω) and damping (ξ), the damping factors are obtained (Equation 8.3) and applied to formulate the damping matrix, C . (Equation 8.4).

$$\xi_i = \frac{\alpha}{2\omega_i} + \frac{\beta\omega_i}{2} \quad (8.3)$$

$$C = \alpha M + \beta K \quad (8.4)$$

Modal Damping

For this method, an arbitrary *diagonal* damping matrix was assumed in the equations-of-motion of the multi-degrees of freedom system, uncoupling the equations for each mode. Thus, the n -independent equations of motion are formulated. The equations of motion simplify to ones similar to the SDOF system 8.5. One may use the same damping ratio for all modes if so desired [26]. This method was used to specify damping for the seismic analyses performed. More information is given in the following section.

Abaqus Damping Models

Several options for specifying damping for the modal superposition method in Abaqus are available:

- Rayleigh damping

- Composite modal damping
- Structural damping
- *Fraction of critical damping*

Fraction of critical damping, (*modal damping* as explained above) was chosen for the seismic analysis. According to [19], the damping in each eigenmode can be specified as a fraction of the critical damping for each mode.

Critical Damping, c_{cr} : As explained in the modal damping section, the modal equations are uncoupled by assuming the diagonal damping matrix, then the equation of motion for one of the eigenmode of a system becomes like the one for the SDOF system (Equation 8.5).

$$m\ddot{q} + c\dot{q} + kq = 0 \quad (8.5)$$

Here, q is modal amplitude, m is mass, c is damping and k is stiffness of a system. The solution to Equation 8.7 is shown in Equation 8.6 below.

$$q = Ae^{\lambda t} \quad (8.6)$$

A is a constant, and λ is given as,

$$\lambda = \frac{-c}{2m} \pm \sqrt{\frac{c^2}{4m^2} + \frac{k}{m}} \quad (8.7)$$

Setting the terms in the square root in Equation 8.7 to zero will yield critical damping, c_{cr} , and is found to be $c_{cr} = 2\sqrt{mk}$. c_{cr} is calculated for each mode; the specified

fractional damping ratio is applied to each c_{cr} to obtain damping for each modal equation.

Damping Used for Seismic Analysis

The values of the damping in the published references (as shown in Table 8.1) are at most approximate. Depending on the type of analysis, these may be acceptable or not. It may be necessary to measure the actual damping ratio of the real structure by experiment.

When damping is small, the damped natural frequency is almost the same as the undamped natural frequency. But the amplitude of response of the structure near resonance may be greatly affected by damping as seen in Figure 8.2.

Figure 8.2: Effect of Damping

For most dynamic analyses, the published data are of sufficient accuracy. If it is suspected that damping effects are causing errors in the model, then a sensitivity analysis may be performed. If the result varies greatly, one may need to obtain the damping experimentally.

In the following seismic analyses, the damping ratio of 5% was used and applied to

each mode using the Abaqus's *Fraction of critical damping (modal damping method)*. Many published values suggest that 5% for the tower analysis is a good approximation as shown in Table 8.1

The results of the transient dynamic analysis are presented after the response spectrum analysis section.

8.3 Response Spectrum Analysis

8.3.1 Introduction

A response spectrum analysis is commonly used to study the response of a structure under seismic loading, especially in the preliminary design stage because of its simplicity. The term *response* here refers to a structure's quantifiable physical behavior such as displacement, velocity, or acceleration subject to a physical input such as an earthquake. Unlike transient dynamic analysis, in a response spectrum analysis, it only seeks the maximum response of the structure without regard to time as the structure is subject to dynamic motion at fixed points [26]. Therefore, the maximum response can be calculated with significantly less time and computational resources compared to the transient analysis. But, the result is only an approximation. Many building codes employ the response spectrum analysis for seismic analysis although they use very simplified representation of building utilizing only a single degree of freedom per floor.

8.3.2 How Does It Work ?

Finding Maximum Response

According to [26], the response spectrum analysis seeks a maximum response of each separate mode then combines the modal maxima in a way that would produce an estimate of maximum response of the structure itself. First, a modal analysis is performed to extract the undamped natural frequencies and modes. Using modal equation (Equation 8.2), the modal displacement, $z_i(t)$, as a function of time is calculated for each mode i . Then, maximum of $z_i(t)$, $z_{i,max}$, is picked for each mode i . The *maximum* physical value of a degree of freedom j associated with mode i , δ_{ji} , can be calculated using the following Equation 8.8.

$$\delta_{ji} = \phi_{ji} z_{i,max} \quad (8.8)$$

The actual maximum physical value for a degree of freedom j , D_j then is found by combining all the δ_{ji} (maximum value from mode i) produced by each mode i . While the maximum response of each mode is known, the relative phase of each mode is unknown. So, a mode combination method is used. Several methods are available; they are briefly conveyed in the following section.

Combining Maximum of Modes

The maximum physical displacement value due to a dynamic load for a particular degree of freedom j , $D_{j,max}$ is calculated by combining δ_j of each mode i . The obvious and intuitive way is to add all the δ_j s produced by the different modes i (for a particular degree of freedom j). This method is called a *sum of absolute*

magnitude. This assumes that all the maxima for each mode i occurs at the same time which results in an overly conservative estimate (the most conservative method). Another well-known method is *Square Root of the Sum of the Squares*. This method assumes that all the maximum modal values are statistically independent. For three-dimensional structures in which large number of frequencies are almost identical, this assumption is not justified [22]. A few methods of combining the modes are available in the Abaqus and are summarized as follows:

- The absolute value method (sum of absolute magnitude)
- The square root of the sum of the squares method (SRSS)
- The ten-percent method
- The complete quadratic combination method (CQC)

The CQC method was chosen for combining modes. It is a fairly new method, formulated based on random vibration theories and has wide acceptance by many engineers for seismic analysis. According to a case study that compared results obtained from the absolute value method, the SRSS, and the CQC to the results from a time history analysis (which is the most accurate method), the CQC method had the least amount of difference [22]. Indeed, according to Abaqus [19], this method improves the estimation of the response of a structure that has closely spaced eigenvalues, which seems to be our case. Also, Abaqus recommends the use of this method for asymmetric buildings.

8.3.3 Response Spectrum as Input Force

In the previous section, the essentials of response spectrum analysis was presented. It uses the modal superposition method to find the maximum of each mode and then combines the maxima using mode combination methods in order to obtain the maximum physical response of the structure for a dynamic input. In this section, the detailed input for the response spectrum analysis is discussed.

Response spectrum is a plot of maximum values of a structure's responses, such as displacement, velocity and acceleration versus natural frequency or period of a single degree of freedom system; it can be used as an input for a response spectrum analysis. Once the response spectrum for a certain forcing function is calculated, it does not need to be recalculated regardless of the number or variety of multi-degrees of freedom structures to which the forcing function is applied. Response spectra of single degree of freedom are applicable to a multi-degree of freedom system by the same forcing function.

In the context of seismic analysis, the response spectrum represents the earthquake motion. Different types of response spectra are available. A *Design spectrum* is a smooth spectrum which uses average of records from several actual earthquake events rather than single particular earthquake record. It is a representative of many earthquakes. A *Site-modified design spectrum* adds the effect of local soil condition and distance to the nearest fault. A site-modified design spectrum was used as the input for the response spectrum analysis of the tower assembly.

Site-modified Design Spectrum

The United States Geological Survey (USGS) provides a software tool that calculates site-specific response spectra for seismic analyses. It can create spectra per different building code specifications and for a specific site accounting for soil conditions as well as the distance to a closest fault. The software is called *USGS Seismic Design Maps and Tools for Engineers* and can be found on their website. For this analysis, site-modified response spectra per *ASCE 7 Standard, Minimum Design Loads for Buildings and Other Structures* were calculated and used as inputs for the response spectrum analyses.

The basis of ASCE-7 spectral acceleration resulted from an earthquake corresponding to a return-period of 2500 years (uniform likelihood of exceedence of 2% in 50 years). This is called *Maximum Considered Earthquake (MCE)*. ASCE-7 defines the maximum considered earthquake ground motion in terms of the mapped values of the spectral response acceleration at short periods, S_s , and at 1 second, S_1 , for site-class B, soft rock. These values may be obtained directly from the map published by USGS [20].

There are six *site-classes* in the ASCE 7 standard. They are based on the average properties of the upper 100 *ft* of soil profile. A brief summary and description for each class are given in Table 8.2. As mentioned, the site-class B is used as a baseline. According to the geotechnical survey for Cal Poly's wind turbine site, the site-class D represents the location's soil condition well.

The response spectra calculated by the USGS software are shown in Figure 8.3 assuming damping of 5%. First, MCE for the wind turbine site was calculated assuming site class B (blue dash-dotted line). The location of the wind turbine site was input by specifying the latitude and longitude. The actual site class is D which gives site

Table 8.2: ASCE 7 Site Classification

Site Class	Site Description
A	Hard Rock
B	Rock
C	Very dense soil and soft rock
D	Stiff soil
E	Soil
F	Soils requiring site-specific evaluation: Clays and soils vulnerable to potential failure

Figure 8.3: Response Spectra per ASCE-7 using USGS software: Blue, dash-dotted line = MCE Spectrum for Site-class B; Red, solid line = Site-Modified (D) MCE Spectrum; Green, dashed line = Site-modified Design Spectrum

coefficients of $F_a = 1.05$, $F_v = 1.538$. These coefficients scale the MCE for site class B for site class D (red, solid line). The site-modified design spectrum, then, is calculated by scaling the response of site-modified MCE spectrum (red, solid line) by $2/3$ (green, dashed line).

8.4 Results of Analyses

As an initial seismic analysis study, the response spectrum analysis was performed using the inputs that are representative of many earthquakes. It was a convenient and quick method to check the response of the tower. The inputs for the analysis are the earthquake acceleration spectra created using methods specified in the building code ASCE-7 (Figure 8.3). After the initial study, the tower was subjected to a more severe earthquake input (Bellcore zone-4 as shown in Figure 8.1) and transient dynamic analysis was performed. Although computationally more costly, this method allowed us to study the response of the tower at each time step as the earthquake progressed. The entire time history was reviewed to find maximum stress and displacement.

8.4.1 Result of Response Spectrum Analysis

First, the response spectrum analysis was performed using the site-modified design spectrum and then using the site-modified MCE spectrum as inputs in Abaqus. The inputs are shown in Figure 8.3 and discussed in the previous section. The results such as stress, S , and displacement, U , for the response spectrum analysis were reviewed. It should be noted that the results in the response spectrum analysis represent the peak magnitude of the output variable.

Using Site-modified Design Spectrum

The site-modified design spectrum in Figure 8.3 was applied in the x-, and y-directions for the analysis. For both cases, the stress level was very low compared to that of the static load cases: installation and 2k-thrust loading.

When the tower was excited in the x-direction, the large contribution of the second mode of the tower (Figure 7.4) was seen as shown in Figure 8.6. The maximum stress was found just above the strut attachment and it was approximately 6.75 *ksi*. The maximum deflection was about 7 *inches* at the top of the tower. The contribution of the fourth mode was observed as well on the strut inducing the bending induced stress on the strut.

When the tower was excited in the y-direction, the first mode contributed largely in the response of the tower as shown in Figure 8.7. The maximum stress was about 7 *ksi* near the bottom of the tower and the maximum deflection was about 11 *inches* at the top of the tower.

Figure 8.4: Response Spectrum Analysis Results for Site-modified Design Spectrum Applied in x-direction: Axial stress, S_{11} , in psi (Left); x-direction Displacement U_1 , in $inches$ (Right)

Figure 8.5: Response Spectrum Analysis Results for Site-modified Design Spectrum Applied in y-direction: Axial stress, S_{11} , in psi (Left); y-direction Displacement U_2 , in $inches$ (Right)

Using Site-modified MCE Spectrum

As mentioned earlier, the MCE spectrum represents a maximum earthquake that has 2 % probability of occurrence in 50 years. It has a higher acceleration level than the site-modified design spectrum (Recall that the design spectrum is a scaled version of the MCE spectrum). The MCE spectrum was applied in the x- and y-directions to see the structure's stress level and deflections. As in the previous case, even with the higher acceleration, the stress level was still low compared to that of the static load cases.

The tower's response was similar to the previous case but with higher stress level and deflection. For the earthquake applied in x-direction, the maximum stress was still acceptable which was approximately 10.1 *ksi*, and the deflection was approximately 10.5 *inches* (Figure 8.6). For the y-direction, the maximum stress was about 10.5 *ksi* and 16.6 *inches* of deflection (Figure 8.7).

Figure 8.6: Response Spectrum Analysis Results for Site-modified MCE Spectrum Applied in x-direction: Axial stress, S_{11} , in *psi* (Left), x-direction Displacement, U_1 , in *inches* (Right)

Figure 8.7: Response Spectrum Analysis Results for Site-modified MCE Spectrum Applied in y-direction: Axial stress, S_{11} , in *psi* (Left), y-direction Displacement, U_2 , in *inches* (Right)

8.4.2 Result of Transient Modal Dynamic Analysis

The Bellcore's earthquake input (Figure 8.1) was applied in the x- and y-direction. It has a higher acceleration level than the ASCE's acceleration inputs used in the response spectrum analysis. The stress and deflection of the tower were calculated at each time step of the input, which was 0.005 seconds. With the time step, it resulted in 6000 time steps. The computational time and effort, therefore, was great in finding the maximum stress and deflection. All 6000 solutions were reviewed to find the maximum stress and the corresponding deflection.

The maximum stress was approximately 17.1 *ksi* with maximum deflection of 14.2 *inches* when the input was applied in the x-direction (Figure 8.8). The result was similar to that of the response spectrum analysis (x-direction) in that the second and the fourth mode were the main contributors of the structure response. For the y-direction, the maximum stress was about 20 *ksi*, and about 7 *inches* for the deflection (Figure 8.9). The stress level was far below the yield strength of the tower, and, again, far below the stress levels of the static analyses performed.

Figure 8.8: Transient Dynamic Analysis Results Earthquake load Applied in x-direction: Axial stress, S_{11} , in *psi* (Left); x-direction Displacement, U_1 , in *inches* (Right)

Figure 8.9: Transient Dynamic Analysis Results Earthquake load Applied in y-direction: Mises stress, $S, Mises$, in *psi* (Left); x-direction Displacement, U_2 , in *inches* (Right)

8.5 Conclusion

The response spectrum analysis and the transient dynamic analysis were performed in order to check the structural integrity the tower assembly for earthquake loads. The wind turbine tower can easily survive the earthquake specified by the ASCE which accounts for the local soil conditions and the distance from the fault line. Furthermore, the tower can survive the earthquake specified by Bellcore, which has a much higher acceleration level. The highest stress level seen from the seismic analysis was 20 *ksi* from the transient dynamic analysis which used Bellcore earthquake as an input. As the results presented, the response of the structure is not nearly as severe as the installation and wind load cases considered in the static analyses, therefore the seismic load is not the driving load for the design.

Chapter 9

Conclusion

9.1 Analyses Results

The structural analyses of the tower for Cal Poly's 3 *kW* wind turbine tower was performed using the finite element method. Cal Poly's tower is a "tilt-up" tower making access to the wind turbine (a nacelle and a rotor assembly) easier. *Abaqus* was used for the finite element analyses. A simplified finite element model that represented the wind turbine system was created using beam, shell and inertia elements for the analysis.

For the static analysis, the 2100 *lbf* thrust load from the worst wind-condition was applied at the top of the tower. The installation case was also simulated by applying the dynamic factor of 2 to the gravitational constant, *g*, on the tower which is on the horizontal position. The tower mast can withstand both load cases with some factor of safety before yielding.

The dynamic analysis comprised of the frequency extraction, and the seismic analysis.

The natural frequencies and the mode shapes of the tower were calculated first. The natural frequencies of the tower (with the wind turbine attached) were not aligned with the possible excitation frequencies such as the rotor frequency or the blade passing frequencies during the normal operation. Therefore, undesirable dynamic excitations were avoided. However, when the turbine starts up or slows down to/from the operating speed, the tower may be excited temporarily.

The results of the modal analysis were also used in the seismic analysis. For the seismic analysis, the response spectrum analysis and the transient dynamic analysis were performed which are based on the modal superposition method. The ASCE's design and MCE acceleration spectrum were considered as the input for the response spectrum analysis which are modified for the installation site accounting for the soil condition and the distance from the fault line. The Bellcore's zone-4 earthquake input was used as the input for the transient dynamic analysis which represents more severe earthquakes than the ASCE inputs. The resulting stress level was still far less than that of the static cases. Therefore, the static cases were the driver for the other mechanical component designs.

From the results of the finite element analysis presented, it can be concluded that the wind turbine tower can withstand both static and dynamic loads considered.

9.2 Future Work

9.2.1 Load Verification

Measuring the loads on the tower using strain gages is recommended in order to verify the finite element model and to obtain load data for fatigue analyses. I suggest

placing strain gages at the location specified in Figure 9.1. The specified locations are suspected to be the high stress areas. The strain gages should be placed around the tower tube so that axial loads, and the torsional load can be determined. The torsional loads will be useful when the thrust is applied in the y-direction. The tower model predicted that some amount of torsional moment exists on strut, ginpole and the lower part of the tower for the load condition. The time-varying loads need to be recorded with the wind speed and direction in order to establish the relationship between them. Therefore, the data need to have the values of strains, the wind speed, and the wind direction along with a time stamp.

Figure 9.1: Recommended Strain-gage Location for Load Measurements

9.2.2 Improvement on Finite Element Model

In the dynamic analysis, damping is an important factor to a structure's response. A further study on the damping could be beneficial to determine the realistic damping value for the tower. The realistic damping values and modeling techniques would increase the accuracy of the analysis results.

The soil has a large influence on the structure's natural frequencies. Further improvement can be made on the finite element model of the wind turbine tower by including the effect of soil and the foundation in the model. It is called the soil-structure interaction (SSI). It is modeled using the spring, damper element at the foundation-soil interface; and the mass of the foundation and part of soil are also included assuming that the soil moves in phase with the foundation [27]. Additionally, inclusion of a simple finite element model that represents the wind turbine using the beam elements could help improve the finite element model too.

Various mechanical components were analyzed using the classical design and the strength of material methods. The calculations are attached as a reference in the appendix that follows. The contents of appendices are summarized as follows:

- Appendix A: Mechanical Component Analysis
 1. Strength analysis of bearings
 2. Strength analysis of ginpole and strut
 3. Strength analysis of various lugs, bearings, and welds
- Appendix B: Bolt Sizing
- Appendix C: Strength Analysis of Flange Welds
- Appendix D: Finite Element Analysis of Mid-Flange

- Appendix E: Cable Load for Installation
- Appendix F: Anchor Bolt Analysis
- Appendix G: Foundation Plans and Drawings

Bibliography

- [1] The European Wind Energy Association, “Economics of wind,” in *Wind Energy, The Facts (WindFacts)*, 2009.
- [2] D. Malcolm, “Windpact rotor design study:hybrid tower design,” tech. rep., National Renewable Energy Laboratory (NREL), April 2004.
- [3] International Electrotechnical Commission (IEC), “homepage.” <http://www.iec.ch>.
- [4] International Electrotechnical Commission (IEC) Technical Committee 88: Wind Turbines, *International Standard IEC61400-2, Wind turbines, Part 2: Design requirements for small wind turbines*. IEC, 2nd ed., 2006.
- [5] The European Wind Energy Association, “Technology,” in *Wind Energy, The Facts (WindFacts)*, 2009.
- [6] K. S. Dahl and P. Fuglsang, “Riso-r-1024 (en): Design of the wind turbine airfoil family riso-a-xx,” tech. rep., Riso National Laboratory, December 1998.
- [7] Ginlong Technologies, *Wind Turbine Permanent Magnet Generator/ Alternator Ginlong Technologies GL-PMG-3500*.
- [8] A. Martinez, F. Martinez, D. Nevarez, and Z. Taylor, “Wind turbine nacelle senior project.” Cal Poly, San Luis Obispo, 2009.

- [9] B. Edwards, “Composite manufacturing of small wind turbine blades,” Master’s thesis, Cal Poly, San Luis Obispo, 2009.
- [10] F. Knox and A. Valverde, “Wind turbine foundation design.” Cal Poly, San Luis Obispo, 2010.
- [11] J. Manwell, J. McGowan, and A. Rogers, *Wind Energy Explained: Theory, design, and application*, ch. 6 Wind Turbine Design. John Wiley Sons Ltd., 2nd ed., 2009.
- [12] B. A. Babcock and K. E. Conover, “Design of cost-effective towers for an advanced wind turbine,” *Wind Energy*, vol. 15, 1994.
- [13] P. Gipe, “Wind turbine tower trends.”
- [14] Det Norske Veritas, Riso National Laboratory, *Guidelines for Design of Wind Turbines*, 2nd ed., 2002.
- [15] L. Global Energy Concepts, “Windpact turbine design scaling studies technical area 3 - self-erecting tower and nacelle feasibility,” tech. rep., National Renewable Energy Laboratory (NREL), March 2001.
- [16] A. Huskey and D. Prascher, “Validation of aeroelastic model of nordtank 500/37,” tech. rep., Riso National Laboratory, November 1997.
- [17] G. C. Larsen and P. Volund, “Validation of aeroelastic model of vestas v39,” tech. rep., Riso National Laboratory, April 1998.
- [18] A. Huskey and D. Prascher, “Tower design load verification on a 1-kw wind turbine,” tech. rep., National Renewable Energy Laboratory (NREL), January 2005.
- [19] Dassault Systemes, *Abaqus Analysis User’s Manual*, 2009.

- [20] B. S. Taranath, *Wind and Earthquake Resistant Buildings: Structural Analysis and Design*, ch. 2 Seismic Design. Marcel Dekker, 1st ed., 2005.
- [21] Bellcore, *Network Equipment-Building System (NEBS) Requirements: Physical Protection (GR-63-Core, section 5)*, 1995.
- [22] E. Wilson, “Dynamic analysis using response spectrum seismic loading,” *CSI Tech Report*, vol. 15, 1994.
- [23] Strand7, *Use of Damping in Dynamic Analysis*, 2011.
- [24] I. Prowell and P. Veers, “Assessment of wind turbine seismic risk: Existing literature and simple study of tower moment demand,” tech. rep., Sandia National Laboratories, March 2009.
- [25] International Electrotechnical Commission (IEC) Technical Committee 88: Wind Turbines, *International Standard IEC61400-1, Wind turbines, Part 1: Design requirements*. IEC, 3rd ed., 2007.
- [26] R. D. Cook, *Finite Element Modeling for Stress Analysis*. John Wiley and Sons, Inc., 1995.
- [27] N. Bazeos, G. Hatzigeorgiou, I. Hondros, H. Karamaneas, D. Karabalis, and D. Beskos, “Static, seismic and stability analyses of a prototype wind turbine steel tower,” *Engineering Structures*, vol. 24, 2002.
- [28] R. Huston and H. Josephs, *Practical Stress Analysis in Engineering Design*, ch. 20 Flanges. CRC Press, 3rd ed., 2009.

Appendix A

Mechanical Component Analysis

Bearing Block Calculation - Bearing Lug, Plate and Welds around the block

Bearing Block

A36 Material Property

$S_y = 36300 \text{ psi}$

$S_u = 58000 - 79800 \text{ psi}$

$E = 29 \text{ Msi}$

$\nu = 0.260$

$\rho = 0.284 \text{ lb/in}^3$

New Material

ASTM A572 Grade-50 Steel	
Ultimate Tensile Strength (ksi)	65.3
Yield Tensile Strength (ksi)	50.0
Poisson's Ratio	0.26
Young's Modulus (Msi)	29.2
Shear Modulus (Msi)	11.6
Bulk Modulus (Msi)	20.3
Elongation at Break	0.21
Density (lbm/in ³)	0.284

Anchor Bolts

FF using $1\frac{1}{2}$ " bolts

$\sigma_b = \frac{B/2}{2A_b}$ At for $1\frac{1}{2}$ " bolt (UNC)

$A_b = 0.969 \text{ in}^2$

$\sigma_b = \frac{20000/2}{2 \cdot (0.969)} = 5160 \text{ psi}$

$\sigma_b < \phi_{\text{proof}}$ Grade 5 UNC (= 74 Kips)

(90% $\phi_{\text{proof}} = 67 \text{ Kips}$)

$\eta = \frac{67}{5} = 13.4$

Preliminary

Using $7/8$ " bolts? $\rightarrow A_t 7/8" = 0.462 \text{ in}^2$

$\sigma_b = \frac{10000}{2 \cdot (0.462)} = 10.8 \text{ ksi} < \phi_{\text{proof, min}} = 85 \text{ ksi}$

SAT Grade 5

$7/8"$ OK, $\eta \approx 9$

* B increased to 35 Kips

$\rightarrow \sigma \uparrow$ by 1.75

Base Weld

$T' = \frac{V}{A} = \frac{B/2}{0.707 \cdot h \cdot l} = \frac{10000}{0.707 \cdot h \cdot (10")}$

Assume $3/16$ " weld leg (L)

$T' = 1886 \text{ psi}$

* B increase to 35 Kips

increased by factor of 1.75

$T' = 1886 (1.75) = 3300 \text{ psi}$

$$\tau' = 1886 \text{ psi}$$

Shear stress allowable,

$$0.40 S_y \text{ base metal} \rightarrow 36 \text{ ksi} (0.40) = 14.4 \text{ ksi}$$

$$0.30 S_{ut} \text{ weld metal} \rightarrow 60 \text{ ksi}, S_{ut} = 60 \text{ ksi}$$

$$0.30 (60 \text{ ksi}) = 18 \text{ ksi}$$

$$n = \frac{14.4}{1.9} = 7.6$$

$$\tau' = (1886)(1.75) = 3300 \text{ psi}$$

$$\tau' < 14.4 \text{ ksi} \rightarrow \text{OK}$$

• Anchor Plate

$$M = 10000 (1.5'') = 15000 \text{ lb-in}$$

$$\sigma_{\max} = \frac{Mc}{I} = \frac{(15000 \text{ lb-in})(0.5'')}{\frac{1}{12} (12'')(1'')^3} = 7500 \text{ psi}$$

$$\tau_{\max} = \frac{3}{2} \cdot \frac{V}{A} = \frac{3}{2} \cdot \frac{10000}{12 \cdot 1} = 1250 \text{ psi}$$

$$S_y = 46 \text{ ksi} \quad \checkmark \quad \text{we are good.}$$

$$\sigma_{\max} = 7500 (1.75) = 13 \text{ ksi}$$

$$\tau_{\max} = 2.2 \text{ ksi}$$

• Side Plate Tension Tear out.

$$\sigma_{t \max} = K_t \sigma_o = K_t \frac{B/2}{(w-d)t}$$

$$K_t \Rightarrow \frac{d}{w} = \frac{9.5}{12.7} = 0.75$$

$$h/w = \frac{6}{12.7} = 0.47$$

$$K_t \approx 2 \text{ from } \left\{ \begin{array}{l} \text{Fig A-15-12 (Pg 1009)} \\ \text{Shigley Ed 8.} \end{array} \right\}$$

$$\sigma_{t \max} = 2 \cdot \frac{10000}{(12.7-9.5)(1'')} = 6250 \text{ psi}$$

If have clearance, use $K_t = 3$ (50% ↑)

$$\text{then } \sigma_{t \max} = 9375 \text{ psi} \Rightarrow S_y (36 \text{ ksi})$$

If thickness is .75", $K_t = 3$

$$\sigma_{t \max} = 12.5 \text{ ksi}$$

with new load

$$\sigma_{t \max} = (1.75)(9.4 \text{ ksi})$$

$$\sigma_{t \max} = 16.5 \text{ ksi} \quad \checkmark$$

• Bearing Weld

↑ $B/2$

$$\tau' = \frac{F}{A_t} = \frac{B/2}{\pi \cdot D \cdot 0.707h}$$

Assume $h = \frac{3}{8}"$

$$\tau' = \frac{10k}{\pi (9.5") (0.707) (\frac{3}{8})} = 1264 \text{ psi} < 14.4 \text{ ksi} \quad (0.4 S_y)$$

$$n = \frac{14.4}{1.26} \approx 11$$

new load

$$\tau' = 2.3 \text{ ksi} \quad \checkmark$$

• Big Tube Base Weld

$$M = 2000 \text{ lb} \cdot (70 \text{ ft}) (12") / \text{ft} \rightarrow V = \frac{2000 (70 \times 12)}{14} = 120 \text{ kip}$$

Including weight $\rightarrow 122.3 \text{ kip}$

$$\tau' = \frac{122300}{2 \cdot (\pi \cdot 8" \cdot 0.707 \cdot h)} \rightarrow \text{with } h = \frac{1}{2}" \Rightarrow \tau' = 6883 \text{ psi}$$

$$h = \frac{3}{8}" \Rightarrow \tau' = 9177.11 \text{ psi}$$

$\frac{1}{2}"$ used in fab

both $> 14.4 \text{ ksi}$

new load

$$\rightarrow \tau' = 12 \text{ ksi} \quad \checkmark$$

Ginpole• Ginpole Buckling

5" OD, 4.5" ID ✓
 $t = 0.25"$

$$A = \pi/4 [5^2 - 4.5^2] = 3.73$$

$$I = \frac{\pi}{64} [5^4 - 4.5^4] = 10.55 \text{ in}^4 \rightarrow K = \sqrt{\frac{I}{A}} = \sqrt{\frac{10.55}{3.73}}$$

$$E = 29 \text{ Msi}$$

$$K = 1.68$$

$$\left(\frac{L}{K}\right)_1 = \sqrt{\frac{2\pi^2 E I}{S_y}} \quad u=1 \text{ round-round}$$

$$= \sqrt{\frac{2\pi^2 \cdot 29 \times 10^6}{36 \times 10^3}} = 126.1 < 146 \text{ long column, (check)}$$

$$\frac{L}{K} = \frac{240.5}{1.68} = 143$$

$$P_{cr} = \frac{\pi^2 E I}{L^2} = \frac{\pi^2 [29 \times 10^6] \pi/64 [5^4 - 4.5^4]}{240.5^2} \approx 52.2 \text{ kips} \checkmark$$

strut

$l = 335.27"$, same cross section

$$P_{cr} \approx 27 \text{ kips} \checkmark$$

max load anticipated $\Rightarrow$ strut = 17 kips
 ginpole = 12.4 kips

Load - 2000 lbf thrust

$$F_{strut} = 9571 \text{ Comp (16P)}$$

$$F_{pin} = 6527.6 \text{ Tens.}$$

$$F_{strut} = 8934 \text{ Comp}$$

$$F_{pin} = 4100 \text{ T.}$$

$$F_s = 8900 \text{ Tens.}$$

$$F_G = 6100 \text{ Comp}$$

$$F_s = 8943 \text{ T.}$$

$$F_G = 3690 \text{ C.}$$

pull-up

• Grip Pole l_{g3}

$$\tau = \frac{F_g/2}{(0.707)(\frac{3}{8})(6)} = 2200 \text{ psi}$$

$$F_{g \text{ max}} \approx 7 \text{ kips}$$

$$\tau' = \frac{3500}{0.707 \cdot 6 \cdot \frac{3}{8}} = 2200 \text{ psi}$$

$$\tau' = \frac{7000}{[(0.707)(0.25)(9.5)] 4} = 1042.2 \text{ psi}$$

$$\tau' \approx 2 \text{ ksi for } F_g = 14 \text{ kips}$$

$$\sigma_{\text{max}} = K_t \sigma_o = K_t \frac{3500}{(0.75)(5-1)''}$$

$$S_y = 36 \text{ ksi}$$

$$P_{\text{max}} = \frac{36 \text{ ksi}}{K_t} (0.75)(4)'' = 32 \text{ kips}$$

$$K_t \rightarrow \frac{h}{w} = \frac{2.5}{5} = 0.5$$

$$\frac{D}{W} = \frac{2}{5} = 0.4$$

$$K_t = 3.14$$

$$\sigma_{\text{max}} = 3966 \approx 4 \text{ ksi, if use } \frac{1}{8}'' \text{ thickness } \sigma_{\text{max}} = 12 \text{ ksi}$$

Strut

$$\tau' = \frac{10000}{[(0.707)(0.25)(10)] 4} = 1414.43 \text{ psi}$$

$$\text{With } F = 17 \text{ kips}$$

$$\tau = 2.4 \text{ ksi (low)}$$

$$\sigma_{\text{max}} = K_t \sigma_o$$

$$K_t \rightarrow \frac{h}{w} \approx \frac{2.5}{5} = 0.5$$

$$\frac{D}{W} \approx \frac{1.25}{5} = 0.25$$

$$\sim 4.7$$

$$\text{Assume } 10k \text{ for } F_g$$

$$\sigma_{\text{max}} = (4.7)(15k/2) = 12.5 \text{ ksi}$$

* new load

$$\sigma_{\text{max}} = 14.2 \text{ ksi}$$

Assume 15K for F_3

$$\sigma_{max} = \frac{(4.7)(15K/2)}{(0.75)(5-1.15)} = \underline{12.5 KSI}$$

$$\frac{w}{L} \approx \frac{1.25}{5} = 0.25'$$

* new load

$$\sigma_{max} = 14.2 KSI$$

$$< F = 17 KIPS$$

Welding

Assume $w = 3/8"$

$$\tau'_1 = \frac{15000 \sin 47}{(0.707)(3/8)(9.86") \times 2} = 2098.3 \text{ psi}$$

$$\tau'_2 = \frac{15000 \cos 47}{"} = 1956.7 \text{ psi}$$

$$\left[\begin{array}{c} \leftarrow b \rightarrow \\ \leftarrow d \rightarrow \end{array} \right] \Rightarrow \left[\begin{array}{l} A = 1.4144hd \\ I_u = \frac{d^3}{6} \\ I = 0.707hI_u \end{array} \right]$$

$$\tau'' = \frac{Mc}{I}$$

$$M = (15000 \sin 47)(3.53)$$

$$c = 9.86/2 = 4.93"$$

$$I = 0.707(3/8) \cdot \frac{9.86^3}{6} = 42.397$$

$$\tau'' = \frac{(15000 \sin 47)(3.53)(4.93)}{42.357} = 4507.3 \text{ psi}$$

$$\tau = \sqrt{2098^2 + (1957 + 4507)^2}$$

$$\tau = 6795.95 \text{ psi} > 14.4 KSI, \text{ OK.}$$

Flanges

$$\tau' = \frac{2000}{\pi D (0.707h)} \approx 370$$

$$\tau'' = \frac{Mr}{I} = \frac{(2000)(14)(7")}{(0.707h) I_u}$$

$$I_u = \pi r^3 \cdot 3/8$$

$$= 5.5K (\approx 5499)$$

$$\tau = \sqrt{370^2 + 5500^2} \approx \underline{5500 \text{ psi}}$$

Weld Flange

$$\tau' = 370 \text{ (same as above)}$$

$$\tau' = \frac{Mr}{I} = \frac{2000 (440) (11.5)}{0.707 \left(\frac{3}{8}\right) \pi (5)^3} = 97k?$$

$$I_u = \pi r^3$$

If 0.5 leg $\rightarrow \tau' = 73k$

If 0.75 leg $\rightarrow \tau' = 49k$

- reinforced ring? (where strut attaches)

$$\tau' = \frac{V}{A} = \frac{15k \sin 47}{(1.414 \pi \cdot h \cdot r) \times 2}$$

Assuming $\frac{3}{8}$ " weld leg

$$\tau' = \frac{15000 \sin 47}{(1.414 \pi) \left(\frac{3}{8}\right) (6) \times 2} = 548.8 \text{ psi}$$

$$\tau'' = \frac{M_c}{I} \quad I_u = \pi r^3$$

Assume one weld carries all the bending moment load

$$\tau'' = \frac{\frac{M}{c}}{\frac{0.707 \left(\frac{3}{8}\right) (\pi) (5.75)^3}{I}} = 3900 \text{ psi}$$

$$\tau \approx 4500 \text{ psi} \geq 14.4 \text{ ksi} \quad \text{OK}$$

low, 17kips ok

- Strut, plate Tear-out

$$\frac{h}{w} = \frac{2.5}{5} = 0.50$$

$$\frac{d}{w} = \frac{1.25}{5} = 0.25$$

$$K_t = 4.3$$

$$\sigma_{max} = \sigma_0 K_t = 4.3 \frac{15000/2}{(0.5)(5 - 1.25)} = 17.2 \text{ ksi}$$

$$\sigma_{max} = 17.2 \text{ ksi} > 36 \text{ ksi} \quad \text{OK}$$

$$u \approx 2.$$

$$F = 17 \text{ kips}$$

$$\sigma_{max} = 19.5 \text{ ksi} \quad \checkmark$$

All tabs

Assume same loading condition $\rightarrow 15000 \text{ kips}$ when lifting up.

$$15k \leftarrow \text{---} \rightarrow 15k$$

$$\tau = \frac{15000}{2A_t} = \frac{7500}{2(0.606)} = 12.4 \text{ ksi}$$

Assume 1" bolt, A578 grades

$$\left\{ \begin{array}{l} A_t = 0.606 \\ \sigma_{proof \min} = 85 \text{ ksi} \end{array} \right.$$

OK

$$\rightarrow 17 \text{ kips}$$

$$\rightarrow \tau = 14 \text{ ksi}$$

Apply to all bolts in strut, girders

Anchor Plate Analysis

Wednesday, January 13, 2010
3:00 PM

The weld → Assume 15Kips directly contribute to Torsioning the weld

Primary shearing $\tau' = \frac{V}{A}$

Assume $\frac{1}{4}$ " leg, Fillet, 4 spot, 2 shown, 2 on the other side

$$\tau' = \frac{15K/2}{0.707(0.25)(18.7) \times 2} = 1134.6 \text{ psi}$$

→ small - OK ✓

Secondary shear $\tau'' = \frac{Tr}{J}$

$$A = 1.41bd, \quad J_u = \frac{d(3b^2 + d^2)}{6}$$

$$r = \left[\left(\frac{18.7}{2} \right)^2 + \left(\frac{2.5}{2} + \frac{0.25}{2} \right)^2 \right]^{1/2} = 9.45"$$

$$\tau'' = \frac{(7500)(9.45")}{(0.707)(0.25) \left[\frac{18.7 \{ 3(2.5 + 0.25)^2 + 18.7^2 \}}{6} \right]}$$

$$\tau'' = 13.11 \text{ ksi} > 14.4 \text{ ksi}, \text{ OK but low F.S.}$$

- plate bolts

Assume that three bolts (solid line) carry all the load (load path)

$$T = \frac{15K}{1.}$$

✓

$$L = \frac{15K}{6A_t}$$

Set S_{prior} to 25 KSI ? ($S_y \geq 50K$) for Grade 2

$$A_t = \frac{15K}{6\tau} = 0.1''$$

$$A_t = 0.1063 \text{ for } 7/16'' \leftarrow \text{pick}$$

$$= 0.1419 \text{ for } 1/2''$$

Appendix B

Bolt Sizing

Bolt Sizing for Wind Turbine Tower

Bolt Stress

Load Case (induces maximum Load)	Location	Tensile Load (kips)	Shear Load (kips)	Nominal Bolt Diameter (in)	Joint Thickness (in)	Stress (ksi)	Bolt Material (proof/yield/ultimate strength, ksi)	Factor of Safety (to Proof Strength)	Fi (kips)*	Torque (in- lbf)
2k thrust	A: Top Flange (bolt group analysis)	1.5	0.25	0.375	1.5 (check w/ Jim)	55.04 (Mises)	Grade 5 (85/92/120)	1.5	2.53	190
2k thrust	B: Mid Flange (bolt group analysis)	17	0.13	0.5	1.5	119.8	ASTM A490 type 3 (120/130/150)	1.25 to ultimate	6.72	672
tilt-up	C: Strut to Tower (single)	0 (lug)	21.40	1	2.25	19.42	ASTM A325 type 3 (85/92/120)	4.4	21.5	4300
tilt-up	D: Ginpole to Tower (single)	0 (lug)	20.46	1	2.25	18.57	ASTM A325 type 3 (85/92/120)	4.6	21.5	4300
tilt-up	E: Strut-Ginpole (single)	0 (lug)	21.40	1	1.75	19.42	ASTM A325 type 3 (85/92/120)	4.4	21.5	4300
tilt-up	F: Ginpole to Rear Anchor (bolt group)	0 (lug)	17.74	1	1.5 (check w/ Jim)	16.10	ASTM A325 type 3 (85/92/120)	5.3	21.5	4300

* Preloaded to
80% of yield strength
of bolt material

2k thrust: 2000 lbf thrust is applied to the wind turbine (71-ft above ground)

tilt-up: case where the tower is pulled by a cable

Factor of safety (dynamic effect) of 2 is included in the load

Hardware List

Location	Bolt	Nut**	Quantity
A: Top Flange (bolt group analysis)	3/8"-16 UNC X 2 1/4" SAE GRADE 5	Not through	8
B: Mid Flange (bolt group analysis)	1/2"-20 UNF X 2 1/4" ASTM A490 Type 3	Heavy Hex (ASTM A563) Grade DH3	16
C: Strut to Tower (single)	1"-8 UNC X 3 3/4" ASTM A325 Type 3	Heavy Hex (ASTM A563) Grade C3	1
D: Ginpole to Tower (single)	1"-8 UNC X 3 3/4" ASTM A325 Type 3	Heavy Hex (ASTM A563) Grade C3	1
E: Strut-Ginpole (single)	1"-8 UNC X 3 1/4" ASTM A325 Type 3	Heavy Hex (ASTM A563) Grade C3	1
F: Ginpole to Rear Anchor (bolt group)	1"-8 UNC X 3" ASTM A325 Type 3	Heavy Hex (ASTM A563) Grade C3	2

** From AISC, Guide to Design Criteria for Bolted and Riveted Joints, Second Edition

Nuts for A325 bolts must be heavy hex and are required to meet ASTM finish, should be used. For bolt Types 1 and 2, galvanized, nut grade DH, galvanized, is required. Nut grade C3 is to be used with bolt Type 3. Grades 2 and 2H nuts, as specified in ASTM A194, and grades D and DH nuts, as specified in ASTM A563, are acceptable alternatives for grade C nuts. Grade 2H nuts (ASTM A194) are an acceptable alternative for grade DH nuts, and type DH3 nuts can be used in place of C3 nuts.

Heavy hex nuts are also required for A490 bolts. Grade DH heavy hex nuts shall be furnished for use with Type 1 and 2 bolts, but grade 2H heavy hex nuts (ASTM A194) are also acceptable. Type 3 A490 bolts require grade DH3 (ASTM A563) heavy hex nuts.

BOLT GROUP ANALYSIS

Using the Elastic Method for up to 25 Total Bolts

Job Name:	Top-flange Bolt	Subject:	Wind Turbine Tower		
Job Number:		Originator:		Checker:	

Input Data:

Number of Bolts, $N_b = 8$

Bolt Coordinates:

[illegible]

Results:

Bolt Reactions (k)

[illegible]

Bolt Group Properties:

Xc =	4.040	in.
Yc =	4.040	in.
Ix =	65.29	in. ²
Iy =	65.29	in. ²
J =	130.57	in. ²
Ixy =	0.00	in. ²
θ =	0.000	deg.

No. of Load Points, N = 1

Load Point Data: Point #1

X-Coordinate (in.) =	4.040			
Y-Coordinate (in.) =	4.040			
Z-Coordinate (in.) =	12.000			
Axial Load, Pz (k) =	0.00			
Shear Load, Px (k) =	0.00			
Shear Load, Py (k) =	2.00			
Moment, Mx (in-k) =	0.00			
Moment, My (in-k) =	0.00			
Moment, Mz (in-k) =	0.00			

Σ Loads @ C.G. of Bolt Group:

$\Sigma P_z =$	0.00	kips
$\Sigma P_x =$	0.00	kips
$\Sigma P_y =$	2.00	kips
$\Sigma M_x =$	-24.00	in-k
$\Sigma M_y =$	0.00	in-k
$\Sigma M_z =$	0.00	in-k

Bolt Reaction Summary:

Rz(max) =	1.49	kips
Rz(min) =	-1.49	kips
Rh(max) =	0.25	kips

BOLT GROUP ANALYSIS Using the Elastic Method for up to 25 Total Bolts			
Job Name:	Mid-flange Bolt	Subject:	Wind Turbine Tower
Job Number:		Originator:	Checker:

Input Data:

Number of Bolts, Nb = 16

Bolt Coordinates:

	Xo (in.)	Yo (in.)
#1:	12.800	6.400
#2:	12.313	8.849
#3:	10.925	10.925
#4:	8.849	12.313
#5:	6.400	12.800
#6:	3.951	12.313
#7:	1.875	10.925
#8:	0.487	8.849
#9:	0.000	6.400
#10:	0.487	3.951
#11:	1.875	1.875
#12:	3.951	0.487
#13:	6.400	0.000
#14:	8.849	0.487
#15:	10.925	1.875
#16:	12.313	3.951

Results:

	Bolt Reactions (k)	
	Axial Rz	Shear Rh
#1:	0.00	0.13
#2:	6.46	0.13
#3:	11.93	0.13
#4:	15.59	0.13
#5:	16.87	0.13
#6:	15.59	0.13
#7:	11.93	0.13
#8:	6.46	0.13
#9:	0.00	0.13
#10:	-6.46	0.13
#11:	-11.93	0.13
#12:	-15.59	0.13
#13:	-16.88	0.13
#14:	-15.59	0.13
#15:	-11.93	0.13
#16:	-6.46	0.13

BOLT GROUP PLOT

Bolt Group Properties:

Xc =	6.400	in.
Yc =	6.400	in.
Ix =	327.68	in.^2
Iy =	327.68	in.^2
J =	655.36	in.^2
Ixy =	0.00	in.^2
θ =	0.000	deg.

No. of Load Points, N = 1

Load Point Data: Point #1

X-Coordinate (in.) =	6.400
Y-Coordinate (in.) =	6.400
Z-Coordinate (in.) =	432.000
Axial Load, Pz (k) =	0.00
Shear Load, Px (k) =	0.00
Shear Load, Py (k) =	2.00
Moment, Mx (in-k) =	0.00
Moment, My (in-k) =	0.00
Moment, Mz (in-k) =	0.00

Σ Loads @ C.G. of Bolt Group:

Σ Pz =	0.00	kips
Σ Px =	0.00	kips
Σ Py =	2.00	kips
Σ Mx =	-864.00	in-k
Σ My =	0.00	in-k
Σ Mz =	0.00	in-k

Bolt Reaction Summary:

Rz(max) =	16.87	kips
Rz(min) =	-16.88	kips
Rh(max) =	0.13	kips

BOLT GROUP ANALYSIS

Using the Elastic Method for up to 25 Total Bolts

Job Name:	gin pole plate	Subject:	Wind Turbine Tower		
Job Number:		Originator:		Checker:	

Input Data:

Number of Bolts, Nb = 2

Bolt Coordinates:

[illegible]

Results:

Bolt Reactions (k)

[illegible]

Bolt Group Properties:

Xc =	6.850	in.
Yc =	0.000	in.
Ix =	0.00	in.^2
Iy =	93.84	in.^2
J =	93.84	in.^2
Ixy =	0.00	in.^2
θ =	0.000	deg.

No. of Load Points, $N =$ 1

Load Point Data: Point #1

X-Coordinate (in.) =	0.000			
Y-Coordinate (in.) =	14.640			
Z-Coordinate (in.) =	0.000			
Axial Load, Pz (k) =	0.00			
Shear Load, Px (k) =	-7.52			
Shear Load, Py (k) =	8.03			
Moment, Mx (in-k) =	0.00			
Moment, My (in-k) =	0.00			
Moment, Mz (in-k) =	0.00			

Σ Loads @ C.G. of Bolt Group:

$\Sigma P_z =$	0.00	kips
$\Sigma P_x =$	-7.52	kips
$\Sigma P_y =$	8.03	kips
$\Sigma M_x =$	0.00	in-k
$\Sigma M_y =$	0.00	in-k
$\Sigma M_z =$	55.09	in-k

Bolt Reaction Summary:

Rz(max) =	0.00	kips
Rz(min) =	0.00	kips
Rh(max) =	8.87	kips

Top Flange - Sample Bolt Calc

Wednesday, September 29, 2010
1:18 PM

Max load at a bolt

$$F_e = 1.5 \text{ kips (Tensile)}$$

$$V = 0.25 \text{ kips (shear)}$$

Joint Stiffness

Assuming long bolt,

$$k_b = \frac{A_b E_b}{l}$$

A_b = Area of bolt shank

E_b = E of bolt

l = length of bolt in the flange

$$k_b = \frac{\pi \cdot \left(\frac{3}{8}\right)^2 \cdot (29.8 \text{ E}^6)}{1.5"} = \underline{\underline{2.194 \text{ E}6 \text{ lbf/in}}}$$

$$k_m = \frac{\pi \cdot E \cdot d}{2 \ln \left[5 \left(\frac{l + 0.5d}{l + 2.5d} \right) \right]}$$

$$E = 28.5 \text{ E}6$$

$$d = 3/8$$

$$l = 1.5$$

$$k_m = 13.71 \text{ E}6 \text{ lbf/in}$$

minimum preload

$$F_{i\min} = F_e \left(\frac{k_m}{k_m + k_b} \right) - F_i$$

= 0 at separation

$$F_{i\min} = F_e \left(\frac{k_m}{k_m + k_b} \right) = 0.862 (1.5 \text{ kips})$$

$$F_{i\min} = 1.29 \text{ kips}$$

Install Torque and Stress

Check Grade 5 $\rightarrow S_y = 92 \text{ ksi}$
 $S_u = 120 \text{ ksi}$
 $S_p = 85 \text{ ksi}$

"Wrench on Stress"

$\rightarrow 80\%$ of yield strength.

Mises $\rightarrow \begin{cases} \sigma_e' = \sqrt{\sigma^2 + 3\tau^2} \\ \tau = \frac{16 \cdot T}{\pi d r^3} \text{ --- Torque, } T = K \cdot d \cdot F_i \\ \sigma = F_i / A_t \end{cases}$ $\swarrow$ Use 0.20 for K

Set $\sigma_e' = 0.8 S_y$ and find F_i

Found $F_i = \underline{\underline{2.53 \text{ k}}}$ (for 80% yield)

Torque = T

$T \approx K \cdot d \cdot F_i$ use $K = 0.20$

$= (0.20) \left(\frac{3}{8} \right) (2.53 \text{ kips})$

$T = \underline{\underline{190 \text{ in-lb}}}$

Check operation load, stresses

$F_b = \left(\frac{K_b}{K_b + K_m} \right) F_e + F_i$

$\frac{K_b}{K_b + K_m} = 0.138$

$F_b = (0.138)(1.5 \text{ k}) + 2.53 \text{ k}$

$F_b = 4.2 \text{ kips}$

$\sigma_b = \frac{F_b}{A_t} = \frac{4.2 \text{ k}}{0.0775} = 54.2 \text{ ksi}$

$\tau = \frac{V}{A_r}$ assuming bolt thread extend into shear plane

$$\tau = \frac{V}{A_r} \quad \text{assuming bolt thread } \dots \dots \dots$$

$$= \frac{0.25k}{0.0678} = 3.69 \text{ ksi (Ave)}, \quad \text{max } \tau = 1.5(3.69 \text{ ksi}) = 5.54 \text{ ksi}$$

$$\sigma' = \sqrt{\sigma^2 + 3\tau^2} = \sqrt{54.2^2 + 3(5.54)^2}$$

$$\boxed{\sigma' = 55.04 \text{ ksi}}$$

$$n_{\text{proof}} = \frac{85 \text{ ksi}}{55.04 \text{ ksi}} = \underline{\underline{1.55}} \quad \checkmark$$

Shear Joints - Sample Bolt Calc.

Wednesday, September 29, 2010
2:22 PM

Applicable to
location
D, E, F, G

Assume : Bolt threads extend into shear plane

$$\tau = \frac{F}{2A_r}$$

Set $\tau = 0.577 S_p \rightarrow S_p$: proof strength

For ASTM A490 $S_p = 120 \text{ ksi}$

A325 $S_p = 85 \text{ ksi}$

For 1" UNC-8 $A_r = 0.551 \text{ in}^2$

$$F_{allow} = \tau \cdot 2A_r = 0.577 S_p \cdot 2 \cdot A_r = 1.154 S_p \cdot A_r$$

A490, 1" $\rightarrow F_{allow} = 76.3 \text{ kips}$

A325, 1" $\rightarrow F_{allow} = 54.1 \text{ kips} \checkmark$

Preload and Torque

"Wrench on Stress" $\rightarrow$ load to 80% yield strength

$$\sigma = \frac{F_i}{A_t} ; \tau = \frac{16T}{\pi d_r^3} ; T = K \cdot d \cdot F_i$$

$$\sigma' = \sqrt{\sigma^2 + 3\tau^2} \rightarrow \text{set to } 0.8 S_y \text{ to find } F_i$$

$$\sigma' = \sqrt{\sigma^2 + 3\tau^2} \rightarrow \text{set to } 0.8 S_y \text{ to find } F_c$$

$$\sigma = F_c \left[\left(\frac{1}{A_t} \right)^2 + 3 \cdot \left(\frac{16 \cdot [k \cdot d]}{\pi d_r^3} \right)^2 \right]^{1/2}$$

$$F_c = \sigma' / [\sim]^{1/2} = \frac{0.80 S_y}{[\sim]^{1/2}}$$

$$F_c = 21.5 \text{ kips}$$

$$T = k \cdot d \cdot F_c = (0.207 \text{ in}) (21.5 \text{ k}) = 4.30 \text{ in-lbf}$$

Appendix C

Strength Analysis of Flange Welds

Welds Calculation for New Flange Design

Monday, May 31, 2010
5:46 PM

LOADS

Extreme wind

$$M = 864000 \text{ in-lbf}$$

$$V = 2000 \text{ lbf}$$

Factor of Safety already included.
in 2000 lbf load.

tilt - up

$$(mg)_{\text{pole}} \rightarrow (mg)_{\text{tower}} = 4500 \text{ lbf.}$$

say ~ 2500 lbf for upper pole

$$M = (500 \times 432) + (2500 \times 420/2)$$

$$M = 741000 \text{ in-lbf}$$

$$V = 3000 \text{ lbf}$$

With Factor of Safety of 1.5 applied to load,

$$M = 1111500 \text{ in-lbf}$$

$$V = 4500 \text{ lbf}$$

New design

New idea:

Detail A:

Mechanical Properties	ASTM A572 Grade 5	ASTM A36 HR
Ultimate Tensile Strength	65.3 Ksi	58-79 Ksi
Yield Tensile Strength	50.0 Ksi	36.3 Ksi
Poisson's Ratio	0.26	0.26
Young's Modulus	29.2 Msi	29.0 Msi
Shear Modulus	11.6 Msi	11.5 Msi
Bulk Modulus	20.3 Msi	20.3 Msi
Elongation at break	21%	20-23%
Density	0.284 lbm/in ³	0.284 lbm/in ³

Pasted from <file:///C:/Users/Tom/Desktop/Thesis%20stuffs/References/A572%20and%20A36%20Steel%20Properties.xlsx>

Detail A:

1/2

1/2

pole wall thickness

Verify welds. $\rightarrow$ Hand calc.

Verify flange $\rightarrow$ FEA, Hand calc?

$$A = 1.414 \pi L r$$

$$I_u = \pi r^3$$

$$I = 0.707 I_m$$

$$\tau' = \frac{V}{A}$$

$$\tau'' = \frac{M_r}{I_x}$$

Load 1: extreme wind (2000lbf Thrust)		Load 2: tilt-up, Factor of Safety of 1.5	
V	1000 lbf	V	2250 lbf
M	432000 in-lbf	M	555750 in-lbf

r	5	in
h	0.5	in
A	11.10553	in ²
I _u	392.6991	in ³
I	138.8191	in ⁴

Weld Stress for Load 2	
Primary Shear	τ' 202.6018 psi
Secondary Shear	τ'' 20017.05 psi
Total Stress	τ 20018.08 psi

r	5	in
h	0.375	in
A	8.329148	in ²
I _u	392.6991	in ³
I	104.1143	in ⁴

Weld Stress for Load 2	
Primary Shear	τ' 270.1357 psi
Secondary Shear	τ'' 26689.41 psi
Total Stress	τ 26690.77 psi

Welds, Bolts Page 2

stress in the weld metal as in the parent metal.

For these ASTM steels, $S_y = 0.5 S_u$. - Shigley Mechanical Engineering Design, Chapter 9 -5 strength of welded joints

Since we are using ASTM 572-50, $S_u = 65 \text{ ksi}$

$S_y = 0.5(65 \text{ ksi}) = 32.5 \text{ ksi}$ according to code.

So permissible stress is 32 ksi

The allowable load on a welded seam can be calculated by the following equation:-

$$P_a = F_{su} \text{ or } F_{ST} (Lt) \text{ --- D2.1}$$

where,

P_a = allowable load in lbs.

F_{su} and F_{ST} from Table D2.2

L = length of welded seams in inches

t = thickness of thinnest material joined by the weld in the case of lap welds between two steel plates or between plates and tubes. (inches)

t = average thickness in inches of the weld metal in the case of tube assemblies, but not to be greater than 1.25 times the thickness of the welded stock.

Material	Heat treatment subsequent to welding	F_{su} , ksi	F_{tu} , ksi
Carbon and alloy steels	None	32	51
Alloy steels	None	43	72
Alloy steels	Stress relieved	50	85
Alloy steels	Stress relieved	60	100
Steels	Quench and temper		
4130	125 ksi	63	105
4140	150 ksi	75	125
4340	180 ksi	90	150

From Bruhn. Analysis of Flt. Structures.

Assume same weld diameter (close)

$$D = 10''$$

$$L = \pi D, \text{ since two welds}$$

$$L = 2\pi D \approx 62.8''$$

$$t = 0.5''$$

$$P_a = Lt S_{all} = (62.8'')(0.5'') S_{all} \Rightarrow \text{Pick } 32 \text{ ksi}$$

$$= \underline{1004800 \text{ lbf}} = 1 \text{ M lbf. ??}$$

$$\sum F \cdot d = M$$

$$\uparrow ?$$

$$F = \frac{M}{D} = \frac{840000}{5}$$

$$F = 168 \text{ kips}$$

$$\sqrt{3k^2 + 168k^2} = 169 \text{ k}$$

$$A = 1.414 \pi \cdot h \cdot r$$

$$I_u = \pi r^3$$

$$A = 1.414 \cdot \pi \cdot 0.5'' \cdot 5''$$

$$= 11,105 \text{ in}^2$$

$$I_u = \pi \cdot r^3 = 392.7 \text{ in}^3$$

$$I = 0.707 h \cdot I_u = (0.707)(0.5)(392.7) = 138.82 \text{ in}^4$$

$$\tau' = \frac{M_r}{I} = \frac{(840,000)(5'')}{138.82} = \underline{\underline{30255 \text{ psi}}} \quad \checkmark$$

$$\tau' = \frac{F}{A} = \frac{2000}{11.1} = 181.8 \text{ (small.)}$$

Appendix D

Finite Element Analysis of Mid-Flange

For bolted flanges, hub stress adjacent to a flange ring is usually the largest stress. So, it is the main concern for flange design [28]. An axis-symmetric flange was modeled in Abaqus to perform a stress analysis on the mid-flange subject to a maximum bolt load, 17 *kips* (from the 2k-thrust load).

D.1 Model

The key features of the flange model are summarized as follows:

- Modeling method: Axis-symmetric analysis
- Included Parts: section of a flange; the adjacent section of the tower mast; and welds between them
- Elements: Axis-symmetric elements (for all parts)

- Kinematic Constraints: Tie (between welds and all other parts)
- Materials: ASTM A572 steel, both elastic and plastic material properties were included in the model (non-linear analysis)
- Boundary Conditions: $U2 = 0$ (global y-direction). It is shown as orange triangles in Figure D.1. It simulated a bolt holding down the flange.
- Loads: A bending moment applied to the flange induces a different reaction load at each bolt. 17 *kips* was the largest bolt load found. It was assumed that the largest bolt load was applied equally around the flange. The equivalent load was calculated as a pressure load, and applied on the tower mast as shown in Figure D.1

D.2 Results

Mises stress, S , *Mises*, and the deflection in the y-direction, $U2$, were reviewed from the results. As predicted, the highest stress was found at the hub adjacent to the flange ring. The stress was a little over 50 *ksi*, which was just above the material's yield point.

The material at the hub with the highest bolt load (17 *ksi*) may yield locally, but it should be fine. It is often presumed that the calculated values exceeding the elastic limit must necessarily be dangerous. However, it may be misleading since it ignores the effect of stress redistribution, and material ductility [28]. In the analysis, we were looking at a section that had the highest load. In reality, the stress would be re-distributed to adjacent locations which have lower loads.

Figure D.1: Result of Finite Element Analysis of Mid-flange: Equivalent Stress, $S, Mises$, in psi (Left); Deflection in y -direction, $U2$, in $inches$ (Right)

Appendix E

Cable Load for Installation

Quasi-Static Analysis of Cable Tension of WTTower

Friday, December 04, 2009
12:01 PM

Assume

- No dynamic effects
(angular acceleration, etc...)
- No friction in the bearing
- No deflection of structure
(no stretching of cable, δ_{max} , ...)
- No wind
- Winch point is at the same level as the bearing

$$\vec{G_{in}-Anc} = \langle s + r_{gin} \cos(\theta + \theta''), -r_{gin} \sin(\theta + \theta'') \rangle$$

$$\|\vec{G_{in}-Anc}\| = \sqrt{(s + r_{gin} \cos(\theta + \theta''))^2 + (r_{gin} \sin(\theta + \theta''))^2}$$

$$\hat{\lambda}_{G_{in}-Anc} = \frac{\vec{G_{in}-Anc}}{\|\vec{G_{in}-Anc}\|} \rightarrow \vec{T} = T \hat{\lambda}_{G_{in}-Anc}$$

$$\sum \vec{M}_O = \vec{0}$$

$$\vec{r}_{cg} \times \vec{W} + \vec{r}_{gin} \times \vec{T} = \vec{0}$$

$$\begin{bmatrix} -r_{cg} \cos(\theta + \theta') & r_{cg} \sin(\theta + \theta') & 0 \\ 0 & 0 & -W \end{bmatrix} \times \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} + \begin{bmatrix} -r_{gin} \cos(\theta + \theta'') & r_{gin} \sin(\theta + \theta'') & 0 \\ 0 & 0 & T \lambda_x \end{bmatrix} \times \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} = \vec{0}$$

$$\begin{vmatrix} \hat{i} & \hat{j} & \hat{k} \\ -r_{cg} \cos(\theta + \theta') & r_{cg} \sin(\theta + \theta') & 0 \\ 0 & 0 & -W \end{vmatrix} + \begin{vmatrix} \hat{i} & \hat{j} & \hat{k} \\ -r_{gin} \cos(\theta + \theta'') & r_{gin} \sin(\theta + \theta'') & 0 \\ T \lambda_x & T \lambda_y & 0 \end{vmatrix} = \vec{0}$$

$$\hat{k} [r_{cg} \cos(\theta + \theta') W] + \hat{k} [-T r_{gin} \lambda_y \cos(\theta + \theta'') - T r_{gin} \lambda_x \sin(\theta + \theta'')] = \vec{0}$$

$$\hat{k}: W r_{cg} \cos(\theta + \theta') = T r_{gin} \lambda_y \cos(\theta + \theta'') + T r_{gin} \lambda_x \sin(\theta + \theta'')$$

$$T = W \frac{r_{cg} \cos(\theta + \theta')}{r_{gin} (\lambda_y \cos(\theta + \theta'') + \lambda_x \sin(\theta + \theta''))}$$

Tower Cable
Tension

* Weight Verification of the wind tower & the turbine assembly

$W = 11.66719 \frac{lb}{in^3}$ from Abaqus Dat file

$$w = \frac{W}{g \cdot \rho_{ft}} \left[\frac{lb}{in^3} \cdot \frac{ft^3}{ft^3} \right]$$

$$w = w \cdot g \cdot 12$$

$$= 11.66719 \cdot \frac{lb}{in^3} \cdot 32.2 \frac{ft}{s^2} \cdot \frac{12}{ft}$$

$$W = 4515.93 \text{ lbf} \rightarrow \text{need verification}$$

$$W = W_{turb} (450 \text{ lbf}) + W_{tower}$$

$$W_{tower} = 4065.93 \text{ lbf}$$

Solid works Assembly volume

$$V_{tower} = 15054.58 \text{ in}^3$$

* A36 Low carbon steel

$$\begin{cases} S_y = 36300 \text{ psi} \\ S_u = 58000 - 79800 \text{ psi} \\ E = 29000 \text{ ksi} \\ \nu = 0.26 \\ \rho = 0.284 \text{ lb/in}^3 \end{cases}$$

$$V_{tower} \cdot \rho = 15054.58 \text{ in}^3 \cdot (0.284 \frac{lb}{in^3}) = 4275.5 \text{ lbf}$$

$$W = W_{\text{turn}} (450 \text{ lbf}) + W_{\text{tower}}$$

$$\underline{W_{\text{tower}} = 4065.93 \text{ lbf}}$$

$$\begin{aligned} W_{\text{tower}} &= 15054.59 \text{ in}^3 (0.284 \text{ lb/in}^3) \\ &= 4275.5 \text{ lbf} \end{aligned}$$

$$\underline{W_{\text{tower}} = 4275.5 \text{ lbf}}$$

will take $W_{\text{tower}} = 4275.5 \approx \underline{4300 \text{ lbf}}$

Static Load to Hold Tower

Monday, January 18, 2010
7:08 PM

Static load required after tipping to hold the tower

Assume cable at 45° wrt mast

$$\sum \vec{M}_O = 0$$

$$-4300 \text{ lbf} (14") + (T \cos 45^\circ) (304") = 0$$

$$T = \frac{(4300)(14)}{(\cos 45^\circ)(304)}$$

$$T = 280 \text{ lbf}$$

Appendix F

Anchor Bolt Analysis

Anchor Bolt Design

Thursday, September 02, 2010
11:35 AM

Loads

$$\begin{aligned} U_{\max} &= \text{max uplift force due to 2000 lbf thrust.} \\ U_{\max} &= 30 \text{ kips } * \text{ (applied to one pile)} \\ S_{\max} &= \text{max shear force due to 2000 lbf thrust.} \\ S_{\max} &= 2 \text{ kips } * \text{ (applied to both pile)} \end{aligned} \quad \left. \begin{array}{l} \\ \\ \\ \end{array} \right\} \begin{array}{l} * \text{ since 2000 lbf Thrust} \\ \text{is conservative load already} \\ \text{No extra margin of safety is} \\ \text{included} \end{array}$$

Methods

1. calculate the l_s (length required for shear load)
2. " " l_t (" " tensile ")
3. Then check bolt strength.

Load per Anchor

Given above, $U_{\max} = 30 \text{ kips}$, applied to one pile (4 anchor bolt)
To be conservative, assume 2 anchors in the pile carries all the load.

$$T = \text{Tensile load per anchor} = 15 \text{ kips}$$

$$S = \text{shear load per anchor} = 1 \text{ kip}$$

Step 1 : Calculate l_s (shear)

F_p : Allowable bearing stress in concrete.

l_s : length of anchor bolt required to resist shear force on bolt.

A_c : Area of concrete compression

D : nominal diameter of bolt. $\leftarrow$ will use 1" bolt.

Allowable Bearing stress

$$F_p = 0.25 f'_c = 0.25 (4 \text{ ksi}) = \underline{1 \text{ ksi}}$$

Since we know D , will check the l_s , whether it can take shear load.

$$l_s = 12 \cdot D = 12 \cdot (1") = 12" \leftarrow \text{this is general rule I think (12x diameter)}$$

$$A_c = l_s D = 12 \text{ in}^2$$

$$\text{Allowable shear load} = \frac{1}{2} F_p A_c$$

$$= \frac{1}{2} (1 \text{ ksi}) (12 \text{ in}^2) = \underline{6 \text{ kips}} < 1 \text{ kips}$$

$\Rightarrow$ each bolt that has 1" diameter, 12 in long
can take 6000 lbf of shear

Step 2 Calculate l_t , Tension

u : allowable bond stress of concrete (ksi)

l_t : length of Anchor bolt req. for resisting tensile load

l : Total length of embedment.

$$u = 4.75 \times 10^{-3/2} (f'_c)^{1/2} / D$$

$$= 4.75 \times 10^{-3/2} (4 \text{ ksi})^{1/2} / 1" = 0.3 \text{ ksi}$$

$$u = 4.75 \times 10$$

Allowable tensile, set equal to our load T .

$$T = u (\pi D) l_t \rightarrow l_t = \frac{T}{u (\pi D)}$$

$$l_t = \frac{15 \text{ kips}}{(0.3 \text{ ksi}) (\pi) (1")} = 15.92" \approx 16"$$

$$l = l_s + l_e = 12" + 16" = 28" \rightarrow \text{Embedment Length}$$

Step 3 Check Bolt.

3441 NW Guam Street, Portland OR 97210
phone: (800)547-6758 | fax: (503) 227-4634
www.portlandbolt.com

Thread Pitch Chart

Coarse Thread Series - UNC				Fine Thread Series - UNF				8-Thread Series - 8UN			
Nominal Size and Threads Per In.	Basic Pitch Dia In.	Section at Minor Dia sq in.	Tensile Stress Area sq in.	Nominal Size and Threads Per In.	Basic Pitch Dia In.	Section at Minor Dia sq in.	Tensile Stress Area sq in.	Nominal Size and Threads Per In.	Basic Pitch Dia In.	Section at Minor Dia sq in.	Tensile Stress Area sq in.
3/8 - 16	0.3344	0.0678	0.0775	3/8 - 24	0.3479	0.0809	0.0878	--	--	--	--
7/16 - 14	0.3911	0.0933	0.1063	7/16 - 20	0.4050	0.1090	0.1187	--	--	--	--
1/2 - 13	0.4500	0.1257	0.1419	1/2 - 20	0.4675	0.1486	0.1599	--	--	--	--
9/16 - 12	0.5084	0.162	0.182	9/16 - 18	0.5264	0.189	0.203	--	--	--	--
5/8 - 11	0.5660	0.202	0.226	5/8 - 18	0.5889	0.240	0.256	--	--	--	--
3/4 - 10	0.6850	0.302	0.334	3/4 - 16	0.7094	0.351	0.373	--	--	--	--
7/8 - 9	0.8028	0.419	0.462	7/8 - 14	0.8286	0.180	0.509	--	--	--	--
1 - 8	0.9188	0.551	0.606	1 - 12	0.9459	0.625	0.663	1 - 8	0.9188	0.551	0.606
1-1/8 - 7	1.0322	0.693	0.763	1-1/8 - 12	1.0709	0.812	0.856	1-1/8 - 8	1.0438	0.728	0.790
1-1/4 - 7	1.1572	0.890	0.969	1-1/4 - 12	1.1959	1.024	1.073	1-1/4 - 8	1.1688	0.929	1.000
1-3/8 - 6	1.2667	1.054	1.155	1-3/8 - 12	1.3209	1.260	1.315	1-3/8 - 8	1.2938	1.155	1.233

Grade Marking	Specification	Material and Treatment	Nominal Size (In.)	Mechanical Properties			Hardness Rockwell	
				Proof Load Min (ksi)	Yield Strength Min (ksi)	Tensile Strength Min (ksi)	Min	Max
One End Yellow								
	SAE J429 Grade 2		1/4" - 1/2"	55	57	74	B80	B100
			5/8" - 1 1/2"	33	36	60	B70	B100
	A193/A320	A191 SS304						

Identification Grade Mark	Specification	Fastener Description	Material	Nominal Size Range (in.)	Mechanical Properties		
					Proof Load (psi)	Yield Strength Min (psi)	Tensile Strength Min (psi)
	SAE J429 Grade 1	Bolts, Screws, Studs	Low or Medium Carbon Steel	1/4 thru 1-1/2	33,000	36,000	60,000
	ASTM A307 Grades A&B		Low Carbon Steel	1/4 thru 4	--	--	
	SAE J429 Grade 2		Low or Medium Carbon Steel	1/4 thru 3/4 Over 3/4 to 1-1/2	55,000 33,000	57,000 36,000	74,000 60,000

Choosing Grade 2, 1" diameter, UNC

$$F_y = 36 \text{ ksi} \quad \text{and} \quad A_t = 0.606 \text{ in}^2$$

• F_v : allowable shear strength in bolt (ksi)

$$F_v = 0.3 F_y = 10.8 \text{ ksi}$$

• F_t : allowable tensile

$$F_t = 0.6 F_y = 21.6 \text{ ksi}$$

or $F_t = 28.0 - 1.6 f_v$
 $= 26 \text{ ksi}$

Choose smaller?

f_v : Actual shear stress in bolt.

$$f_v = \frac{S}{A_g}$$

$$A_g = \frac{\pi D^2}{4} = 0.7854 \text{ in}^2$$

(Gross area of Anchor bolt)

$$f_v = \frac{1 \text{ kip}}{0.7854} = \underline{1.27 \text{ ksi}} < F_v \text{ (Good!)}$$

f_t : Actual tensile stress

$$f_t = \frac{T}{A_t} = \frac{15 \text{ kips}}{0.606} = \underline{24.8 \text{ ksi}}$$

well $f_t \geq F_t^{\sim \text{allowable}} (= 21.6 \text{ ksi}) \rightarrow \text{No good...}$

because I assumed that 2 anchor carries all the tensile (out of 4)

if assume 4 anchor carries equal load

$$f_t = \frac{7.5 \text{ kips}}{0.606 \text{ in}^2} = 12.4 \text{ ksi} < F_t.$$

Appendix G

Foundation Plans and Drawings

NOTES:
1. ALL UNITS ARE FEET UNLESS OTHERWISE SPECIFIED
2. SEE ATTACHED HUBBEL-CHANCE CATALOG FOR MORE INFORMATION ON EXPANDING ANCHORS AND RODS

SIZE	B		
FOUNDATION LAYOUT			
UNIT: FEET	SCALE: 1/256	SHEET 1 OF 1	

LEVEL TOP PAD
AND
BROOM FINISH

PAD

MAIN PILE (2X)

DETAIL A
SCALE 1 : 16

CENTERLINE FOR ALIGNMENT OF
SUPPORT PILES AND EXPANDABLE ANCHORS
AS SHOWN IN "FOUNDATION LAYOUT"
PAY ATTENTION TO THE ALIGNMENT OF
L-ANCHOR BOLTS WITH RESPECT TO
THIS CENTERLINE

AS EXPLAINED IN "MAIN PILE, SUPPORT PILE"
FORMED EDGE OF SQUARE IS PARALLEL TO
CENTER LINE

SIZE	DWG. NO.	REV
A	MAIN PAD AND PILE	
UNIT: INCHES	SCALE: 1:64	WEIGHT:
1	2	3
4	5	1

NOTE:

1. USE CONCRETE COMPRESSIVE STRENGTH OF 4KSI
2. USE #4 REBAR
3. SEE "MAIN PAD AND PILE" FOR COMPLETE ASSEMBLY
4. PAD MUST BE LEVELED
5. BROOM FINISH

SIZE	DWG. NO.	REV
A	PAD	
SCALE: 1:48	UNIT: INCHES	SHEET 1 OF 1

DO NOT SCALE DRAWING

APPLICATION

3.0" SPACING
BETWEEN CONCRETE
WALL AND REBAR/SPIRAL
ALL AROUND

#3 SPIRAL
4" SPACING
9.5'(114") HEIGHT

#9 REBARS (8X)
EQUAL SPACING
9.5' (114") HEIGHT

SECTION A-A

PAY ATTENTION TO
RELATIVE LOCATION OF
ANCHOR BOLT AND REBARS

NOTES

1. USE CONCRETE COMPRESSIVE STRENGTH OF 4KSI
2. USE GRADE 60 STEEL FOR COMPRESSIVE REINFORCEMENT
3. USE GRADE 40 STEEL FOR SPIRAL REINFORCEMENT
4. USE 1"X36"X4"X6" ASTM F1554-55 GALVANIZED L-ANCHOR BOLTS FROM PORTLAND BOLT
SEE ATTACHED PORTLAND BOLT'S ANCHORBOLT SPECIFICATION FOR MORE INFORMATION
5. TWO MAIN PILES ARE IDENTICAL
6. TWO SUPPORT PILES: FRONT (UPWIND) PILE; REAR (DOWNWIND) PILE HAVE IDENTICAL FEATURES
AS MAIN PILE EXCEPT FOR DEPTH AND L-ANCHOR BOLT
 1. FRONT (UPWIND) PILE HAS DEPTH OF 4' (48") AND
REAR (DOWNWIND) PILE HAS DEPTH OF 8'(96")
 2. USE 3/4"X18"X4"X6" ASTM F1554-55 GALVANIZED L-ANCHOR BOLT (PORTLAND BOLT)
FOR BOTH FRONT (UPWIND) PILE AND REAR (DOWNWIND) PILE, ALSO
SAME BOLT PLACEMENT AS SHOWN HERE

NOTES ON ANCHOR BOLT ALIGNMENT

1. PROTRUDING ENDS OF ANCHOR BOLTS ON EACH PILE FORM A SQUARE.
EDGE OF EACH SQUARE MUST BE PARALLEL TO CENTER LINE SHOWN IN
"FOUNDATION LAYOUT"
2. REFER TO "FOUNDATION LAYOUT" AND "MAIN PAD AND PILE" FOR MORE INFORMATION
3. TOLERANCE OF ANCHOR BOLT PLACEMENT
 1. +/-0.125"HORIZONTALLY (SIDE-TO-SIDE)
 2. +/- 0.5" VERTICALLY (HEIGHT)

SIZE	DWG. NO.	REV.
A	MAIN PILE, SUPPORT PILE	
SCALE:1:48	UNIT: INCHES	SHEET 1 OF 1